

Sherlock User's Guide

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R1
January 2025

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2025 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. Getting Started with Sherlock 1
1.1. Welcome to Sherlock 1
1.2. Start-Up and Configuration 2
1.2.1. Sherlock Licensing 2
1.2.2. Standard Folder Locations 2
1.2.3. For Linux Users 3
1.2.4. Optional Software Compatibility 4
1.2.5. Initial Testing and Configuration 4
1.2.5.1. Sherlock Client Console 4
1.2.6. Memory Management 5
1.2.7. 3D Viewer Compatibility 5
1.3. Compatibility with Earlier Versions of Sherlock 5
1.4. Sherlock Settings Dialog 6
2. Introductory Tutorials 7
2.1. Tutorial: Project Overview 8
2.1.1. Main Window 8
2.1.2. Importing a Project 8
2.1.3. Project Life Cycle 10
2.1.4. Circuit Card Information 12
2.1.4.1. Layer Viewer 13
2.1.4.2. Input Data 15
2.1.5. Analysis 17
2.1.6. Results 19
2.1.6.1. Score Card 19
2.1.6.2. Life Prediction Chart 20
2.1.6.3. Project Report 20
2.1.6.4. Saving Results 22
2.1.7. Closing a Project 23
2.2. Tutorial: ODB++ Project Creation 23
2.2.1. Importing an ODB++ Archive 24
2.2.2. Project Name and Properties 30
2.3. Tutorial: Reviewing Part Properties 30
2.3.1. Managing Part Properties 32
2.3.2. Filtering and Selecting Parts 33
2.3.3. Changing Part Properties 34
2.3.3.1. The Part Property Viewer 34
2.3.3.1.1. The Part Editor 36
2.3.3.2. The Package Chooser 37
2.3.3.3. Back to the Part Editor 37
2.3.4. Part Source Priority 39
2.3.5. Changing Properties for Multiple Parts 39
2.3.6. Confirming Parts 42
2.3.7. Updating Component Layers 42
2.4. Tutorial: Layer Viewer 43
2.4.1. Layer Viewer Overview 43
2.4.2. Launching the Layer Viewer 43
2.4.3. Reviewing Layers 44
2.4.4. Layer Panning and Zooming 46
2.4.5. Component Editor 47

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.5.1. Editing Part Orientation and Location 48
2.4.5.2. Overlapping Parts Warning 48
2.4.5.3. One-Click Editing 49
2.4.5.4. Editing Part Properties 49
2.4.5.5. Graphically Filtering and Selecting Parts 50
2.4.6. Board Outline Editor 51
2.4.7. Other Layer Viewer Features 52
2.4.7.1. Component and Lead Identification 53
2.4.7.2. Camera Tool 54
2.4.7.3. Distance Tool 55
2.4.7.4. Generate Trace Models 56
2.5. Tutorial: Ad Hoc Project Creation 56
2.5.1. Project Creation 57
2.5.2. Circuit Card Creation 58
2.5.3. Adding CAD Files 59
2.5.4. Importing a Bill of Materials 61
2.5.5. BGA Part Properties 62
2.5.6. Importing Pick & Place Data 64
2.5.7. Importing Copper Layers 65
2.5.7.1. Unknown Materials in the Stackup 67
2.5.8. Insulated Metal Substrate Stackup 68
2.5.9. Additional Stackup Tools 69
2.5.10. Reviewing the Board Outline 72
2.5.11. Reviewing Component Layers 73
2.5.11.1. Missing Components 74
2.5.11.2. Components in the Wrong Place 74
2.5.12. Reviewing Drill Holes 75
3. User Reference 77
3.1. Sherlock Settings 77
3.1.1. Advanced Settings 78
3.1.2. Beta Features Settings 80
3.1.3. Color Settings 81
3.1.4. Data Store Settings 82
3.1.5. Debug Settings 83
3.1.6. FEA Analysis Settings 84
3.1.7. Launcher Settings 84
3.1.8. Meshing Settings 85
3.1.9. Part Library Settings 87
3.1.10. Report Settings 88
3.1.11. Score Settings 88
3.2. User Interface Overview 89
3.2.1. Main Window 89
3.2.2. Project Tree 90
3.2.3. Main Menu 90
3.3. Single and Multi-Project Mode 92
3.3.1. Multi-Project Mode 93
3.3.1.1. Closing Projects 93
3.3.2. Single-Project Mode 94
3.3.2.1. Launching Sherlock 94
3.3.2.1.1. Creating a Custom Project Shortcut 95
3.3.2.2. Project Menu 95

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3.2.2.1. Opening, Importing, and Library Loading 96
3.3.2.2.2. Recent Projects Menu 96
3.3.3. Changing Project Directory 96
3.4. Circuit Card Assembly Files 97
3.4.1. Circuit Card Information 98
3.4.2. CCA File Properties Viewer 99
3.4.3. Creating a Circuit Card 100
3.4.4. Adding Files to a Circuit Card Assembly 100
3.4.5. Attaching Informational (Info) Files 103
3.4.6. CCA File Format Specifications 103
3.4.6.1. CAD File (IPC2581, ODB XML) 105
3.4.6.2. Copper Layer Files 105
3.4.6.2.1. Copper Layer (CADIF, Gerber, ODB XML, ODB++) 105
3.4.6.2.2. Copper Layer (EDB, IPC2581) 105
3.4.6.2.3. Copper Layer (Image) 105
3.4.6.2.4. Copper Layer (Trace) 106
3.4.6.3. Cutout Files 106
3.4.6.3.1. Cutout File (ODB XML) 106
3.4.6.3.2. Cutout File (ODB++) 106
3.4.6.4. Drill Files 107
3.4.6.4.1. Drill File (CADIF, IPC2581, ODB XML, ODB++) 107
3.4.6.4.2. Drill File (Excellon) 107
3.4.6.4.3. Drill File (Gerber) 108
3.4.6.5. ICT Fixture Files (CSV, Excel) 108
3.4.6.6. Icepak Thermal Map (TMAP) 109
3.4.6.7. Info (Excel, PDF, Text, Word) 109
3.4.6.8. Laminate Layer Files 109
3.4.6.8.1. Laminate Layer (CADIF, Gerber, ODB XML, ODB++) 109
3.4.6.8.2. Laminate Layer (EDB, IPC2581) 109
3.4.6.8.3. Laminate Layer (Image) 110
3.4.6.8.4. Laminate Layer (ODB XML) 110
3.4.6.9. Mount Point Files (CSV, Excel) 110
3.4.6.10. Net List Files 111
3.4.6.10.1. Net List (CADIF) 111
3.4.6.10.2. Net List Import (CSV, Excel) 111
3.4.6.10.3. Net List (IPC, IPC2581) 111
3.4.6.11. Other Layer (Gerber, IPC2581, Image, ODB++) 111
3.4.6.12. Outline Layer Files 111
3.4.6.12.1. Outline Layer (CSV, Excel) 112
3.4.6.12.2. Outline Layer (Gerber, IPC2581) 112
3.4.6.13. Parts List Files 112
3.4.6.13.1. Parts List (CSV, Eagle, Excel) 112
3.4.6.13.2. Parts List (IPC2581, ODB++) 113
3.4.6.14. Pick and Place Files 113
3.4.6.14.1. Pick & Place (CADIF) 113
3.4.6.14.2. Pick & Place (CSV) 113
3.4.6.14.3. Pick & Place (DELIMITED) 114
3.4.6.14.4. Pick & Place (Eagle) 114
3.4.6.14.5. Pick & Place (Excel, FIXED) 114
3.4.6.14.6. Pick & Place (IPC2581) 114
3.4.6.14.7. Pick & Place (ODB XML) 114

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4.6.14.8. Pick & Place (ODB++) 114
3.4.6.15. Silk Screen Files 115
3.4.6.15.1. Silk Screen (CADIF, Gerber, ODB XML, ODB++) 115
3.4.6.15.2. Silk Screen (IPC2581) 115
3.4.6.16. Solder Mask Files 115
3.4.6.16.1. Solder Mask (CADIF, Gerber) 115
3.4.6.16.2. Solder Mask (IPC2581) 116
3.4.6.16.3. Solder Mask (ODB XML, ODB++) 116
3.4.6.16.4. Test Point (CSV, Excel) 116
3.4.6.17. Thermal Map Files 116
3.4.6.17.1. Thermal Map (CSV, Excel) 116
3.4.6.17.2. Thermal Map (Image) 117
3.4.6.18. Unknown 117
3.4.6.19. Wire Bond (CSV, Excel) 117
3.5. Layer Viewer 118
3.6. Importing Projects and ECAD Archives 118
3.6.1. How to Import a Project into Sherlock 119
3.6.1.1. Importing Sherlock Projects (.zip) 119
3.6.1.2. Importing Other File Types 120
3.6.2. Import-Compatible File Types 123
3.6.3. IPC-2581 Import Tutorial 124
3.6.3.1. Importing an IPC-2581 File 125
3.6.3.2. IPC-2581 Supported Elements 129
3.7. Exporting Projects 131
3.7.1. Exporting Sherlock Projects (.zip, .dfr-result) 131
3.7.2. Exporting CCAs to Ansys Icepak / Electronics Desktop (.aedb) 132
3.7.3. Links to Other Types of Exports 132
3.8. Stackups and Copper Layers 133
3.8.1. Importing Copper Layers 133
3.8.2. Stackup Tools 133
3.9. Laminate Management 133
3.9.1. Laminate Manager 134
3.9.2. Laminate Manager Icon Ribbon 135
3.9.3. Filtering Laminates 136
3.9.4. Selecting Multiple Laminates for Editing, Exporting 136
3.9.5. Adding and Copying Laminates 137
3.9.5.1. Adding a New Laminate 137
3.9.5.2. Copying a Laminate 138
3.9.6. Deleting Laminates 138
3.9.7. Exporting Laminates 138
3.9.8. Importing Laminates 140
3.9.9. Laminate Editor 141
3.9.9.1. Laminate Editor Interface 142
3.9.9.2. Editing Laminates 144
3.9.10. Editing Multiple Laminates Simultaneously 145
3.10. Solder Management 145
3.10.1. Solder Manager Listing 146
3.10.2. Solder Editor 146
3.10.2.1. Selecting a Solder to Edit 146
3.10.2.2. Using the Solder Editor 147
3.10.3. Temperature-Dependent Material Properties 148

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.10.3.1. CTE and Elasticity Modulus Calculators 149
3.10.3.2. Adding and Copying Solders 150
3.10.3.3. Deleting Solders 150
3.10.4. Export Solders 151
3.10.5. Import Solders 152
3.11. Material Management 153
3.11.1. Material Manager Listing 153
3.11.2. Material Editor 155
3.11.2.1. Selecting a Material to Edit 155
3.11.2.2. Using the Material Editor 155
3.11.2.2.1. Material Editor Icon Ribbon 157
3.11.2.2.2. Material Editor Property Tabs and Fields 157
3.11.3. Temperature-Dependent Material Properties 158
3.11.3.1. CTE and Elasticity Modulus Calculators 159
3.11.3.2. Adding and Copying Materials 163
3.11.3.3. Deleting Materials 163
3.11.4. Export Materials 164
3.11.5. Import Materials 165
3.12. User Data Files and Documents 166
3.12.1. User Defined Data Files 167
3.12.1.1. Shared Data Files 168
3.12.1.2. Material Properties 169
3.12.1.2.1. Temperature-Dependent Material Properties 170
3.12.1.3. Laminate Properties 171
3.12.1.3.1. Temperature-Dependent Laminate Properties 172
3.12.1.4. Laminate Glass Properties 172
3.12.1.5. Solder Properties 173
3.12.1.6. Package Properties 176
3.12.1.7. Package Images 177
3.12.1.8. Part Type Properties 178
3.12.1.9. PTH Quality Factors 179
3.12.1.10. Die Process Node Acceleration Factors 179
3.12.1.11. Manufacturer Names 180
3.12.2. User Document Files 181
3.12.3. User Properties File (user.properties) 181
4. Managing Parts in Sherlock 183
4.1. Parts List Overview 183
4.1.1. How Sherlock Generates the Parts List 184
4.1.2. The Importance of Reviewing Part Properties 185
4.1.2.1. Parts Requiring Special Consideration 186
4.1.3. The Parts List Interface 186
4.1.3.1. Main Features of the Parts List 186
4.1.3.2. How to Select Parts 187
4.1.3.3. The Icon Ribbon 187
4.1.3.4. Data Source Color Codes 190
4.1.4. Customizing the Parts List 191
4.1.5. Filtering and Selecting Parts 196
4.1.6. Updating Component Layers 196
4.2. Parts List Management Tools 197
4.2.1. The Part Properties Viewer 197
4.2.1.1. Opening the Part Properties Viewer 198

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.1.2. Part Properties Viewer Interface 198
4.2.1.3. Part Property Viewing/Editing Options 199
4.2.2. The Part Editor 200
4.2.2.1. Opening the Part Editor 200
4.2.2.2. Part Editor Interface 201
4.2.2.3. The Part Editor and Package Properties 202
4.2.2.4. Scope of Changes 203
4.2.3. The Multi-Part Editor 203
4.2.3.1. Opening the Multi-Part Editor 204
4.2.3.2. Multi-Part Editor Interface 204
4.2.3.3. Scope of Changes 205
4.2.4. The Package Chooser 206
4.2.5. Part Source Priority 207
4.2.6. Confirming Parts 210
4.2.7. Filtering the Parts List (Advanced) 211
4.2.7.1. Column Filters 211
4.2.7.1.1. Column Filter - String Matching 212
4.2.7.1.2. Column Filtering - Comparison Operators 213
4.2.7.1.3. Column Filtering - Multi Column 214
4.2.7.1.4. Column Filtering - Boolean Expressions 214
4.2.7.1.5. Column Filtering - Meta-Functions 215
4.2.7.2. Advanced Filter 216
4.2.7.2.1. Advanced Filter - String Matching 216
4.2.7.2.2. Advanced Filtering - Comparison Operators 217
4.2.7.2.3. Advanced Filtering - Boolean Expressions 218
4.2.7.2.4. Advanced Filtering - Meta-Functions 219
4.2.7.2.5. Advanced Filtering - Numeric Expressions 220
4.2.7.2.6. Advanced Filtering - Filter Management 221
4.2.7.2.7. Advanced Filtering - Property Chooser 222
4.2.8. Exporting the Parts List 223
4.2.9. Importing the Parts List 224
4.2.9.1. Import Options 224
4.2.9.2. Column Mappings 226
4.2.9.3. Addressing Import Errors 226
4.3. Building Custom Parts 229
4.4. Part Library 230
4.4.1. Parts Library Menu 231
4.4.2. Creating a Part Library 231
4.4.3. Part Library Management 233
4.4.3.1. Viewing Part Properties 234
4.4.3.2. Editing Properties for a Single Part 236
4.4.3.3. The Part Editor and Package Properties 236
4.4.3.4. Editing Properties for Multiple Parts 237
4.4.3.5. Part Visual Previewer 237
4.4.3.6. Package and Lead Properties 238
4.4.3.7. Adding, Copying or Deleting Parts 238
4.4.4. Customized Search Results 239
4.4.5. Filtered Search Results 243
4.4.5.1. Column Filters 243
4.4.5.1.1. Column Filter - String Matching 244
4.4.5.1.2. Column Filtering - Comparison Operators 245

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.1.3. Column Filtering - Multi Column 246
4.4.5.1.4. Column Filtering - Boolean Expressions 246
4.4.5.1.5. Column Filtering - Meta Functions 247
4.4.5.2. Advanced Filter 248
4.4.5.2.1. Advanced Filter - String Matching 249
4.4.5.2.2. Advanced Filtering - Comparison Operators 250
4.4.5.2.3. Advanced Filtering - Boolean Expressions 252
4.4.5.2.4. Advanced Filters - Meta Functions 253
4.4.5.3. Advanced Filtering - Numeric Expressions 254
4.4.5.3.1. Advanced Filtering - Filter Management 255
4.4.5.3.2. Advanced Filtering - Property Chooser 256
4.4.6. Update Part Library from Parts List 257
4.4.6.1. Part Library Interface with Parts List 258
4.4.7. Update Parts List from Part Library 259
4.4.7.1. Export Part Data 260
4.4.8. Import Part Data 261
4.4.9. Recover Part Library 262
4.5. Part Validation 263
4.5.1. Input Data 263
4.5.2. Analysis Options 264
4.5.3. Analysis Results 266
4.5.3.1. Summary Results 266
4.5.3.2. Table of Individual Part List Results 267
4.5.3.3. List of Issues Found 268
4.5.4. Component Analysis Part Validation 268
4.6. Approved Vendor List (AVL) 268
4.6.1. AVL Menu Options 269
4.6.2. Maintaining the AVL 269
4.6.2.1. Importing AVL Data 269
4.6.2.2. Importing Multiple Files 272
4.6.2.3. Editing AVL Part Data 273
4.6.2.4. Deleting AVL Part Data 273
4.6.2.5. Exporting AVL Part Data 273
4.6.3. Using The AVL 274
4.6.3.1. Searching the Part Library 274
4.6.3.2. Updating the Parts List 275
4.6.3.3. Matching Criteria 276
4.6.3.4. Duplication Handling 276
4.6.3.5. Part Property Updates 276
4.6.3.6. Validating the Parts List 277
4.7. Part Wizard 278
4.7.1. Part Number Recognition 279
4.7.2. Viewing Part Wizard Patterns 281
4.7.3. Editing Part Wizard Patterns 282
4.7.4. Editing Part Wizard Default Properties 284
4.7.4.1. Supported Operators 285
4.7.5. Editing Part Wizard Pattern Fields 286
4.7.6. Editing Part Wizard Field Codes 288
4.7.7. Managing Part Wizard Patterns 290
4.7.7.1. Adding / Copying a Part Wizard Pattern 290
4.7.7.2. Deleting Part Wizard Patterns 291

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.7.3. Exporting Part Wizard Patterns 291
4.7.7.4. Importing Part Wizard Patterns 291
4.7.8. Bulk Pattern Testing 292
4.7.9. Bulk Test Results 293
4.7.10. Bulk Pattern Matching 294
4.8. Sherlock Part Properties 294
4.8.1. A - Part Properties 295
4.8.2. B - Properties 295
4.8.3. C - D Properties 299
4.8.4. E - J Properties 300
4.8.5. L - Properties 301
4.8.6. M - Properties 306
4.8.7. N - O Properties 308
4.8.8. P - Properties 308
4.8.9. Q - R Properties 311
4.8.10. S - Properties 312
4.8.11. T - Y Properties 314
4.9. Package Management 315
4.9.1. Package Manager Listing 316
4.9.2. Selecting a Package to Edit 319
4.9.3. Using the Package Editor 320
4.9.4. Adding and Copying Packages 322
4.9.5. Deleting Packages 323
4.9.6. Package Export and Import 323
4.9.7. Column Mappings 325
4.9.8. Package Previewer 326
4.10. Tables: Required Part Properties per Analysis Type 326
4.10.1. Required Part Properties for: Component Failure Analysis, DFMEA, and PTH Fatigue 327
4.10.2. Required Part Properties for: Semi-Conductor Wearout, Solder Fatigue, Thermal Derating, and Part Validation 330
4.10.3. Required Part Properties for Each FEA Analysis 333
5. Life Cycle Management 339
5.1. Tutorial Life Cycle 340
5.2. Phase Definitions 341
5.3. Project Life Cycles 343
5.3.1. Event Definitions 344
5.3.2. Thermal Event 344
5.3.3. Shock Event 347
5.3.4. Random Event 350
5.3.4.1. Random Vibe Profile 351
5.3.4.2. Shock Response Spectrum 353
5.3.5. Harmonic Event 354
5.3.6. Load Direction and PCB Orientation 355
5.3.7. Life Cycle Editing 357
5.3.7.1. Adding Life Cycle Phases 357
5.3.7.2. Removing a Life Cycle Phase 357
5.3.7.3. Duplicating a Life Cycle Phase 357
5.3.7.4. Copy and Paste a Life Phase 357
5.3.7.5. Adding a Life Cycle Event 358
5.3.7.6. Duplicating a Life Cycle Event 358
5.3.7.7. Copy and Paste a Life Cycle Event 358

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.7.8. Deleting a Life Cycle Event 358
5.3.8. Life Cycle Import / Export 358
5.3.8.1. Exporting a Life Cycle 359
5.3.8.2. Importing a Life Cycle 359
5.3.9. Life Cycle Manager 359
5.3.9.1. Life Cycle Definition Files 359
5.3.9.2. Launching the Life Cycle Manager 360
5.3.9.3. Life Cycle Manager Window 360
5.3.9.4. Life Cycle Manager Menu Options 361
5.3.9.4.1. Create Life Cycle 361
5.3.9.4.2. Open Life Cycle 362
5.3.9.4.3. Close Life Cycle 362
5.3.9.4.4. Generate Report 362
6. Failure Analysis 363
6.1. Design Failure Mode and Effects Analysis (DFMEA) 363
6.1.1. Managing DFMEA Data 364
6.1.1.1. Updating DFMEA Data from Parts List 364
6.1.1.2. Manually Organizing DFMEA Data 367
6.1.1.3. Manually Adding Part Data 368
6.1.1.4. Adding Default Part Failures 369
6.1.1.5. Adding Default Pin Open Failures 370
6.1.1.6. Adding Default Pin Short Failures 371
6.1.1.7. Manually Adding Failure Modes 372
6.1.1.8. Editing / Removing DFMEA Data 372
6.1.2. Customizing DFMEA Data 373
6.1.2.1. Custom DFMEA Properties 373
6.1.2.2. Custom Default Failure Modes 375
6.1.2.3. Default Part Failures 376
6.1.2.4. Default Pin Open Failures 376
6.1.2.5. Default Pin Short Failures 377
6.1.3. Exporting DFMEA Data 378
6.1.3.1. Template Files 378
6.1.3.2. Exporting DFMEA Data 378
6.1.3.3. Template Variables 380
6.1.3.4. Templates Lists 383
6.1.3.5. Nested Templates Lists 385
6.1.3.6. Page Headers 386
6.1.4. Importing DFMEA Data 386
6.1.4.1. Re-Importing a DFMEA Spreadsheet 386
6.1.4.2. Importing Custom DFMEA Properties 389
6.1.4.3. Importing Pre-Existing Spreadsheets 390
6.1.4.4. Import Errors 391
6.1.5. DFMEA Results 392
6.1.5.1. Generating DFMEA Results 392
6.1.5.2. DFMEA Results 392
6.1.5.3. DFMEA Layers 395
6.1.6. Net List 395
6.2. Component Failure Analysis 396
6.2.1. Input Data 396
6.2.1.1. Required Vendor Inputs 397
6.2.2. Analysis Properties and Analysis Results 399

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.2.1. Analysis Properties 399
6.2.2.2. Analysis Results 399
6.3. Plated Through Hole (PTH) Fatigue Analysis 399
6.3.1. Input Data 400
6.3.2. Analysis Properties 401
6.3.3. Analysis Results 402
6.4. Semiconductor Wearout 408
6.4.1. Input Data 409
6.4.1.1. Option 1: Prediction and User Field Data 409
6.4.1.1.1. Semiconductor Wearout Tab 409
6.4.1.1.2. Die Tab 410
6.4.1.2. Option 2: Temperature Specific and Acceleration Factors 410
6.4.1.2.1. Semiconductor Wearout Tab 410
6.4.1.2.2. Die Tab 410
6.4.1.2.3. Electrical Tab 410
6.4.1.2.4. Test Tab 410
6.4.1.3. Option 3: Test Results and Acceleration Factors 410
6.4.1.3.1. Semiconductor Wearout Tab 411
6.4.1.3.2. Die Tab 411
6.4.1.3.3. Electrical Tab 411
6.4.1.3.4. Test Tab 411
6.4.1.4. Option 4: Temperature Specific and Calculated Factors 411
6.4.1.4.1. Semiconductor Wearout Tab 411
6.4.1.4.2. Die Tab 412
6.4.1.4.3. Electrical Tab 412
6.4.1.4.4. Test Tab 412
6.4.1.5. Option 5: Test Results and Calculated Factors 413
6.4.1.5.1. Semiconductor Wearout Tab 413
6.4.1.5.2. Die Tab 413
6.4.1.5.3. Electrical Tab 413
6.4.1.5.4. Test Tab 413
6.4.2. Acceleration Factors 414
6.4.2.1. Acceleration Factors Manager Listing 414
6.4.2.2. Viewing and Editing Acceleration Factors 414
6.4.2.3. Modifying Acceleration Factors 415
6.4.3. Documents 416
6.4.4. Analysis 417
6.4.4.1. Analysis Results 418
6.4.4.1.1. Analysis Computations 418
6.5. Solder Fatigue Analysis 419
6.6. Solder Fatigue Analysis Module 420
6.6.1. Input Data 420
6.6.2. Solder Fatigue Properties Dialog 421
6.6.3. Pad Properties 422
6.6.3.1. How Sherlock Models Component Pads 422
6.6.3.2. Updating Pad Properties 423
6.6.4. Editing Solder Properties 424
6.6.5. Supported Solder Models 425
6.7. Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis 426
6.7.1. Workflow Summary 427
6.7.2. TMAP File Specifications 431

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.8. Solder Fatigue Tool 433
6.9. Thermal Derating 435
6.9.1. Input Data 435
6.9.2. Analysis Options 436
6.9.3. Analysis Results 437
6.9.3.1. Summary Results 437
6.9.3.2. Score Distribution Chart 438
6.9.3.3. Event Score Distribution Chart 439
6.9.3.4. Table of Individual Results 440
6.9.3.4.1. Event Scores 440
6.9.3.4.2. Part Scores 441
6.9.3.4.3. Part / Event Scores 441
6.9.3.5. Layer Viewer 442
7. Finite Element Analysis (FEA) Overview 443
7.1. FEA Integration 444
7.2. FEA Engine 444
7.2.1. Selecting the FEA Engine 445
7.2.2. FEA Analysis Settings 446
7.2.2.1. Engine Properties 447
7.2.2.2. Analysis Properties 448
7.2.2.3. Analysis Color Settings 450
7.2.2.4. Strain Color Settings 450
7.2.2.5. Stress Color Settings 451
7.2.2.6. Mesh Settings 451
7.3. Data Sources 451
7.3.1. Generated Data Source 451
7.3.2. Imported Data Source 452
7.3.3. Strain Map Import 452
7.4. FEA Models 453
7.4.1. Bonded Model 453
7.4.2. FEA Modeling Properties 454
7.4.3. Generate 3D Model 455
7.4.4. Export FEA Model 458
7.4.5. Run FEA Analysis 462
7.5. FEA Processing 463
7.6. FEA Results 464
7.6.1. Summary Panel 465
7.6.2. Tabular Results 466
7.6.3. Graphical Result Layers 468
7.6.4. 3D Results File 471
7.6.5. Log Files 471
8. FEA Modeling Guide 473
8.1. FEA- PCB Modeling 473
8.1.1. PCB Models 474
8.1.1.1. Uniform Model 475
8.1.1.2. Layered Model 475
8.1.1.3. Uniform Elements Model 475
8.1.1.4. Layered Elements Model 476
8.1.2. PCB Modeling Properties 476
8.1.3. PCB Material Management 477
8.1.4. PCB Model Tradeoffs 478

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.2. FEA- Mount Points and Fixtures 479
8.2.1. Mount Point & Fixture Types 480
8.2.2. Mount Point Management 481
8.2.2.1. Viewing Mount Points 481
8.2.2.2. Adding Mount Points 482
8.2.2.3. Modifying Mount Points 483
8.2.2.4. Mount Point Properties 484
8.2.2.5. Mount Point Multi-Edit 486
8.2.2.6. Saving Mount Point Changes 486
8.2.2.7. Mount Point Results 487
8.2.3. Fixture Management 487
8.2.3.1. Viewing Fixtures 487
8.2.3.2. Adding Fixtures 488
8.2.3.3. Modifying Fixtures 488
8.2.3.4. Using Mount Points as Fixtures 488
8.2.3.5. Saving Fixture Changes 489
8.3. FEA- Cutouts and Drill Holes 489
8.3.1. Cutouts 489
8.3.1.1. Cutout Viewing 489
8.3.1.2. Cutout Editing 490
8.3.1.3. Cutout Properties 491
8.3.2. Drill Holes 492
8.3.2.1. Drill Hole Files 493
8.3.2.2. Drill Hole Modeling 494
8.4. FEA- Trace Modeling 495
8.4.1. Tutorial Project 496
8.4.2. Accessing the Trace Modeling Context Menu 496
8.4.3. Trace Model Generation 497
8.4.4. Advanced Trace Modeling Settings 499
8.4.5. Trace Model Viewing 500
8.4.6. Trace Model Import 503
8.4.7. Trace Model and Sketch Export 503
8.4.7.1. Trace Model Export 503
8.4.7.2. Trace Sketch Export 506
8.4.8. Trace Reinforcement Export 506
8.4.8.1. Trace Reinforcement Model Export Automation (*.wbjn) 509
8.4.9. Importing the STEP File into Ansys Workbench or Ansys Discovery 510
8.4.10. Trace Model Resolution 512
8.4.11. Trace Model Analysis 513
8.5. FEA- Image File Layers 514
8.5.1. Preparing Image Files 515
8.5.2. Adding Image File Layers 516
8.5.3. Image File Alignment 517
8.5.3.1. Aligning Images in the Layer Viewer 517
8.5.3.2. Aligning Images Using the Mapping Editor 519
8.5.4. Converting a Copper Layer File to an Image File 520
8.5.5. Image Layer Stackup Data 521
8.5.6. Image Layer Trace Model 522
8.6. FEA- PCB Modeling Regions 523
8.6.1. Modeling Region Example 524
8.6.1.1. Model 1 - Large Elements 524

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.1.2. Model 2 - Small Elements 525
8.6.1.3. Model 3 - Large and Small Elements 526
8.6.2. Modeling Region Editor 526
8.6.2.1. Modeling Region Properties 527
8.6.2.2. PCB Modeling Properties 529
8.6.2.3. Trace Modeling Properties 530
8.6.2.4. Graphical Editing 530
8.6.2.5. Modeling Region Restrictions 531
8.6.3. Using Modeling Regions 532
8.6.4. Common Examples 532
8.6.4.1. Simple Regions 532
8.6.4.2. Nested Regions 534
8.6.4.3. Regional Trace Modeling 535
8.6.4.4. Excluded Regions 537
8.6.5. Hybrid Modeling 538
8.6.5.1. Bonded Uniform 540
8.6.5.2. Bonded Layered 541
8.6.5.3. Bonded Trace Layered 541
8.6.5.4. Bonded Layered Elements 542
8.6.5.5. Bonded Trace Elements 542
8.6.5.6. Sweep Uniform 543
8.6.5.7. Sweep Layered 544
8.6.5.8. Sweep Trace Layered 544
8.6.5.9. Sweep Layered Elements 545
8.6.5.10. Sweep Trace Elements 546
8.6.6. Hybrid Models with Sub-Assemblies 547
8.6.7. Creating a CCA from a Modeling Region 550
8.7. FEA- Part Modeling 552
8.7.1. Layer Viewer Component Selection 553
8.7.2. Component Size, Location, and Orientation 553
8.7.3. Editing Part Properties 554
8.7.4. 2D Component Outline 554
8.7.5. 3D Block Model 558
8.7.6. Corner Orientation 559
8.7.7. Corner Face Restrictions 560
8.7.7.1. Merged PCB Models and Potting Regions 560
8.7.7.2. Round Part Corners and PCB Contact 560
8.7.8. 3D Part Model Viewer 561
8.7.9. FEA Model Properties 562
8.8. FEA- Lead Modeling 563
8.8.1. Lead Properties 564
8.8.2. Lead Modeling Analysis Properties 565
8.8.3. Advanced Lead Meshing 566
8.8.3.1. Overview of Advanced Lead Meshing 567
8.8.3.2. Through Thickness and Aspect Ratio 568
8.8.3.3. Lead Bend Radius and Lead Bend Package Radius 569
8.8.3.4. Frequently Asked Questions About Advanced Lead Meshing (ALM) 571
8.8.4. BGA Solder Ball Properties 572
8.8.5. Surface Mount Lead Modeling 573
8.8.5.1. C-Lead / J-Lead Model 573
8.8.5.2. Gullwing Model 574

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.5.3. Stub Model 575
8.8.5.4. L Lead Model 577
8.8.5.5. Lead Placement 579
8.8.6. Through-Hole Lead Modeling 581
8.8.6.1. SIP Layout 582
8.8.6.2. SIP-SINK Layout 583
8.8.6.3. DIP Layout 584
8.8.6.4. AXIAL Layout 585
8.8.6.5. CIRCULAR Layout 588
8.8.6.6. GRID Layout 588
8.8.7. Solder Ball Modeling 590
8.8.7.1. Solder Ball Shape Properties 591
8.8.7.2. Solder Ball Layout Properties 592
8.8.7.2.1. Full Ball Pattern 592
8.8.7.2.2. Perimeter Ball Pattern 593
8.8.7.2.3. Channel Ball Pattern 594
8.8.7.2.4. Island Ball Pattern 594
8.8.7.2.5. Corner Solder Balls 594
8.8.7.2.6. Automatic Consistency Checks 596
8.8.8. 2D Lead Viewing 596
8.9. FEA- Heat Sink Modeling 598
8.9.1. Heat Sink Modeling 598
8.9.1.1. Heat Sink Layouts 598
8.9.2. Heat Sink Modeling Properties 602
8.9.3. Creating Heat Sinks 603
8.9.4. Updating Heat Sinks 604
8.9.5. Heat Sink Attachment Points 604
8.9.6. Viewing Heat Sink 3D Model 605
8.9.7. Heat Sink Analysis 605
8.9.7.1. Heat Sink Results 606
8.10. FEA- Wire Bonds 607
8.10.1. Adding Wire Bonds 608
8.10.2. 3D Model Viewer 610
8.10.3. Editing Wire Bonds 611
8.10.4. Wire Bond Export 612
8.10.5. Wire Bond Import 613
8.10.6. 3D FEA Modeling 614
8.10.7. Wire Bond Results 616
8.11. FEA- Potting & Staking 616
8.11.1. Tutorial Project 617
8.11.2. 2D Modeling 617
8.11.3. PCB Shape 619
8.11.4. Rectangular Shape 620
8.11.5. Circular Shape 621
8.11.6. Slot Shape 622
8.11.7. Polygonal Shape 623
8.11.8. 3D Modeling 624
8.11.9. Potting and Part Elements 625
8.11.10. Potting and Lead Elements 625
8.11.11. Potting & Staking Examples 626
8.12. FEA- Mechanical Parts 629

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.12.1. Mechanical Part Viewing 630
8.12.2. Mechanical Part Editing 631
8.12.3. Mechanical Part Properties 632
8.12.4. Mechanical Part Multi-Edit 634
8.12.5. Mechanical Part Attachment Points 635
8.12.6. FEA Model 636
8.13. FEA- Sub-Assembly Analysis 636
8.13.1. Tutorial Review 638
8.13.2. Baseline Analysis 639
8.13.3. Adding an Assembly 641
8.13.4. Defining Assembly Points 643
8.13.5. Multiple Assemblies 645
8.13.6. Assembly Results 647
8.14. Advanced BGA Modeling 648
8.14.1. Creating the Detailed Component 650
8.14.2. Properties of Detailed BGA Component 651
8.14.2.1. CCA Properties 651
8.14.2.2. Overmold-Potting Region Properties 652
8.14.2.3. Die Properties 653
8.14.2.4. Mount Point - Ball Properties 654
8.14.3. Exporting the Detailed Component to Ansys Mechanical 654
9. FEA Analysis Types 655
9.1. FEA- Natural Frequency 655
9.1.1. Natural Frequency Input Data 655
9.1.2. Natural Frequency Analysis Properties 656
9.1.2.1. Generated Data Source 656
9.1.2.2. Imported Data Source 658
9.1.3. Natural Frequency Analysis Results 659
9.1.3.1. Summary Panel 659
9.1.3.2. Graphical Results Layers 660
9.1.3.3. 3D Model / Results Viewing 660
9.1.3.4. Log Files 660
9.2. FEA- In-Circuit Testing (ICT) 661
9.2.1. ICT Input Data 661
9.2.1.1. ICT Test Fixtures 662
9.2.1.2. ICT Test Points 662
9.2.1.2.1. Adding a Test Point 663
9.2.1.2.2. Modifying Test Point Properties 663
9.2.1.2.3. Saving Test Point Changes 664
9.2.1.2.4. Importing Test Point Locations 664
9.2.1.2.5. Importing Test Points From Pick & Place Files 666
9.2.1.3. ICT Mesh 667
9.2.2. ICT Analysis Properties 667
9.2.2.1. Generated Data Source 668
9.2.2.2. Imported Data Source 669
9.2.3. ICT Analysis Results 670
9.2.3.1. Summary Panel 670
9.2.3.2. ICT Table 671
9.2.3.3. Graphical Results Layers 672
9.2.3.4. 3D Model / Results Viewing 673
9.2.3.5. Log Files 674

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3. FEA- Mechanical Shock 674
9.3.1. Input Data 675
9.3.1.1. Mechanical Shock Events 676
9.3.1.2. Mount Points 676
9.3.2. Mechanical Shock Analysis Properties 676
9.3.2.1. Generated Data Source 676
9.3.2.2. Imported Data Source 678
9.3.2.3. Strain Map Data Source 680
9.3.2.4. LS-DYNA Imported Data Source 680
9.3.3. Mechanical Shock Analysis Results 682
9.3.3.1. Summary Panel 682
9.3.3.2. Tabular Mechanical Shock Results 684
9.3.3.3. Life Prediction Curve 685
9.3.3.4. Graphical FEA Scores 686
9.3.3.5. Graphical FEA Results 687
9.3.3.6. 3D Model / Results Viewing 687
9.3.3.7. Log Files 687
9.4. FEA- Harmonic Vibration 688
9.4.1. Harmonic Vibe Input Data 689
9.4.1.1. Harmonic Events 689
9.4.1.2. Reuse Modal Results Setting 689
9.4.1.3. Mount Points 690
9.4.2. Harmonic Vibration Analysis Properties 690
9.4.2.1. Generated Data Source 690
9.4.2.2. Imported Data Source 692
9.4.3. Harmonic Vibration Analysis Results 694
9.4.3.1. Summary Panel 694
9.4.3.2. Harmonic Vibe Tabular Results 695
9.4.3.3. Harmonic Vibe Life Prediction Curve 696
9.4.3.4. Harmonic Vibe Graphical Results Layers 696
9.4.3.5. Harmonic Vibe 3D Model / Results Viewing 697
9.4.3.6. Log Files 698
9.5. FEA- Random Vibration 698
9.5.1. Random Vibe Input Data 699
9.5.1.1. Random Events 700
9.5.1.2. Single Modal Analysis Setting 700
9.5.1.3. Mount Points 700
9.5.2. Random Vibration Analysis Properties 701
9.5.2.1. Generated Data Source 701
9.5.2.2. Imported Data Source 702
9.5.2.3. Strain Map Data Source 703
9.5.3. Random Vibration Analysis Results 704
9.5.3.1. Summary Panel 704
9.5.3.2. Random Vibration Tabular Results 705
9.5.3.3. Random Vibe Life Prediction Curve 706
9.5.3.4. Random Vibe Graphical Results Layers 707
9.5.3.5. Random Vibe 3D Model / Results Viewing 707
9.5.3.6. Log Files 708
9.6. FEA- Thermal Mech (Thermo-Mechanical) 708
9.6.1. Thermal Mech and Material Properties 709
9.6.2. Thermal Mech and Input Data 710

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.6.3. Overview of Thermal Mech Analyses 711
9.6.4. Thermal Mech Properties Dialog 713
9.6.5. Types of Thermal Mech Analyses 717
9.6.5.1. Thermal Events with Part Temperature Rise 718
9.6.5.2. Thermal Events without Part Temperature Rise 719
9.6.5.3. Thermal Events with Thermal Map (CSV) 719
9.6.5.4. Thermal Events with Thermal Map (Image) 722
9.6.6. Summary of Analysis Combinations 727
9.6.7. Thermal Mech and Mount Points 728
9.6.8. Part Temp Rise and Thermal Map FAQ 728
9.6.9. Imported Data Source 729
9.6.10. Thermal Mech Analysis Results 730
9.6.10.1. The Results Tables 730
9.6.10.2. Layer Viewer and 3D Results Viewer 738
9.7. FEA- Virtual Accelerometer 741
9.7.1. Accelerometer Editor 741
9.7.1.1. Adding an Accelerometer 741
9.7.1.2. Modifying An Accelerometer 743
9.7.1.3. Removing an Accelerometer 744
9.7.1.4. Accelerometer Changes 744
9.7.2. Accelerometer Results 744
9.7.2.1. Displacement Results 745
9.7.2.2. Acceleration Results 746
9.7.2.3. Number of Data Points 747
9.8. FEA- Ansys Workbench™ Integration 747
9.8.1. Exporting a CCA from Sherlock to Workbench 749
9.8.1.1. Exporting ECAD to Mechanical CAD Automation 750
9.8.1.1.1. Shells and Beam Element Order 751
9.8.2. Ansys Workbench Geometry and Materials Import 752
9.8.2.1. Manually Running Workbench Journal File 752
9.8.2.2. Add Additional Geometries 753
9.8.3. Performing the Analysis 754
9.8.3.1. When to Use Multiple Models for Multiple Analyses 754
9.8.3.2. Setting Up the Project Schematic 754
9.8.3.3. Sherlock-to-Workbench Model Update Notifications 755
9.8.3.4. Settings and Parameters Required for a Valid Results File 756
9.8.3.4.1. Set Uniform Units 756
9.8.3.4.2. Use Supported Element Types 757
9.8.3.4.3. Review Mesh Properties 757
9.8.3.4.4. Update Contacts 757
9.8.3.4.5. Preserve Named Selections 757
9.8.3.4.6. Define Boundary Conditions 758
9.8.3.4.7. Define Loads 758
9.8.3.5. Natural Frequency Analysis 758
9.8.3.6. Settings by Analysis Type 759
9.8.3.6.1. Vibration Analysis 759
9.8.3.6.2. Mechanical Shock Analysis 760
9.8.3.6.3. ICT Analysis 760
9.8.4. Importing Analysis Results into Sherlock 761
9.8.4.1. Setting Nodal Forces in Workbench 761
9.8.4.2. Step-by-Step Procedure for Importing Results 761

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.8.4.3. Resolving Possible Import Issues 764
9.9. FEA- Ansys Icepak Integration 765
9.9.1. Step 1: Exporting the .tcl File from Sherlock 765
9.9.2. Step 2: Creating the Icepak Project TZR File 767
9.9.3. Step 3: Modify or Add Additional Geometries 769
9.10. Strain Mapping in Sherlock 769
9.10.1. Strain Map Feature: Capabilities and Limitations 770
9.10.2. Adding Strain Map Files (Image or CSV) to Sherlock 770
9.10.2.1. Adding Strain Map Image Files 770
9.10.2.2. Adding Strain Map CSV/Excel Files 775
9.10.3. Random Vibe Analysis Using Strain Maps 777
9.10.4. Harmonic Vibe Analysis Using Strain Maps 780
9.10.5. Mechanical Shock Analysis Using Strain Maps 781
9.10.6. Running a Strain Map Analysis on an Assembly 783
9.10.7. Results of the Strain Map Analysis 784
10. Viewing and Managing Analysis Results 785
10.1. Sherlock Results Viewer 785
10.1.1. The Results Viewer and Memory Management 786
10.1.2. Viewing Current Results 786
10.1.3. Saving Results 787
10.1.4. Renaming / Deleting Results 788
10.1.5. Viewing Saved Results 788
10.1.6. Exporting Saved Results 790
10.1.7. Console Window 790
10.2. Sherlock 2D Viewer 791
10.3. Sherlock 3D Viewer 791
10.3.1. Launching the 3D Viewer 791
10.3.2. Viewing and Configuring the 3D Results 792
10.3.3. 3D Viewer's Main Menu and Toolbar 795
10.3.4. Exporting 3D Images 798
10.3.5. The 3D Viewer and Memory Management 799
10.4. Reporting and Data Export 799
10.4.1. Generating Report Files 800
10.4.1.1. Customized Reports 801
10.4.2. Exporting Graphical Data 804
10.4.2.1. Exporting Data 804
10.4.2.2. Changing Plot Properties 804
10.4.3. Exporting Tabular Data 806
11. Integration with other Ansys Products 809
12. API Services 811

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

2.1. Copper Layers 67
3.1. Sample Net List Data Showing Required Columns 111
3.2. Sample Outline Layer, X and Y Coordinates 112
3.3. Import-Compatible Project Files 119
3.4. IPC-2581 Element Requirements 129
3.5. IPC-2581 Specification Support 130
3.6. User Defined Data Files 167
3.7. Material Properties File 169
3.8. Laminate Properties File 171
3.9. Laminate Glass Properties File 172
3.10. Solder Data File, Example 173
3.11. Solder Data File, Required Data 173
3.12. Solder Data File, Optional Data 174
3.13. Package Properties File 176
3.14. Package Properties File, Optional Data 177
3.15. Part Type Properties File 178
3.16. Part Type Properties File, Optional Data 178
3.17. PTH Quality File 179
3.18. PTH Quality File, Optional Data 179
3.19. Default PTH Quality Factors 179
3.20. Acceleration Factors Data File 180
3.21. Manufacturer File 180
4.1. String Matching Operators 213
4.2. Column Filtering Comparison Operators 214
4.3. Column Filtering Boolean Operators 215
4.4. Column Filtering Meta-Functions 215
4.5. String Matching Operators 217
4.6. Comparison Operators 217
4.7. Advanced Filtering Boolean Operators 218
4.8. Advanced Filtering Meta-Functions 219
4.9. Advanced Filtering- Numeric Expressions 220
4.10. Advanced Filtering- Numeric Functions 220
4.11. Advanced Filtering- Input Shortcuts 221
4.12. Column Filter String Matching Operators 244
4.13. Column Filter Comparison Operators 246
4.14. Column Filter Boolean Operators 247
4.15. Column Filter Meta Functions 247
4.16. Advanced Filters: String Matching 249
4.17. Advanced Filters: Comparison Operators 250
4.18. Advanced Filters: Boolean Operators 252
4.19. Advanced Filters: Meta Functions 253
4.20. Advanced Filters: Numeric Operators 254
4.21. Advanced Filters: Numeric Functions 254
4.22. Advanced Filters: Filter Management 255
4.23. Part Information for 08055A101JAT2A 278
4.24. Parts Wizard- Supported Operators 285
4.25. Supported Pattern Matching Fields 287
6.1. DFMEA Viewer, Keyboard Navigation 366
6.2. dfmeaProperties.csv 373

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3. Hardcoded DFMEA Properties 374
6.4. Add Rows: dfmeaProperties.csv 374
6.5. dfmeaFailure.csv 376
6.6. Template Variables (Default Opens) 376
6.7. Template Variables (Default Shorts) 377
6.8. Internally Defined Template Variables 381
6.9. DFMEA Data Properties 382
6.10. Template Lists 383
6.11. Processing Options 392
8.1. 10mm ICT Analysis 478
8.2. Shape Types 486
8.3. Shape Types 492
8.4. Strain Map Results (5mm Mesh) 525
8.5. Strain Map Results (1mm Mesh) 525
8.6. Strain Map Results (1mm and 5mm Mesh) 526
8.7. Region Shapes 529
8.8. PCB Modeling Properties 533
8.9. Trace Modeling Properties 535
8.10. Modeling Region Properties 539
8.11. Board Modeling Properties 547
8.12. Sub-Assembly Modeling Properties 548
8.13. Modeling Properties 550
8.14. Corner Shapes 557
8.15. Circular Component Properties 557
8.16. SMT Lead Types 573
8.17. Stub Model Lead Configurations 576
8.18. Lead Layouts 579
8.19. Lead Layouts (continued) 580
8.20. Lead Layout Properties 581
8.21. Ball Shape Properties 591
8.22. Solder Ball Layout Properties 592
8.23. Full Ball Pattern Properties 593
8.24. Perimeter Solder Pattern Properties 593
8.25. Channel Ball Pattern Properties 594
8.26. Island Solder Pattern Properties 594
8.27. Corner Ball Count Patterns 595
8.28. Heat Sink Modeling Properties 598
8.29. Arc Shapes 610
8.30. Mechanical Part Shape Types 633

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Getting Started with Sherlock

Topics Covered in this Chapter

1.1. Welcome to Sherlock
1.2. Start-Up and Configuration
1.3. Compatibility with Earlier Versions of Sherlock
1.4. Sherlock Settings Dialog

1.1. Welcome to Sherlock

The Ansys Sherlock™ electronics reliability prediction software is a physics-based engineering simulation that provides fast and accurate life predictions for electronic hardware at the component, board and system levels in early design stages. Among its many capabilities, Sherlock can translate ECAD files to FEA files in minutes, predict time-to-failure for an entire PCBA– down to each component and connection, and turn stresses (electrical, thermal and mechanical) into a prediction of product lifetime. Sherlock integrates seamlessly with Ansys Workbench, Ansys Icepak, and Ansys Mechanical.

The following links provide the information you need to start using Sherlock:

- Read the Sherlock Release Notes on the Ansys Help website.
- For detailed information on licensing, see the Ansys Licensing Guide on the Ansys Help website.
- For detailed information on software installation, see the following sections of the Ansys Installation Guide:
  - Sherlock Installation Guide for Windows
  - Sherlock Installation Guide for Linux
- If you are running Sherlock on Linux, see the section below, For Linux Users (p. 3).
- A list of third-party software used by Sherlock has been packaged with your Ansys installation. For standard installations, you will find the list in the following location:

  ```
  C:\Program Files\ANSYS Inc\v251\sherlock\Open Source
  ```

  A complete software bill of materials is available upon request.
- Tutorials designed to introduce you to Sherlock's capabilities:
  - Tutorial: Project Overview (p. 8)
  - Tutorial: ODB++ Project Creation (p. 23)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Tutorial: Reviewing Part Properties (p. 30)
  - Tutorial: Layer Viewer (p. 43)
  - Tutorial: Ad Hoc Project Creation (p. 56)

- For advanced users, Sherlock APIs are available online at the Ansys DevPortal where you will find:
  - **Closed source APIs utilizing gRPC**—a high performance, universal RPC (Remote Procedure Call) framework, allowing those with programming experience to access some of the Sherlock application's internal functions, making it easier, for example, to integrate Sherlock's capabilities with other applications.
  - **PySherlock:** PyAnsys is a collection of open source Python libraries designed for engineers to enhance the capabilities of Ansys products, including the Sherlock electronics reliability prediction software. These Python client libraries enable efficient data manipulation, automation, and customization of workflows, empowering engineers to gain deeper insights and to streamline simulations. The project fosters a collaborative environment within the engineering community, encouraging knowledge sharing and community-driven development. With comprehensive documentation and vibrant community support, developers and engineers can maximize the capabilities of PyAnsys and contribute to its growth.

1.2. Start-Up and Configuration

In this section, the following topics are covered:

1.2.1. Sherlock Licensing
1.2.2. Standard Folder Locations
1.2.3. For Linux Users
1.2.4. Optional Software Compatibility
1.2.5. Initial Testing and Configuration
1.2.6. Memory Management
1.2.7. 3D Viewer Compatibility

1.2.1. Sherlock Licensing

For information on Ansys licensing, refer to the Ansys Licensing Guide.

1.2.2. Standard Folder Locations

Windows

The following Sherlock folder locations are for installations running on Windows. Folder locations for Linux are listed in the section below.

In Windows, Sherlock uses two standard folder locations for its files: the Ansys installation folder and the Sherlock user data folder. The Sherlock installation folder contains all the common files needed to run Sherlock on any system. Usually the installation folder is located in the following location (64-bit platforms):

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c:\Program Files\Ansys Inc\251
```

Although you may change this location at installation time, no files in the installation folder need to be changed after installation.

The Sherlock user data folder is created in the following location

```
c:\Users\USERID\AppData\Roaming\Sherlock
```

where USERID is the Windows user ID. The Sherlock user data folder contains all log and property files. This is also the default location for the user project folder in older versions of Sherlock prior to the implementation of the default project directory setting. These folders are only visible in Windows Explorer if the option to display hidden files is selected in Windows. Alternately, you can type the full path into Windows Explorer.

You can set the default project directory in **Settings > General Settings > Advanced** where your projects will be created. The user project folder holds design, result, and report files created by a given user. (See Project Management (p. 92) for more details.)

Linux

On Linux systems, the location of the Ansys installation folder:

```
/ansys_inc/v251
```

Although you may change this location at installation time, no files in the installation folder need to be changed after installation.

Sherlock user settings (where $Home is your home directory):

```
$HOME/.sherlock
```

The user data folder contains all log and property files. User data files and documents default location:

```
$HOME/.sherlock
```

Default project folder:

```
$HOME/Sherlock/projects
```

You can set the default project directory in **Settings > General Settings > Advanced** where your projects will be created. The user project folder holds design, result, and report files created by a given user. (See Project Management (p. 92) for more details.)

1.2.3. For Linux Users

This user's guide was written with Windows users in mind. Where the Linux version of Sherlock differs, you will find explanatory notes in the relevant sections of this user's guide. Below is a list of the most important differences:

- Be sure to refer to the Sherlock Installation Guide for Linux.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Abaqus and NX Nastran solvers are not supported by Sherlock when running on Linux.
- On Linux, there is no separate client console window. For its output, Sherlock uses the terminal window which was used to launch Sherlock.
- By default, you cannot open Sherlock files by double-clicking them. You must manually associate file types to specific applications within Linux. The procedure for doing this depends on the windowing environment you are using.
- In Linux, you can launch certain features using commands in the operating system:
  - To launch the 3D Viewer, use `runSherlock -3dViewer`
  - To launch the Results Viewer, use `runSherlock -resultViewer`
  - To launch the Life Cycle Manager, use `runSherlock -lifecycle`
  - To launch Sherlock in single-project mode, use `runSherlock -singleProject`

1.2.4. Optional Software Compatibility

The following may also be used with additional Sherlock license features enabled:

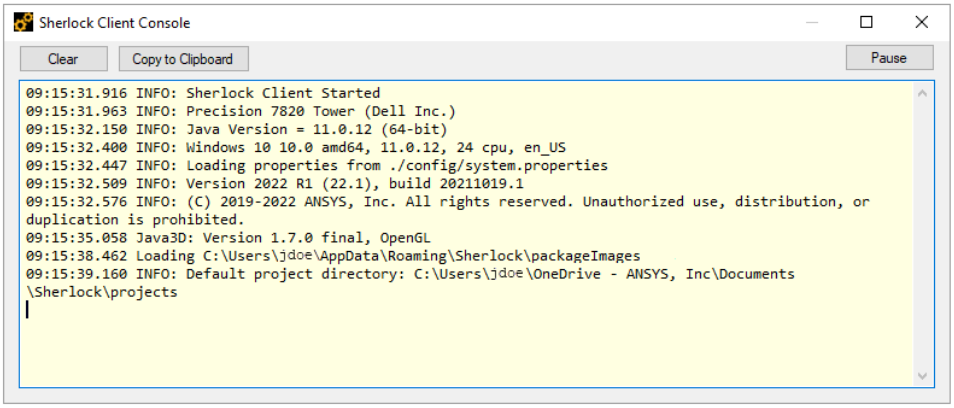
- Abaqus (Windows only)
- NX Nastran (Windows only)

1.2.5. Initial Testing and Configuration

You can perform the following simple tests to confirm that Sherlock is installed properly and that all required software components are configured properly.

1.2.5.1. Sherlock Client Console

When running on Windows, the Sherlock Client Console opens when Sherlock is started. You can also open it manually by selecting **Help > Show Console.** As shown below, the Client Console window displays key file system properties.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linux

On Linux, there there is no separate client console window. For its output, Sherlock uses the terminal window used to launch the application.

1.2.6. Memory Management

For information on configuring memory settings in Sherlock, refer to Launcher Settings (p. 84).

1.2.7. 3D Viewer Compatibility

If your video card does not support Sherlock's 3D Viewer, try these troubleshooting steps:

- Update your graphics driver.
- In some cases, modifying your graphics driver settings can resolve the issue. If an Intel HD graphics card is being used, right-click your **Desktop**, then **Graphics Properties**, then **3D**. Look for the OpenGL options to enable Vertical Sync, also known as Asynchronous Flip.
- If you are running Sherlock on a remote desktop or a virtual machine, you might be able to resolve the issue by enabling 3D graphics acceleration on the virtual/remote machine.

Laptops

3D Viewer compatibility issues can arise on laptops that have both an internal Intel graphics adaptor and a second adapter such as an NVIDIA graphics card. Configuring the laptop so it always uses the NVIDIA card often resolves the issue. To force the laptop to use the NVIDIA card only, do the following:

1. Right-click Windows desktop.
2. Select the **NVIDIA Control Panel** from the context menu.
3. Navigate to **3D Settings** > **Manage 3D Settings**.
4. Select NVIDIA for the preferred graphics processor.
5. Restart Sherlock.

1.3. Compatibility with Earlier Versions of Sherlock

> **Warning:**
>
> Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate analysis results, read the following information carefully.

In the unlikely event you create, open, or import a project in the current version of (Sherlock 2025 R1) and then open that same project and run an analysis in an earlier version, the FEA analysis might be incorrect. To correct this issue, do the following in the *earlier* version of Sherlock: Open the project, navigate to **Settings** > **Meshing**, and click the **Apply & Close** button. Then repeat the affected analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4. Sherlock Settings Dialog

This content has been moved to the *User Reference* chapter. See Sherlock Settings (p. 77).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Introductory Tutorials

The following tutorials provide a guided tour through Ansys Sherlock's basic features and workflows. If you are new to Sherlock, this is an excellent place to begin. It's a hands-on introduction where you will work with actual project files that have been packaged with your Sherlock software. Each tutorial builds on the others before it, so you should work through them consecutively, especially if this is your first time using Sherlock.

Tutorials:

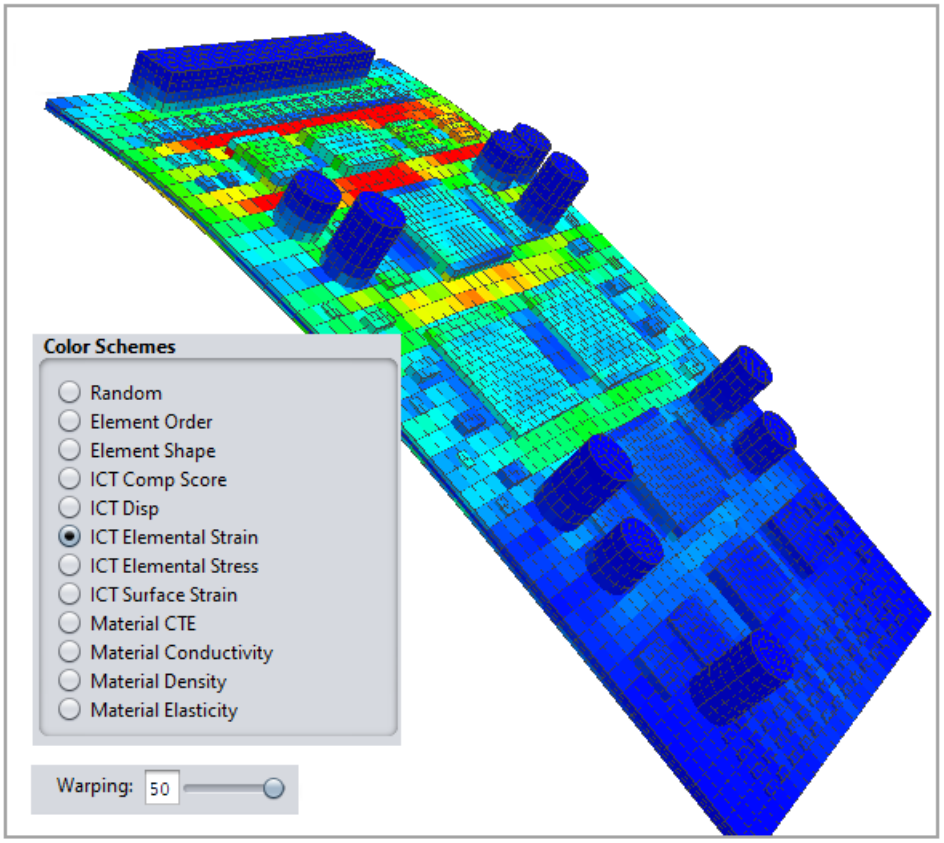
2.1. Tutorial: Project Overview

2.2. Tutorial: ODB++ Project Creation

2.3. Tutorial: Reviewing Part Properties

2.4. Tutorial: Layer Viewer

2.5. Tutorial: Ad Hoc Project Creation



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1. Tutorial: Project Overview

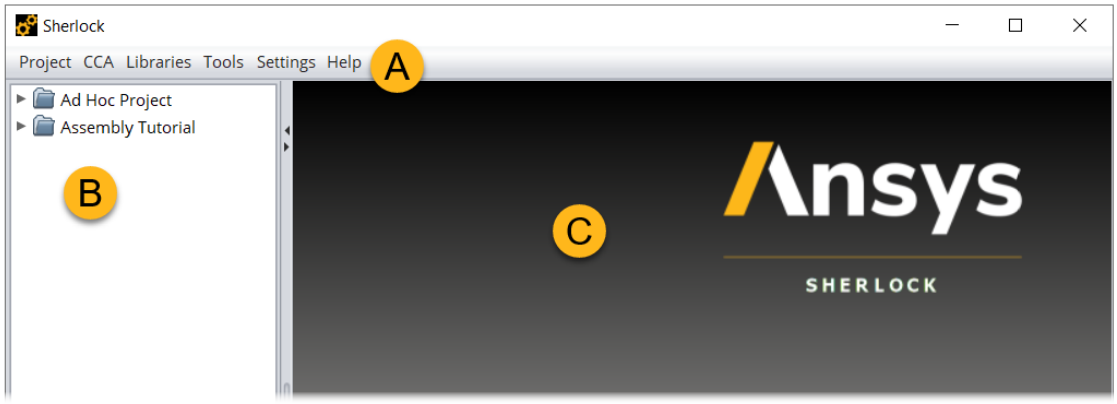
Ansys Sherlock organizes design files, inputs and analysis results as **Project Folders** that can be managed inside of the Sherlock application and shared between Sherlock users. This lesson shows you how to import a Sherlock project and provides an overview of the various files and displays associated with a project. Subsequent lessons will show you how to create projects, maintain design files, modify inputs and perform analysis tasks.

In this section, the following topics are covered:

2.1.1. Main Window
2.1.2. Importing a Project
2.1.3. Project Life Cycle
2.1.4. Circuit Card Information
2.1.5. Analysis
2.1.6. Results
2.1.7. Closing a Project

2.1.1. Main Window

When Sherlock is executed and your license file has been properly validated, the Sherlock main window will appear.



The main window is divided into three sections, with (A) the **Main Menu** along the top, (B) the **Project Tree** along the left side, and (C) the **Content Panels** occupying the rest of the window. There is a vertical divider between the Project Tree and the Content Panels that allows you to make the Project Tree panel wider or narrower to suit your purposes. You can also resize the main window at any time using standard window controls. Since no projects exist when Sherlock is first installed, the Project Tree is empty, waiting for you to import or create one or more projects.

2.1.2. Importing a Project

You will begin this tutorial by importing a completed project, one which has all the necessary design files, inputs defined, analysis tasks performed, and results generated. To import a project, select **Project** from the **Main Menu** and select **Import Project**. In the **Import Project** dialog (A), do the following:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For **Project Name**, enter **`Tutorial`** (or any name you wish).
- While not necessary, you may also choose to create a **Project Category**.
- Press the **Browse** button to locate the project archive file to be imported. If you installed your Ansys software in the default location, you will find the `.zip` archive here:

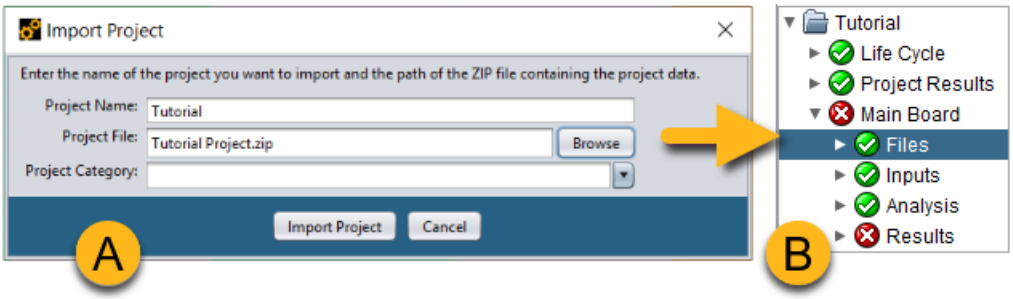
  Windows:

  ```
  C:\Program Files\Ansys Inc\v251\sherlock\Tutorial Project.zip
  ```

  Linux:

  ```
  /ansys_inc/v251sherlock/Tutorial Project.zip
  ```

  After locating the `.zip` file using the file browser, the **Import Project** dialog will look like this (A):



- Now click the **Import Project** button to import all files associated with the tutorial project.

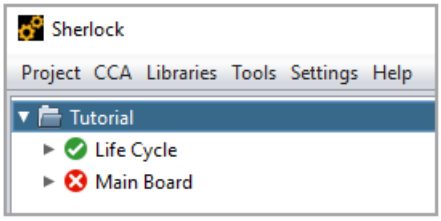
Note:

Depending on the size of the project archive file and the types of information being imported it may take a minute or so to load the archive file. During that time, a progress dialog will be displayed.

When the import process has completed, the **Tutorial Folder** will be added to the **Project Tree** (B, above) and will be fully expanded to allow quick access to all project files, inputs and results. All folders in the **Project Tree** (including the project folder itself) can be expanded or collapsed as needed. You can expand or collapse a folder either by left-clicking the triangle icon to the left of the folder name or by right-clicking the folder name and selecting **Expand** or **Collapse** in the context menu.

For practice, collapse the various project folders until the **Project Tree** looks like the tree shown below. As you can see, the project consists of two basic sets of information, the **Life Cycle** definition and the **Main Board**. In this project, only a single printed circuit card is defined, but projects may contain more than one board. The information associated with the Main Board includes the **Files, Inputs, Analysis**, and the **Results.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

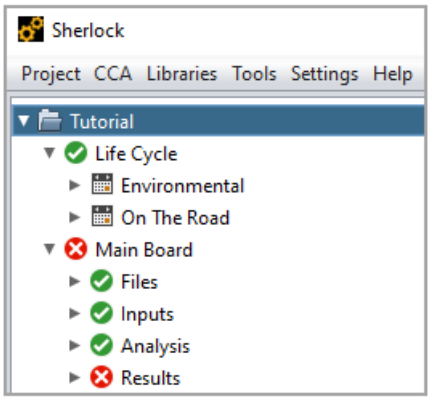


Note:

The available menu items may differ from what is reflected in these tutorials depending on licensed options.

The **Project Tree** not only organizes the information about each project, but it also serves as a to-do list. Specifically, the icons displayed next to each folder indicate whether some user action is required to either correct a problem or provide additional input. Sherlock does its best to eliminate the need for user inputs, but there are still times when user actions are required. In the tutorial project, most inputs have been entered and all analyses have been performed, so you see a green check mark alongside most folders indicating that no additional work is required.

The sections which follow explain the information and files associated with each of the **Project Tree** folders.

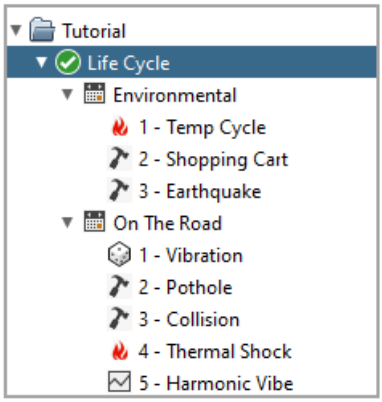


2.1.3. Project Life Cycle

The project **Life Cycle** defines the desired reliability goals for the project and the various environmental stresses that the circuit cards will see over the project lifetime. Sherlock can define a wide range of **Life Cycles**, from simple to complex, which are then used by the analysis modules to predict circuit card reliability. As such, it is important for you to become familiar with how modifications made to the **Life Cycle** definitions affect the analysis results. This lesson provides an overview of the various definitions. Subsequent lessons will describe how the **Life Cycle** definitions are used by the analysis modules.

Expand the **Life Cycle** folder to show all its definitions as shown below. The Life Cycle consists of two Phases called **Environmental** and **On The Road** represented by folders. In turn, each Phase consists of multiple Events representing the various environmental stresses the project will experience during all or part of the associated Phase. Such a hierarchical definition of Phases and Events is very flexible, allowing users to define a wide range of **Life Cycles.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

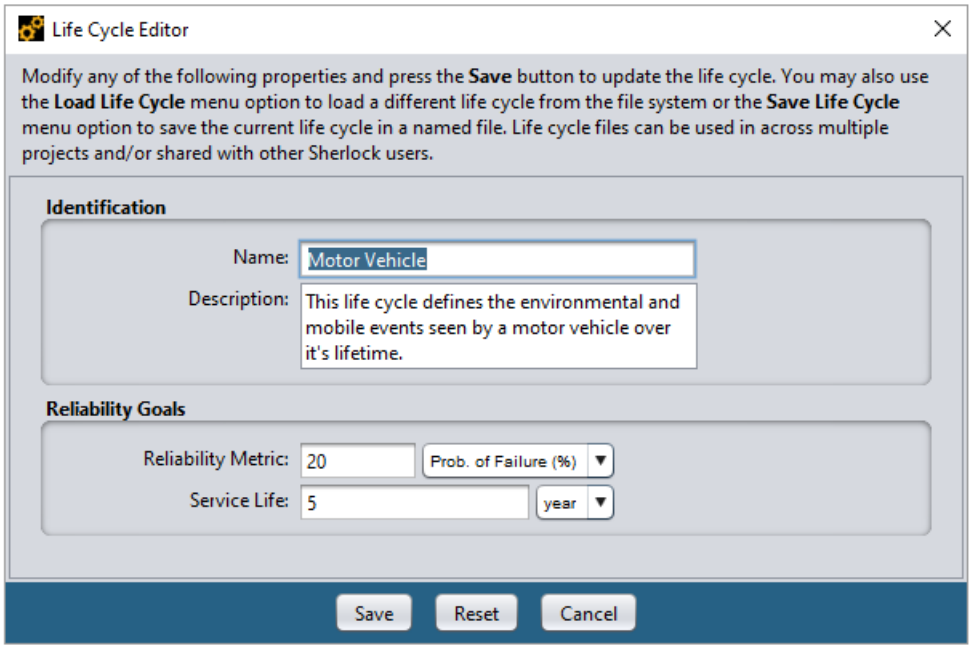


Note:

The definitions in the tutorial **Life Cycle** were contrived to show various aspects of the Sherlock analysis process and results. Don't use the tutorial **Life Cycle** as is for any real circuit card analysis.

Life Cycle definitions can be exported to the file system using the **Save Life Cycle** menu option available by right-clicking on the **Life Cycle** folder in the **Project Tree** and can be subsequently imported using the **Load Life Cycle** menu option. This allows you to maintain different Life Cycle definitions over a long period of time, easily assign a given Life Cycle to a given project, and share Life Cycles across many projects and users.

You can examine the overall project goals and how often the Phases and Events will be seen by the circuit card being analyzed. Right-click the **Life Cycle** folder and select **Edit Properties** from the context menu to show the **Life Cycle Editor.**



As you can see, the **Life Cycle** has a name and description (that is displayed in the **Project Report**) and two reliability goal properties (that are used by all analysis modules). Press **Cancel** to close the dialog without modifying any of the properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Now right-click **Life Cycle** in the **Project Tree** and select **View Life Cycle.** Afterwards, select the **Events** sub-tab at the bottom of the **Life Cycle** window to view a summary of all Phase Events.



The **Events** sub-tab is broken into four panels, summarizing the four different types of Events that can be defined. In this case, one of the Thermal Events (Environmental – 1 – Temp Cycle) spans the entire service life, while the other (On The Road – 4 – Thermal Shock) only spans a small percentage of the service life. The Mechanical Shock and Random Vibe Events only occur a small number of times, which represents a very small percentage of the five year service life.

A subsequent lesson (see Tutorial Life Cycle (p. 340)) will show how to modify Life Cycle Phases and Events. At this point, you can close the **Life Cycle** tab in the **Content Panel** and collapse the **Life Cycle** folder in the **Project Tree** in order to focus on the next project folder.

2.1.4. Circuit Card Information

Ansys Sherlock maintains various information about each circuit card assigned to a given project, including:

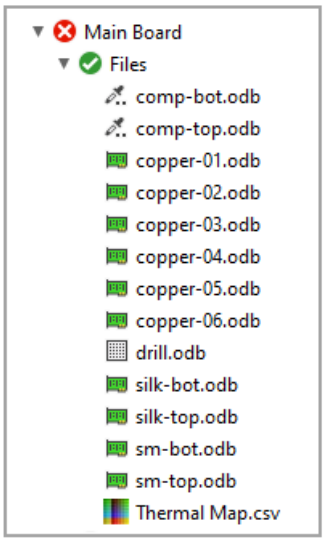
- Electronic design files (such as BOM, layer, drill, and layout files)
- Input data derived from design files (such as the parts list and stackup file)
- Tabular and graphical results for each analysis performed *and*
- Circuit card results summaries (such as the Score Card, Life Predictions, and Reports)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Project Tree** organizes all the circuit card information, providing quick access to both high-level and low-level information. Expand the **Main Board** entry in the **Project Tree** to view all the information associated with your tutorial board.

As seen earlier, the icons associated with each entry show either the type of information to be found, indicate that a user action is required or that a problem has occurred. In this tutorial example, all files have been processed successfully, all inputs have been entered and reviewed, and all analysis tasks have been executed, so you see green check marks and associated file type icons everywhere.

If you examine the **Files** folder, you can see the collection of design files that have been imported into Sherlock from which input data is derived. Shown below are two Pick and Place files providing the top and bottom locations, six copper layer definition files, one drill hole file, two silk screen layers, two solder mask layers, and a map of thermal data for various components.

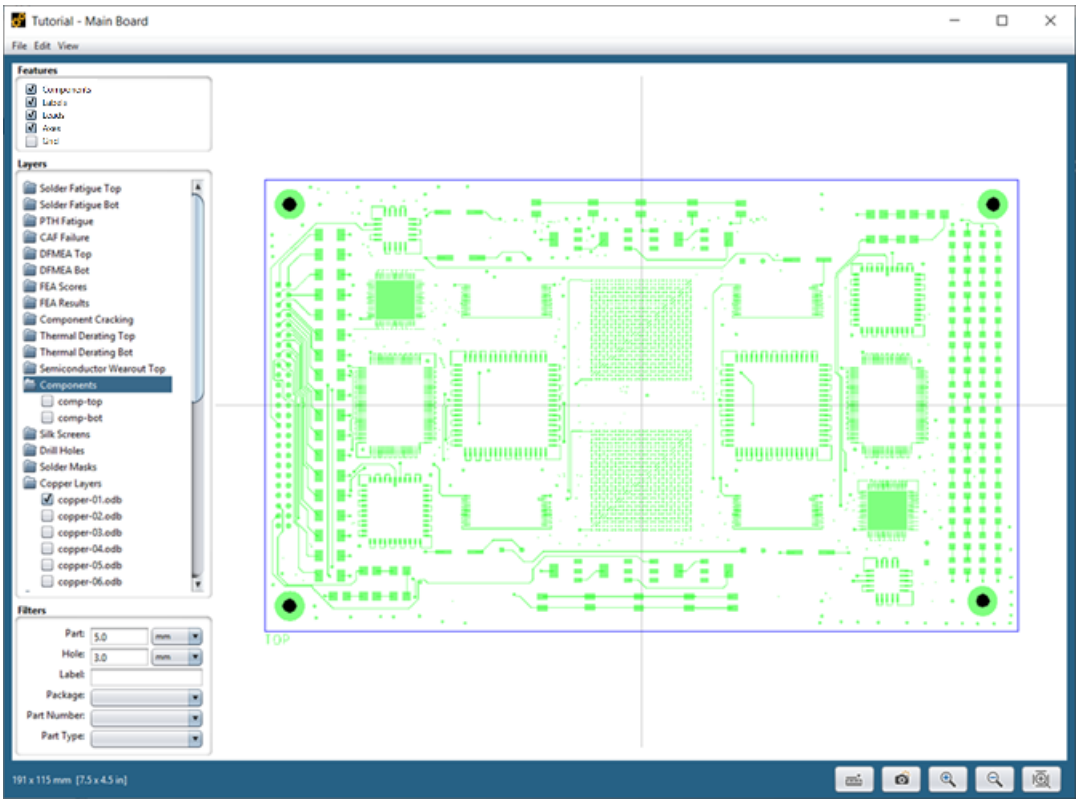


Sherlock knows how to parse and process an extensive list of file formats. In this case, most of the files came from an ODB++ archive of design files for the **Main Board**, but you are free to add files (such as the Thermal Map CSV file) that come from other sources. Subsequent lessons will explain how various design files can be processed by Sherlock.

2.1.4.1. Layer Viewer

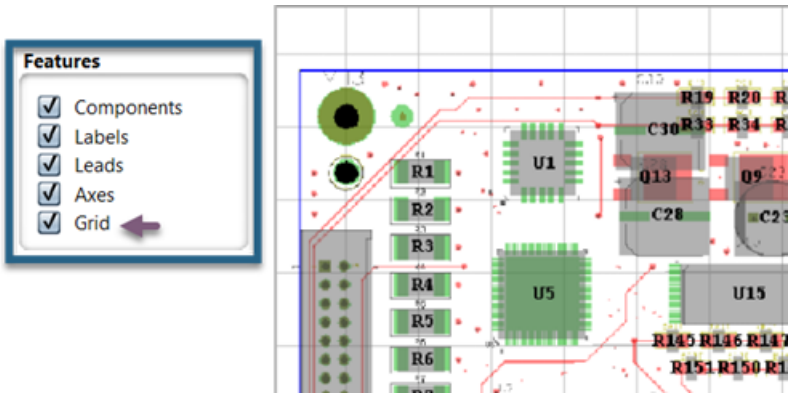
Aside from providing a listing of all design files, the **Project Tree** can also be used to quickly view the design files in various formats. For example, if you double-click the **copper-01.odb** entry in the **Project Tree**, the Sherlock Layer Viewer window will appear as shown here:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Layer Viewer displays one or more graphical layers associated with the selected circuit card. In this case, the top copper layer (the green traces) is in view, along with the board outline (the blue outer box). Although you can launch the Layer Viewer by selecting a single design file in the **Project Tree**, you can easily add or remove any number of layers to the display by selecting or deselecting the layer name on the left side of the Layer Viewer window. You can also filter graphical features (such as components and/or their labels), filter parts by size or label, and filter holes by size.

You can select the **Grid** option to superimpose a grid over the layer as shown below. The combination of layer and feature filtering makes the Layer Viewer an excellent way to examine circuit card design details and analysis results.

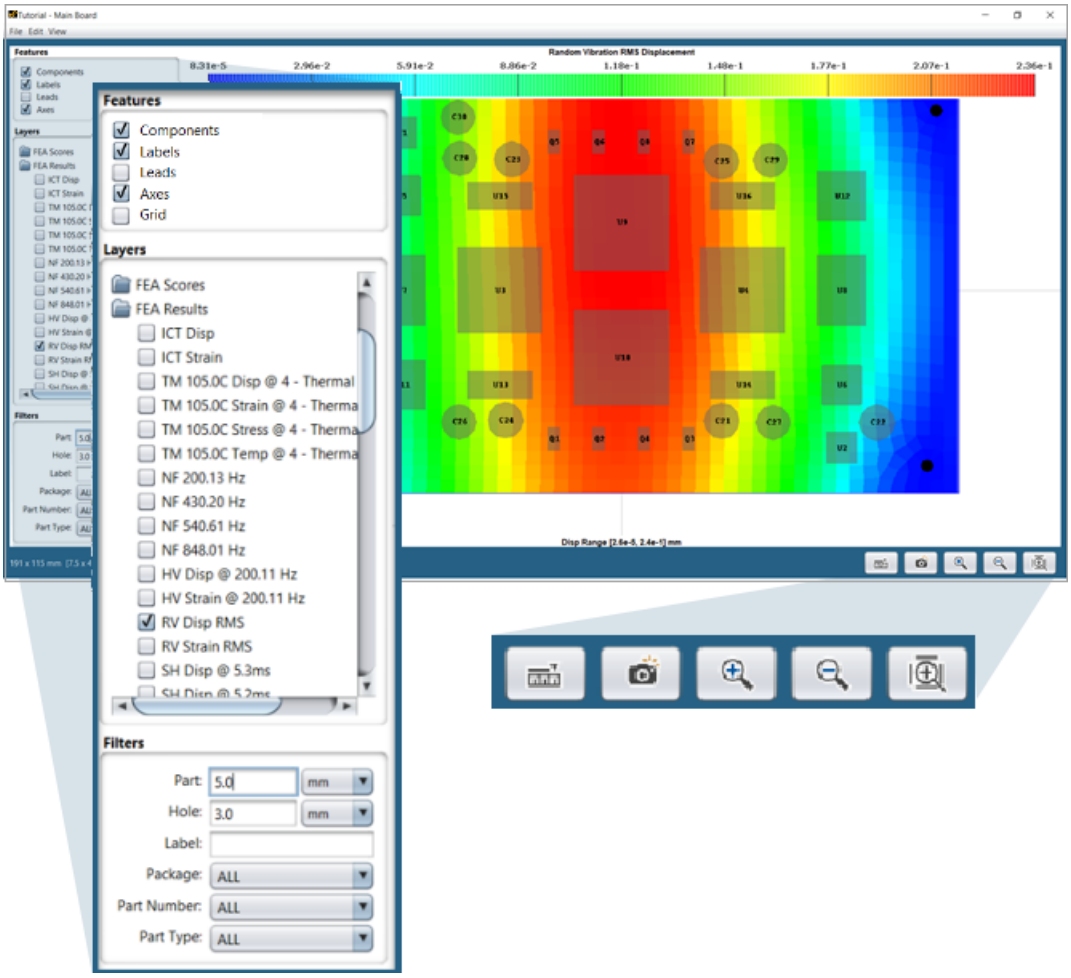


For example, the display shown below results from selecting the **RV Disp RMS** layer in the **FEA Results** folder, along with the **comp-top** layer in the **Components** folder, and **Mount Points** in the **Mechanical** Folder.

In the Layer Viewer shown below, the following settings have been selected: Display all component labels, filter all parts smaller than 5 mm, and filter all holes less than 3 mm (except mount points).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the buttons in the lower right corner of the Layer Viewer to export a snapshot to an image file, to zoom in, zoom out, and zoom to fit. You can also zoom in by double-clicking anywhere in the Layer Viewer and pan the display by dragging the mouse while holding the left mouse button down. Additionally, you may measure the distance between any two points with the distance tool.



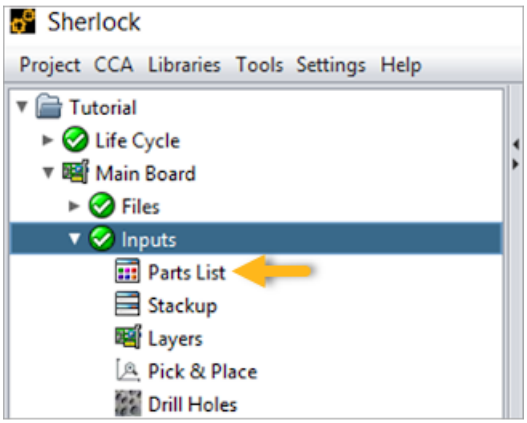
Take some time to click through the various design and result layers provided with the tutorial, and you'll quickly see the usefulness of the Layer Viewer. Subsequent lessons will show how to modify design files and input data using the Layer Viewer. For additional information, see Tutorial: Layer Viewer (p. 43).

2.1.4.2. Input Data

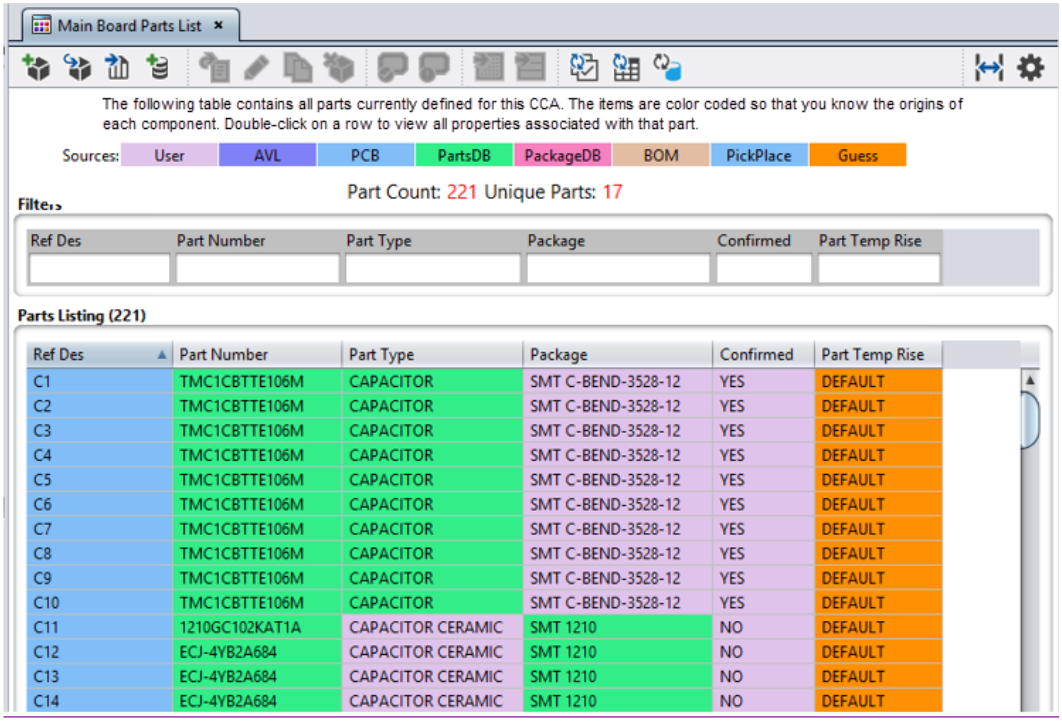
Design files provide Sherlock with graphical representations of the circuit card, but they also provide a wealth of data related to the circuit card components and the circuit card itself. Such data is used as input to the analysis modules, depending on their specific needs, and therefore represents a critical part of the process performed by Sherlock.

If you expand the **Inputs** entry in the **Project Tree** as shown below, you'll see entries for the **Parts List**, **Stackup data**, **Layers**, **Pick & Place**, and **Drill Holes**. The last three entries (**Layers**, **Pick & Place**, and **Drill Holes**) are simply convenient links to the Layer Viewer and need no further discussion. The first two, however, display two very important data displays, namely the **Parts List** and **Stackup Data**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Double-click the **Parts List** entry to display the **Main Board Parts List** in the **Content Panel** as shown here:



Note:

The columns shown in the **Parts List** above might be different than the columns you currently see in the Sherlock application. This is normal. You can add and delete columns to suit your needs. Click the ⚙ icon in the upper right of the table to customize what columns you want Sherlock to display. Refer to The Part Properties Viewer (p. 197) for additional information.

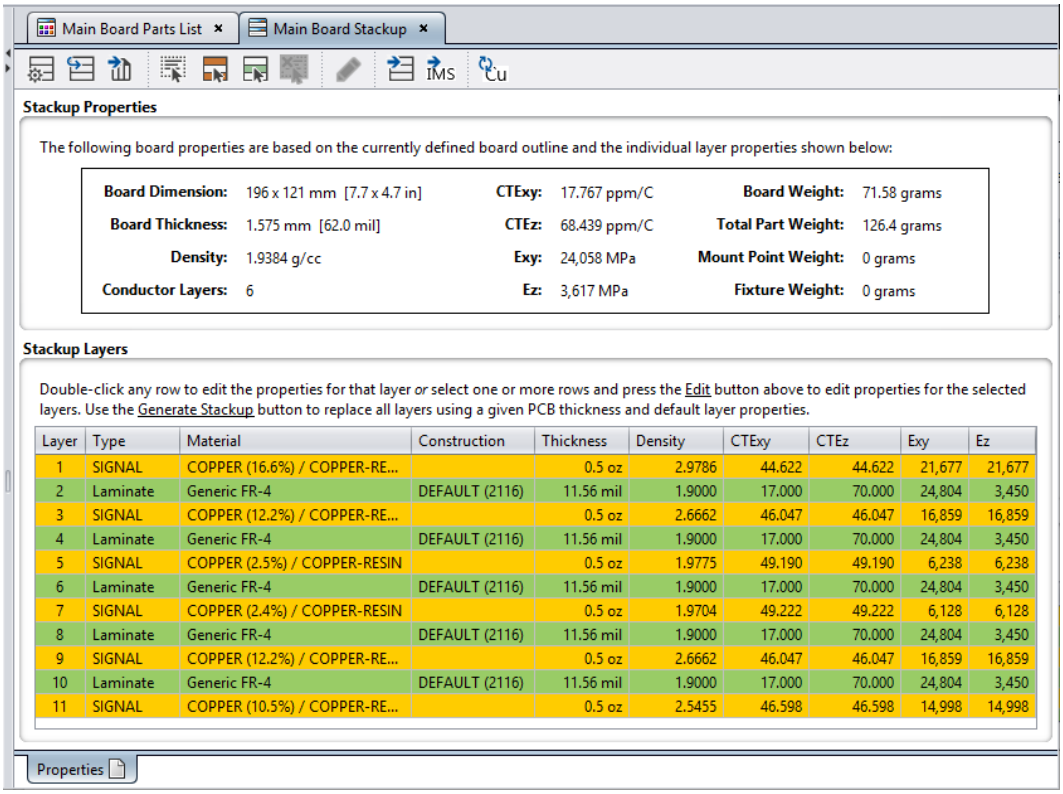
For the Sherlock tutorials, make sure the following columns are displayed: **Ref Des**, **Part Number**, **Part Type**, **Package**, and **Confirmed**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As the name implies, the **Parts List** shows a list of all parts (that is, components) associated with the circuit card. The **Parts List** is color coded to indicate where the various data was found (data can be derived from multiple, conflicting sources) and whether you have reviewed the data for correctness. The **Parts List** supports several features that allow you to drill-down to specific part properties (Sherlock maintains many dozens of properties per part, as well as different property values from different sources), modify part properties as needed, filter parts for viewing/editing purposes and export part properties to a CSV file for further processing. Subsequent lessons (specifically the lessons covering how to import and create projects) describe in detail the **Parts List** features and how to use them in Sherlock.

The **Parts List** holds information about each of the parts used on a circuit card, but the data associated with the circuit card itself is maintained as Stackup Data.

The **Stackup** tab shows the design and mechanical properties associated with each circuit card layer (as derived from the design files, entered manually by the user, or imported) and the aggregate mechanical properties derived automatically by Sherlock for the circuit card. These mechanical properties are used primarily by the Finite Element Analysis (p. 443) module and will be covered in more detail in that lesson. Sherlock gives you the tools to create, edit, import, and export stackup data. Refer to the section Additional Stackup Tools (p. 69) in the Tutorial: Ad Hoc Project Creation (p. 56) to learn more about the stackup editing tools.



2.1.5. Analysis

All the design files and input data are maintained for only one purpose, to allow the Sherlock analysis modules to predict the reliability of the circuit card. For the most part, analysis tasks can be executed at any time once a minimal amount of data has been entered. Clearly, if only minimal data has been entered and/or the data has not been fully verified, then the analysis results may be suspect; nonetheless, Sherlock will still generate the results. It is ultimately up to you to ensure that the input data is as correct as possible or required.

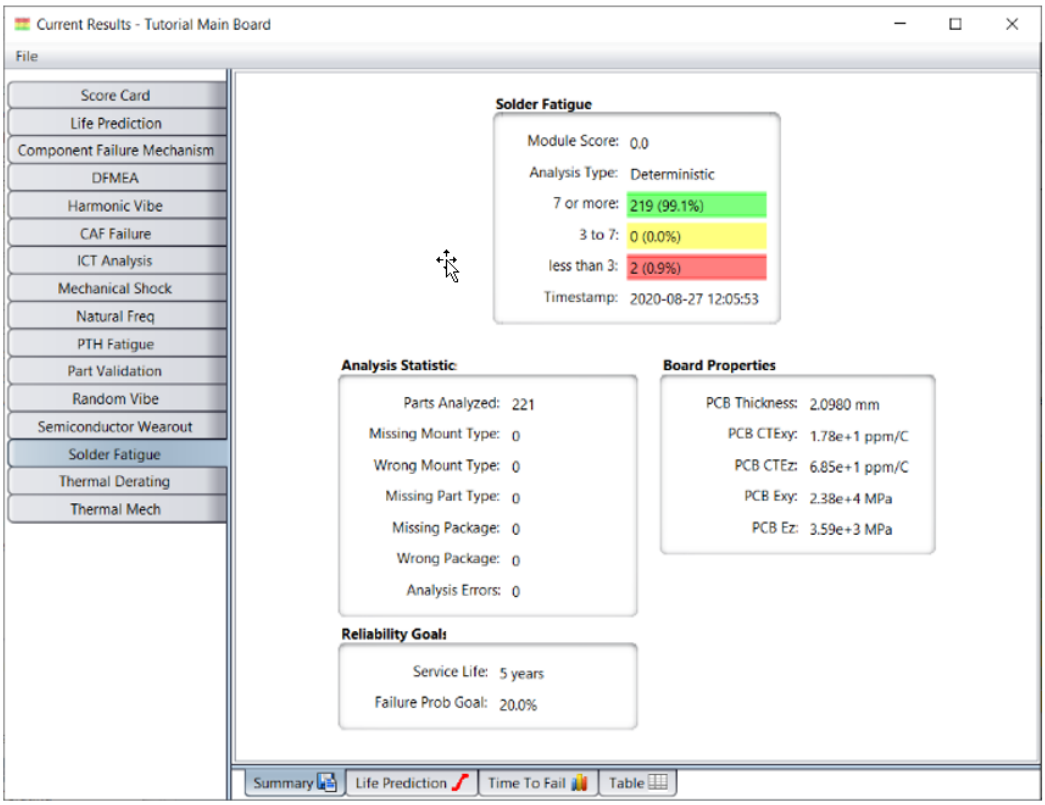
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

That said, Sherlock does automatically take some actions to ensure that the inputs and results maintained in a project are consistent. Specifically, Sherlock automatically clears analysis results, score cards, Life Prediction curves, and any previously generated report whenever key input data is modified. For example, if you add or remove a **Thermal Event** to the **Life Cycle**, then any analysis result that depends on thermal Events will be cleared. Similarly, if you move the location of a part by editing the **Parts List** or by using the **Component Editor** in the Layer Viewer, then Sherlock will automatically clear all results that depend on part location.

The results generated by each analysis module may differ in various ways depending on the type of analysis being performed, but they also show similar types of results, such as:

- Summary panel showing key input values and results
- Table of detailed result data (per part, per hole, for example)
- Life Prediction curve (Deterministic analysis modules only)
- Result data distribution charts

For example, double-click the **Solder Fatigue** entry in the **Project Tree** to display the **Main Board Solder Fatigue** tab in the Sherlock Results Viewer as shown below. The **Summary Panel** provides an overview of the scores generated for each component analyzed, the key inputs used for the analysis, and statistics about the analysis process itself. A module score of 0.0 reveals there are major concerns for this circuit card regarding solder fatigue failures.



The **Time to Fail** and **Table** sub-tabs at the bottom of the **Solder Fatigue** results panel provide views of the analysis results, which can be exported as image or CSV files. Additionally, to see the Life Prediction Chart, select the **Table** sub-tab, right-click anywhere on the table, and select **View Life Prediction**.

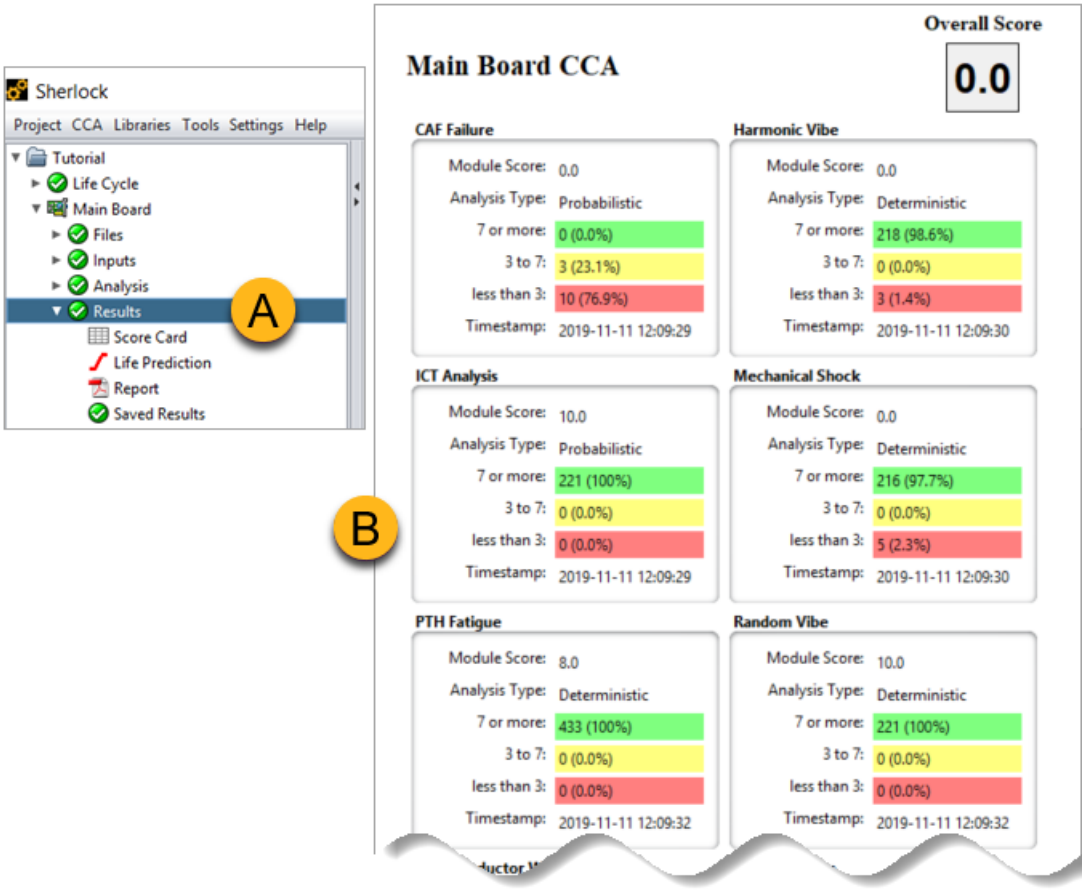
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Take some time to click through the various tutorial results provided by each of the analysis modules. When viewing a tabular display, you can right-click any row for additional features. When viewing graphical charts, you can also right-click anywhere in the chart for additional features. These topics will be discussed in detail in subsequent lessons.

2.1.6. Results

The primary purpose of Ansys Sherlock is to generate analysis results. This section reviews how the analysis results are summarized in Sherlock as a **Score Card**, **Life Prediction** chart and a printable **PDF Report**. To begin, expand the **Results** folder (A, below) to reveal the underlying project results. Keep in mind, for simplicity, the tutorial project only has a single circuit card defined. Nonetheless, there are plenty of results to examine.

The icons next to each folder entry indicates that our results are ready and waiting to be reviewed. If results were not available or a problem occurred while generating them, then a red X icon would be displayed next to the unavailable item.



2.1.6.1. Score Card

Double-click the **Score Card** entry to display the **Project Results** tab in the **Content Panel**. Shown above (B) is a portion of the tutorial project score card.

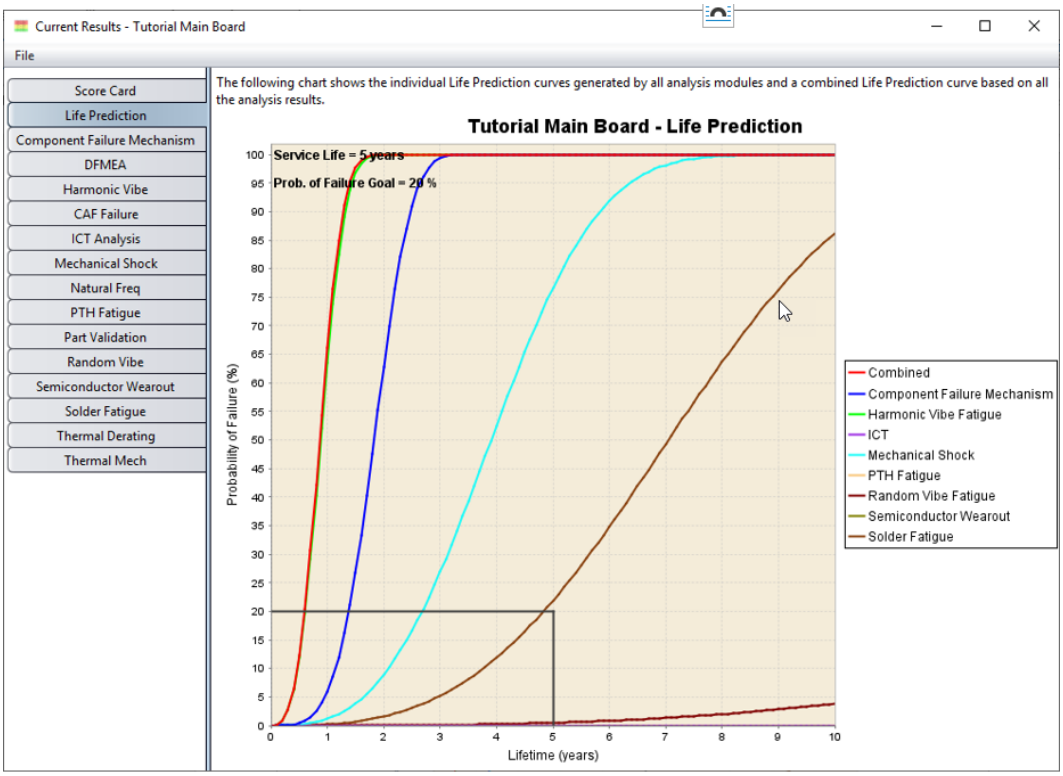
The **Score Card** is a collection of score cards, one each for the various analyses performed on the circuit card. The individual scores for each analysis task and a break-down of component scores are provided. The individual scores are then combined to produce an **Overall Score** for the project.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, the Main Board has some significant problems with an Overall Score of **0.0** – not a production-ready circuit card.

2.1.6.2. Life Prediction Chart

The Life Prediction chart shows a summary of all analysis results that predict the reliability of circuit card holes or components, as well as a graphical display of the reliability goals specified in the **Life Cycle** definition.



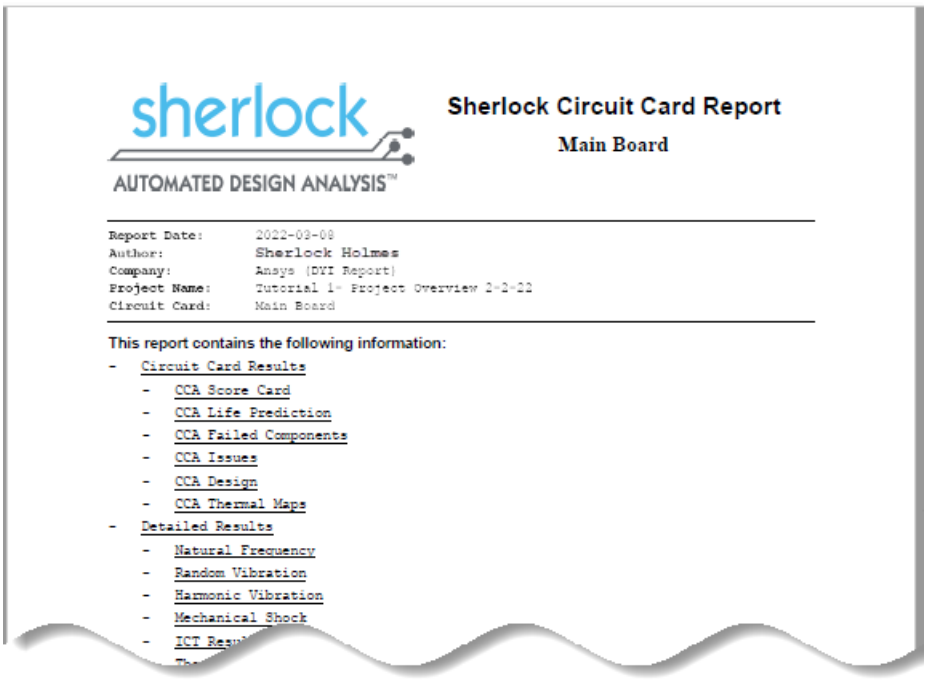
Here, the chart shows that the Main Board will not meet the life cycle reliability goals. At the 5-year point, failures related to Random Vibration Fatigue, Mechanical Shock, Harmonic Vibration Fatigue, In-Circuit Test (ICT), and Plated Through Hole Fatigue (PTH Fatigue) are predicted to be well below the desired goal. Failures related to Solder Fatigue and Semiconductor wearout are predicted to be within the desired goal. The Combined Failure curve shows that the Main Board will simply not meet its overall reliability goals.

2.1.6.3. Project Report

It's often said that a job isn't complete until the paperwork is done. Sherlock helps as much as possible by generating a printable **PDF Report** containing as much or as little as you want about the project. Aside from choosing various options from a long list of options, you only need to press a button to generate and/or view the report.

We've already generated a report for the tutorial project, so let's examine it first by double-clicking on the **Report** entry in the **Results** folder. At that point, your favorite PDF viewer should appear, displaying the Sherlock report, as shown here.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



sherlock
AUTOMATED DESIGN ANALYSIS™

Sherlock Circuit Card Report

Main Board

Report Date:	2022-03-08
Author:	Sherlock Holmes
Company:	Ansys (DYI Report)
Project Name:	Tutorial 1- Project Overview 2-2-22
Circuit Card:	Main Board

This report contains the following information:

- Circuit Card Results
  - CCA Score Card
  - CCA Life Prediction
  - CCA Failed Components
  - CCA Issues
  - CCA Design
  - CCA Thermal Maps
- Detailed Results
  - Natural Frequency
  - Random Vibration
  - Harmonic Vibration
  - Mechanical Shock
  - ICT Resu

As you page through the report you'll see **Circuit Card Results** as well as detailed results and problem areas determined by each of the analysis modules.

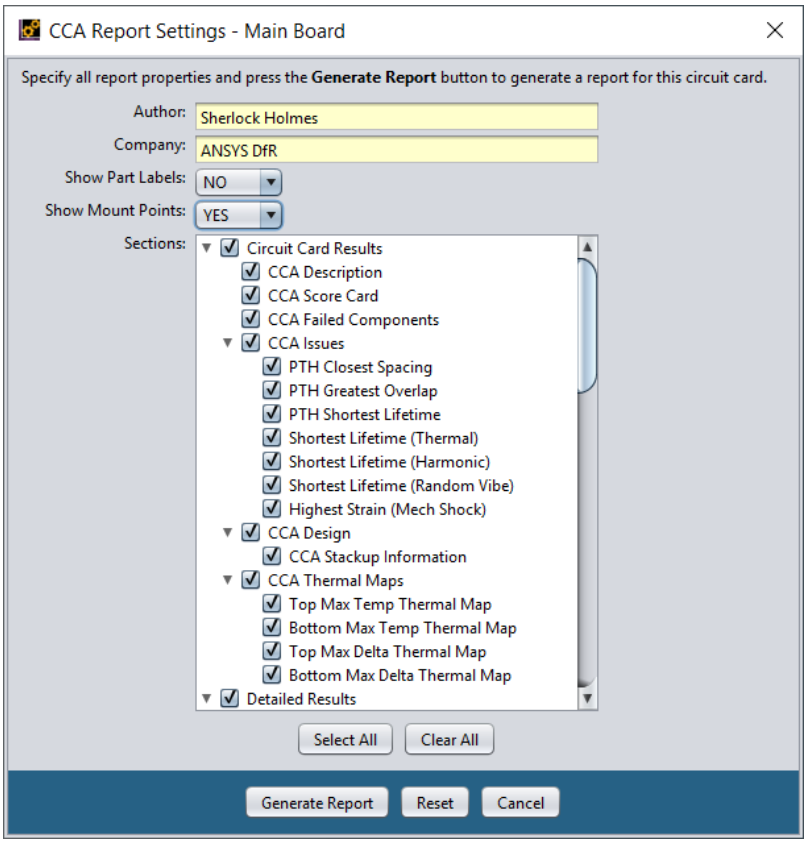
The report provided with the tutorial project is 51 pages. If all the report options are selected for a circuit card report, then a report could be very long. It's up to you to decide how much or how little you want to include in each report. Fortunately, you can change your mind at any time and generate as many different reports as needed.

Note:

Sherlock only stores the last project report generated for subsequent viewing. If you need different project reports for different purposes or need to maintain an archive of reports over time for a given project - either save a copy of the PDF report in a separate location, or save the results using the Sherlock Results Viewer (described later in the tutorial).

To see how project reports are generated, open the **Report Properties** dialog by right-clicking the **Report** entry in the **Project Tree** and selecting the **Generate Report** menu option. In the figure below, the report consists of a sequence of optional sections. By simply selecting or deselecting sections you can customize the report to suit your purposes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

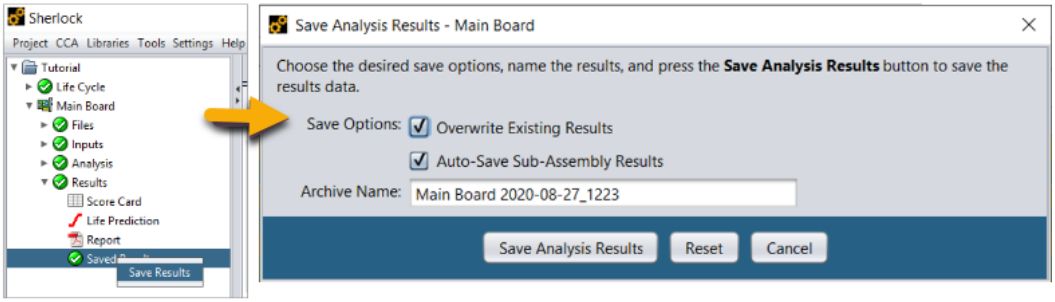


For convenience, if a high-level tree entry is deselected, then all lower-level entries in that sub-tree will *not* appear in the report, regardless of their individual selection setting. This allows you to exclude large sections temporarily without losing all lower-level selection settings.

For now, you can close the **Results** tab in the **Content Panel** and collapse the **Results** folder in the Project Tree.

2.1.6.4. Saving Results

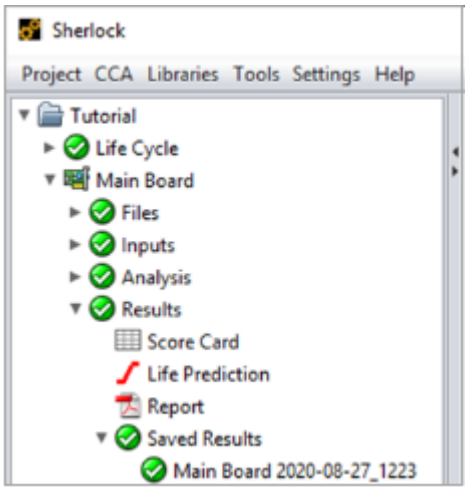
The circuit card report only contains the results applicable to the given circuit card and may also contain all the selected analysis layers generated for each circuit card. **Circuit Card Results** can be saved for later comparison and reporting purposes. Right-click the **Results** folder and select the **Save Results** menu option.



A new message will appear asking you to name the analysis results. You may also choose to overwrite existing results or auto save subassembly results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After entering a name (Main Board 2020-08-27_1223, in this case) for the results, click **Save Analysis Results.** A snapshot of the current results is saved to the **Saved Results** folder.



2.1.7. Closing a Project

As shown above, you can expand or collapse any folder in the **Project Tree** as needed, including the project folder itself. When a project folder is collapsed, all the entries associated with that project are hidden from view, but internal resources associated with the project are still maintained. If you want to keep a Sherlock project available, but not consume any resources then you can close a project by right-clicking on the project folder and selecting the **Close Project** menu option. When a project is closed, the project name is displayed in the **Project Tree,** but all entries are hidden from view and no resources are used in Sherlock. Additionally, the Layer Viewer and any content panels associated with the project will be automatically closed. You can re-open a closed project at any time by right-clicking the project folder and selecting the **Open Project** menu option (which will be the only available menu option).

Since you are done with the tutorial project overview, go ahead and close the project. The tutorial will let you know in subsequent lessons when you will need to access it again.

Summary

This concludes the overview of Sherlock's key capabilities by examining a completed tutorial project. The lessons which follow describe how to import and create Sherlock projects, including detailed discussions of the critical data validation process. You'll see many of the same displays discussed in this lesson but with more detailed explanations. Upcoming lessons also describe each of the analysis modules in more detail, the inputs required by them, and the iterative analysis process (entering data, performing analysis, reviewing results, and repeating) that can be used to improve the accuracy of your reliability predictions over time.

2.2. Tutorial: ODB++ Project Creation

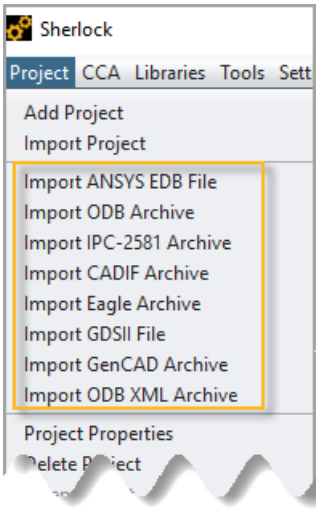
Topics:

2.2.1. Importing an ODB++ Archive
2.2.2. Project Name and Properties

There are two primary ways to create a Sherlock project:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. By importing data from an archive such as an ODB archive or one of the other file types listed in the **Project** menu. (See Importing Projects and ECAD Archives (p. 118) for a complete list of all supported file types and how to work with them.)



2. Or by creating a project consisting of an ad hoc collection of layer files, drill files, and data files. This method will be described later in the Tutorial: Ad Hoc Project Creation (p. 56).

This lesson is dedicated to the first method, specifically, importing a project from an ODB++ archive (or ODB XML archive). Later tutorials and chapters will reference this project, so you should walk through this tutorial even if you do not expect to work with ODB++ archives. Subsequent lessons will address the additional steps needed to create an ad-hoc project.

In this section, the following topics are covered:

2.2.1. Importing an ODB++ Archive

2.2.2. Project Name and Properties

2.2.1. Importing an ODB++ Archive

ODB++ archives consist of a standard collection of folders and data files used to describe almost everything needed to design and analyze circuit cards. As such, they represent the easiest method for creating a Sherlock project. In fact, Sherlock is capable of automatically scanning the archive looking for all the data that is required for analysis purposes.

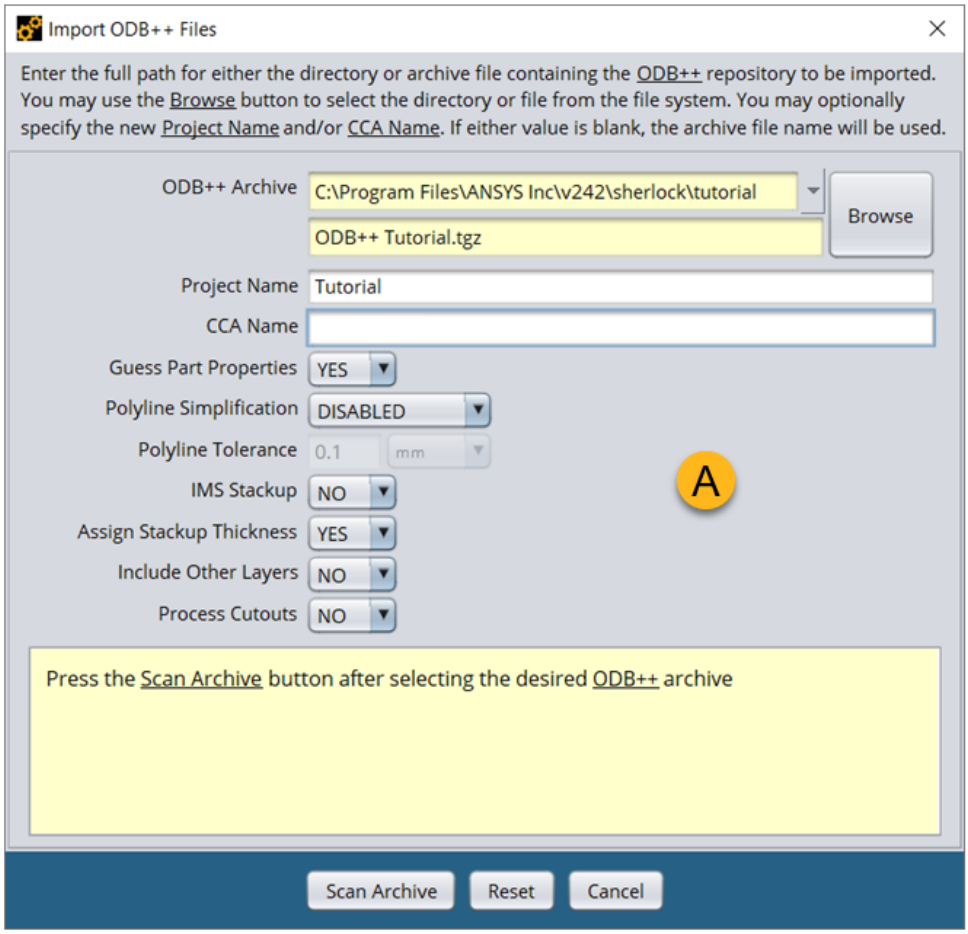
Note:

Sherlock's import feature is compatible with ODB++ version 8.1 and earlier. However, this compatibility only applies to the features and specifications of ODB++ that are utilized in Sherlock.

ODB++ archives are usually transferred as a single compressed file with a suffix of `.tgz` or `.tar.gz` or an uncompressed file with a suffix of `.tar`. It is also possible that the ODB++ archive has already been unpacked into a directory containing various sub-folders and data files. Finally, an ODB++ archive can be formatted as a single XML file, which is called an ODB XML Archive. Sherlock can import ODB++ archives in any of these forms using basically the same process.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To import an ODB++ archive, select the **Import ODB Archive** or **Import ODB XML Archive** option from the **Project** menu, at which point a dialog will be displayed allowing you to select the archive to be imported.



1. Press the **Browse** button to locate the archive file. The ODB++ tutorial archive file is located in the `tutorial` folder of the Sherlock installation directory:

   Windows:

   `C:/Program Files/Ansys Inc/v251/sherlock/tutorial`

   Linux:

   `/ansys_inc/v251/sherlock/tutorial`

   The file name for this tutorial is `ODB++ Tutorial.tgz`.

2. Keep the default settings of the file, as shown above (A):

   - The **Guess Part Properties** option will provide initial values for parts added to a CCA.
   - **Polyline Simplification**: Leave DISABLED. See the section Importing Other File Types (p. 120) for an explanation of this feature.
   - **Polyline Tolerance**: See the section Importing Other File Types (p. 120) for an explanation of this feature.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **IMS Stackup**: For this tutorial, set **IMS Stackup** to **NO**. (The tutorial project does not contain an IMS board.)

  Tip:

  The **IMS Stackup** option affects how Sherlock processes the IMS (insulated metal substrate) stackup data. Typically, if there is an IMS in the imported file, you want to select **YES**.

  – If you select **YES**, Sherlock creates a three-layer IMS stackup whether the import contains an IMS or not. Sherlock ignores the number of layers defined in the imported file.

  – If you select **NO**, it gets imported as a standard stackup, whether the import contains an IMS or not. The number of layers is defined by the data in the imported file.

  Later, you can use Sherlock's stackup editing tools to modify the IMS layers.

  For more information on Sherlock's IMS tools, see the section Insulated Metal Substrate Stackup (p. 68) in the tutorial *Ad Hoc Project Creation*.

- The **Assign Stackup Thickness** option will use the copper and laminate layer thickness values found in ODB++ archive attributes for the given layer. When disabled, Sherlock will automatically assign computed thickness values for each layer.

  Note:

  When importing an ODB++ archive with adjacent laminate files (that is, no copper layer between them), Sherlock combines the adjacent laminate files into a single file and assigns it to that layer of the Sherlock stackup. The thickness of the combined layer will be the sum of the thicknesses of the individual laminate layers.

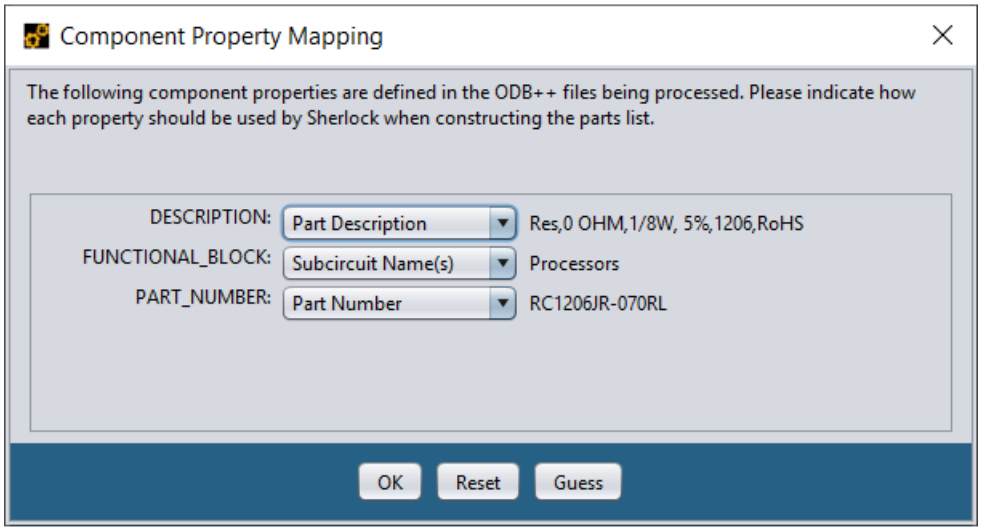
- The **Include Other Layers** option will add all found layers to the project. Layers which are not used directly by Sherlock will be included as **Other Layers** in the Sherlock Layer Viewer.

- The **Process Cutouts** option will attempt to process cutouts from the ODB++ file. These will appear as a cutouts layer in the Sherlock Layer Viewer.

3. After selecting the archive file, press the **Scan Archive** button to determine the types of data that can be automatically retrieved from the archive. After the ODB archive has been scanned, the **Component Property Mapping** dialog will appear.

4. The **Component Property Mapping** dialog, shown below, is an important, but sometimes overlooked, dialog that allows you to map various components/part properties included in the ODB++ archive to corresponding Sherlock property fields. Sherlock automatically attempts to match appropriate fields when they are found, but most CAD tools allow circuit card designers to create their own property names and fill them with arbitrary data, so it is not always possible to determine which field should be used.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

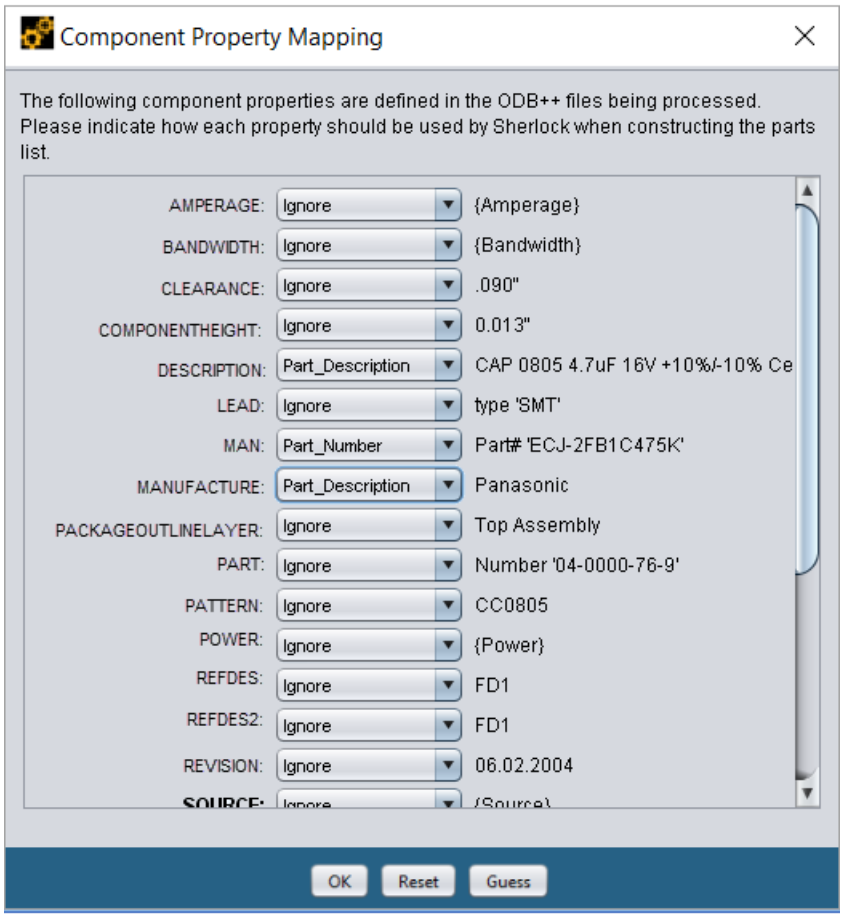
In this tutorial archive, only three properties are defined, and the names are easily matched with Sherlock property names. In a real-world archive, there can be dozens of property names, in which case you'll need to determine which, if any, should be mapped to Sherlock properties.

The bold-face label is the property name defined in the ODB++ archive (for example, **DESCRIPTION**). The pull-down choice list shows the Sherlock property names that can be matched (for example, **Part Description**) and the right-side notation shows example data pulled from each of the property fields found in the ODB++ archive (for example, Res, 0 OHM, 1/8W, 5%, 1206, RoHS).

You may map the ODB++ properties to Sherlock properties however you think is best. In fact, you can map multiple ODB++ properties to the same Sherlock property if you can't decide which is best. In such cases, all the ODB++ property values assigned to the same Sherlock property will be concatenated, separated by commas.

For example, consider the following set of ODB++ properties found in another archive, as shown here.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



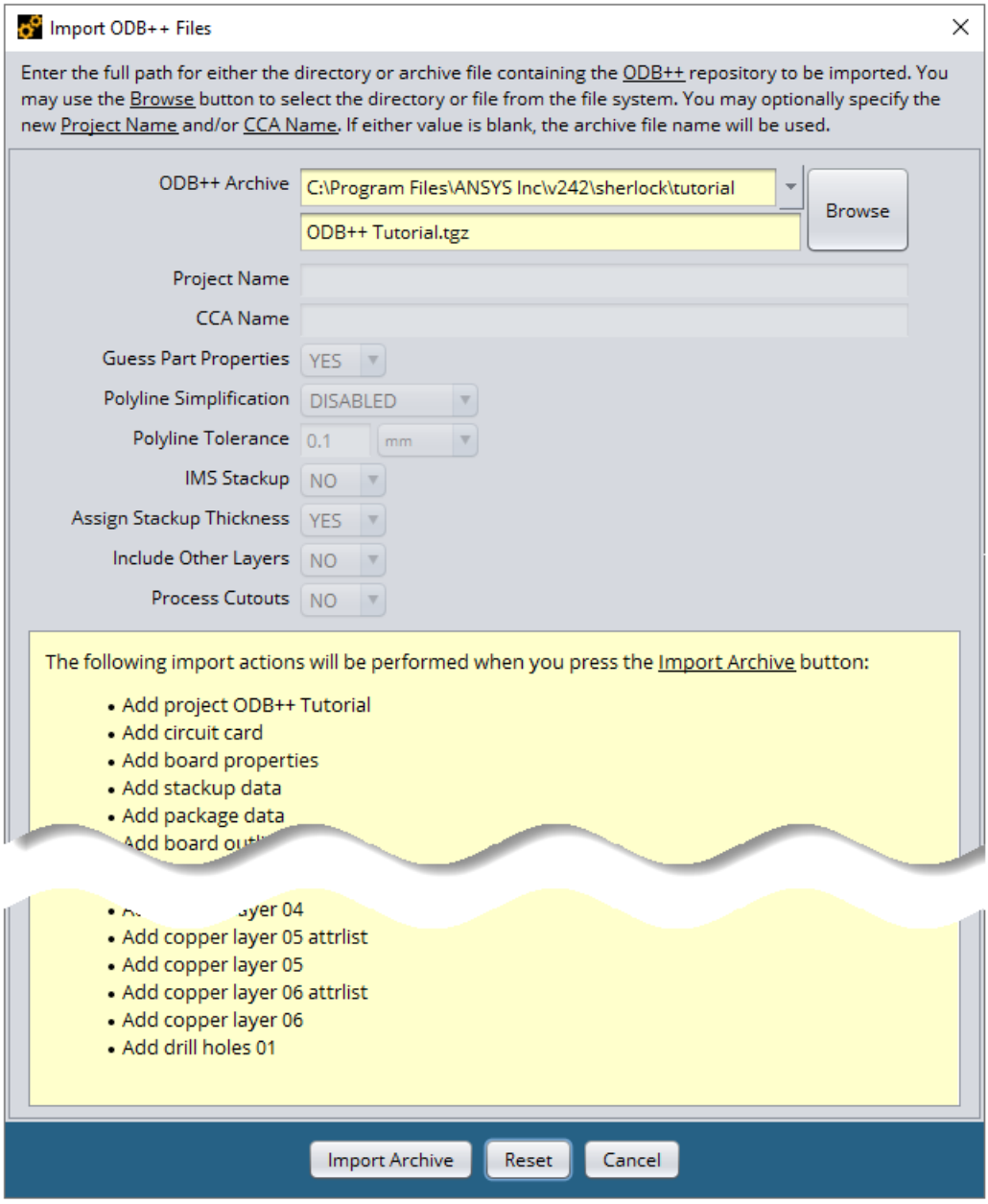
In this case, 24 different properties are defined in the ODB++ archive. Sherlock automatically mapped **DESCRIPTION** to "Part_Description" but didn't recognize the other properties. It decided that the **MANUFACTURE** property is also valuable, so the user assigned it to the **Part_Description** property as well.

After scanning the list of properties, the user found two different properties that could be used as part numbers, the **PART** and **MAN** properties. The **MAN** part number looks more useful than the other, so he assigned that ODB++ property to the Sherlock **Part_Number** property.

5. After you've reviewed the various properties and have decided that they are mapped appropriately, press the **OK** button to continue the import process.

6. After the part properties have been mapped, the **Import ODB++ Files** dialog will display the complete list of data that can be extracted automatically from the archive, as shown here:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this typical case, Sherlock has found a complete set of files, including package information, stackup data, board outline, drill holes and all the important layers. In some cases, various files may be missing from the ODB++ archive. Although the lack of such files may prevent certain types of analysis, Sherlock will still import whatever it can. Missing files can be added subsequently or a more complete ODB++ archive can be re-imported later without losing changes made in Sherlock.

Note:

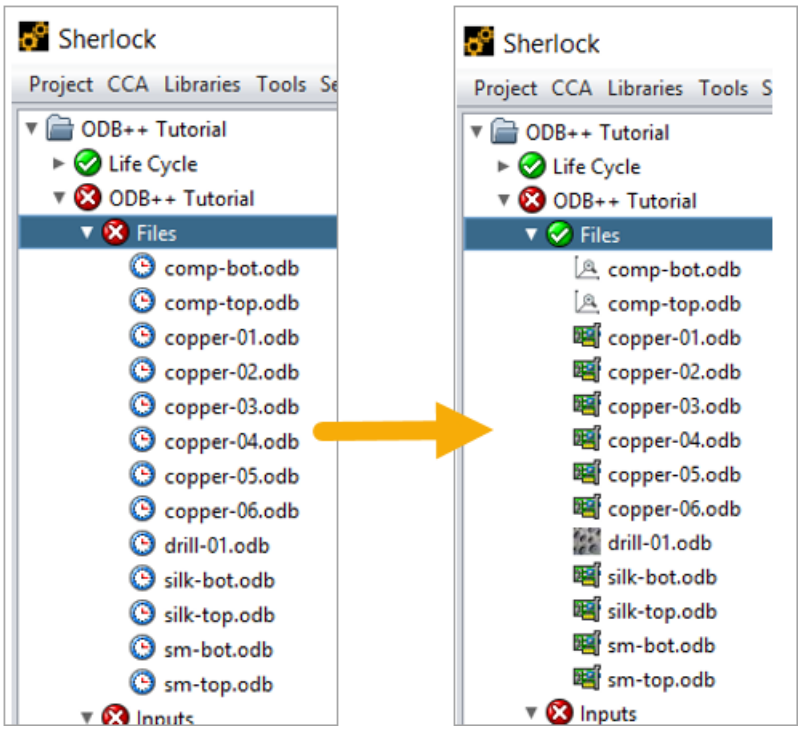
ODB++ archives are generated from sophisticated CAD tools. If you're given an ODB++ archive that is missing some key files, you may want to ask the person who generated the file to generate another ODB++ archive containing the necessary data.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Press the **Import Archive** button to start the import process, at which point the import dialog will close and the project navigation tree will be updated to include all the files being imported.

   Left Image (below): While each ODB++ file is being imported and processed a clock icon will be displayed next to the file name in the navigation tree.

   Right Image (below): When all of the ODB files have been successfully imported the clock icons will be replaced by an icon representing the board type. If any errors were generated while processing the files, then the file will be highlighted with an orange **"?"** icon.



At this point, all the useful data from the ODB++ archive has been imported into a Sherlock project.

2.2.2. Project Name and Properties

By default, the project name used during the ODB++ archive import is the same as the currently selected project in the Project Tree. This allows an ODB++ archive to be imported as another circuit card into an existing project. If no project is currently selected, or to import the ODB++ archive as a new project, clear out the project name in the import dialog. When empty, the ODB++ archive name will then be used as the project name. If you'd like to change the project name after it has been created, right-click the **Project Folder** and select **Project Properties** to change the name and enter an optional description.

By default, the ODB++ archive name is used as the circuit card name. You may also right-click the circuit card folder and select **Circuit Card Properties** to change the circuit card name after import, as well as other circuit card properties (to be discussed in another lesson).

2.3. Tutorial: Reviewing Part Properties

In previous lessons, we've seen how Ansys Sherlock streamlines the process of importing circuit card and part properties from various types of CAD files. Such automated processes significantly decrease

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the amount of time needed to prepare for circuit card analysis, but there is one important step that requires careful attention, namely reviewing all the part properties imported from the CAD files. Even if Sherlock successfully imports all the data from the CAD file, it is still very important to review all of the components in the Parts List to ensure that the information is accurate. Many properties contained in CAD files are simply passed along as whatever string values were entered by the user. Sometimes those values are correct and sometimes not. Since PCB designers and manufacturers don't always use those properties, they may not be maintained properly.

Sherlock relies on several critical properties, such as package names and descriptions in order to make educated guesses about the parts being analyzed. It has several built-in heuristics designed to extract as much information as possible from such data fields, by comparing what it finds to a collection of internal databases in an attempt to standardize property values whenever possible. Unfortunately, the task is not always possible, especially when key data is missing from the CAD files themselves, so it's your responsibility to review the part properties and correct them as needed to ensure accurate analysis results. This section describes the mechanisms provided by Sherlock to streamline this verification process and provide examples of some common data modifications.

Sherlock analysis tasks can be run at any time, with or without correct part properties, but the accuracy of the results depends on the accuracy of the properties themselves. So, don't be afraid to work in a cyclic fashion: review properties, run analysis, update properties, run analysis, etc. There's no harm in doing a quick and dirty analysis immediately after importing the CAD file, but just remember that the results may not be as accurate as possible. As you refine the part properties, the analysis results will become more accurate.

Note:

Sherlock doesn't require an extensive data input phase before starting analysis. Some analysis tasks may not run without key inputs, but for the most part Sherlock will make assumptions and select defaults so that you can get started quickly. As you refine the inputs, the analysis will be more accurate for your project.

In this section, the following topics are covered:

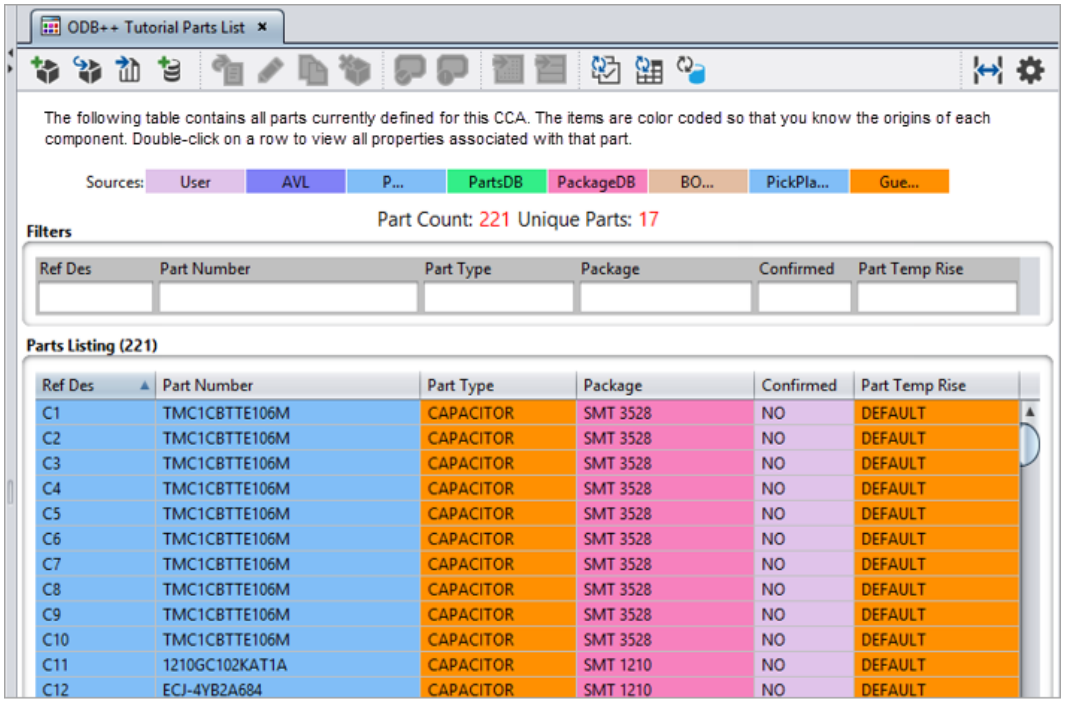
2.3.1. Managing Part Properties
2.3.2. Filtering and Selecting Parts
2.3.3. Changing Part Properties
2.3.4. Part Source Priority
2.3.5. Changing Properties for Multiple Parts
2.3.6. Confirming Parts
2.3.7. Updating Component Layers

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.1. Managing Part Properties

Regardless of how the data was initially obtained, all part information maintained by Sherlock is contained in an internal **Parts List**. The properties contained in the **Parts List** are used as input to all analysis tasks and are used to generate the graphical displays associated with the circuit card. Part properties span a wide range of characteristics, including size, location, orientation, identification, electrical, and quality. Such a wide range of characteristics is not often found in one set of CAD files, so Sherlock combines properties from multiple sources and keeps track of the various sources for each part property to ensure that it always has the most accurate information to be used as input to the analysis tasks. The data you enter manually is added to this mix as needed to correct any deficiencies with the data extracted from CAD files. So, let's take a look at the **Parts List** obtained by importing the ODB++ Tutorial archive.

Double-click the **Parts List** entry in the project navigation tree to display the **Parts List** maintained by Sherlock for this circuit card, as shown here:



Note:

The columns shown in the **Parts List** above might be different than the columns you currently see in the Sherlock application. This is normal. You can add and delete columns to suit your needs. Click the ⚙ icon in the upper right of the table to customize what columns you want Sherlock to display. Refer to Customizing the Parts List (p. 191) for additional information.

For the Sherlock tutorials, make sure the following columns are displayed: **Ref Des**, **Part Number**, **Part Type**, **Package**, and **Confirmed**.

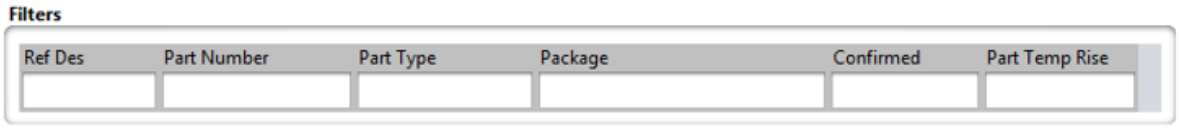
The **Parts List** shows a tabular summary of all parts contained in the circuit card. Each of the high-level part properties are color-coded to indicate where the data was obtained, and an icon is used to indicate the validity of the data. For example, in the **Parts List** shown above, the **Ref Des** (reference

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

designator), **Part Number** and **Location** information for each of the capacitors listed was obtained from the Pick & Place data extracted from the ODB++ archive, as denoted by the blue cell color. The **Package** information was obtained from the **Package Database**, as denoted by the magenta color. **Part Type** information was guessed by Sherlock as indicated by the orange color. In all cases, the information is known, but the part has not been **Confirmed** by the user. That is, Sherlock can start analyzing the data, but the analysis will not be considered complete until you have verified that the information is indeed correct.

2.3.2. Filtering and Selecting Parts

Before working with the **Parts List**, there are a couple of user interface features that will make life easier. Directly above the **Parts List** itself is a row of six **column filters** that can be used to quickly prune down the list of parts being displayed.



If a value is entered into one of the filter boxes, then only those parts with a cell entry matching that value will be displayed. For example, if you enter Q into the **Ref Des** filter and press **Enter**, then only parts starting with a reference designator of Q will be displayed.

If values are entered into two or more filter boxes, then only those parts that match ALL of the filter conditions will be displayed. For example, if Q is entered into the **Ref Des** filter and SOT is entered into the **Packaging** filter, then only parts Q1 through Q8 will be displayed.

Note:

See the Part List Management (p. 197) user guide section for a complete description of the filtering capabilities provided for the **Parts List**.

The other convenient user interface feature is the ability to select one or more parts from the **Parts List** and then modify all selected parts. Selections are made using the following standard mouse conventions:

- Left-click the mouse on any row in the **Parts List** to select a single part
- Left-click any row and drag the mouse up or down to select a range of parts
- Shift-Left-click to select all rows between the previously selected row and the current row
- Control-Left-click to toggle row selection
- Double-click any row to view all properties associated with the part

The **Parts List** provides various ways for viewing and modifying part properties, allowing users to find the mechanism best suited to their tastes and purposes. In the following exercises, we'll make use of each of these mechanisms so you can decide for yourself which works best for you.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

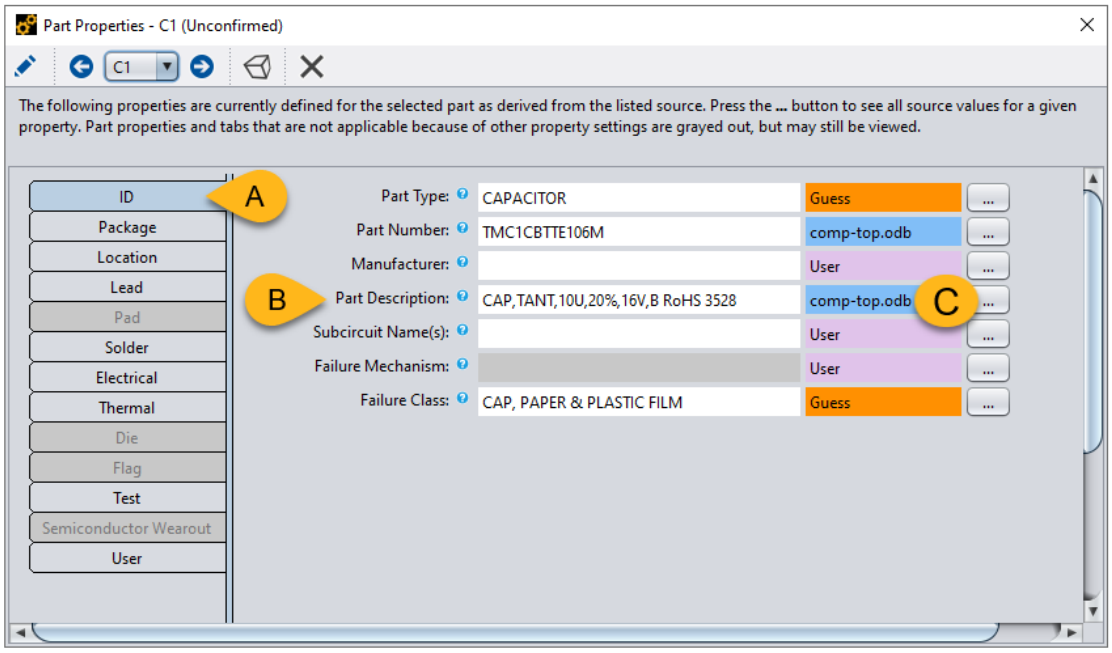
2.3.3. Changing Part Properties

One of the most common changes made in the **Parts List** involves the package designation assigned to each part. This is necessary because of the large number of standard package types used in the industry and because such information is not always correctly defined in the ODB++ properties. Typically, the package information is loosely specified in the **Description** property associated with each part. As such, Sherlock attempts to guess the package information, but it isn't always correct.

For example, the **Parts List** currently shows the package designation for capacitor C1 as SMT 3528, but that's not exactly right, so let's correct it. First, double-click the **C1** row to display the **Part Property Viewer**.

2.3.3.1. The Part Property Viewer

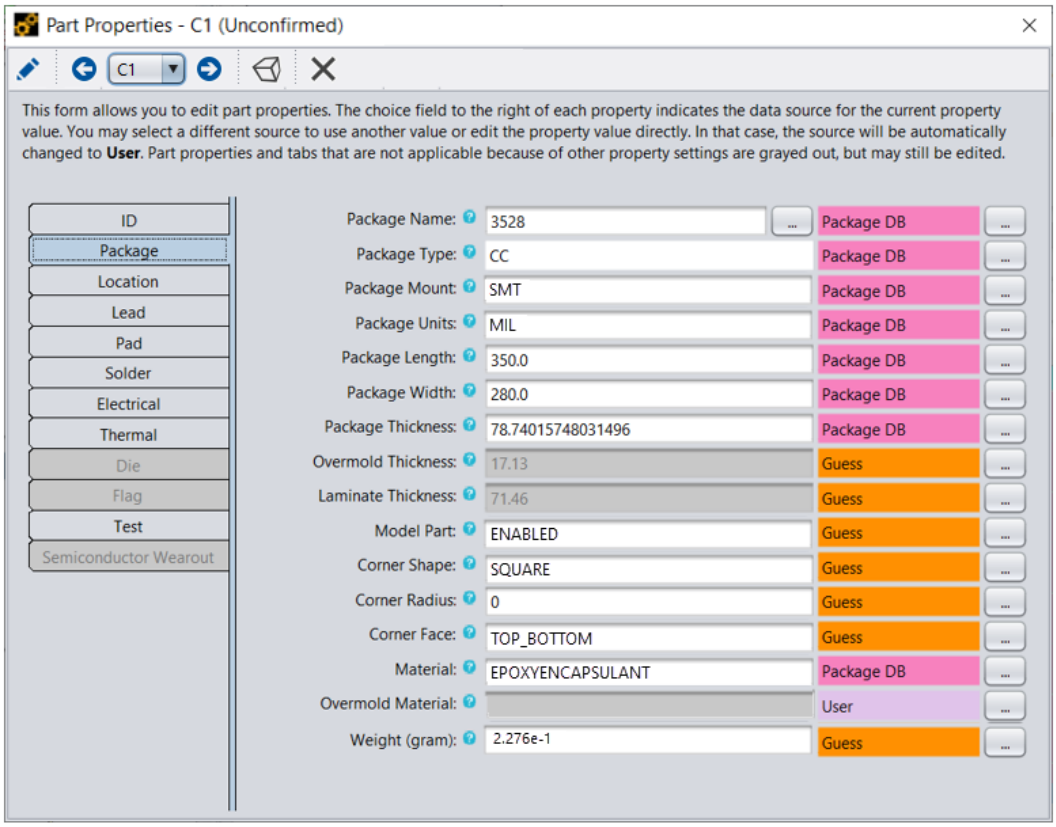
The **Part Property Viewer** displays all the properties currently known for the selected part, with all properties organized by tabs. The Package Type determines which fields and tabs are enabled.



The **ID** shows the extremely valuable **Part Description** property, which is used by Sherlock to guess other part properties. In this case, it is the 3528 designator that Sherlock used as the package designator. The blue box to the right of the description field indicates that the description property itself was obtained from the **comp-top.odb** file extracted from the ODB++ archive, which is denoted as Pick & Place data by the blue color (using the same color-coding scheme used in the Parts List itself).

Select the **Package** tab to view all the package properties associated with capacitor C1.

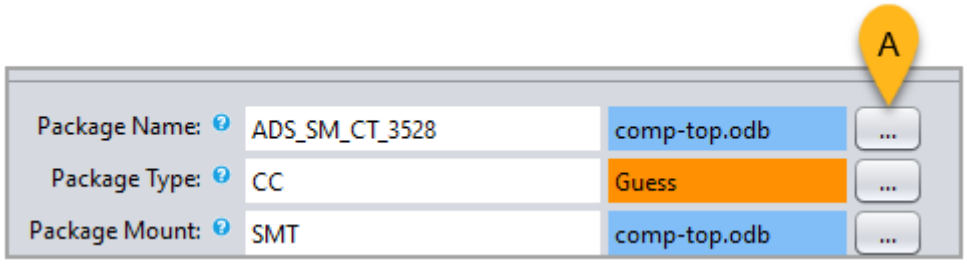
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this case, the properties obtained from the **Package DB** data are shown because Sherlock was able to associate the 3528 from the part description with a chip-component. In a way this was a guess by Sherlock because it didn't have the exact package name but found something close to a package. However, in this case, this is not the correct selection. Note that properties for which no data has been obtained are given a **User** source designation because only you can provide that information at this point.

It is important to note that Sherlock may find multiple values for a given part property from the various data sources processed. In such cases, Sherlock uses predefined priorities to determine which value should be used. You can override this process simply by entering the desired value manually.

For example, select the **"..."** button farthest to the right of the **Package Name** property to show all the values obtained by Sherlock for that property.



As shown in the image above, Sherlock found the custom package type ADS_SM_CT_3528 in the ODB component file. Given the input data found in the ODB++ file, Sherlock's guess is close, but not correct.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set the proper package type, press the **Edit the selected part** icon ( ) located at the top of the or close the **Part Property** and press the **Edit Selected** button at the bottom of the **Parts List**. In either case, the **Part Editor** will be displayed.

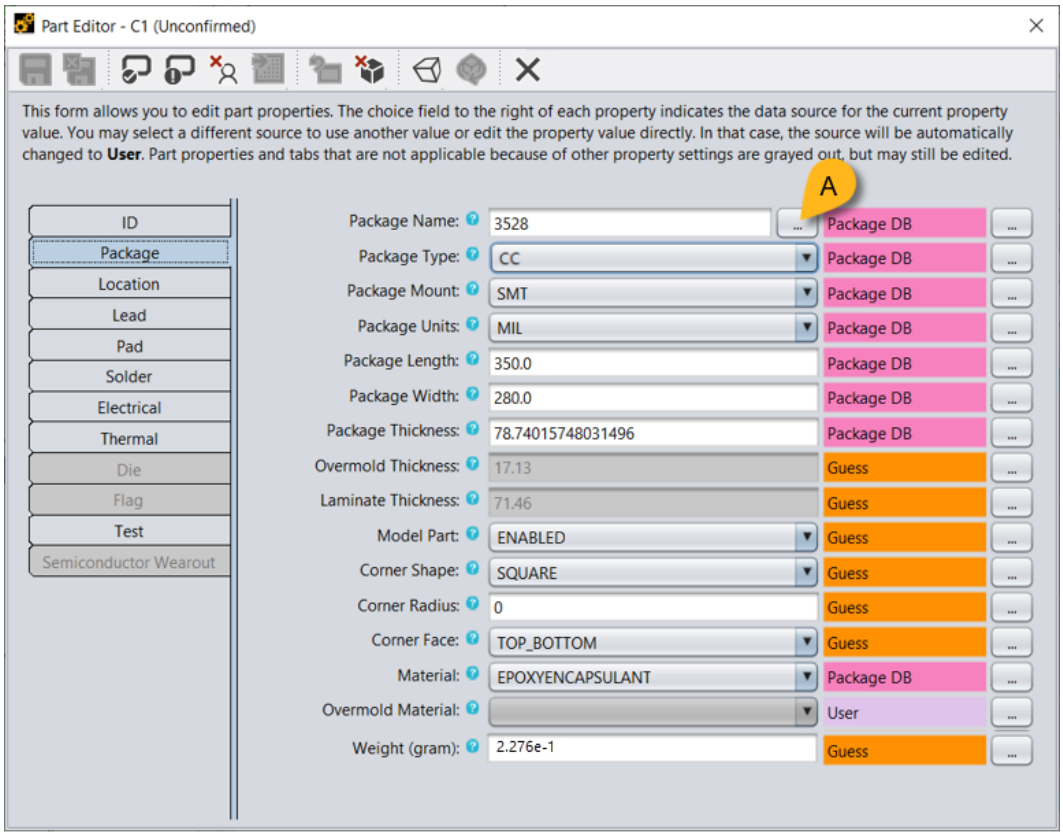
2.3.3.1.1. The Part Editor

The **Part Editor** looks like the **Part Property** , except that it allows you to edit the various property values instead of just viewing them.

Note:

Helpful things to know about Sherlock's **Part Editor**:

- The **Package Type** determines which fields and tabs are active in the **Part Editor**.
- Click the information icon for an explanation of any of the part properties. (An example is shown below, A.)

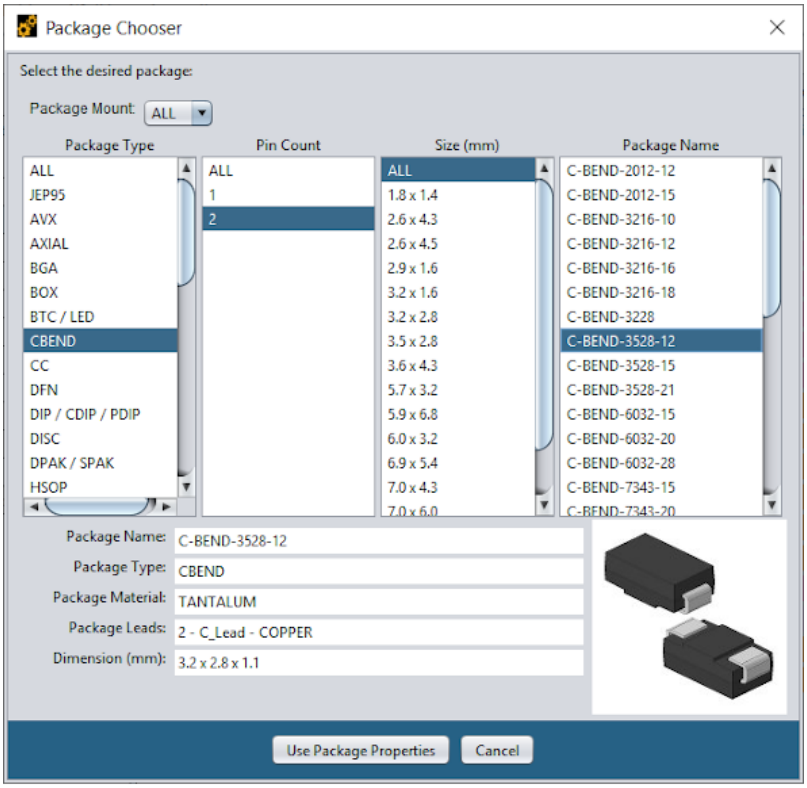


If you know the exact standard **Package Name**, you could simply enter it into the **Package Name Field.** However, given the large number of possible package names, it is easier to press the ... button (A, above) that is immediately adjacent to the **Package Name Field** to display the **Package Chooser**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.3.2. The Package Chooser

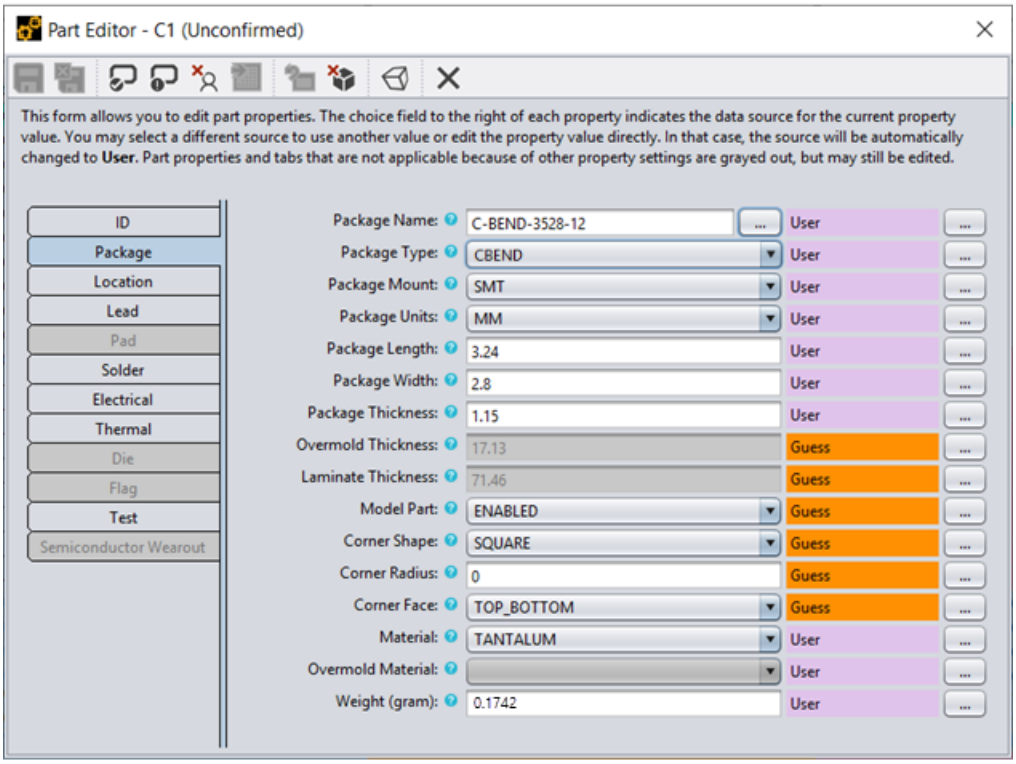
The **Package Chooser** presents a hierarchical list of all standard packages known by Sherlock, organized by **Package Type**, **Pin Count**, and **Size**. In this case, we know that capacitor C1 is a CBEND package type and that there are multiple package names with 3528 in them. By examining the package properties associated with each of those packages, we determine (or guess) that C-BEND-3528-12 is the desired package name. After selecting that name from the list, press **Use Package Properties** to copy all package properties to the **Part Editor**. (See the Package Management (p. 315) section for more information on managing **Packages**.)



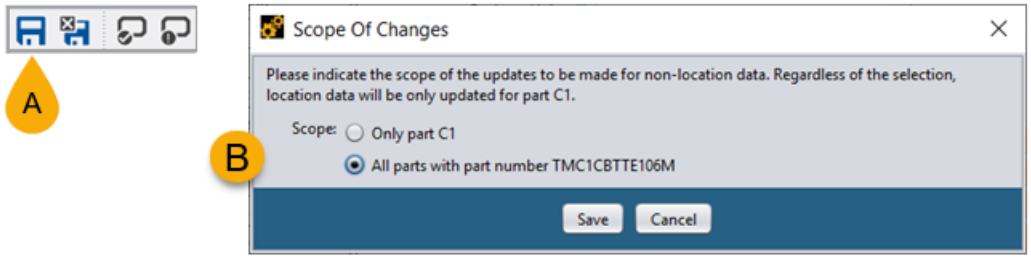
2.3.3.3. Back to the Part Editor

When you return to the **Part Editor,** you will see that not only the **Package Name** value was updated but also several other package properties. Since the user selected the package name and properties, they are all designated as **User** modifications to the **Parts List** for future reference.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Press the **Save the Part Editor Changes** button (A, below) at the top of the **Part Editor** ribbon to apply all property updates and close the **Part Editor** dialog. Since multiple parts have the same Part Number, Sherlock will display the following dialog (B) to allow you to apply the property updates to just the current part or to all parts with the same part number.



In this case, we want to apply the changes to all capacitors with that part number, so we select the second option and press **Save**. The **Scope of Changes** dialog will be dismissed, and the **Parts List** will be updated to reflect our package name change.

Although the first time through the property editing process it might seem complicated, it is a relatively simple process:

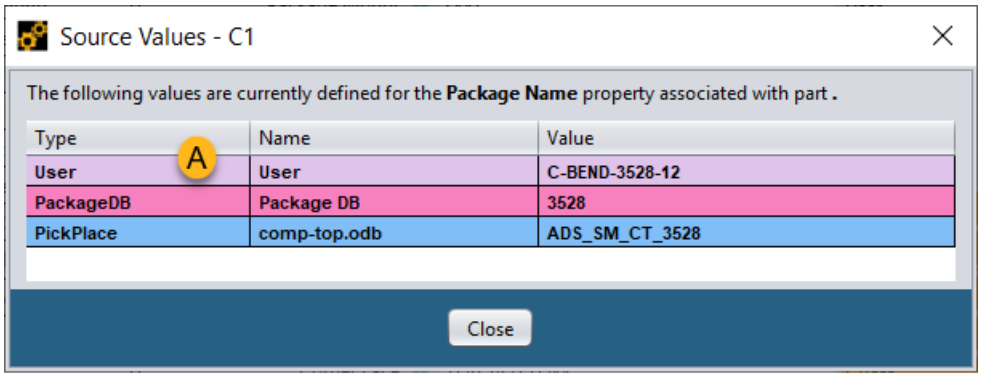
1. Select a part to be updated from the **Parts List.**
2. Update one or more part properties using the various pull-downs and selection dialog windows.
3. Apply the updates to the selected part or all parts with the same part number.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.4. Part Source Priority

As noted in the previous section, Sherlock may have found multiple sources of values for any given part property. These values and sources may be reviewed for any given part property by selecting the **"..."** button located in the **Part Editor** and **Part** dialog windows that is farthest to the right of the desired field. When selected, the **"..."** button will open the **Source Values** dialog for the selected field.

When in the **Part Property** , the **Source Values** dialog is in read-only mode and only a reference to the values assigned to the given part property from the known sources. When the **Part Editor** is active, this dialog may be used to change the value of a part property to that of another source. In the previous section, the package name of part C1 was changed. Opening the **Source Values** dialog window for the package name from the **Part Editor** will now reflect the following:



The values listed in this dialog are ordered by priority. Since we have changed the assigned value, a new source of **User** has been added. **User** source values always receive the highest priority. If for some reason we wish to change to a different value there are a couple of methods for doing this.

The first method is simply to close this dialog and manually enter the desired value for the part property. If for some reason we want to leave the user source in place but want to just select one of the other sources, open the **Source Values** dialog in the **Part Editor** then double-click the row in the table for the source value desired. This will automatically update the **Part Editor** with the selected value and source.

If multiple sources of data are available for a given part property, then it is also possible to remove those sources by right-clicking on the row and selecting **Remove Source Value** from the menu. The table will update. Once the **Save** button on the **Source Values** dialog is selected, then the **Part Editor** may update the selected value if the value just removed was previously selected. This is also the method that can be used to clear user values, also known as user overrides, for a given part property.

User values for all part properties on a given part can be cleared at once by selecting the **Clear Overrides** button in the **Part Editor** for the given part. Any user assigned value to all part properties is removed and the values reverted to the next highest priority value available.

2.3.5. Changing Properties for Multiple Parts

The update method described above started by selecting a single part, modifying part properties and then applying those changes to that part or all parts with the same part number. That is a convenient method for updating multiple parts with the same part number, but you may also need to update

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the same property for parts with different part numbers. For that reason, Sherlock allows you to select multiple parts for updating using a slightly different Part Editor.

Multi-part editing begins by choosing a set of parts from the **Parts List.** For example, if you enter the following Parts List filter values

Ref Des: C

Package: TH

you'll see a list of 5 capacitors, C21 through C25, all of which have the same Package type TH ADS_CPCYL1_D400_LS200_034. The "TH" denotes them as through-hole parts and the package designation, although well understood by CAD designers, is not a standard package type in the **Sherlock Package Database**.

Note:

Through-hole parts always require scrutiny in the Parts List because of the lack of standard package designations, their different shapes and the different ways that CAD tools designate their location. Whenever you create a Sherlock project, you should always examine each of the through-hole parts.

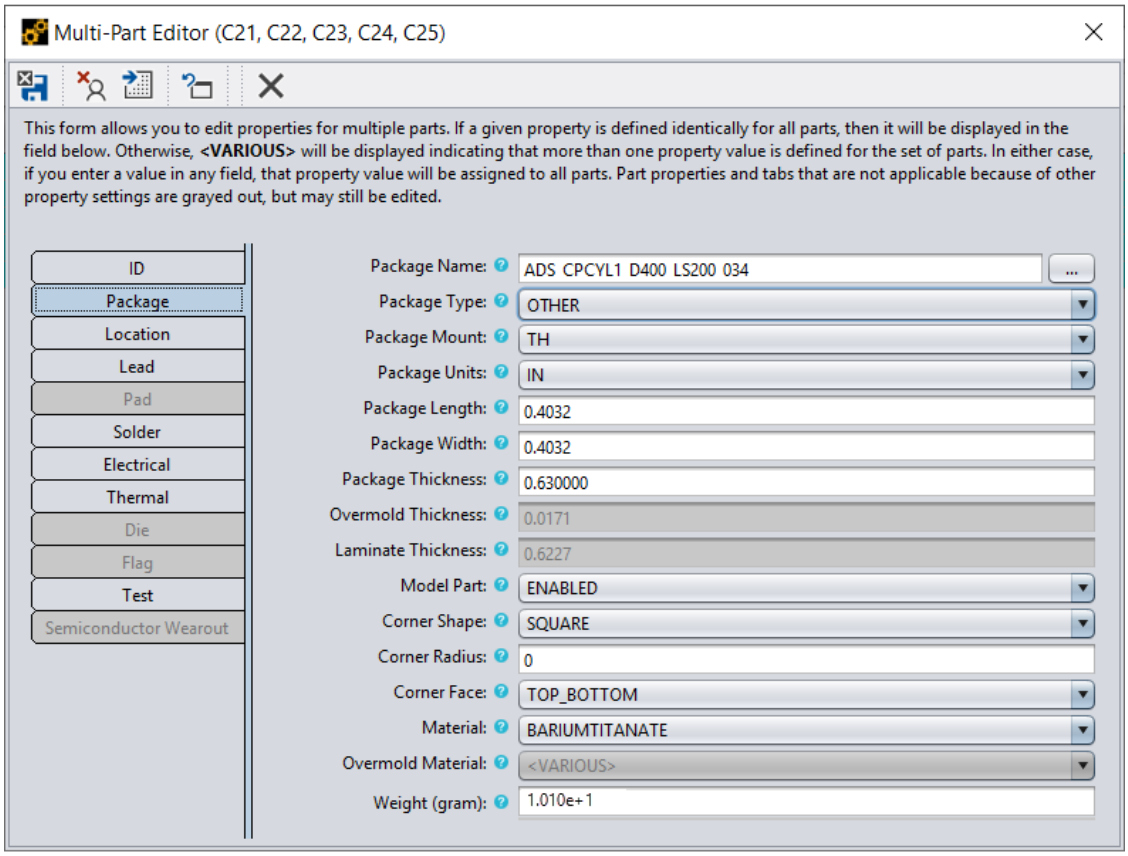
In order to change the package designation for these parts, let's use the multi-part editing method. (Yes, they have the same Part Number, but this is a tutorial). Start by selecting all five rows in the **Parts List** using the mouse, either by:

- Left-click the first row and drag the mouse down the list
- Left-click the first row and then Shift-Left-click the last row *or*
- Left-click any row and press Control-A on the keyboard

To edit properties for all 5 parts, press the **Edit the Selected Parts** button at the top of the Parts List (denoted by a pencil), or Right-Click any selected row and select **Edit Selected Parts** from the mouse menu. In either case, the **Multi-Part Editor** will be displayed, listing all the selected parts as shown below.

The **Multi-Part Editor** is similar to the **Parts Editor**, except for two important differences. First, the Multi-Part Editor does NOT display the source for each property values, because different parts may have different sources. More importantly, the **Multi-Part Editor** displays <VARIOUS> in some property fields to indicate that the values assigned to the specific parts are either different or not defined at all. In this case, for example, all the <VARIOUS> values shown are because those properties are not defined at all, relying on default settings. The other property values, however, are shown, indicating that all 5 parts have the same value for those properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aside from the display differences, the **Multi-Part Editor** is used to update part properties just like the **Parts Editor**.

Since this is a large, through-hole capacitor, you must manually update a couple package properties. You can deduce from the package name provided by ODB++ that it has a 400 mil diameter, which is specified correctly in the **Package Length** and **Package Width** fields. By default, Sherlock assumes all components are square, so you need to update the following properties so that it will be displayed and analyzed as a round part:

Corner Shape: ROUND

Corner Radius: 0.2

Note:

By using a Corner Radius that is half the part length/width, you force the entire part to be round. **Corner Shape** settings of **Miter** and **Round** can be used in conjunction with the **Corner Radius** setting to yield a large variety of different shapes sufficient for most analysis purposes.

With those changes made, press the **Save the Part Editor Changes and Exit the Editor** button to update all five parts. Notice, in this case you are not prompted to determine which parts to be updated. The **Multi-Part Editor** always updates all the selected parts, regardless of their Part Numbers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.6. Confirming Parts

You may have noticed the last column in the **Parts List** labeled **Confirmed**.

The status values of **YES** or **NO** provide a simple to-do mechanism for users when reviewing the Parts List. When a project is initially created and data is loaded from CAD files, all the parts marked as **NO** indicate that the user must review the part and confirm the part properties. As you review the properties and confirm that they are correct, you should select the **Confirm All Part Properties** button at the top of the Part Editor ribbon. From that point on, the part should have **YES** under the **Confirmed** column. You can also un-confirm a part at a later time if necessary.



A = Confirm all part properties.

B = Unconfirm all part properties.

Note:

Disciplined use of the confirmation process will ensure more accurate analysis results.

2.3.7. Updating Component Layers

At this point, you have made several modifications to the **Parts List** and you are ready to start reviewing the various layers associated with the circuit card. However, before you move on there is one last important, but often overlooked, step. In order to see your part property changes reflected in the layer displays, you need to update the component layers. To do this, press the **Update the CCA Layers** button (C, below) located on the **Parts List** ribbon. After the component layers are updated, the buttons will be disabled (indicating that no part changes are pending) and you can move along to view the layers.



Note:

We can't remind you too many times: **If you modify something in the Parts List, be sure to press the UPDATE CCA LAYERS button before moving on.** Otherwise, when you start looking at the component layers and analysis results, you might not see what you expect to see. If there are no layers to be updated, the icon will be greyed out.

Sherlock will automatically update the component layers if you attempt to close the **Parts List** display or Sherlock itself after making one or more changes. In that case, a progress dialog will be displayed indicating the processing being performed. If, for any reason, you want to update the component layers directly, right-click the **Parts List** entry in the **Project Tree** and select the **Update Layers** menu option.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4. Tutorial: Layer Viewer

Chapter Sections:

2.4.1. Layer Viewer Overview
2.4.2. Launching the Layer Viewer
2.4.3. Reviewing Layers
2.4.4. Layer Panning and Zooming
2.4.5. Component Editor
2.4.6. Board Outline Editor
2.4.7. Other Layer Viewer Features

2.4.1. Layer Viewer Overview

The **Parts List** contains lots of details that are best displayed in tabular form and processed using tabbed forms, but circuit cards are intrinsically graphical, as are the analysis results associated with them. As such Ansys Sherlock provides a sophisticated Layer Viewer with a collection of graphical tools that can be used to review, analyze, and update circuit card information. This section describes how the Layer Viewer can be used to review and modify the circuit card information imported directly from CAD files in preparation for analysis. In other lessons, you will see how the Layer Viewer is used to modify and review other types of information about the circuit card.

The Layer Viewer consists of a separate window that provides its own menu bar, control and content panels. There is only one Layer Viewer per circuit card, but you may display one or more Layer Viewers for different circuit cards in order to compare them as needed. You may also display the Layer Viewer side-by-side with the main Sherlock window in order to compare content panels in the main window, such as the **Parts List**, with the graphical information displayed in the Layer Viewer.

The Layer Viewer includes a list of available layers on the left side of the display grouped into folders of similar items. The layers included may change depending on which analysis results are available. The only layers displayed in the main window of the layer viewer are those which are selected with from the checkbox and from which the folder is expanded. If a layer is selected but the folder is not expanded, the layer will no longer be displayed. For folders which have many layers, the ability to quickly select or unselect all layer items exists by right-clicking the folder menu item and selecting **Select All** or **Unselect All.** Folders are expanded and collapsed with a single-left-click of the folder or from the appropriate menu option by right-clicking the folder.

Note:

Unless otherwise noted, this lesson makes use of the ODB++ Tutorial (p. 23) project as created in previous lessons.

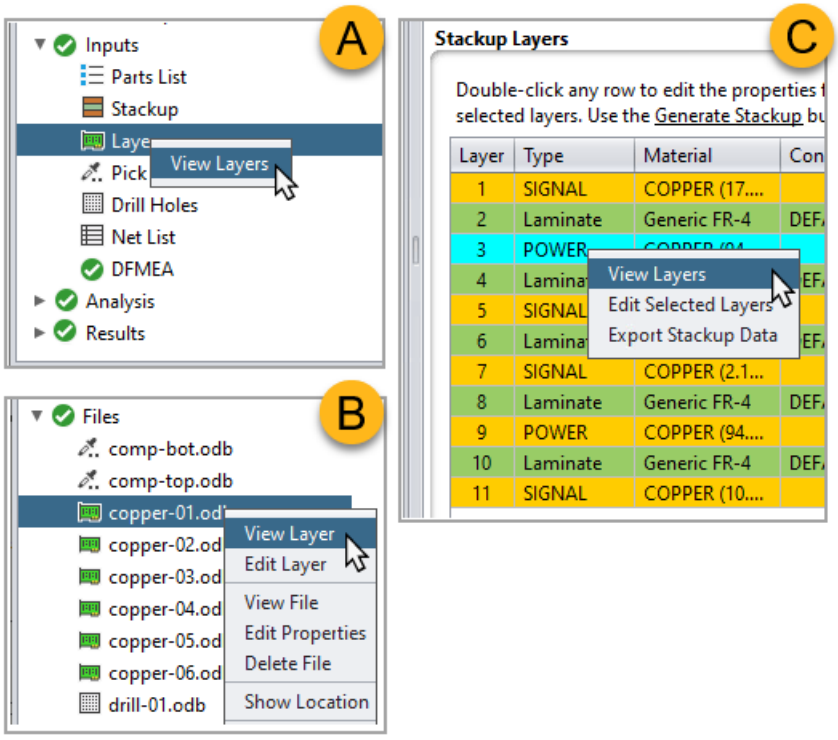
2.4.2. Launching the Layer Viewer

The Layer Viewer can be launched in several ways:

- The most direct method is to double-click the **Layers** entry located in the **Inputs** folder for the circuit card being reviewed (A, below).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

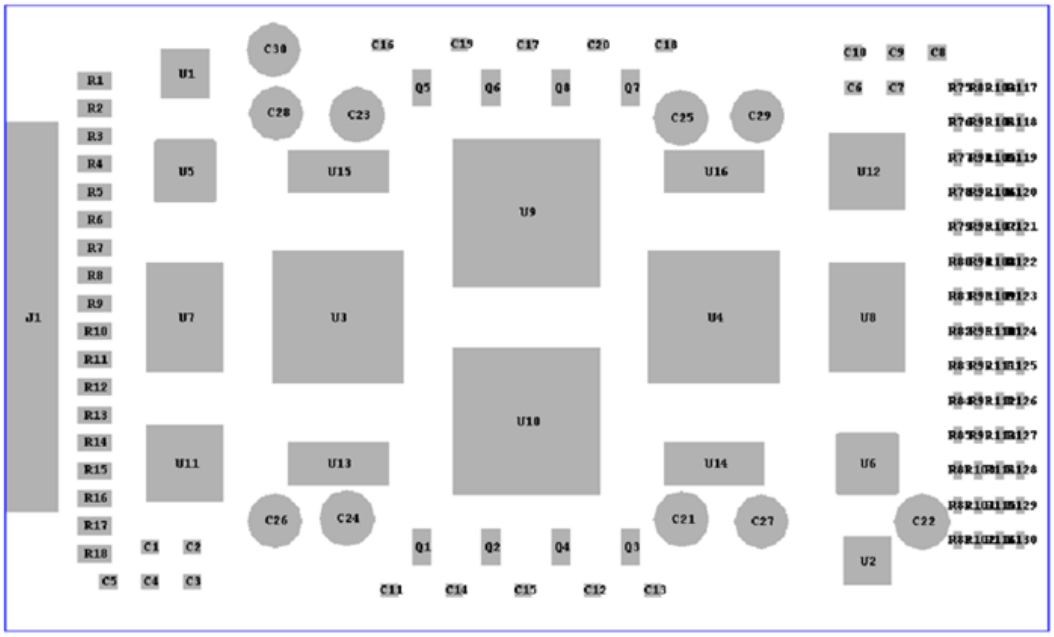
- You can also double-click any of the CAD files listed in the **Files** folder to open the Layer Viewer with specific layers pre-selected for display (B).
- In the **Stackup Layers** table you can right-click a layer and select **View Layers** (C).



2.4.3. Reviewing Layers

To start the review process, double-click the **Layers** entry in the **Inputs** folder of the **Project Tree** to launch the Layer Viewer. If the **Components** folder in the **Layers Tree** located on the left-side of the window is not expanded, click the **Components** folder to expand its contents. At that point, the top and bottom component layers will be displayed. Likewise, if the **Other Layers** folder is not expanded, click the **Other Layers** folder to display the board **Outline**. To reduce a little of the clutter, deselect the **comp-bot** layer so you can focus on the top components. The remaining layers should look similar to this:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Component labels are displayed by default, but sometimes they get in the way (especially for small parts on complex circuit cards). For that reason, the **Components** and **Labels** feature selections are provided in the top left-hand corner of the Layer Viewer. You can toggle those check boxes at any time to hide/display the component outlines and labels according to your processing needs. For now, keep both the component outlines and labels visible so you can review all the parts.

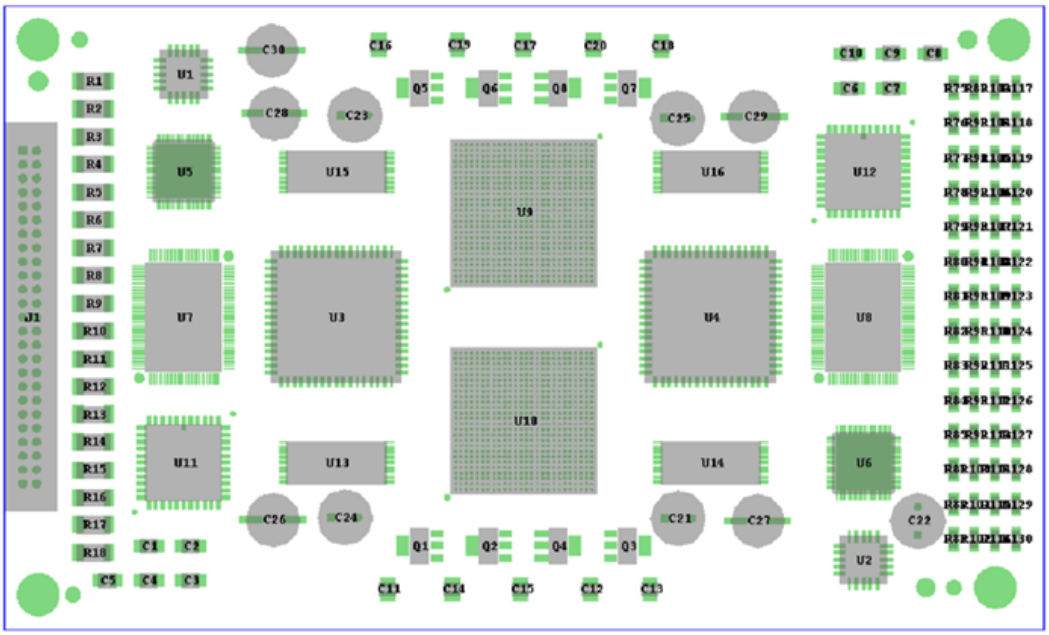
Note:

Flexibility is indeed a double-edged sword. Sherlock provides various controls to allow you to hide graphical components so you can focus on other things. But you must remember to un-hide things when you're done. **If you think something is missing from the display, double-check the filter settings.**

The three most common properties to review in the Layer Viewer are part size, location and orientation. Most CAD files do a very good job keeping these properties consistent but depending on which package types are assigned in the **Parts List**, some parts can have issues. The easiest way to check for all three of these issues is to compare the component layer outlines to another layer, such as the corresponding Solder Mask.

To start the review process, single-click the **Solder Masks** folder in the **Layers Tree** and then de-select the sm-bot.odb layer, if it is selected, then select the top solder mask layer as shown here:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The first thing to notice is that all your parts are basically in the right place and appear to be the right size. But, first looks can be deceiving, so poke around a little and see what you find.

2.4.4. Layer Panning and Zooming

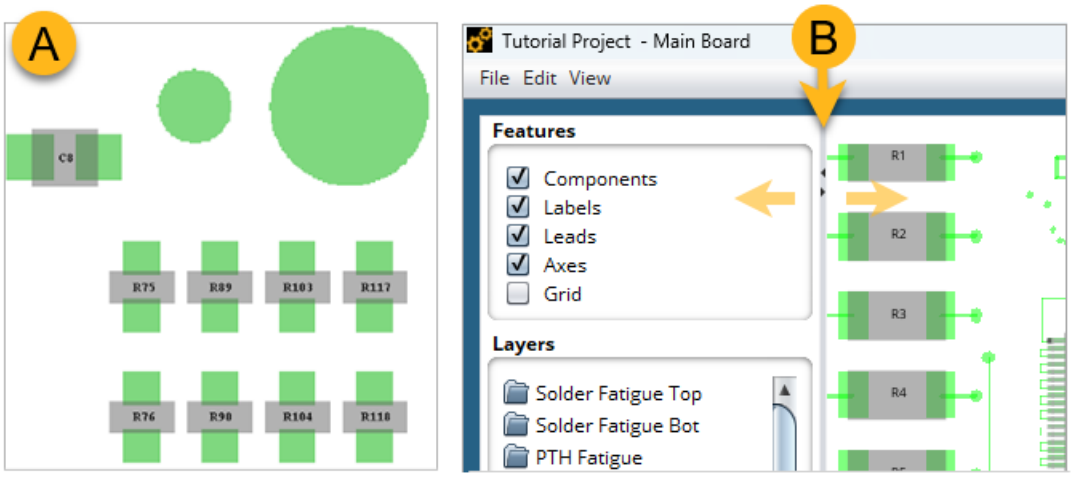
Let's start by zooming in on the collection of SMT resistors located along the right-side of the circuit card. To zoom into that region, you can double-click anywhere on the circuit card, press the **Zoom In** icon (A, below) in the bottom right-hand corner of the window, or use your mouse scroll wheel.



You can pan across the layer by left-clicking the mouse anywhere in the layer viewer and dragging the mouse in any direction. The layer will move inside of the viewable window.

With a little practice you should be able to zoom around to any part of the circuit card, such as the upper right-hand corner as shown here (A).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tip:

If you wish, you can add an adjustable divider to the Layer Viewer (B, above) which separates the list of filter options on the left and the graphical display on the right. To activate this feature, ENABLE **Layer Viewer Divider** under **Advanced Settings** in **Sherlock Settings**. See Advanced Settings (p. 78).

After zooming in, you can see that the orientation of the SMT resistors is off by 90 degrees. This is often the case with chip components because some CAD tools use vertical representations of the components, while others use horizontal. The **Sherlock Package Database** assumes a horizontal orientation, so sometimes there is an orientation mismatch. Fortunately, such issues are easily resolved in Sherlock.

2.4.5. Component Editor

To make changes to one or more components, select **Edit Components** from the **Edit** menu located at the top of the Layer Viewer window. At that point, the **Component Editor Mode** will be indicated along the bottom of the Layer Viewer and five buttons will be displayed. As long as the Component Editor is active, you can select one or more components, edit them graphically or edit their part properties, and then update the **Parts List**.

Note:

The buttons shown at the bottom of the Layer Viewer window during edit mode may be hidden if the window is too narrow. If that happens, you'll need to maximize the window or increase its size while editing.

In order to make changes in the **Component Editor** you must first select one or more parts to be changed. Part selection is done graphically using the following set of mouse/key operations:

- SHIFT-Left-click any part to select only that part.
- SHIFT-Left-click and drag the mouse to select all parts within a rectangular region.
- Control-Left-click any part to toggle selection of that part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Control-Left-click and drag the mouse to toggle selection of all parts within a region.
- SHIFT-Double-Left-click any part to select only that part and display part properties.
- Right-click outside all selected parts to clear the current selections.

Part selection takes a little practice, but you'll soon become comfortable with the selection process.

2.4.5.1. Editing Part Orientation and Location

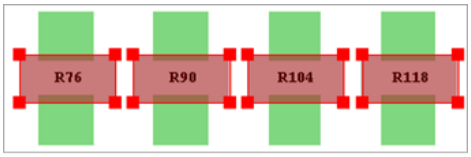
Let's perform a basic graphical edit operation by rotating one of the resistors by 90 degrees.

Select part R75 by left-clicking with the mouse anywhere inside of the part outline (the gray box) while holding down the SHIFT key.

Then, right-click anywhere inside the selected region to display the **Edit Menu**. Scroll down and select **Rotate 90** to change the part orientation, as shown here.



To change the orientation of multiple resistors, left-click and drag the mouse over the part outlines, while holding down the SHIFT key. After all desired parts are selected, right-click in any of the selected parts to display the **Edit Menu** and select **Rotate 90** to rotate all selected parts.



To change the orientation of all the resistors on the right-side of the circuit card, zoom out so that they all appear on the display and then select all of them by pressing the SHIFT key and dragging the mouse over all of them. Select **Rotate 90** from the right-mouse menu to change their orientation.

Note:

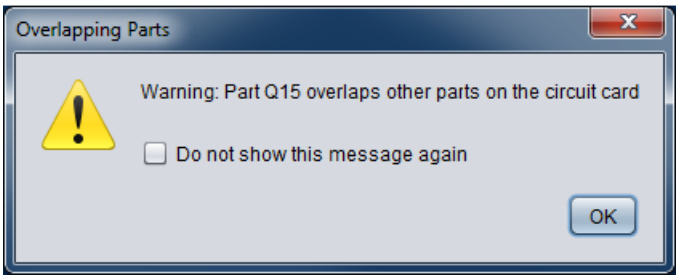
We've rotated parts a couple different ways, so you may have one or two out of phase with the others. If so, select them individually, **Rotate 90** and fix the problem.

When you're done editing the parts, right-click anywhere in the layer viewer to clear the selections and then press the **Apply** button at the bottom of the Layer Viewer to update the Parts List and continue editing parts.

2.4.5.2. Overlapping Parts Warning

When you clicked the **Apply** button previously, you may have seen a warning dialog appear:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This warning is generated by Sherlock when updating parts in the Layer Viewer and Sherlock has detected that some parts may overlap each other on the PCB. You may use the information here regarding the parts listed to investigate and make changes if necessary. If you never wish to see this warning dialog again, select the check box in the dialog before selecting the **OK** button.

For the purposes of this tutorial simply check **OK** as this part should be corrected by the time the tutorial is completed.

2.4.5.3. One-Click Editing

If you only want to edit a single part using one of the **Edit Menu** functions, you can combine the **select** and **modify** operations by **right-clicking** inside of the part outline and selecting the edit operation. The right mouse button automatically selects only the designated part and applies the modification only to that part.

2.4.5.4. Editing Part Properties

While reviewing the component layers, it is often necessary to view or edit part properties to resolve an issue. The **View Properties** and **Edit Properties** menu options are provided for such situations. For example, a closer examination of parts U5 and U6 indicates that the size may be slightly off. The parts cover the corresponding solder mask regions, but perhaps a little too much. Let's investigate and see if something must be changed. Start by right-clicking U5 and selecting **Edit Properties** from the edit menu.

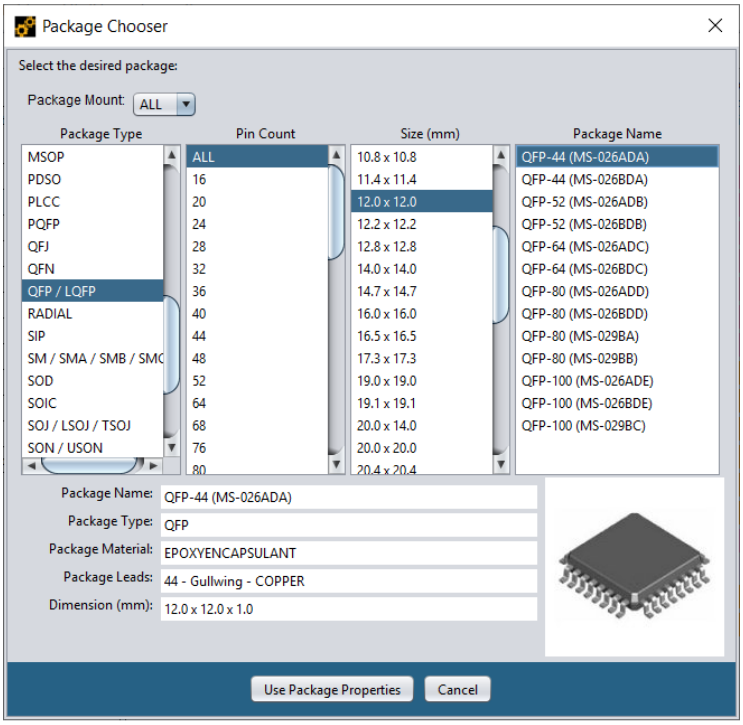
Note:

If you made any pending changes to the layer, you will be instructed to apply all such changes when the **Edit Properties** option is chosen. In that case, press the **Apply** button at the bottom of the Layer Viewer and then re-select the **Edit Properties** menu option to proceed.

The **Edit Properties** menu option displays the same **Part Editor** that you used to edit part properties in the **Parts List**, thereby allowing you to conveniently change any part property. In this case, select the **ID** tab to examine the Part Description value, which contains QFP-80, and then the **Package** tab to examine the Package Name value, which is QFN-80 (MO-220VRRD). The package ID indicates that U5 is a QFP, but the Package Name and Type incorrectly indicate this part as a QFP.

Select the **"..."** button immediately to the right of the **Package Name** value to display the **Package Chooser** shown below. Then choose QFP as the package type and look for the correct **Package Name**. The list of QFP-80 packages shows that you may have to do some research outside of the information CAD files.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



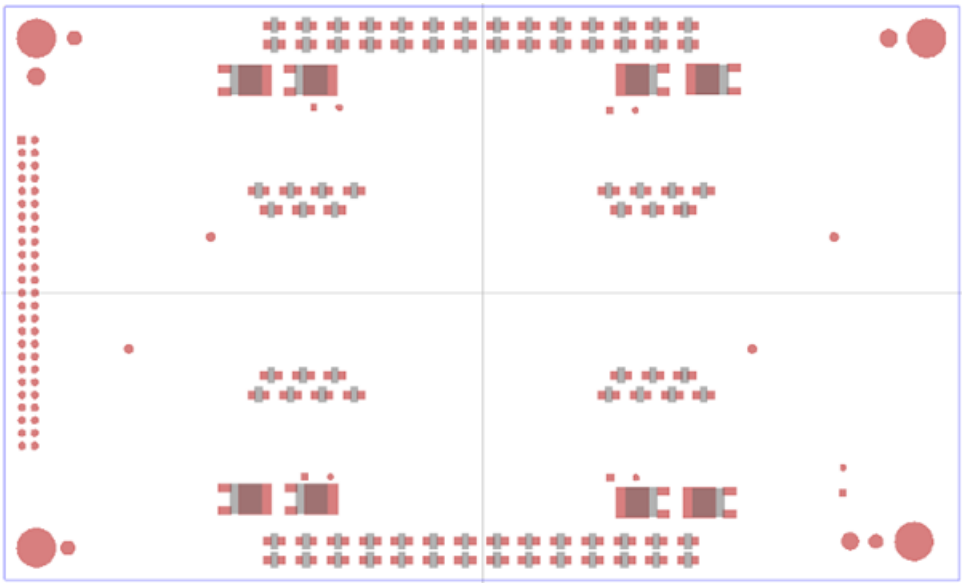
Specifically, QFP-80 packages come in various sizes. Sherlock attempts to select the proper size based on the package dimensions provided in the CAD file, but that's not always accurate. In fact, the dimensions provided by the CAD tool represent the overall footprint of the part, not the actual package size. For analysis purposes, Sherlock needs the package size, but that it can't always be determined automatically from the CAD dimensions. In this case, parts U5 and U6 are 12mm square components, so you need to select “12.0 x 12.0” from the **Size** list and the first package name displayed for that size in the **Package Name** list. Press **Use Package Properties** and **Save** to update both parts. The Layer Viewer display will be automatically refreshed showing that the parts now lie inside of the solder mask regions, as desired.

2.4.5.5. Graphically Filtering and Selecting Parts

With all the top components reviewed, switch your attention to the bottom components as follows:

- De-select the **comp-top** and **sm-top.odb** layers
- Select the **comp-bot** and **sm-bot.odb** layers
- De-select **Labels** to see only the component outlines and solder masks

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



As with the SMT resistors found on the top side, you can see that all the resistors on the bottom side are also rotated by 90 degrees. You want to select all those resistors and then apply the **Rotate 90** function to fix the problem. You've already used one method for selecting groups of parts, namely pressing the **Control key** and dragging the mouse over each group of resistors to add them to the selected group. However, there is another useful approach that combines label filtering and mouse selections to streamline the process.

In the lower left-hand corner of the Layer Viewer is a panel titled **Filters**, which includes the **Label** filter. To see how it works, turn labels back on (using the checkbox at the top of the window), enter "R" in the **Label filter field** at the bottom of the window and press **Enter**. At that point, all the "Q" parts that were previously displayed will be hidden. In fact, all parts not starting with "R" will be hidden. Now, press the **SHIFT** key and drag the mouse over the entire board to select all remaining parts. Right-click any of the parts (you may need to zoom in) and choose **Rotate 90** to rotate all the resistors. Finally, right-click anywhere on the Layer Viewer to clear the current selections.

Parts may also be filtered based on package type, part number, and part type by making a selection in each of the desired filters for the part properties to filter on. To clear any of these filters, simply select the **ALL** item in the filter.

Finally, clear the **Label filter** to show all components and press the **Close** button to exit the **Component Editor**. Your component layers are now ready for analysis.

2.4.6. Board Outline Editor

Sherlock allows you to edit the circuit board outline. In the Layer Viewer, select **Edit Board Outline** in the **Edit** menu, and the **Outline Editor** appears at the bottom of the display.



- **Draw Outline** (A): Press the **Draw Outline** icon to delete the existing outline (if any) and draw a new outline by left-clicking the mouse at one or more points on in the editor panel. When you are

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

done entering points, right-click the mouse over any point to automatically connect the last point entered to the starting point of the outline. The mouse operations supported in this mode include:

  – Left-click one or more times to add points.

  – Right-click to close the polygon and save the outline.

- **Edit Outline** (B): Press the **Edit Outline** icon to edit the existing outline. In edit mode, you can move an existing outline vertex by dragging it with the left mouse button. To insert a new vertex after a selected vertex, hold the SHIFT key and drag the vertex to the desired location. To delete an existing vertex, hold the CTRL key and left-click the vertex. To exit edit mode, right-click anywhere in the editor panel and select **Done Editing** from the context menu. The mouse operations supported in this mode include:

  – Left-Drag to move a vertex.

  – SHIFT-Left-Drag to split and drag vertex.

  – CTRL-Left-Click to delete vertex.

  – Right-Click any vertex to enter its location.

- **Simplify Outline** (C): When this function is invoked, Sherlock simplifies the board outline by reducing the number of points which comprise the outline. This can resolve meshing issues that happen when too many vertices define an outline. When you click the **Simplify Outline** icon, a dialog appears, asking you to specify the **Polyline Tolerance** (default is 0.1 mm). Any point that is within the specified distance of another point will be removed.

- **Save** (D): Saves the outline to the file system and updates all layer displays.

- **Reset Outline** (E): Resets all changes made to the outline since the editor was invoked.

- **Reset and Close** (F): Reset the board outline to its last saved state and close the Outline Editor.

- **Outline Editor Help** (G): Opens a help window with a detailed explanation of the Outline Editor.

Tip:

The tutorial *Ad Hoc Project Creation* provides a practical example of the Outline Editor in use. See Reviewing the Board Outline (p. 72).

2.4.7. Other Layer Viewer Features

The Layer Viewer has several other helpful features described in this section:

2.4.7.1. Component and Lead Identification

2.4.7.2. Camera Tool

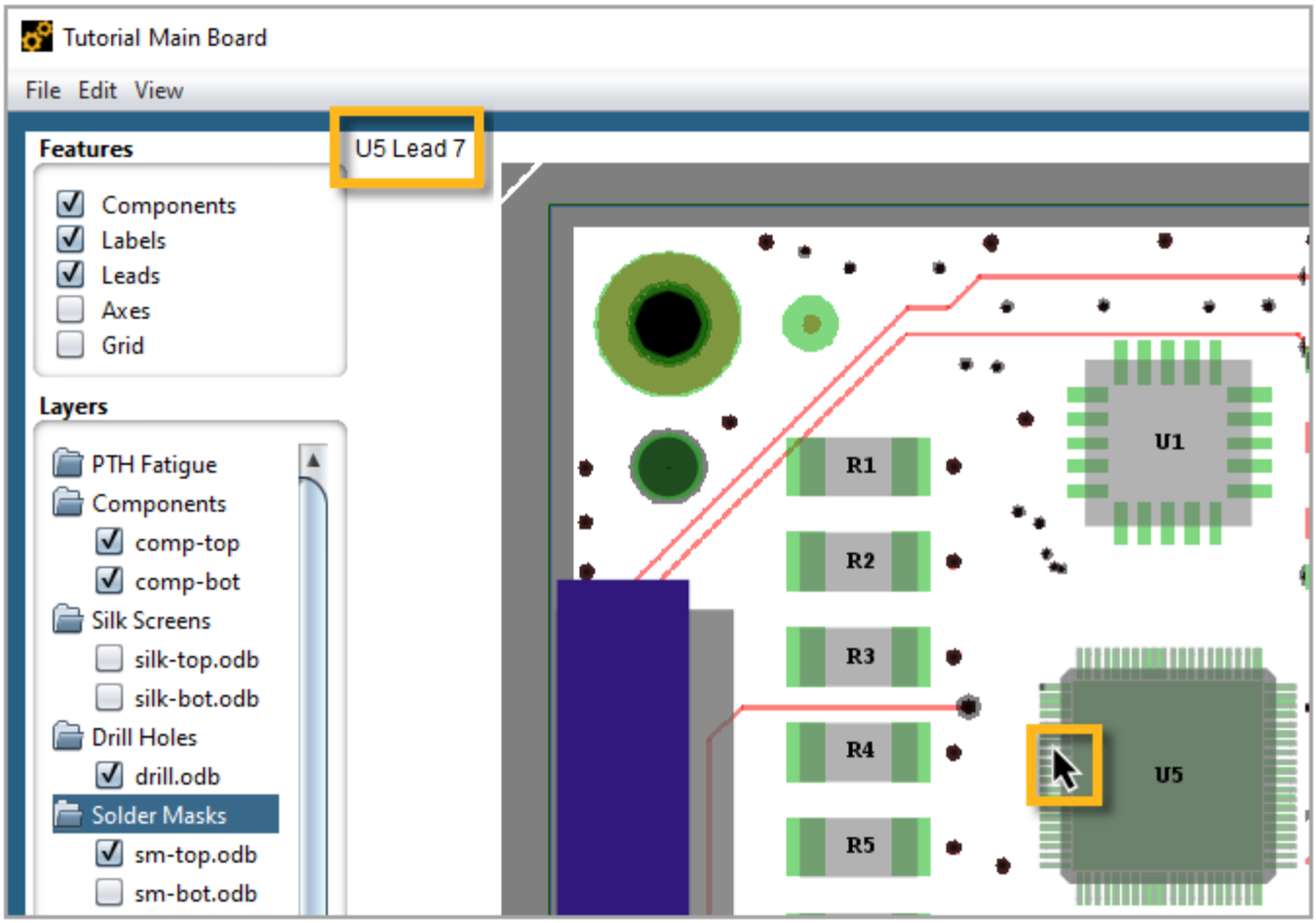
2.4.7.3. Distance Tool

2.4.7.4. Generate Trace Models

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.7.1. Component and Lead Identification

As shown in the image below, when you hover over a component, Sherlock will display the component reference designator in the upper left of the display. If you hover over a lead or a solder ball, Sherlock will also display the lead or ball number.



Note:

In order for this feature to work in projects created in Sherlock version 22.2 or earlier, you must first update the CCA layers from the Parts List: Right-click **Parts List** in the **Project Tree** and select **Update Layers**. Or click the **Update CCA Layers** icon in the **Parts List** window.

If you have enabled **Labels** and **Leads** under the **Features** filter, ball numbers for any BGA component are displayed when you zoom in close as shown below. If the BGA balls are not visible, verify **Ball Modeling** is activated for the part. Use the **Parts Editor** (p. 200) to change this setting.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2.4.7.2. Camera Tool

Much of Sherlock's usefulness comes from the numerous graphical layers generated to show analysis results. With just a few displays, it can show issues across hundreds of components. When combined with the ability to pan and zoom all layers, such information allows you to see with your own eyes where the problems lie. For example, the image below (A) shows a close-up view of analysis data from a Harmonic Vibration Analysis, along with the top component, top solder mask, and top copper layers.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The image itself was generated by panning and zooming the display to focus on the desired region and then using the **Camera Tool** to export the display to an image file.

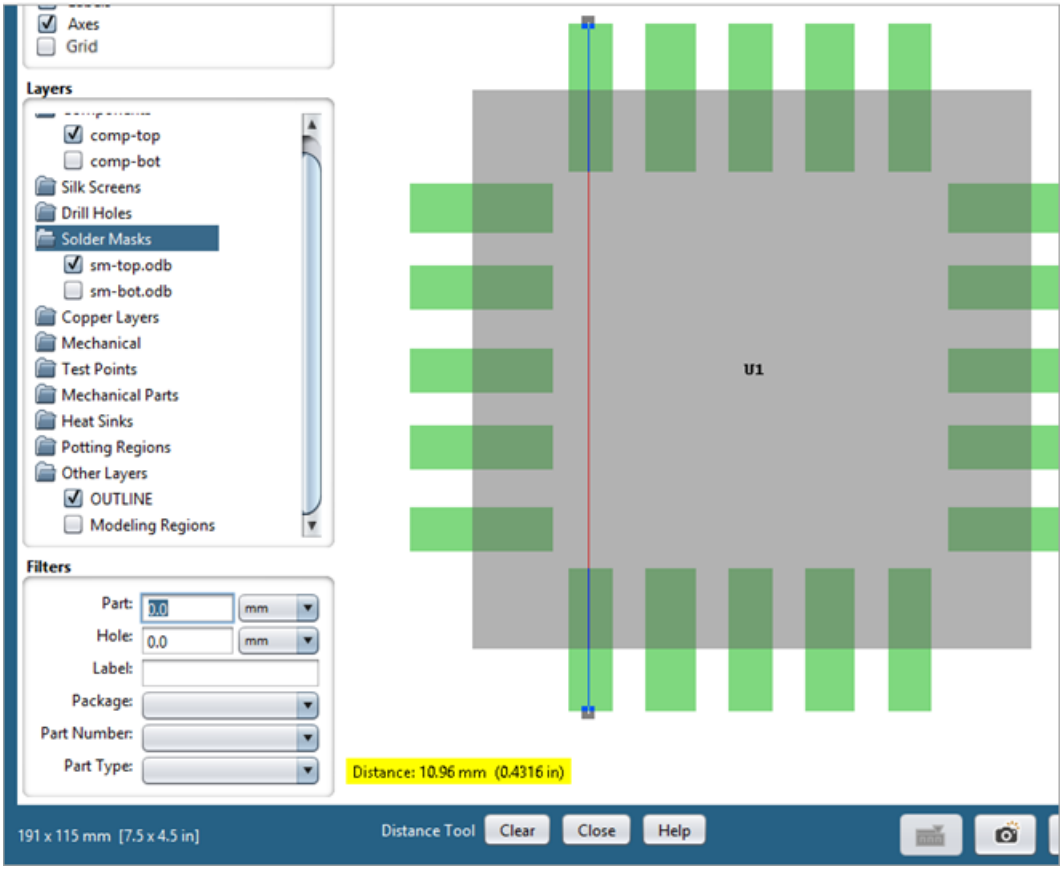
To use the **Camera Tool**, press the **Camera Icon**  located in the bottom right-hand corner of the Layer Viewer. At that point, a dialog will appear (B, above) allowing you to specify the scale factor, folder, file name and file type to be used.

After entering the required information, press the **Export File** button to create the image file.

2.4.7.3. Distance Tool

The **Distance Tool** is provided in the Layer Viewer to allow users to measure the distance between any two points.

To start using the **Distance Tool**, simply press the **Ruler Icon** button  in the lower right corner of the Layer Viewer. At that point, the control buttons for the **Distance Tool** will be shown in the bottom center of the Layer Viewer, as shown here:



To measure the distance between any two points, left-click anywhere in the layer display area while holding down the **SHIFT** key to set the first location. Then, move the mouse to the second location and left-click again. After the second location is specified, the distance between the two points will be displayed in the yellow box in both Metric and English units. The example above shows that the maximum distance covered by the U1 solder pads in the vertical direction is 10.96 mm.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

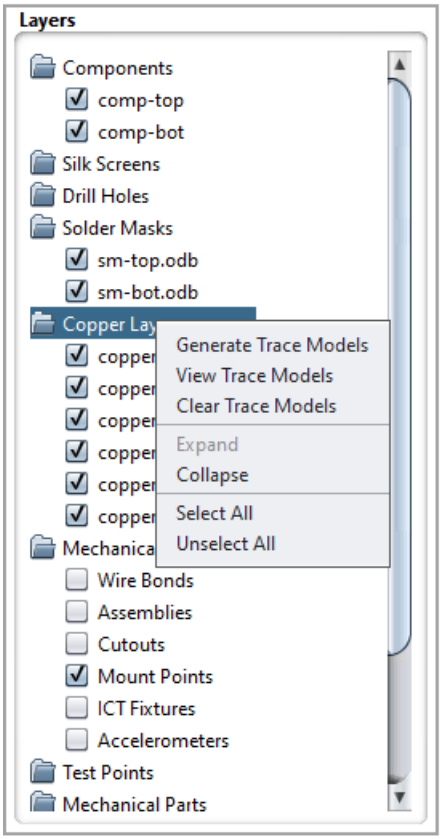
Repeat the process as many times as needed to measure other distances. When you no longer need the distance tool function, press the **Close** button. All filtering and zoom capabilities are available when using the **Distance Tool** to facilitate measuring various parts of the circuit card.

Note:

The **Distance Tool** will NOT be rendered in any image file exported using the **Camera Tool**.

2.4.7.4. Generate Trace Models

You can access Sherlock's Trace Modeling features from the Layer Viewer. Simply right-click the **Copper Layers** folder or one of the subfolders beneath it. Trace Modeling is a way to generate an FEA model of a board's signal traces. For a detailed discussion of Trace Modeling, see the section FEA- Trace Modeling (p. 495). For a general overview of FEA, see Finite Element Analysis (FEA) Overview (p. 443).



2.5. Tutorial: Ad Hoc Project Creation

In previous lessons you learned how to import CAD data from an ODB++ archive, how to update part properties as needed and how to ensure that all component locations, sizes and orientations are correct. These are the basic steps required to create any Sherlock project regardless of the CAD files used to initially import the data. This lesson describes the overall process used to create a Sherlock project from an ad hoc collection of CAD files typically provided to PCB manufacturer's for production purposes. For example, CAD tools typically generate a collection of layer files (such as Gerber formatted files) to describe

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

all copper layers, silk screens and solder masks needed to create the circuit card, as well as one or more drill files specifying the size and location of all holes in the circuit card. For assembly purposes, a Bill of Material (BOM) and Pick & Place file may also be generated, indicating the types of components on the circuit card and their location.

All of this CAD data is useful for analysis purposes and Sherlock is designed to import as much of it as possible. Unfortunately, there are literally dozens of file formats used in the manufacturing of circuit cards, many of which describe only a piece of the puzzle. In some cases, some of the critical information, such as drill sizes, may only be provided in human-readable form (for example, a PDF image of the circuit card design). Nonetheless, Sherlock knows how to read a large number of file formats (more added all the time) and also allows users to create Comma Separated Value (CSV) files using any modern spreadsheet application to import critical data into Sherlock. In this lesson will cover the most common file formats and point out where other formats can be used.

In this section, the following topics are covered:

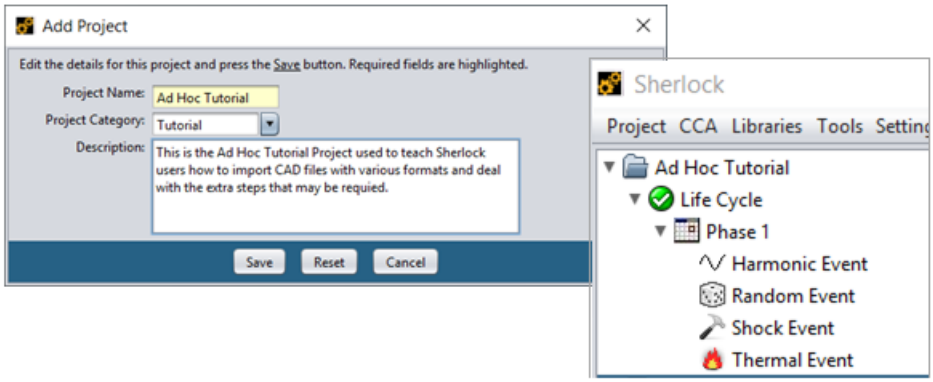
2.5.1. Project Creation
2.5.2. Circuit Card Creation
2.5.3. Adding CAD Files
2.5.4. Importing a Bill of Materials
2.5.5. BGA Part Properties
2.5.6. Importing Pick & Place Data
2.5.7. Importing Copper Layers
2.5.8. Insulated Metal Substrate Stackup
2.5.9. Additional Stackup Tools
2.5.10. Reviewing the Board Outline
2.5.11. Reviewing Component Layers
2.5.12. Reviewing Drill Holes

2.5.1. Project Creation

Before you create a new ad hoc project, clean up the project navigation tree by closing the previously created ODB++ Tutorial project. To do so, right-click the **ODB++ Tutorial Folder** in the Project Tree and select **Close Project** from the context menu. At that point, the project folder will be closed, as indicated by a closed file cabinet icon. All the files associated with a closed project are still maintained in the user project directory, but the Project Tree is trimmed so that only the project name itself appears. You can open the project at any time (thereby expanding the folder in the Project Tree) by right-clicking and selecting the **Open Project** menu option.

To create a new project, select the **Add Project** option from the **Project** menu available in the main window. Enter a unique **Project Name** and a short project **Description**. The project description will be included in all Sherlock analysis reports, so plan accordingly. Optionally, you may choose to add a **Project Category**, but this is not necessary.

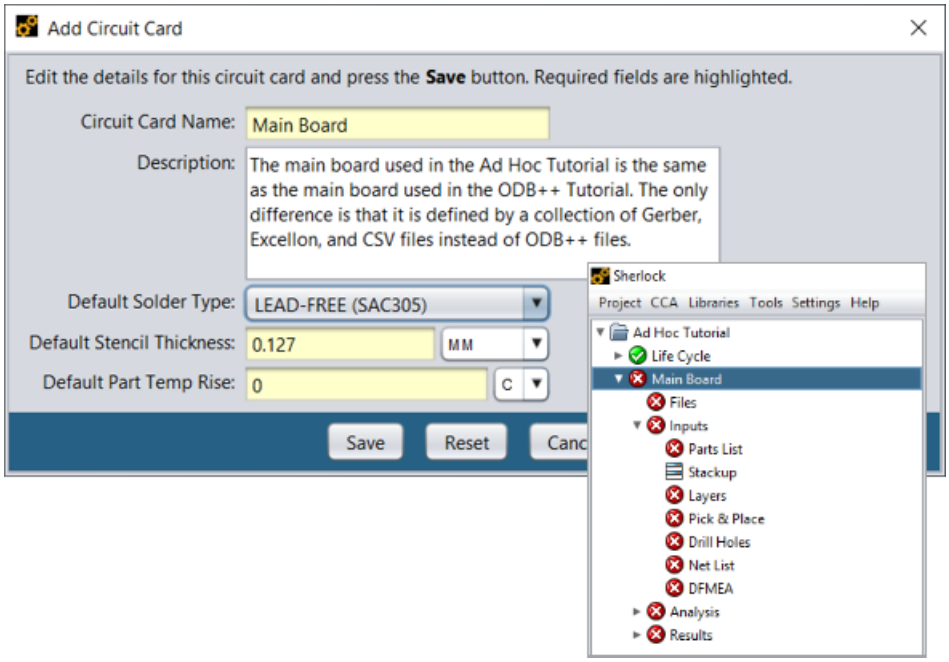
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Press the **Save** button to create the project, which will be displayed immediately in the Project Tree. Note that only a default **Life Cycle** has been created for the project. More on this later.

2.5.2. Circuit Card Creation

The real project begins when you add a Circuit Card to the project. To do this select the **Add Circuit Card** menu option from the **Project** menu or right-click the project folder in the Project Tree and select the **Add Circuit Card** menu option. In either case the following dialog will be displayed, allowing you to enter the basic circuit card information.



As with the project properties, you must enter a unique name and an optional description. For circuit cards, you also need to enter some default part properties used by various analysis modules in case those properties aren't defined for one or more parts. (This document will discuss the specific use of these properties in later lessons that cover the analysis modules.)

When the **Save** button is pressed, the **Circuit Card** will appear as a sub-folder in the project folder. Double-click the folder to view the default contents.

Much like the project creation, you have a lot of default folders and entries, most of which are marked with missing data (we haven't imported anything yet) or warnings (we haven't run analysis yet). These

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

indicators serve as a to-do list to remind you of what must be done. You can view the error messages by right-clicking on each entry and selecting **Show Error** to determine what remains to be done.

2.5.3. Adding CAD Files

With a project and circuit card framework in place, you can start adding CAD files and importing data from those files. Unlike the ODB++ archive import process, which automatically scans the archive and extracts everything of value, with ad hoc projects you need to designate the files to be used and then specify the format of those files so that Sherlock can extract the data and use it appropriately.

To begin the import process, you need to add some files. To do so, right-click the **Main Board** circuit card folder (or the **Files** folder below it) and select the **Add File(s)** menu option. At that point, a file system dialog will appear allowing you to locate the directory or ZIP archive file containing the files to be added. The CAD files to be used for this lesson are contained in the `Ad Hoc Tutorial.zip` archive file located in the `tutorial` sub-directory of the Sherlock installation directory:

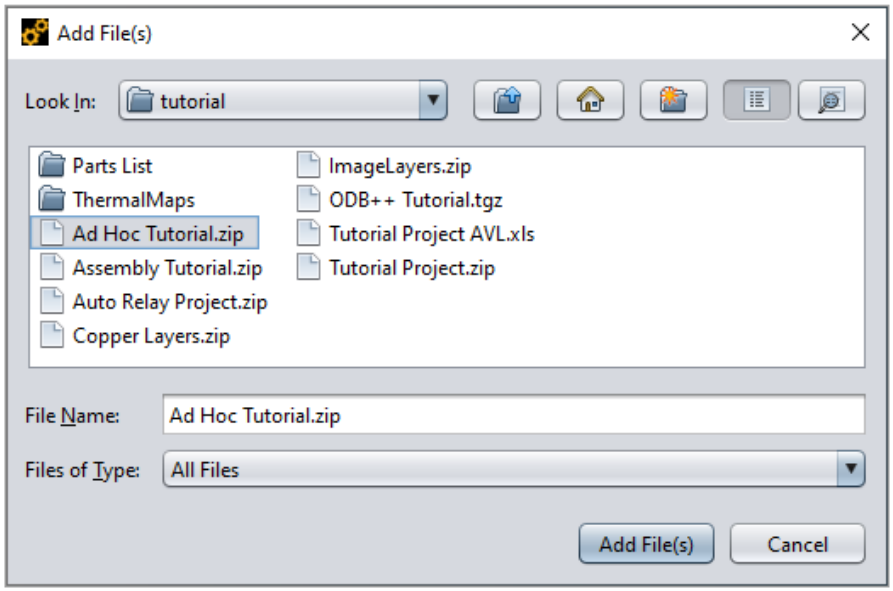
Windows:

```
C:/Program Files/Ansys Inc/v251/sherlock/tutorial
```

Linux:

```
/ansys_inc/v251/sherlock/tutorial
```

Navigate to that directory, select and open the ZIP archive file. Afterwards, click the **Add File(s)** button to add the contents of that archive to the circuit card.



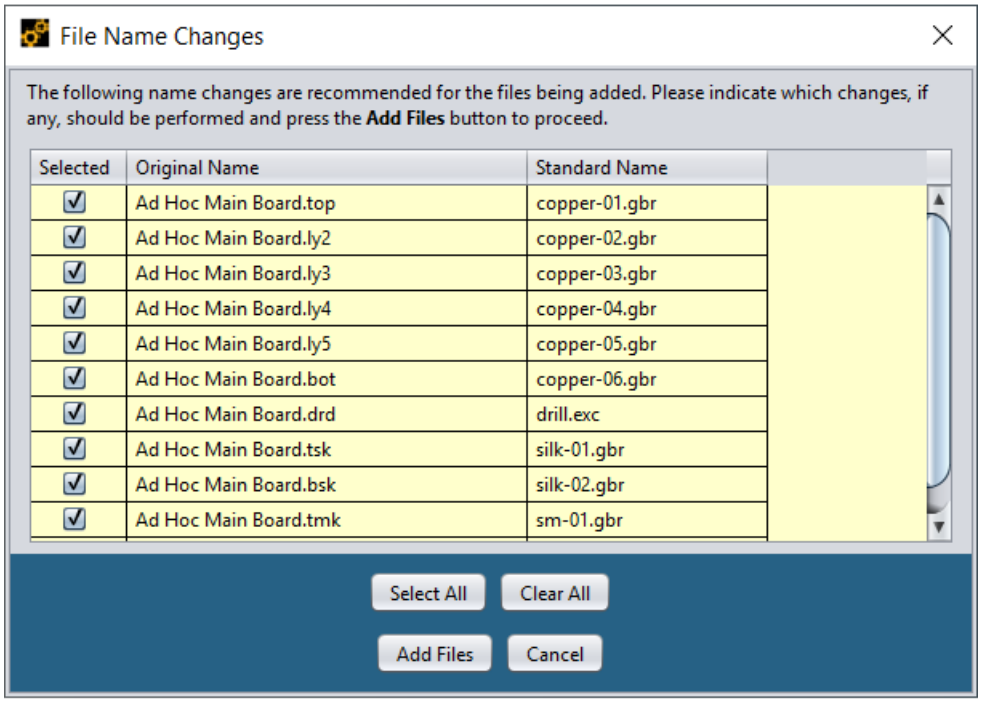
Note:

You can execute the **Add File(s)** command as many times as necessary to find files in different locations or at a later time when additional files are obtained. When files are added to a Sherlock project, a copy of the original file is made and stored in the Sherlock User directory. As such, you are free to do whatever you want with the original files without affecting Sherlock.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

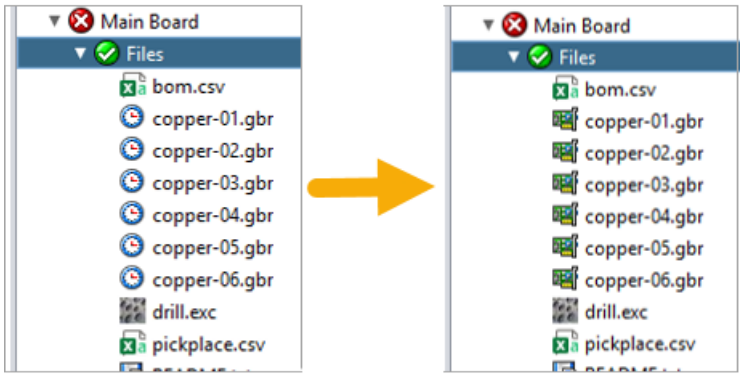
When the **Add File(s)** button is pressed, Sherlock examines the given file names to determine as much as possible in preparation for the import process. Since many CAD packages use cryptic file name prefixes, suffixes and formats, if Sherlock recognizes the file name format then it recommends a simpler, more easily understood standard name for the file, as shown below.

You are free to accept any of the name changes or reject them all. In either case, when the **Add Files** button is pressed in the File Name Changes window, all of the files specified in the first dialog will be added to the circuit card and all of the file names selected in the name change dialog will be re-named.



Immediately after all the files have been added, the Project Tree will be updated to show the new files. If Sherlock recognized the file format a **clock icon** will be shown (see below, left), indicating that Sherlock is currently parsing and importing data from that file. If Sherlock did not recognize the file format, then a red X will be displayed indicating that you need to specify the file type manually.

After each file has been imported, an icon associated with that file type will be displayed (see below, right). If something went wrong or a red X will appear in its place.



After the Ad Hoc Tutorial files have been imported, two CSV files appear as spreadsheet files. This is correct; however, you need to select a more specific file type for each of these so that Sherlock can

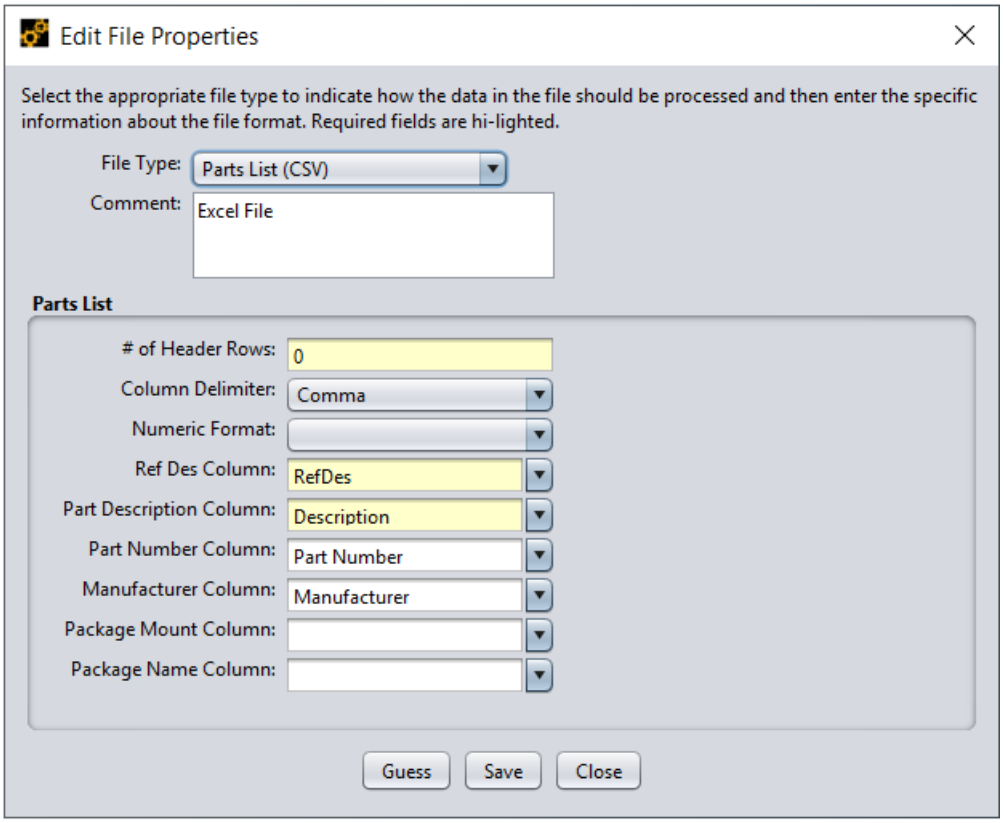
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

make use of the data. When loading files from a ZIP archive, the archive may also contain files that you don't need, such as the **README.txt** file here. Simply right-click the file name and select **Delete File** if you wish to remove it.

2.5.4. Importing a Bill of Materials

The first and most important file to be processed is the **Bill of Materials (BOM) File**. In this case, the BOM is formatted as a CSV file as indicated by the `.csv` suffix. To verify this, right-click the **bom.csv** filename in the Project Tree and select the **View File** menu option. Sherlock will display the file contents as text in the content panel, along with a secondary tab showing the error associated with this file.

To specify the file type and import the data, right-click again on the **bom.csv** filename but select the **Edit Properties** menu option this time. A dialog will appear that allows you to select the file type and enter any type-specific properties required.



Select **Parts List (CSV)** from the **File Type** pull-down list and the type-specific property form will be displayed. Sherlock has already scanned the CSV file to determine the number of header rows and the column names. Such guesses are usually right, but you should check them anyway by viewing the CSV file in the content panel.

In addition to guessing the column names, Sherlock also attempts to guess which columns should be mapped to Sherlock part properties. In this tutorial, the CSV file uses some standard column names,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

so Sherlock successfully guessed how to map all the columns. In real-world examples, lots of different and unusual column names are used, so you may have to select the desired mapping yourself.

Note:

The type-specific entry forms use color-coding to denote required or optional fields (as do most Sherlock input forms). Fields displayed with a pale-yellow background are required, while fields shown with a white background are optional.

With the desired column mapping specified, press the **Save** button to start importing data from the BOM file. A clock icon will be displayed next to the file in the Project Tree while it is being processed. If the file is processed successfully, a BOM icon will be displayed. Otherwise, the error icon will be displayed.

With the BOM file successfully processed, you can double-click the **Parts List** entry in the **Inputs Folder** to review the **Parts List.** This document already covered how to review and update the Parts List in a previous lesson, so that won't be repeated here. However, there are some special issues that arise when processing ad hoc CAD files that need to be examined.

The first thing to notice in the **Parts List** is that there is no **Location** data assigned to any parts. That's because you've only processed the BOM so far and location data is contained in Pick & Place files. You will fix that problem when you import the Pick & Place file.

Another issue involves the lack of package information for certain components. Unlike an ODB++ archive which defines a package type for all parts, Sherlock must guess package information from the BOM based solely on the part description. If no such package can be determined, it is left empty to remind you that you must manually enter that information. This issue almost always arises with through-hole parts, for which textual package designations are not well standardized.

Note:

You may have noticed on the BOM file property form that there was an entry labeled **Package Column.** So, if the CSV file contained that information, Sherlock could have imported it, but in this tutorial file no such column was defined.

As an exercise, review the parts list data imported from the BOM file and update the following properties (most of which were covered in the previous lesson):

- Parts C1 – C10, change the package type to "C-BEND-3528-12".
- Parts C21 – C25, set the mount type to "TH" and set the proper dimension and shape.
- Parts C26 – C30, set the mount type and package type based on the description.
- Part J1 – Leave this alone for now; the tutorial will return to it later.

2.5.5. BGA Part Properties

There is one **Parts List** modification required for Ad Hoc projects that is related to BGA components. When a BGA part is imported from an ODB++ archive, Sherlock has access to the pin locations for that part. Given the pin locations, Sherlock can not only deduce that the part is a BGA component,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

but also the exact grid arrangement. Unfortunately, such information is not provided in a BOM, so you must designate BGA the part properties yourself, as demonstrated below.

If you enter XCV in the Part Number filter of the Parts List two parts, U9 and U10, will be displayed with no package information imported from the BOM. Select either of the parts and press the **Edit Selected** button at the top of the **Parts List** ribbon to examine the part properties. Select the **ID Tab** first and examine the **Part Description Field** to see that it includes "676" and "BGA" in the description. Now, select the **Package Tab** and note that the **Package Name Field** is empty - something you need to correct.

Since there are no standard BGA package designators, Sherlock uses its own standard to recognize BGA package names. Quite simply, any package name that includes "BGA" is recognized as a BGA component. (Why makes things harder than necessary?) So, you can simply enter "BGA-676" in the **Package Name Field** to designate the part as a BGA component. Of course, you must also enter the package dimensions. A quick Internet search for the Part Number (XCV405E-6FG676I) reveals that it's a 27mm square part. After updating the Package Length and Package Width, press **Save the part editor changes** button to update both parts, but remain in the **Parts Editor.**

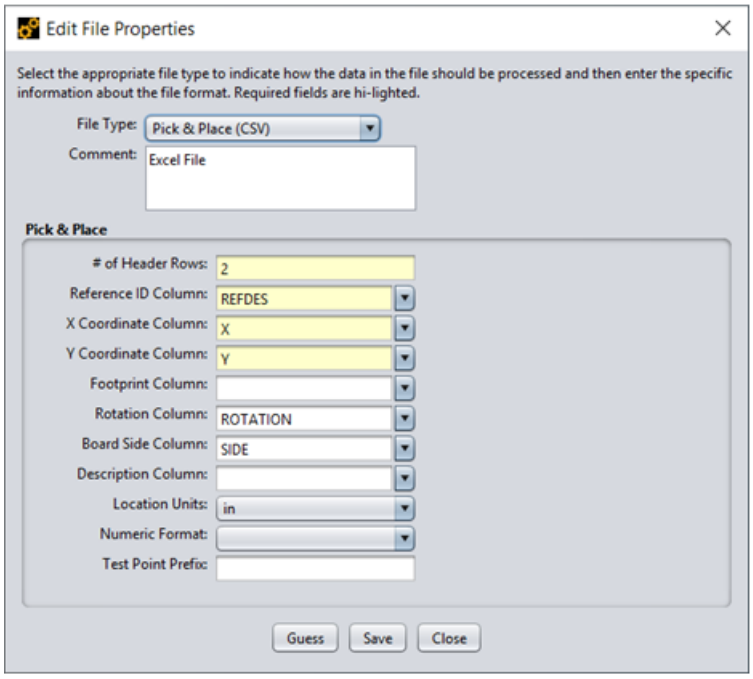
After changing the **Package Name** to "BGA-676" and applying the change, the **Ball** and **Die Tabs** become active in the **Parts Editor**, indicating that Sherlock recognizes this part as a BGA component. Select the **Ball Tab** and enter "676" in the **Ball Count Field.** The other default BGA properties are correct, so simply **Save** the changes for both parts and return to the Parts List.

You're done making **Parts List** changes at this time, so remember to press the **Update Layers** button at the bottom of the Parts List to update the component layers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.5.6. Importing Pick & Place Data

After importing the BOM, the next logical step is to import the Pick & Place data, so that Sherlock will know the location of each component. To do so, right-click the **pickplace.csv** file name in the Project Tree and select **View File** to display the text file. You can see that it is indeed a CSV file, so you can proceed with the import process. Right-click again on the **pickplace.csv** file name and select **Edit Properties** from the menu. The **Edit File Properties Dialog** will appear, allowing you to select **Pick & Place (CSV)** as the file type, thereby displaying the following property form:



As was done with the BOM file, Sherlock automatically scanned the Pick & Place CSV file to determine the number of header rows, the column names and the mapping of column names to Sherlock part properties. Remember, Sherlock can only guess at the column mapping, you need to review each assignment (and possibly examining the data file) to determine if the column mapping is correct. Once you've determined that everything is correct, press **Save** to start the import process.

After the Pick & Place file has been successfully processed, the Parts List will be automatically updated to show that Location data is now known for all parts. In some situations, the Pick & Place data file may not contain locations for all parts, in which case the **Location** entry will be RED for those parts. The document will deal with missing parts and missing part locations later in this lesson.

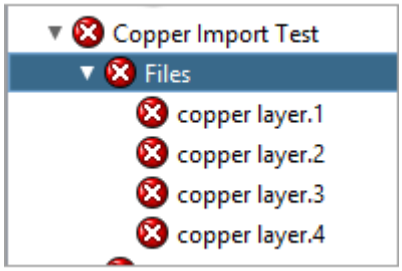
After you import the various circuit card layers, review the component locations using the Layer Viewer.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.5.7. Importing Copper Layers

When the **Add Files** process recognizes copper layer files, it automatically imports each layer and updates the stackup data accordingly (like the process used when importing layers from an ODB++ archive). If, however, Sherlock does not recognize the file type automatically, then you'll need to specify the proper file type and where the layer is located (top, bottom or somewhere in the middle).

Since the **Main Board** copper layer file formats were all recognized by Sherlock, you will create another circuit card and add some copper layers manually to that circuit card. To that end, right-click the **Ad Hoc Tutorial folder** in the Project Tree and select the **Add Circuit Card** menu entry to add a new circuit card named **Copper Import Test.** You're only going to use this circuit card to practice loading copper layers, so feel free to use all the default properties.



After the new circuit card has been added to the Project Tree, right-click the **Copper Import Test** folder and select **Add File(s)** from the menu. Using the file chooser, navigate to the **tutorial** subdirectory of the Sherlock Installation Directory (usually **c:\Program Files\Ansys Inc\v251\sherlock**) and select the **Copper Layers.zip** file to be imported.

Since that ZIP file contains completely non-standard file names, Sherlock will not recognize any of them and you will have to import them manually. However, before you can designate where those layers reside, you need to modify the default Stackup data assigned by Sherlock. To do so, simply double-click the **Stackup** entry located in the **Inputs** folder for this circuit card.

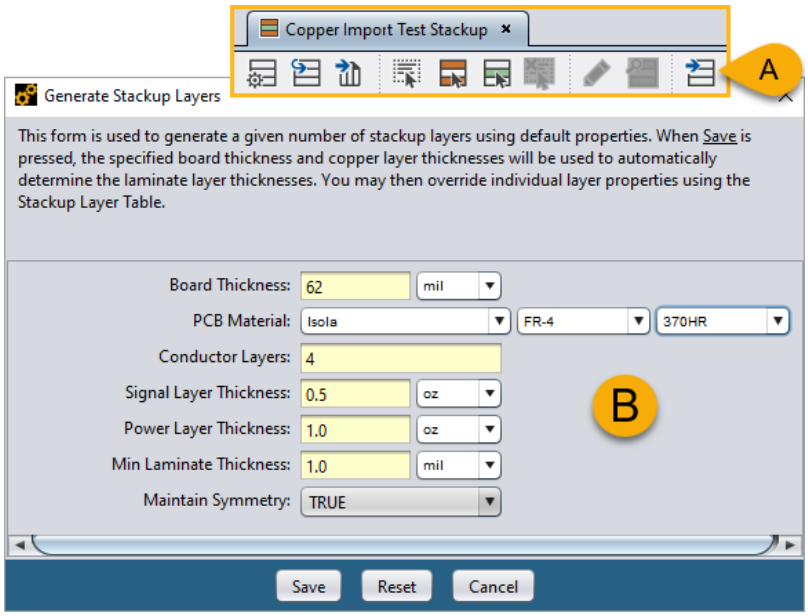
For this case you will have to first enter the board dimensions before modifying the Stackup data as none of the imported files provide Sherlock the necessary information to understand the board dimensions. For the purposes of this tutorial, enter most any valid dimensions you wish as it won't be used.

The default Stackup data says that the board only has two copper layers, but you know that it's actually a four- layer board, so you need to update the Stackup data.

Layer	Type	Material	Construction	Thickness	Density	CTExy	CTEz	Exy	Ez
1	SIGNAL	COPPER (50.0%) / COP...		0.5 oz	5.3500	33.800	33.800	58,250	58,250
2	Laminate	Generic FR-4	DEFAULT (2116)	60.6 mil	1.9000	17.000	70.000	24,804	3,450
3	SIGNAL	COPPER (50.0%) / COP...		0.5 oz	5.3500	33.800	33.800	58,250	58,250

Press the **Generate a New Stackup** button (A, below) at the top of the **Copper Import Test Stackup** ribbon to generate the proper number of layers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Generate Stackup Layers** dialog (B) allows you to specify the copper and laminate properties for the board, as well as the board thickness. It then generates each layer accordingly, automatically selecting the laminate layer thicknesses to ensure that the specified board thickness is achieved. After entering the following information

Number of Conductor Layers: 4

PCB Material: Isola, FR-4, 370HR

press **Save** to generate the circuit card layers, resulting in a board with the desired 4 copper layers as well as 3 laminate layers automatically generated and defaulted with the given properties.

Layer	Type	Material	Construction	Thickness	Density	CTExy	CTEz	Exy	Ez
1	SIGNAL	COPPER (50.0%) / COPPER-RESIN		0.5 oz	5.3500	33.800	33.800	58,25	58,250
2	Laminate	370HR	DEFAULT (2116)	19.73 mil	1.9000	14.000	45.000	26,12	3,450
3	SIGNAL	COPPER (50.0%) / COPPER-RESIN		0.5 oz	5.3500	33.800	33.800	58,25	58,250
4	Laminate	370HR	DEFAULT (2116)	19.73 mil	1.9000	14.000	45.000	26,12	3,450
5	SIGNAL	COPPER (50.0%) / COPPER-RESIN		0.5 oz	5.3500	33.800	33.800	58,25	58,250
6	Laminate	370HR	DEFAULT (2116)	19.73 mil	1.9000	14.000	45.000	26,12	3,450
7	SIGNAL	COPPER (50.0%) / COPPER-RESIN		0.5 oz	5.3500	33.800	33.800	58,25	58,250

You should note that the copper percentage values are currently listed as default values (50%). These will be updated automatically when the copper layers are imported.

With the updated Stackup data, you can now import the copper layers. For each of the files listed below, right-click the file name in the Project Tree, select **Edit Properties** from the menu, select the designated file type and Copper Layer position in the file property form, and press **Save** to import the file.

Note:

Sherlock is a multi-tasking application, which means that you can have multiple processing tasks running in the background at the same time. So, feel free to start importing all files as fast as you can fill out the file properties forms.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 2.1: Copper Layers

File Name	File Type	Copper Layer
copper layer.1	Copper Layer (Gerber)	Top Layer
copper layer.2	Copper Layer (Gerber)	Layer 2
copper layer.3	Copper Layer (Gerber)	Layer 3
copper layer.4	Copper Layer (Gerber)	Bottom Layer

Note:

After each copper layer file is successfully imported, the corresponding copper layer row in the Stackup Table will be updated to reflect the actual percentage of copper contained on that layer. In addition, the mechanical properties of the board will be updated to reflect the changes made to the layer properties.

When all the copper layers have been imported, double-click any of them to give them a quick review for correctness.

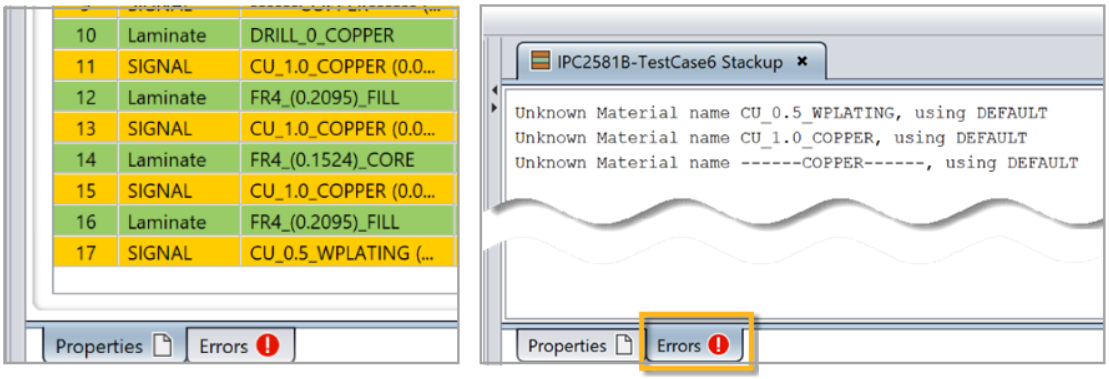
Note:

Currently, the effective in-plane stackup CTE calculation is based on a volume average. The method works for most laminate layers. However, for stack-up constructions with softer laminates, the method tends to cause the overall in-plane CTE to be lower than it should be. This has been noted as a current limitation in the software.

2.5.7.1. Unknown Materials in the Stackup

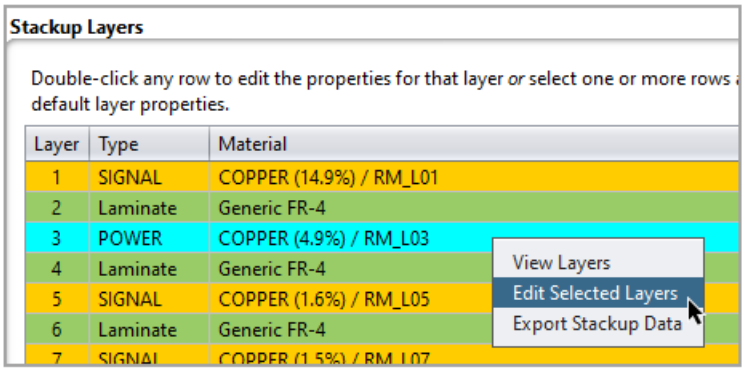
If Sherlock finds a material in the imported stackup that is not in the Material Library, you will receive an error message so you can remedy the problem. The error message will appear in several places:

- In the Console
- In an **Errors** tab that will appear at the bottom of the **Stackup Layers** display as shown below.
- In the **Project Tree**. The **Stackup** node will have the error icon next to it.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To correct the issue, you can right-click the problem layer in the **Stackup Layers** display and select **Edit Selected Layers** as shown below. If needed, you can add a new entry in the Material Library using the Material Manger (p. 153). Once you have corrected the issue, the **Errors** tab will go away, and the **Stackup** node in the **Project Tree** will return to normal.

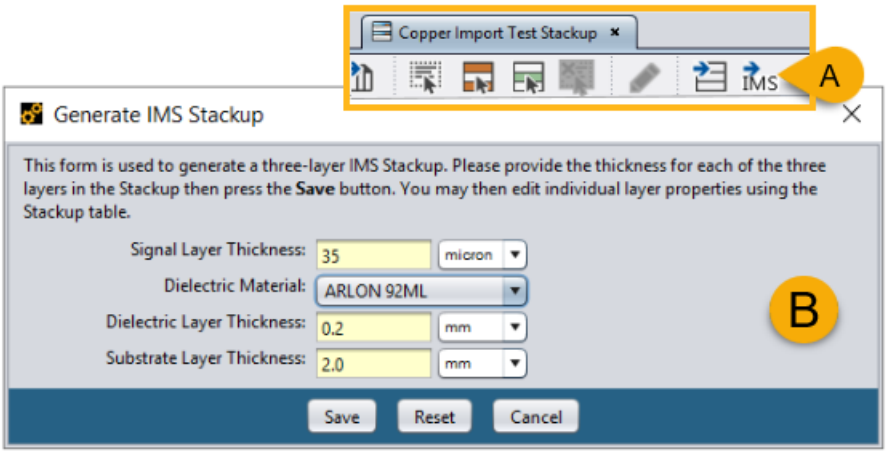


2.5.8. Insulated Metal Substrate Stackup

Sherlock includes the ability to define an insulated metal substrate (IMS) stackup which differs from the standard FR4 stackup in that it defines only three layers. The top layer is a copper layer, the second layer is a dielectric layer, and the third layer is a substrate layer. To have Sherlock generate an IMS stackup, press the **Generate IMS Stackup** button (A) on the Stackup ribbon.

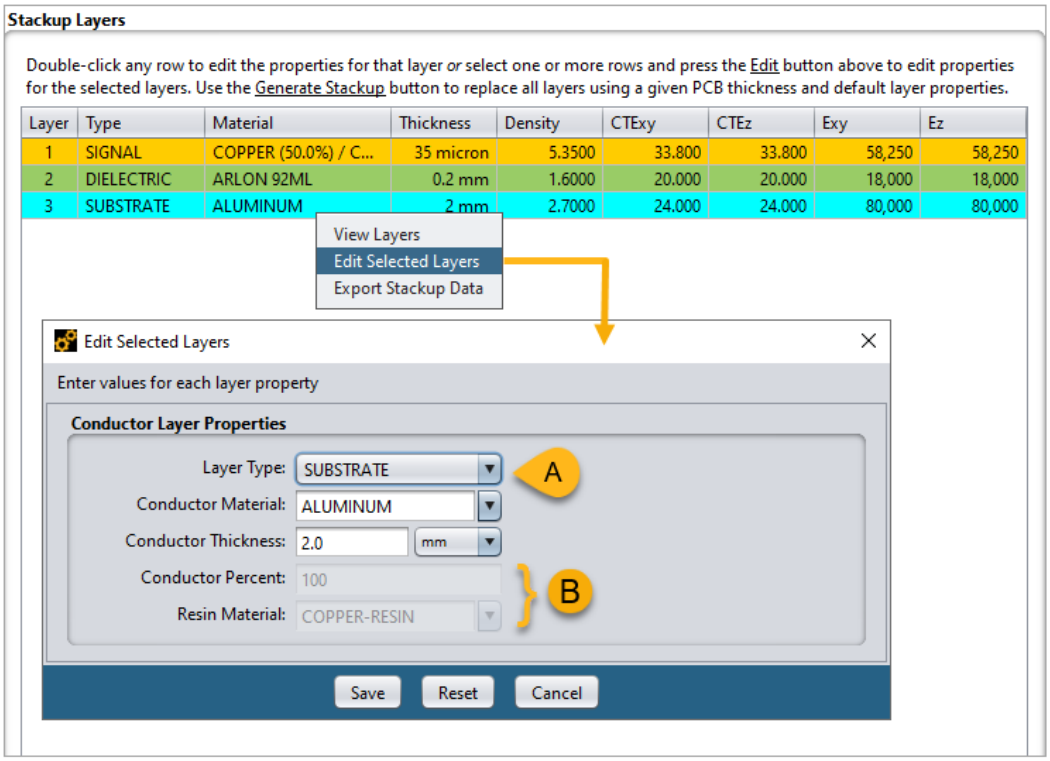
Note:

When you import an ODB++ archive, you can select how you want Sherlock to process the IMS data. See the IMS Stackup option (p. 26) in the tutorial *ODB++ Project Creation*.



The **Generate IMS Stackup** dialog (B) will appear which will assist in the creation of the IMS stackup. After selecting the desired stackup properties, click the **Save** button and Sherlock will generate the new stackup. Once generated, each of the stackup layers may be edited by right-clicking the desired layer and selecting **Edit Selected Layers** from the menu (see image below).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

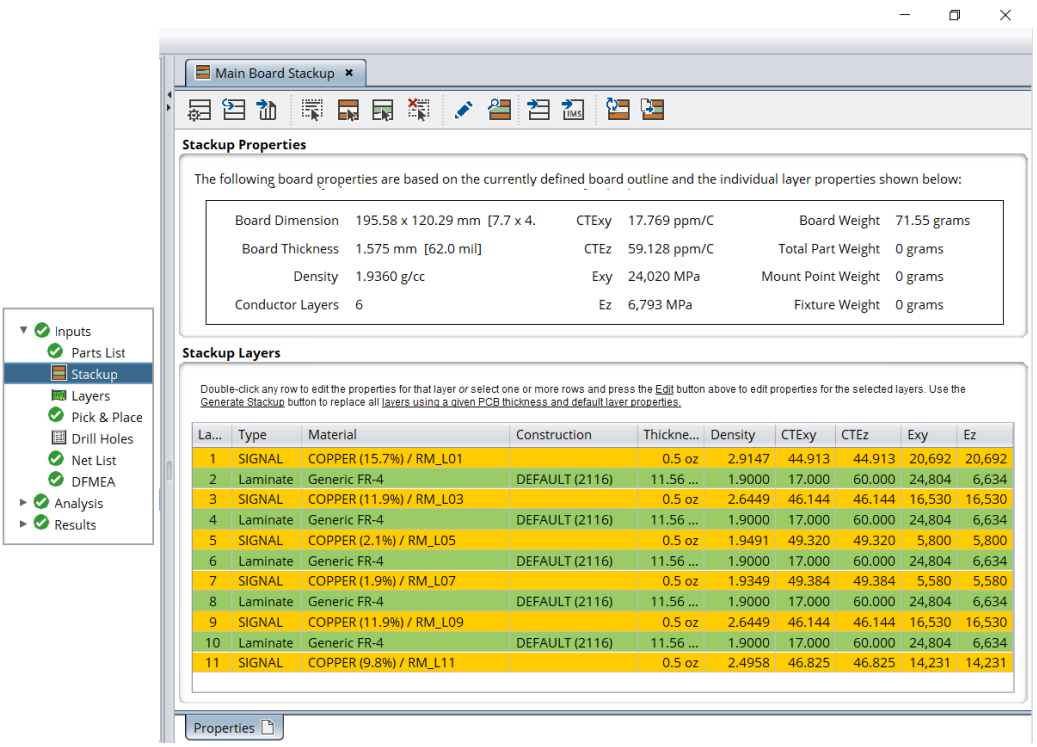


As shown in the image above, the bottommost layer of the IMS stackup represents the metal substrate, typically made of copper or aluminum. **SUBSTRATE** is assigned to the **Layer Type** (A, above). When **SUBSTRATE** is the **Layer Type**, the **Conductor Percent** and **Resin Material** options are grayed out (B), with **Conductor Percent** set at **100**.

2.5.9. Additional Stackup Tools

After you have created or imported the initial stackup layers for your board, you can always come back later to the Stackup Layers Table to do further editing. Simply double-click the **Stackup** entry in the **Project Tree** to display the **Main Board Stackup** tab in the **Content Panel.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Stackup Layer tools allow you to add, export, or edit layer properties:

Edit Stackup Properties

Import New Stackup: Allows you to select a Stackup file to replace the current Stackup.

Export Stackup Data: Allows you export the Stackup data as a CSV file. You can export all data or select specific rows and columns.

Select All Layers in Stackup: Helpful when you want to edit or export all layers shown on the table.

Select All Conductor Layers in Stackup

Select All Laminate Layers in Stackup

Clear All Selections

Edit Selected Row(s) in Stackup

View the Selected Layers in the Layer Viewer

Generate a new Stackup: This tool opens a dialog which allows you to configure and generate a stackup which replaces existing layers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Generate a new IMS Stackup: This tool opens a dialog which allows you to generate a three-layer IMS (Insulated Metal Substrate) Stackup. See the section above, Insulated Metal Substrate Stackup (p. 68).



Reset the Percentages in the Stackup: Resets the copper percentages to the values calculated at the initial stackup generation. This operation does *not* reparse the copper layer files. To reparse the files, use the operation below, **Recompute Copper Percentage**. If no file exists, the copper percentage is reset to the system defaults for the given layer.

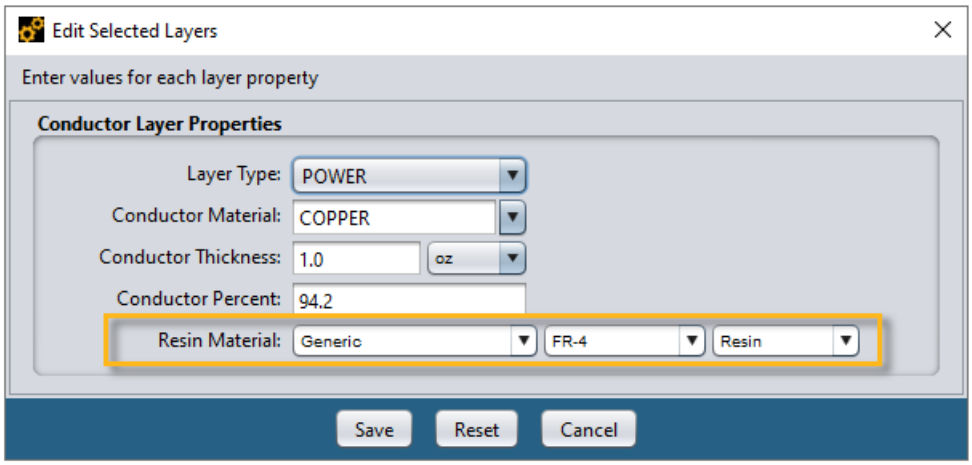


Recompute Copper Percentage: Recomputes the copper percentage for each layer based on the data in its layer file. If no file exists, the copper percentage is not updated for the given layer.

The section above, Importing Copper Layers (p. 65), provides examples on how the stackup tools can be used.

Note:

Beginning with Sherlock 22.2, there has been a change in the Stackup editor. When you are editing the **Resin Material** field of a copper layer, the Stackup tool now references the Laminate Library instead of the Material Library.



The default resin material is now Generic FR-4 Resin. The laminate Generic FR-4 Resin has the same material properties as the previously used COPPER-RESIN material. If you have defined a stackup in an earlier version of Sherlock, it will be updated to use the new default resin material. When this is done, Sherlock will display a warning on the console.

Since you're done with this exercise and you already have these layers defined in the **Main Board**, you can delete the **Copper Import Test** circuit card by right-clicking its entry in the Project Tree and selecting **Delete Circuit Card** from the menu.

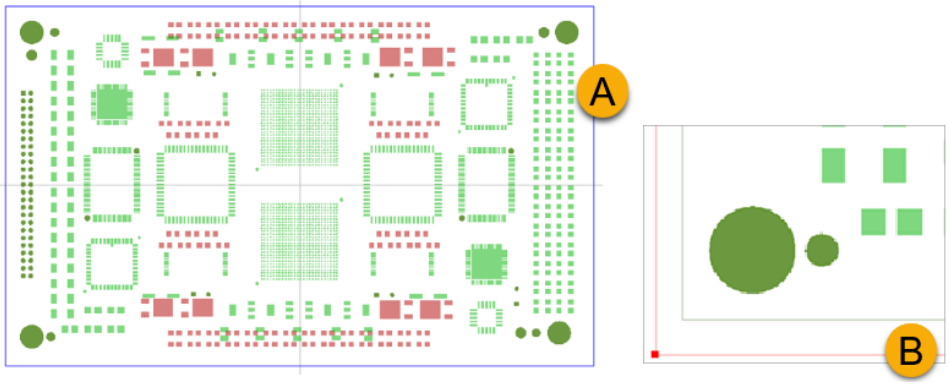
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.5.10. Reviewing the Board Outline

Back to the **Main Board:** Every PCB in Sherlock needs a board outline defined. Sherlock will typically be able to automatically determine the board outline when provided file information such as pick and place data, however if that data is not available, the board outline must be defined. If there is a red X next to the **Layers** entry in the **Inputs** folder, hover the mouse over that entry to reveal the error or right-click the entry and select **Show Error** from the menu. When the board outline must be defined, simply double-click the **Layers** entry to launch the Layer Viewer.

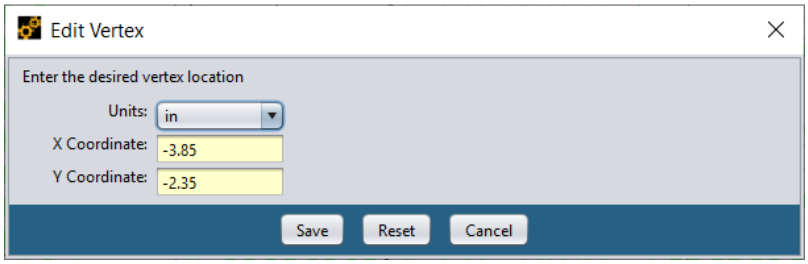
If there is no board outline defined, then the **Missing Board Outline** (A, below) will be displayed to allow the rectangular dimensions of the board to be provided.

If you provided the board dimensions or guessed the board outline based on all component, silk screen, solder mask and copper layers already processed, further refinement may be necessary, especially if the board isn't a rectangular shape. To do so, select **Edit Board Outline** from the **Edit** menu at the top of the Layer Viewer, then press the **Edit Outline** button at the bottom of the Layer Viewer. Upon entering edit mode, the outline will change to red and the outline nodes will be displayed as filled red squares (B, below).



Left-click any node and drag it with the mouse to the desired location. Note that the node location is shown in the upper-left hand corner of the layer display as you drag the node for convenience. After you are done moving one or more nodes, right-click anywhere outside the nodes and select **Done Editing** to terminate edit mode. You may also press the **Done** button at the bottom of the Layer Viewer.

If you know the exact location of the board outline nodes (easily found by looking at the board layout diagram) you can use another approach to enter them exactly. Start by zooming out to see the entire outline. Then, press the **Edit Outline** button to start edit mode. Now, instead of left-clicking a node and dragging it into position, right-click the node and select **Edit Location** from the menu to display the **Edit Vertex** dialog. That dialog allows you to enter the precise coordinates for the node. In this case, you're looking at the lower-left corner of the board outline. Since the board dimensions are 7.7 x 4.7 inches and it is centered at (0, 0), you know exactly where each corner should be located.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Press **Save** to update the node location and repeat for all other board outline nodes. When done, right-click anywhere outside the nodes and select **Done Editing** to update the board outline.

When you're done making all outline changes, press the **Save** button at the bottom of the Layer Viewer to make the changes permanent.

Note:

Extremely accurate placement of board outline nodes and components is not really important for analysis purposes. Sherlock automatically snaps all part locations and board outline nodes to a virtual 1 mm grid spacing before Finite Element Analysis. As long as parts don't overlap, placing them anywhere within 1 mm of their actual location is good enough.

Tip:

For a complete explanation of all the board outline editing tools, see Board Outline Editor (p. 51). Also, you can click the **Help** icon in the **Board Outline Editor**.

2.5.11. Reviewing Component Layers

With the board outline in place and all files imported, you can focus on reviewing the size, location and shapes of all components. The component review process for ad hoc projects is the same as that for ODB++ projects, with just a couple extra things to examine. Start by reviewing the same components that had to be modified in the previous lesson focused on the ODB++ project. It's left as an exercise for you to make the following changes that you made in the previous lesson:

Top Side Changes

- Parts R75 – R130, rotate by 90 degrees.
- Parts U5 – U6, change package type to QFP-80 (12x12).
- Parts U7 – U8, change package type to QFP-100 (20x14) and rotate by 90 degrees.
- Parts U13 – U16, change package type to TSOP-32 (18.4x8.0).

Bottom Side Changes

- Rotate all resistors by 90 degrees.
- Move all DPAK transistors to their proper locations.

In addition to these changes, there are some common issues to examine related to the use of Gerber and Pick & Place files. Specifically, you want to look for the following problems:

- Components that are not displayed in any of the component layers.
- Components that are not located in the proper position.

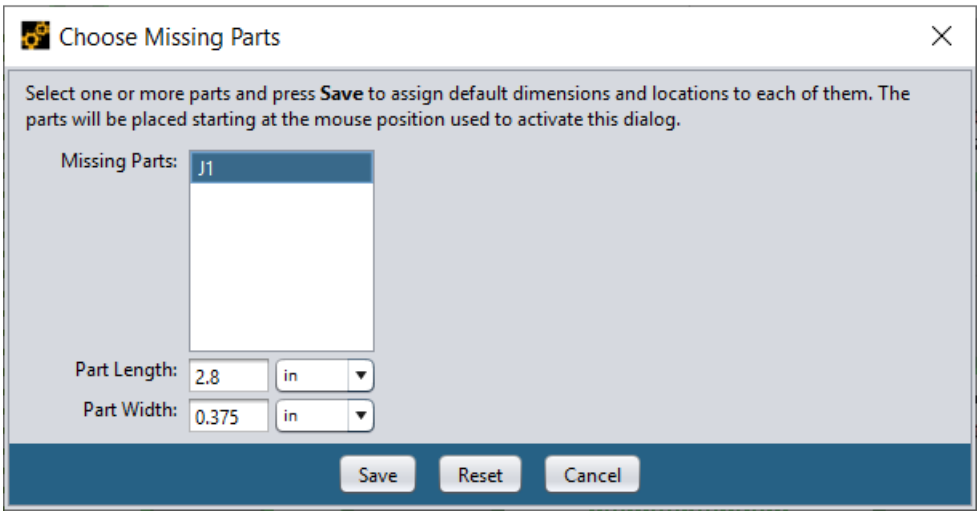
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.5.11.1. Missing Components

Components missing from the component layers are almost always missing because there are no dimension or location properties assigned to that part. In some cases, the location data is simply not provided in the Pick & Place file (often happens with through-hole parts). In other cases, no package data is known, and the user forgot to review the Parts List when importing the BOM. Fear not, however, Sherlock provides a convenient mechanism for handling such issues.

You may recall, the J1 part remained unmodified when you reviewed the **Parts List**, even though it was missing dimensional data. You can now correct that problem in the **Component Editor**, while at the same time placing it where it belongs.

Select the top component and solder mask layers for display, while in the Component Editor mode. Then, right-click the mouse outside of all component outlines near the left side of the board (roughly where the J1 connector belongs) and select the **Add Missing Parts** menu option. At that point, the **Choose Missing Parts** dialog will appear, showing a list of all parts for which size and/or location data is missing. In this case, only the J1 part is listed, but there could be a lot more missing parts in real-world examples.



Select one or more of the parts listed and then enter the dimension data to be used for those parts. Usually, you would select parts of the same type, but you can process them however you want.

Press **Save** to assign the size data to those parts and display them on the layer in the location where the right-mouse click took place. The parts will be automatically selected so that you can immediately start dragging them to their proper locations.

Once you're done moving those parts, press **Apply** to save the changes to the **Parts List**. At that point, you can repeat the process to select another batch of missing parts, set their size and move them into position.

2.5.11.2. Components in the Wrong Place

Since Pick & Place files don't provide Sherlock with the actual outline of any components, the location provided for a given part could be anywhere relative to the part outline. Fortunately, for surface mount parts, the location is almost always the center position of the part, but for through-hole parts the location is usually the pin 1 location and since Sherlock doesn't have the part outline, it doesn't know where that location should be. Moreover, most BOM and/or Pick & Place files don't

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

even specify if the part if surface mount or through-hole. So, Sherlock assumes it is dealing with surface mount parts and that the location provided is the center position. This works most of the time (because most parts are now surface mounted) but means that users creating Ad Hoc projects need to position through-hole parts manually.

In this tutorial, the only through-hole parts are the J1 connector on the left side of the board and the C21 through C25 round capacitors located on the top and bottom sides of the board. You fixed the J1 location in the previous exercise, so it remains to move the capacitors to their proper location. To position them properly, simply select each of them and drag them into position, using the top solder mask and/or top silk screen as your guide. Feel free to zoom in for more accurate placement.

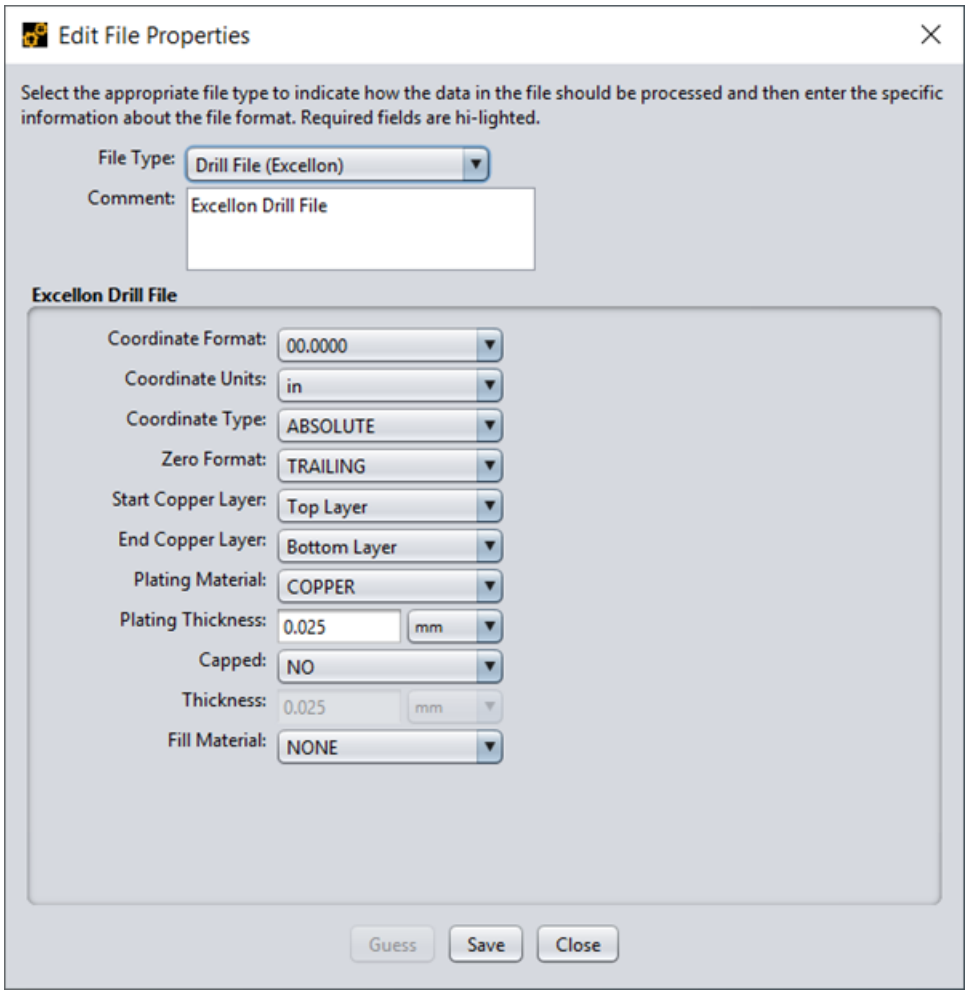
When you're done moving the through-hole parts, remember to press the **Save** button at the bottom of the **Component Editor** to update the **Parts List** and close edit mode.

2.5.12. Reviewing Drill Holes

The last review task associated with ad hoc projects concerns drill hole sizes and placement. Unlike ODB++ archives which use consistent coordinate specifications in all files, the coordinate systems used by drill files and layer files in an ad hoc project may be different. The most common difference involves how decimal values are stored in Excellon drill files. To see if your drill file has been imported correctly, single-click the **Drill Holes** folder in the Layer Viewer to expand the drill hole layers. At that point, you'll see a cluster of holes being displayed between the U9 and U10 components in the center of the board. Clearly, something is wrong and must be corrected. The key thing to notice is that the holes look like they are centered properly (that is, all files use the same origin location), but they appear to be scaled smaller. Such a situation usually indicates that you chose the wrong numeric format when you imported the file.

To correct the problem, right-click the **drill.exc** file in the **Project Tree** of the main window and select **Edit Properties.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



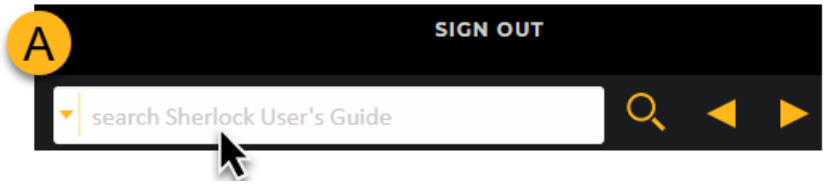
You can see from the file properties form that you initially specified a format of 00.0000 for coordinates (the default format). Since the hole positions are less than expected, you need to increase the number of integer parts in the format by selecting the 000.000 coordinate format. For example, if Sherlock previously read the string "123456" and parsed it as "12.3456" using the 00.0000 format, now Sherlock will parse that string as "123.456", multiplying all coordinates by a factor of 10. Also, ensure that the **Coordinate Units** are specified as **in** since you know the whole project uses non-metric values. Press **Save** to try that setting and re-parse the hole locations (and sizes). After the file processing has completed, re-display the drill holes in the Layer Viewer and you'll see them in the proper locations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: User Reference

In this chapter, you will find a topical list of some of Sherlock's features.

If you don't find the topic you are looking for in the list below, it's probably covered in another section of the *User's Guide*. Refer to the table of contents (p. 1) or use the Ansys search feature (A) at the top of the web page to locate the content you're looking for.



Tip:

If you are new to Sherlock, working through the Introductory Tutorials (p. 7) is an ideal way to become acquainted with Sherlock's features.

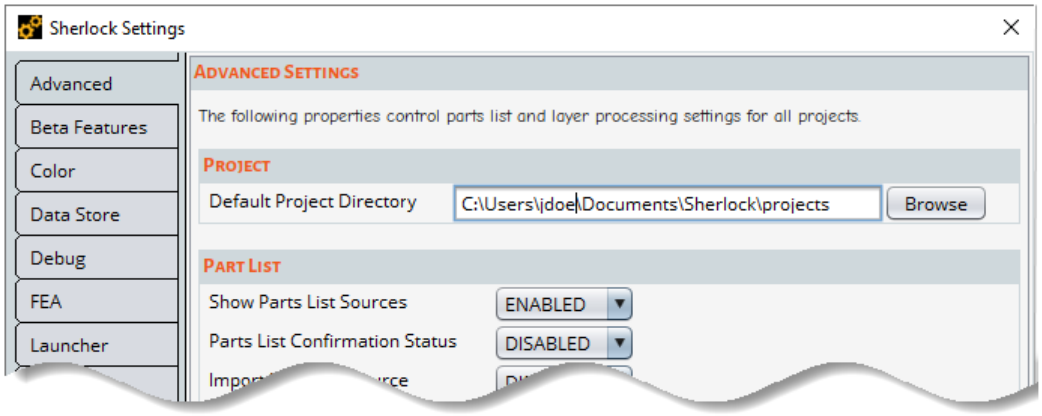
Topics Covered in this Chapter:

3.1. Sherlock Settings
3.2. User Interface Overview
3.3. Single and Multi-Project Mode
3.4. Circuit Card Assembly Files
3.5. Layer Viewer
3.6. Importing Projects and ECAD Archives
3.7. Exporting Projects
3.8. Stackups and Copper Layers
3.9. Laminate Management
3.10. Solder Management
3.11. Material Management
3.12. User Data Files and Documents

3.1. Sherlock Settings

You can access **Sherlock Settings** by clicking **Settings** on the **Main Menu** and then selecting any of the listed entries.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



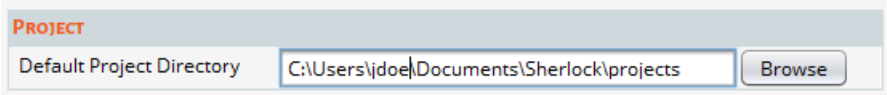
- Advanced Settings (p. 78)
- Beta Features Settings (p. 80)
- Color Settings (p. 81)
- Data Store Settings (p. 82)
- Debug Settings (p. 83)
- FEA Analysis Settings (p. 84)
- Launcher Settings (p. 84)
- Meshing Settings (p. 85)
- Part Library Settings (p. 87)
- Report Settings (p. 88)
- Score Settings (p. 88)

3.1.1. Advanced Settings

The **Advanced Settings** dialog has several sections, each explained below.

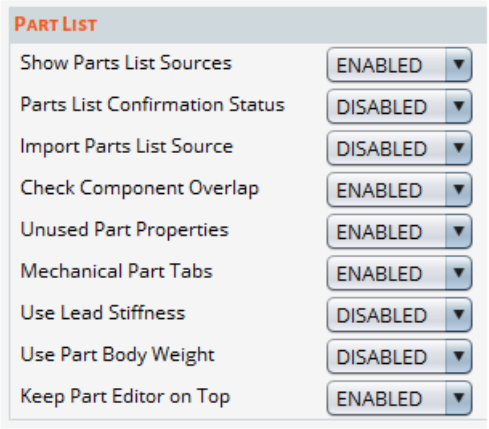
Project

Default Project Directory: Select the default directory where Sherlock projects are saved.



Part List

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Show Parts List Sources: When ENABLED, Part Property sources will be displayed in the **Parts List** and the **Part Editor**. See Part Source Priority (p. 39).

Parts List Confirmation Status: When ENABLED, the confirmation status of a part's properties is displayed in the parts list. See Confirming Parts (p. 42).

Import Parts List Source: When ENABLED, you will have the option to prioritize the source of the imported part properties as **User** instead of **BOM** in the **Parts List Import** dialog. See Importing the Parts List (p. 224).

Check Component Overlap: When ENABLED, Sherlock will display a warning when it detects overlapping components in a circuit card assembly.

Unused Part Properties: This setting affects how the **Part Properties Viewer** and **Editor** display properties that are not applicable to a selected part. When ENABLED (default), these unused properties are displayed and editable. Select DISABLED if you want unused part properties to remain visible but not editable. Select HIDDEN to hide unused part properties altogether. See Part Property Viewing/Editing Options (p. 199) for more information.

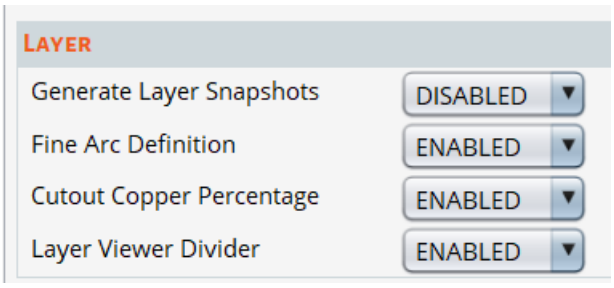
Mechanical Part Tabs: When ENABLED, all **Part Editor** tabs are enabled for Mechanical Part (p. 629) component types.

Show Lead Stiffness: When ENABLED, Lead Stiffness will be displayed in the **Part Properties Viewer** and the **Part Editor**. (See Managing Part Properties (p. 32) and The Part Editor (p. 200).) See also the lead stiffness entry in Sherlock Part Properties (p. 294).

Show Part Body Weight: When ENABLED, Sherlock will display part body weight in the **Part Editor** and use it in FEA analysis. If this property is not displayed, Sherlock will automatically compute the weight.

Keep Part Editor on Top: When ENABLED, all open Part Editor and Viewer dialogs remain on top of the main Sherlock application window.

Layer



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Generate Layer Snapshots: This feature (when ENABLED) generates image snapshots when importing layers. This improves **Layer Viewer** performance, but it may reduce the resolution of large layer models.

Fine Arc Definition: When ENABLED, Sherlock generates smoother arcs definitions when converting Gerber, ODB++, and IPC-2581 files.

Cutout Copper Percentage: Enabling this setting will subtract the area of cutouts from the total PCB area when Sherlock computes the copper percentage of a layer.

Layer Viewer Divider: ENABLING this feature adds a moveable window divider to the Layer Viewer, separating the filter options on the left side and the graphical content on the right side. See Layer Panning and Zooming (p. 46).

Life Cycle



Unsaved Results Alert: When ENABLED, Sherlock will display an alert if you are in danger of losing unsaved analyses when you choose to apply Life Cycle changes.

General



Beta Features: When ENABLED, the **Beta Features** tab appears in the **Sherlock Settings** window. See below, Beta Features Settings (p. 80).

Start Task Monitor: When ENABLED, the Sherlock task monitor will open automatically when the Sherlock application launches.

Precise Unit Conversion: When enabled, Sherlock uses more precise methods of performing unit conversions. This may result in slower file processing and analysis.

Trace Modeling

See Advanced Trace Modeling Settings (p. 499) for details on this section of the **Advanced Settings** dialog.

3.1.2. Beta Features Settings

By default, the **Beta Features** tab is not displayed in the **Sherlock Settings** window. To activate the tab, do the following: In the **Sherlock Settings** window, click the **Advanced** tab. Then under **General**, select ENABLED for **Beta Features** (see above (p. 80)).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Beta Features** panel allows you to enable or disable individual beta features available in Sherlock.

> **Warning:**
>
> Before enabling any beta feature, read the Beta Features Disclaimer.



3.1.3. Color Settings

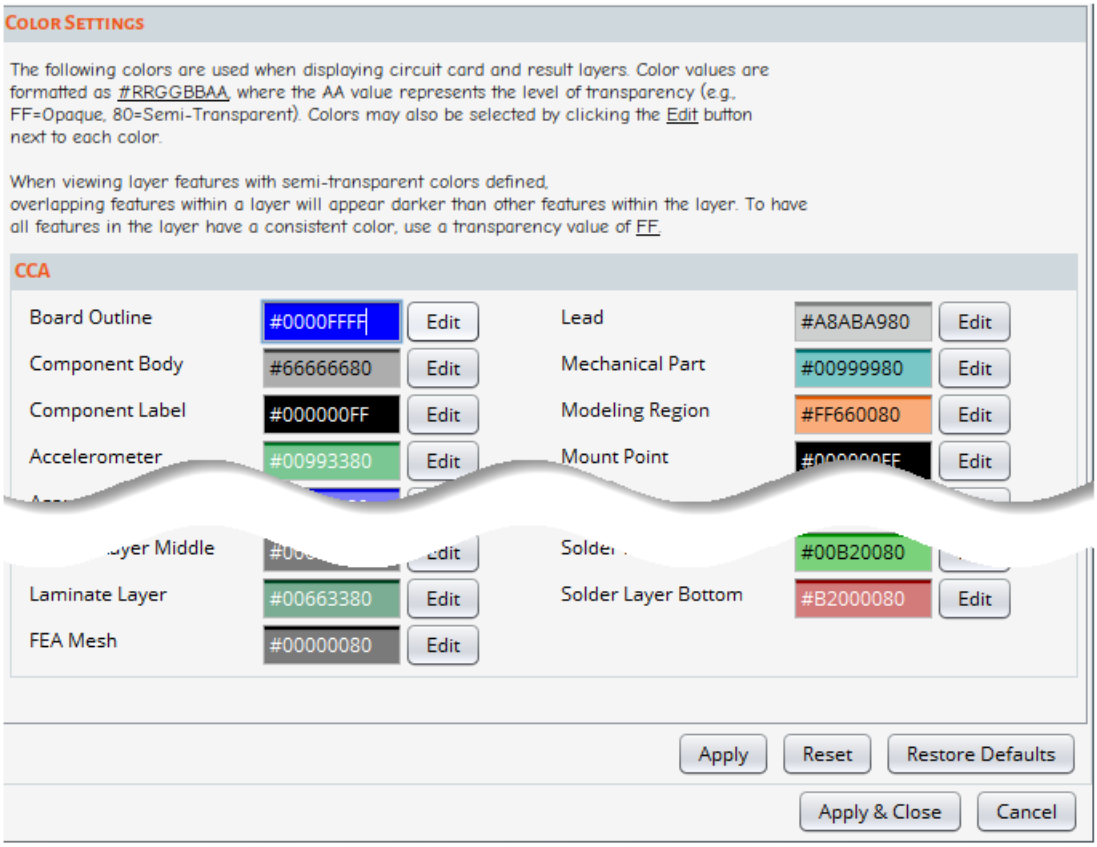
The **Color Settings** affect how circuit card and result layers are displayed. These settings can be configured with the color of your choosing. Color values are formatted as #RRGGBBAA, where AA represents the level of transparency (FF=Opaque, 80= Semi- Transparent). Colors may also be selected by clicking the **Edit** button next to each color.

> **Note:**
>
> Transparency settings and the **Layer Viewer** (p. 277): When objects are defined with semi-transparent colors and they overlap within the same layer, they will appear darker than other layer objects. If you prefer that all layer objects have a consistent color, use a transparency value of FF (opaque).

Use the buttons in the dialog to save or restore settings:

- **Apply:** Applies and saves changes to color assignments.
- **Reset:** Resets color assignments to their last saved values.
- **Restore Defaults**: Restores color assignments to default values. When you click the **Restore Defaults** button, Sherlock will prompt you to confirm your decision. If you click **Yes**, all color assignments will return to their default settings. *Once confirmed, this action cannot be undone.*

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

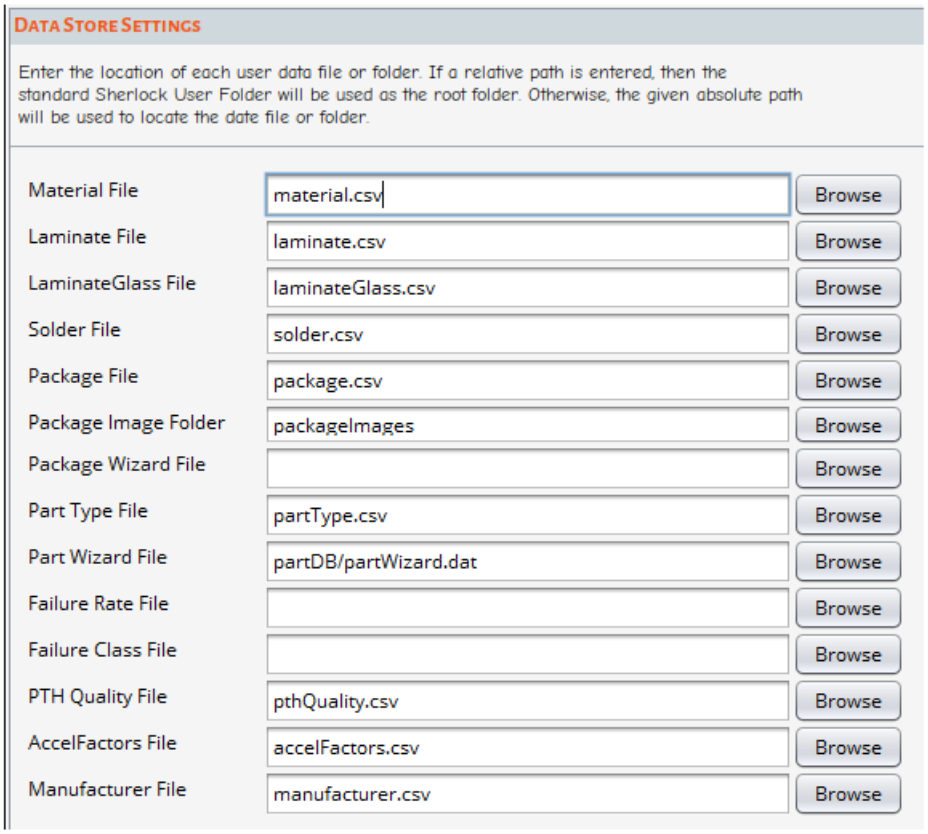


3.1.4. Data Store Settings

The **Data Store Settings** control the location of each user data file or folder. If a relative path is entered, the standard Sherlock User Folder will be used as the root folder. Otherwise, the specified absolute path will be used to locate the data file or folder.

The user data files or folders listed in the **Data Store Settings** are: the Material File, Laminate File, LaminateGlass File, Solder File, Package File, Package Image Folder, Part Type File, Part Wizard File, PTH Quality File, AccelFactors File, Manufacturer File, and the Meshed Part Folder.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

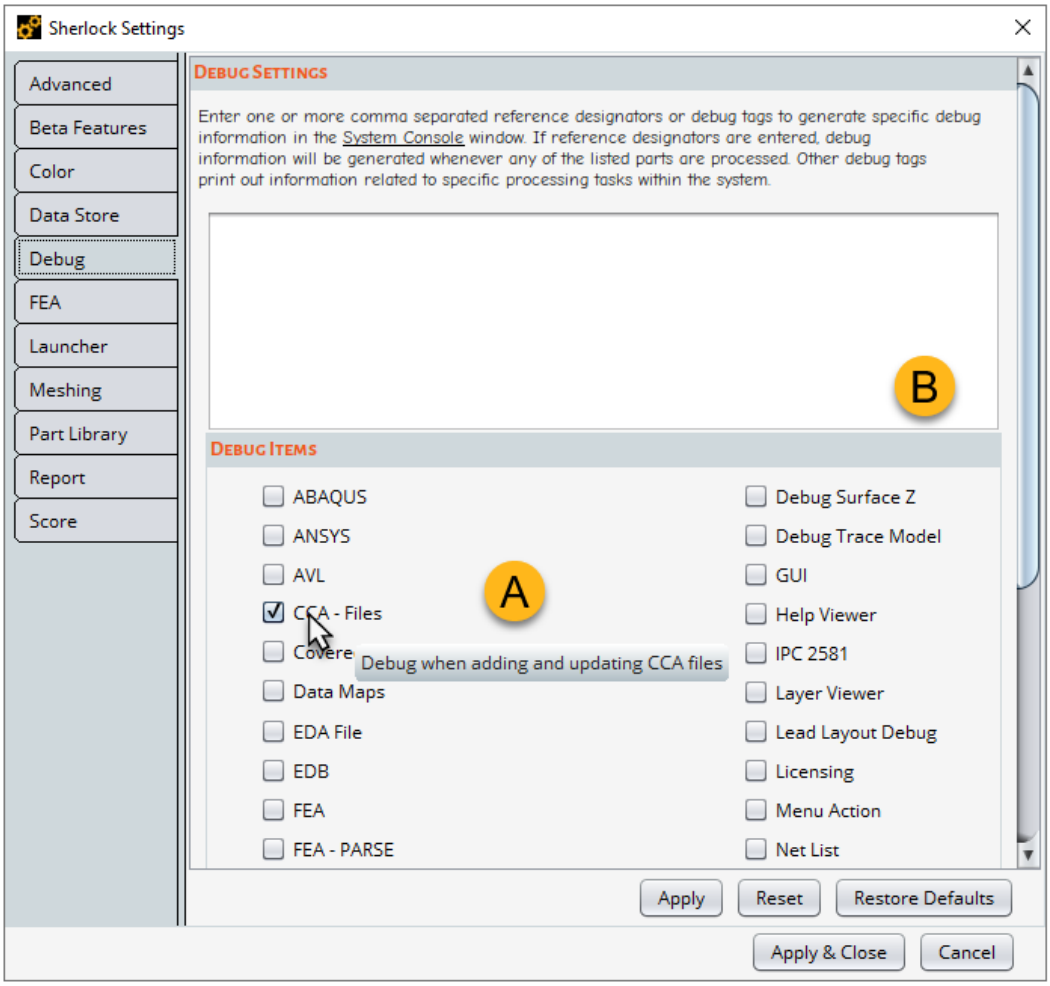


3.1.5. Debug Settings

The **Debug Settings** window allows you to activate debug tags which generate debugging information in Sherlock's Client Console. The tags are listed in the **Debug Items** panel (A, below). Hover over a tag to see a tool tip that briefly explains its function. To activate a tag, click the appropriate check box and click the **Apply** or **Apply & Close** button.

The text field (B, below) serves a couple of functions: You can manually enter a debug tag not listed on the **Debug Items** panel. Also, you can have Sherlock generate debugging information for a specific part. Simply enter its Reference Designator in the text field. Separate multiple entries with a comma. Sherlock will provide details about the component(s) in the Console during an FEA or Solder Fatigue analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



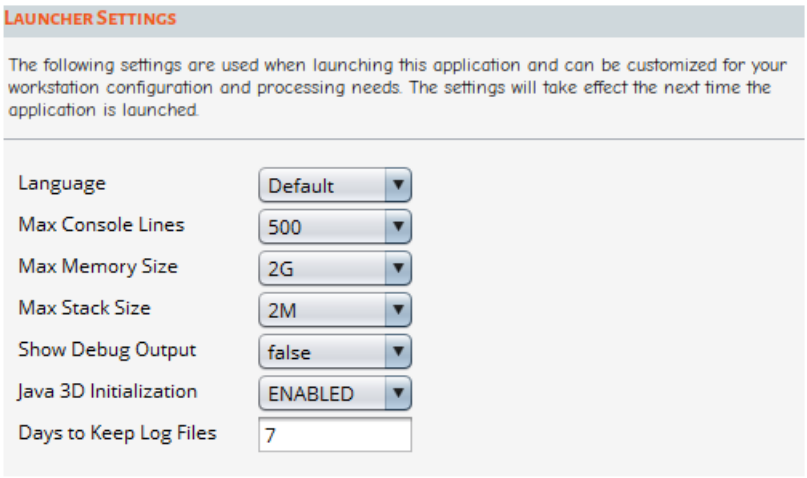
3.1.6. FEA Analysis Settings

For a detailed explanation, see FEA Analysis Settings (p. 446).

3.1.7. Launcher Settings

The **Launcher Settings** are used when launching Sherlock. Any setting changes will take effect the next time the application is launched.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Language: Select the language used in the application. Sherlock is currently supported in English, Japanese, and Korean. See limitation below.

Note:

The language fonts in Sherlock do not support one another. For example, if you name a project using Japanese characters and then change the **Language** to **English**, the Japanese characters will not be visible, and the resulting text will not be correct.

Max Console Lines: Specify the maximum number of lines to retain in the Console (p. 4).

Max Memory Size: Specify the maximum amount of memory Sherlock uses for processing. These settings apply also to Sherlock's 3D Viewer and the Results Viewer clients. The settings do not affect memory usage at start-up, however, but only during high-demand processing as needed. Remember, you must restart Sherlock for any changes to take effect. For specific information on memory usage during FEA process, see FEA Processing (p. 463).

Max Stack Size: Specify the maximum Java stack size. Typically, this setting does not need adjusting.

Show Debug Output: This enables additional application debug messages to be displayed on the Sherlock Console (p. 4).

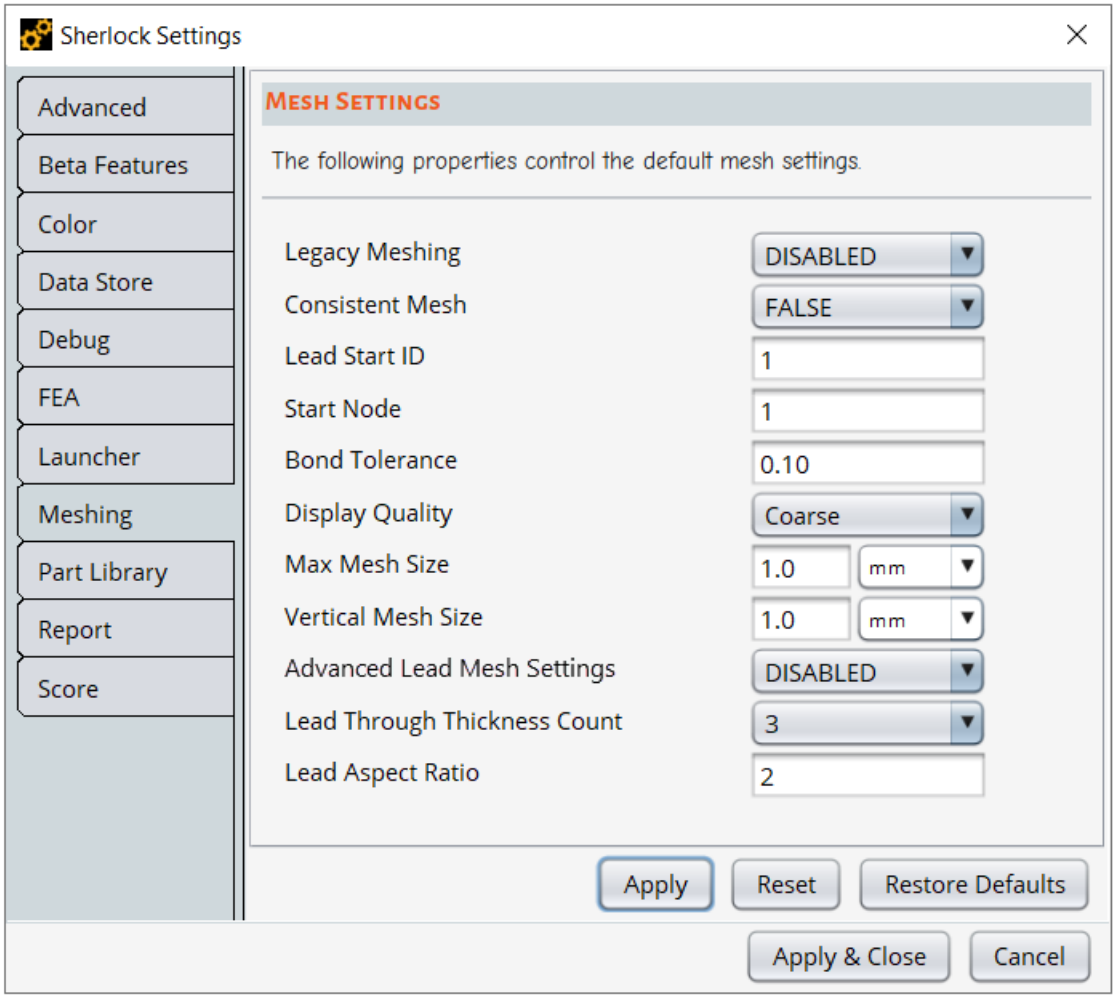
Java 3D Initialization: When DISABLED, Sherlock will not initialize the 3D system prior to needing it. Leaving it ENABLED reduces the initial startup time when using the Part Editor and Viewer. This should only be DISABLED if Sherlock generates a "GDI Generic Driver use detected" message.

Days to Keep Log Files: Sets the number of days Sherlock preserves the log files.

3.1.8. Meshing Settings

To access the Mesh Settings within Sherlock, select **Settings > Meshing** from the main menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Legacy Meshing: By default, Sherlock uses Ansys Meshing (Bonded). If you set **Legacy Meshing** to ENABLED, you have the option of using Sherlock's legacy meshing (Merged) when configuring a model for Finite Element Analysis.

Consistent Mesh: When ENABLED, Sherlock generates the same mesh for a given circuit card when meshing parameters and geometry are unchanged.

Lead Start ID: This value sets the ID of the first lead in the FEA Model. **Lead Start ID** does not affect the mesh. It only impacts the numbering of the node. The default value is 1.

Start Node: This value sets the ID of the first node in the FEA Model. **Start Node** does not affect the mesh. It only impacts the numbering of the node. The default value is 1.

Bond Tolerance: Sherlock uses Constraint Equations (CE) to establish bonded contacts. This value sets the bond tolerance in mm when generating FEA bonds. The default value is 0.10 mm.

Display Quality: This option sets the display quality of the 2D Graphical Result Layers. Options include **Coarse** and **Fine**. When **Coarse** is selected, each triangular or quad face in the mesh will be rendered using a single color representing the average of the three or four corner values. The **Fine** setting results in smoother edges between colored regions by using smaller triangles and quads using the midpoint method. This setting has no effect on the FEA results. It simply changes the appearance of

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the 2D Results Layer. The default is **Coarse**. See Graphical Result Layers (p. 468) to see how **Display Quality** affects the 2D Graphical Result Layers.

Max Mesh Size: This value sets the default mesh size, indicating the maximum length and width of an element. This setting is only used if there are no values in the analysis mesh settings.

Vertical Mesh Size: Sets the default vertical mesh size. This setting is only used if there are no values in the analysis mesh settings.

Advanced Lead Mesh Settings: When ENABLED, the Advanced Lead Mesh Settings give you more options for modeling leads such as the ability to specify how many elements define the thickness and width of the lead. Once enabled, Sherlock applies the ALM settings to the model when generating the 3D FEA model, when running an FE analysis, or when generating a 3D model of a component in the **Part Viewer/Editor**. Enabling this feature increases processing time, however. By default, Advanced Lead Meshing is DISABLED. For a detailed explanation of Advanced Lead Meshing, see Advanced Lead Meshing (p. 566).

Lead Through Thickness Count: The setting applies only when **Advanced Lead Mesh Settings** is enabled. It sets the number of elements through the lead thickness and width when Sherlock models a lead. The default value is three, and the maximum is five. See note below.

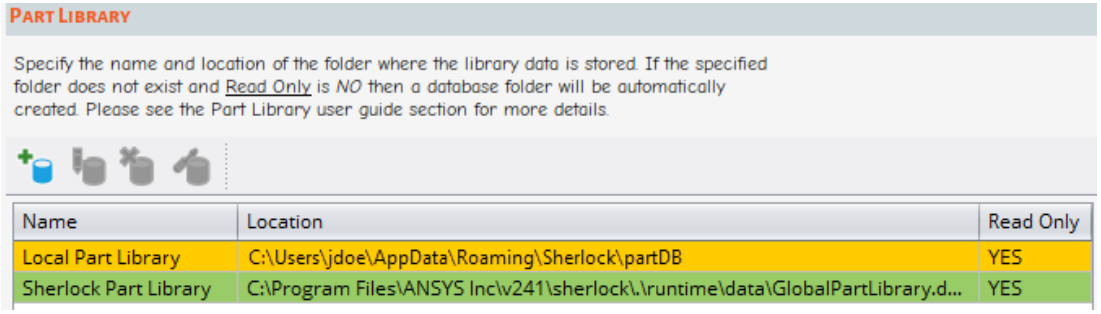
Lead Aspect Ratio: The setting applies only when **Advanced Lead Mesh Settings** is enabled. When Sherlock generates a lead model, the Lead Aspect Ratio (LAR) determines the ratio between the elements' thickness and height. The default value for the LAR is two and the maximum is ten. See note below.

Note:

You can also set the **Lead Through Thickness Count** and the **Lead Aspect Ratio** in the **Lead Modeling** tab of the **Analysis Properties** window when setting up your FE analysis. Until you run your first analysis with **Advanced Lead Meshing** enabled, Sherlock defaults to the values you have chosen in the **Sherlock Settings** window. For a more detailed explanation of Advanced Lead Meshing and guidance on choosing optimal settings, see Advanced Lead Meshing (p. 566).

3.1.9. Part Library Settings

Here you can specify the name and location of the folder where the library data is stored. If the specified folder does not exist and **Read Only** is NO, then a database folder will be automatically created.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1.10. Report Settings

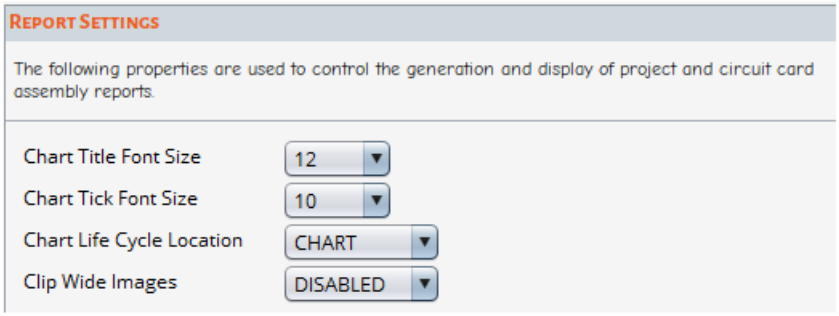
The **Report Settings** control how project and circuit card assembly reports are displayed. See Generating Report Files (p. 800).

Chart Title Font Size: This value specifies the font size used in chart titles.

Chart Tick Font Size: This value specifies the font size used in chart axis labels.

Chart Life Cycle Location: This value specifies the location of the service life goal in life prediction charts. Options include CHART or SUBTITLE.

Clip Wide Images: DISABLED by default. When ENABLED, Sherlock clips Layer images that are wider than their height to preserve space.



3.1.11. Score Settings

Use the **Score Settings** dialog to set threshold levels for score-related results. The thresholds you can customize are **Result OK**, **Result Warning**, and **Result Error** which correspond to Green, Yellow, and Red respectively. You can see a sample score card here (p. 437).

Score Settings apply to the following analyses: Thermal Derating, PTH Fatigue, Harmonic Vibration, Random Vibration, Mechanical Shock, Solder Fatigue, Semiconductor Wearout, and ICT Analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

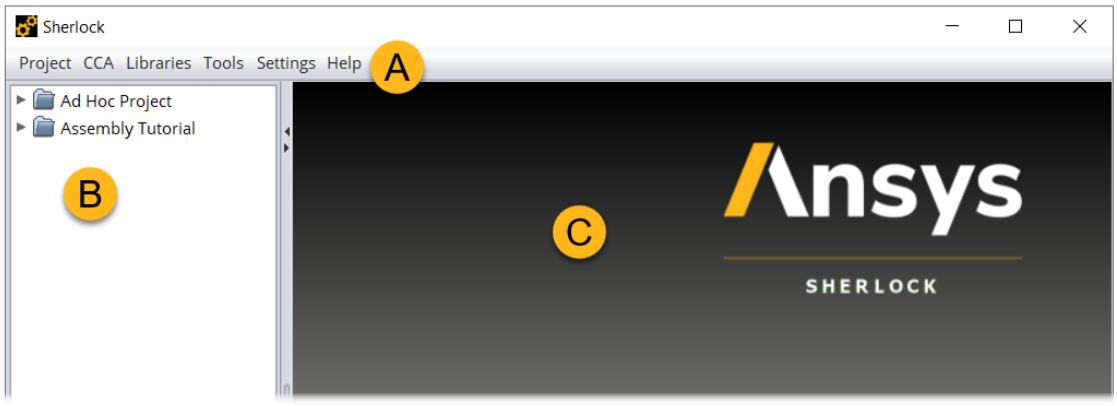


3.2. User Interface Overview

Topics:

3.2.1. Main Window
3.2.2. Project Tree
3.2.3. Main Menu

3.2.1. Main Window

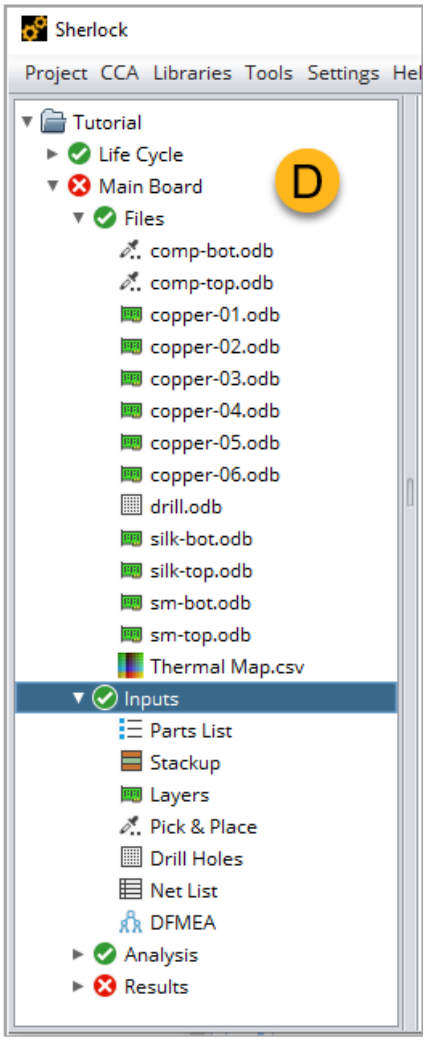


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sherlock's **Main Window** is divided into three sections, with (A) the **Main Menu** along the top, (B) the **Project Tree** along the left side, and (C) the **Content Panels** occupying the rest of the window. There is a vertical divider between the **Project Tree** and the Content Panels that allows you to make the **Project Tree** panel wider or narrower to suit your purposes. You can also resize the **Main Window** at any time using standard window controls. When Sherlock is first installed, the **Project Tree** is empty until you import or create your first project.

3.2.2. Project Tree

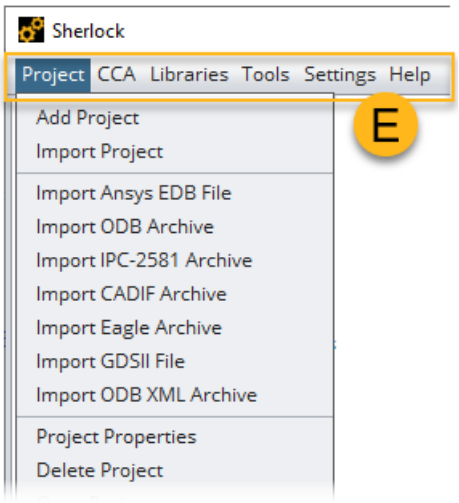
You will find a guided explanation of Sherlock's **Project Tree** (D) in the second tutorial, *ODB++ Project Creation*. See Importing a Project (p. 8) and the sections that follow.



3.2.3. Main Menu

This section provides brief explanation of Sherlock's **Main Menu** system (E) as well as links to sections where you will find a more detailed treatment of each menu item.

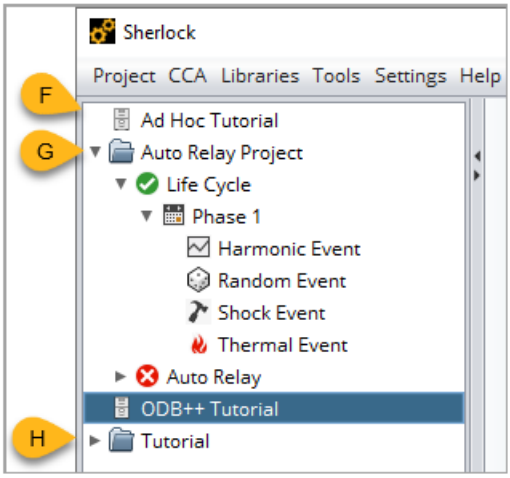
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Project menu

- **Add Project**: Click this item to start a new project in Sherlock. Typically, you choose this option when manually building a project by adding individual CCA files as opposed to importing a single project file. See Project Creation (p. 57) in the tutorial *Ad Hoc Project Creation* where the **Add Project** feature is demonstrated.
- **Import Project**: Click to import a Sherlock project (`.zip`). See How to Import a Project into Sherlock (p. 119).
- **Import Ansys EDB** to **Import EDB XML Archive**: Use these menu items to import a project from one of the industry-standard file types. See Importing Projects and ECAD Archives (p. 118).
- **Project Properties**: Click to open the **Edit Project Details** window where you can edit the Project Name, Project Category, and Description. See Project Name and Properties (p. 30).
- **Delete Project**: To delete an entire project from the **Project Tree**, highlight the project and then select this menu option. Sherlock will ask you to confirm your decision. You *cannot* undo this operation. Only open projects can be deleted. (Open projects have the folder icon, not the file cabinet icon.)
- **Open Project:** Click this option to open a closed project. An open project (G, below) has the folder icon displayed, and the project resources are loaded into Sherlock's system. See **Close Project** below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Close Project**: Click this option to close a project and conserve Sherlock application resources. A closed project has the file cabinet icon (F, above). For comparison, a *collapsed* project folder (H) hides the project subfiles but is still using application resources.
- **Export Project**: Click to export a Sherlock project (`.zip` or `.dfr-result`). See Exporting Sherlock Projects (.zip, .dfr-result) (p. 131).
- **Add Circuit Card**: Click to add a new circuit card to your project. See Circuit Card Creation (p. 58) in the *Ad Hoc Project Creation* tutorial.
- **Import Circuit Card**: Import a circuit card that has been exported from Sherlock.
- **Project Report**: View, generate, or clear a Project Report. See Reporting and Data Export (p. 799).
- **Clear Project Results**: Clear unsaved project results.

3.3. Single and Multi-Project Mode

Ansys Sherlock can be used in two different modes: **Multi-Project** and **Single-Project Mode**. In **Multi-Project Mode**, Sherlock displays all the projects stored in the Sherlock **User Data Folder** in the **Project Navigation Tree**, allowing users to manage one or more projects in the same application. In **Single-Project Mode**, Sherlock displays only a single project in the **Project Navigation Tree**, but that project can be stored anywhere in the file system. This section describes how each of these modes can be used to manage Sherlock projects.

In this section, the following topics are covered:

3.3.1. Multi-Project Mode
3.3.2. Single-Project Mode
3.3.3. Changing Project Directory

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3.1. Multi-Project Mode

By default, Sherlock launches in Multi-Project Mode when you start Sherlock from the **Windows Start Menu**. (The Sherlock icon is located within the Ansys folder on the **Windows Start Menu**.) When operating in Multi-Project Mode, Sherlock allows you to manage all projects located in the projects sub-folder located here:

```
C:\Users\USERID\Documents\Sherlock\projects
```

where USERID is your Windows user ID. You can change the default project directory in Advanced Settings (p. 77).

For Linux users, the default, multi-project folder is located here:

```
$HOME/Sherlock/projects
```

3.3.1.1. Closing Projects

Projects can be opened or closed when operating in **Multi-Project Mode**. When a project is opened, Sherlock loads all the information needed to display the **Project Tree** entries for that project, as well as project-specific data stores, such as the **Parts List** and **Stackup** data. Furthermore, as project data is examined, Sherlock may retain additional data in memory, such as recently viewed 2D layers. Over time, the amount of memory used by a project increases and if multiple projects are open, and each project contains multiple circuit cards, the amount of memory consumed may be quite large.

To help reduce memory usage and trim down visual clutter in the **Project Navigation Tree**, you should close projects when not currently being used. This is done simply by right-clicking the project node in the navigation tree and selecting **Close Project** from the context menu. At that point, Sherlock will release all memory used by the project and will display only the project node in the navigation tree. You can re-open a closed project by right-clicking the project node and selecting **Open Project** from the context menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3.2. Single-Project Mode

Single-Project Mode provides an alternative way to launch the Sherlock application. Single-Project Mode allows users to store the project folders anywhere in the file system and manage them with standard file system tools such as Windows Explorer. Existing projects in the **Projects** folder of the Sherlock **User Data Folder** can be copied or exported to other locations and opened individually when using Sherlock in **Single-Project Mode**. Single-Project Mode not only reduces the visual clutter in the **Project Navigation Tree** (because only a single project is displayed), but it also allows you to do side-by-side comparisons of two projects by launching two instances of Sherlock. **Single-Project Mode** is the default mode used when Sherlock is executed by double-clicking the **Sherlock Single Project** desktop icon or by selecting **Sherlock Single Project** from the Windows Start Menu.

Note:

For Linux users: By default, you cannot open Sherlock files by double-clicking them. You must manually associate file types to specific applications within the operating system.

3.3.2.1. Launching Sherlock

All Sherlock project folders contain a project link file named **a.dfr-project**. The project link file doesn't contain any data, it serves solely to identify a Sherlock project folder and provide a mechanism for easily launching Sherlock in Single-Project Mode.

Note:

A project link file will be automatically created by Sherlock whenever it opens a project folder created by an older version of Sherlock.

For convenience, Sherlock can be launched in **Single-Project Mode** using any of the following methods (Windows version only):

- Double-click a project link file.
- Drag a project link file onto the Sherlock desktop shortcut icon.
- Drag a project folder onto the Sherlock desktop shortcut icon.
- Use the desktop shortcut **Sherlock Single Project.**
- Use the Windows **Start Menu** item **Sherlock Single Project**, *or*
- Create one or more custom desktop shortcuts to open specific projects.

The first five launch methods are self-explanatory and will not be discussed further. The last launch methods involve the creation or modification of Windows desktop shortcuts.

Linux Users

To launch Sherlock in single-project mode in Linux, use the command:

```
runSherlock -singleProject
```


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to open a specific project in single-project mode directly from the command line, use the command:

```
runSherlock -singleProject/path-to-project-directory
```

where "path-to-project directory" is the path name of the directory containing your project.

By default, you cannot open Sherlock files by double-clicking them in Linux. You must manually associate file types to specific applications within the operating system. The procedure for doing this depends on the windowing environment you are using.

3.3.2.1.1. Creating a Custom Project Shortcut

In Windows, the easiest way to create a custom desktop shortcut for a specific project is to copy the standard Sherlock desktop shortcut and modify the **Target** entry to indicate the project to be opened. For example, create a copy of the Sherlock shortcut on the desktop, right-click the new shortcut and select **Properties** from the context menu. Select the **General** tab and edit the shortcut name to indicate the project being opened (for example, My Project). Then, select the **Shortcut** tab and append the full path of the project folder to the **Target** entry. For example, if the project folder is located at:

```
C:\test\My Project
```

then you should modify the **Target** entry to be:

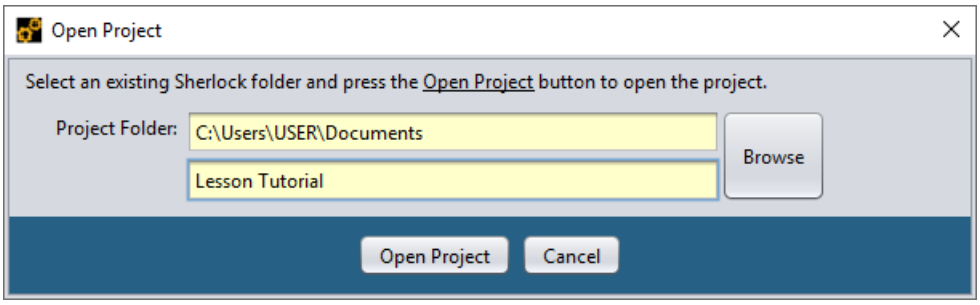
```
C:\Program Files\Ansys Inc\v251\sherlock\SherlockClient.exe
C:\test\My Project
```

Press the **Apply** or **OK** button to save the changes and then double-click the **icon** to launch Sherlock in **Single-Project Mode**, automatically loading the designated project.

3.3.2.2. Project Menu

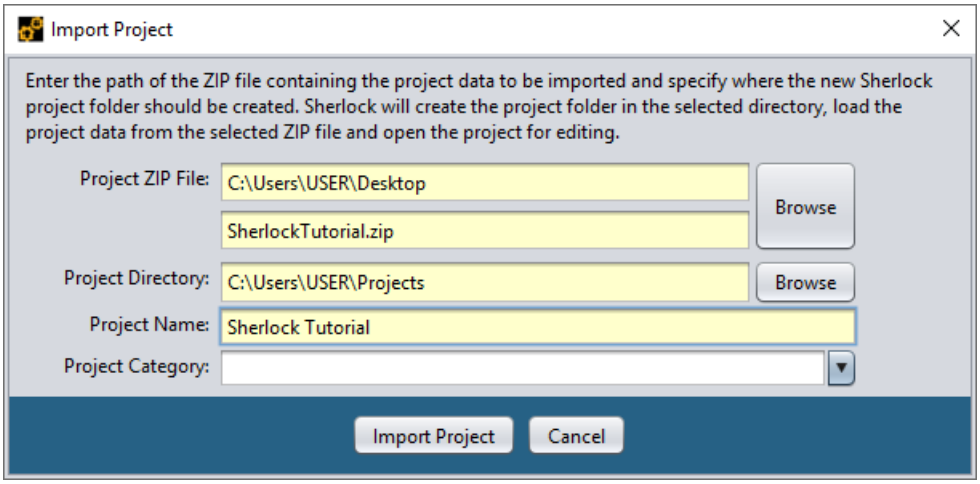
When operating Sherlock in **Single-Project Mode**, the **Project Menu** behaves differently than when used in **Multi-Project Mode**. Specifically, if a project is currently open, then all menu options that can be used to load a project (that is, **Open Project**, **Create Project**, **Import Project**, for example) are disabled. In that case, you will only be allowed to export or close the project from the **Project Menu**. If you close the project, using the **Project > Close Project** menu option, then all memory will be released, the project navigation tree will be cleared, and the menu options associated with loading a project will be enabled.

To open an existing project folder, select **Project > Open Project**.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The dialog that appears allows you to browse the file system to locate the project folder to be opened. Once the project folder is selected, click **Open Project** to load the project into Sherlock. As a convenience, the **Open Project** dialog remembers the last project folder location designated so that you can quickly return to your collection of projects, wherever they may be in the file system.



When creating a new project or importing a project using Single-Project Mode, the location of the project directory for the new or imported project is specified in a similar manner. For example, to import the Tutorial project provided in the Sherlock Installation Folder, select **Project > Import Project** from the Main Menu. You may also specify a Project Category which is used to help organize projects by category in the new open project screen.

In this case, the import dialog contains two **Browse** buttons. The first is used to locate the ZIP file to be imported and the other is used to indicate where the project folder should be created. In the example shown above, the **Lesson Tutorial** project folder will be created on the desktop by importing the contents of the **Tutorial Project.zip** project archive located in the **tutorial** sub-folder of the **Ansys Installation Folder**.

3.3.2.2.1. Opening, Importing, and Library Loading

When opening or importing a project in **Single-Project Mode**, libraries (such as material, laminate, or solder libraries) that were included with a prior export will automatically be loaded if they are also associated with the new project.

3.3.2.2.2. Recent Projects Menu

In **Single-Project Mode**, the **Project Menu** contains a menu item named **Recent Projects**. Any recently opened projects appear in this menu as a convenient shortcut to open those projects. This menu item contains up to ten recent projects. The reset this list, select the **Clear History** menu item.

3.3.3. Changing Project Directory

To view your default project directory, select **Settings > General Settings** from the Sherlock **Main Menu** and view the **Advanced** side tab. Under **Project**, you can view your **Default Project Directory** or **Browse** to a new directory.

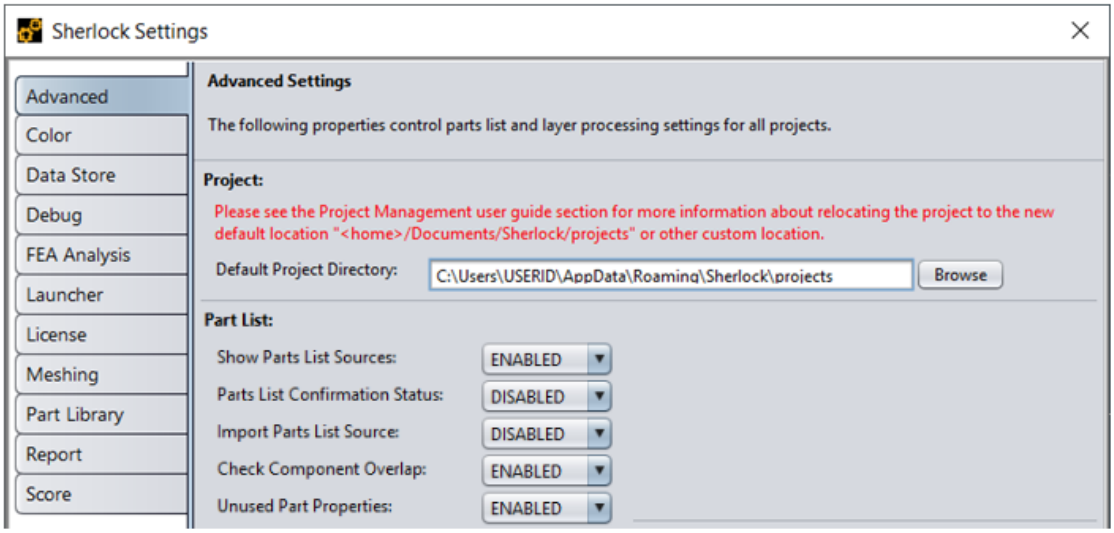
For new users, the **Default Project Directory** will be:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Windows version: `C:\Users\USERID\Documents\Sherlock\projects`

Linux: `$HOME/Sherlock/projects`

For users who are upgrading from a pervious version of Sherlock, you may see the message in red below.



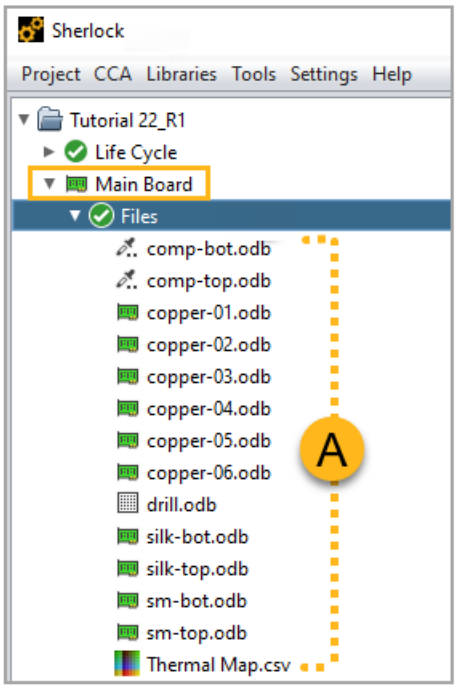
To change the default project directory, click the **Browse** button and select the intended location.

3.4. Circuit Card Assembly Files

As described in the Sherlock Tutorial Lesson 06 – Ad Hoc Project Creation (p. 56), Sherlock supports many file formats for importing data to define a circuit card assembly (CCA). Many of the supported file formats are governed by formal specifications such as ODB++, Gerber, and IPC-2581. However, Sherlock allows importing of other types of files such as CSV, Excel spreadsheets, and image files to define various aspects of a CCA. This section describes those files, their required format, and how to add the files to Sherlock.

Circuit card files (A, in the image below) are listed in the **Files** folder or in a **CCA** folder in Sherlock's **Project Tree**. In the example shown, the CCA is named **Main Board**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this section, the following topics are covered:

3.4.1. Circuit Card Information

3.4.2. CCA File Properties Viewer

3.4.3. Creating a Circuit Card

3.4.4. Adding Files to a Circuit Card Assembly

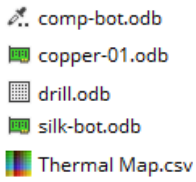
3.4.5. Attaching Informational (Info) Files

3.4.6. CCA File Format Specifications

3.4.1. Circuit Card Information

Sherlock maintains various information about each circuit card assigned to a given project, such as electronic design files (BOM, layer, drill, and layout files for example), input data derived from design files (such as the parts list and stackup file), tabular and graphical results for each analysis performed, and circuit card result summaries (such as the Score Card, Life Predictions, and Reports) The **Project Tree** organizes all the circuit card information, providing quick access to both high-level and low-level information.

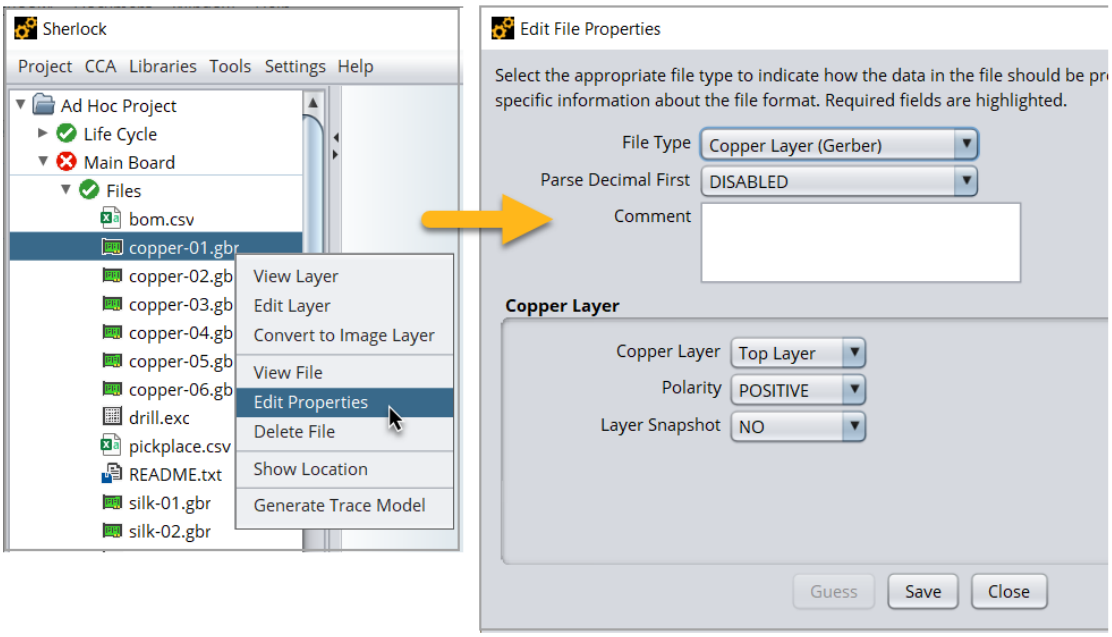
The icons associated with each entry in the **Project Tree** show the type of information to be found. In the image below, for example, several icons are shown: the icon for a components (bottom) file, a copper layer file, a drill file, a silk screen file, and a thermal map.



You can view and edit the properties of a CCA file by opening its **Edit File Properties** dialog as shown below. Simply right-click the circuit card file in the **Project Tree** and choose **Edit Properties** from

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the context menu. Each CCA file-type has its own set of properties that appear in the **Edit File Properties** dialog. For an overview of file-types and their properties, see the section below, CCA File Format Specifications (p. 103).



Tip:

When reviewing CCA file properties, opening and closing the **Edit File Properties** dialog for each file can be time consuming. To view all the properties at once for all files under a specific CCA, use the CCA File Properties Viewer (p. 99) as explained below.

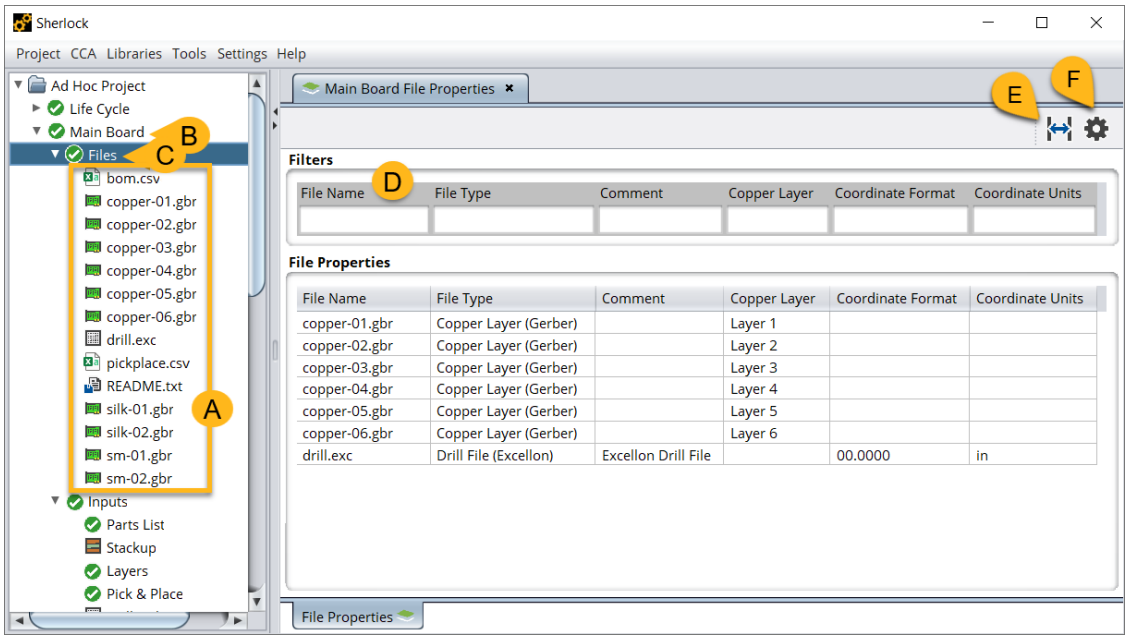
3.4.2. CCA File Properties Viewer

Sherlock allows you to view in table form the properties for copper, laminate, and drill files under a **CCA/Files** node (see A in the image below). It makes reviewing CCA file properties much easier than examining each file individually. The table is for *viewing* properties only. To *edit* a property, use the **Edit File Properties** dialog for the CCA file as explained above in Circuit Card Information (p. 98).

There are two ways you can open the table:

- Right-click the CCA in the **Project Tree** (**Main Board**, B, in the image below) or the **Files** subfolder (C) and select **File Properties** from the context menu.
- Or, highlight a CCA or any of its subfolders in the **Project Tree**, and then select **Main Menu > CCA > File Properties**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can customize the **File Properties** table in several ways:

- **Filtering**: You can filter the table contents by using the filter fields (D, above). For a detailed explanation on how to use the filtering feature in Sherlock's various tables, see Filtered Search Results (p. 243) under the section *Part Library*.
- **Resize Columns**: To resize the column widths to fit the current window, click the **Resize** icon (E, above).
- **Add or Remove Columns**: Clicking the gear icon (F, above) opens the **Customize File Properties** dialog. There, you can add or remove which columns you want displayed.

3.4.3. Creating a Circuit Card

For instructions on creating (rather than importing) a circuit card, see Circuit Card Creation (p. 58) in the tutorial *Ad Hoc Project Creation*.

3.4.4. Adding Files to a Circuit Card Assembly

Summary

Adding a file or files to the CCA of your project requires three steps, which will be explained in detail.

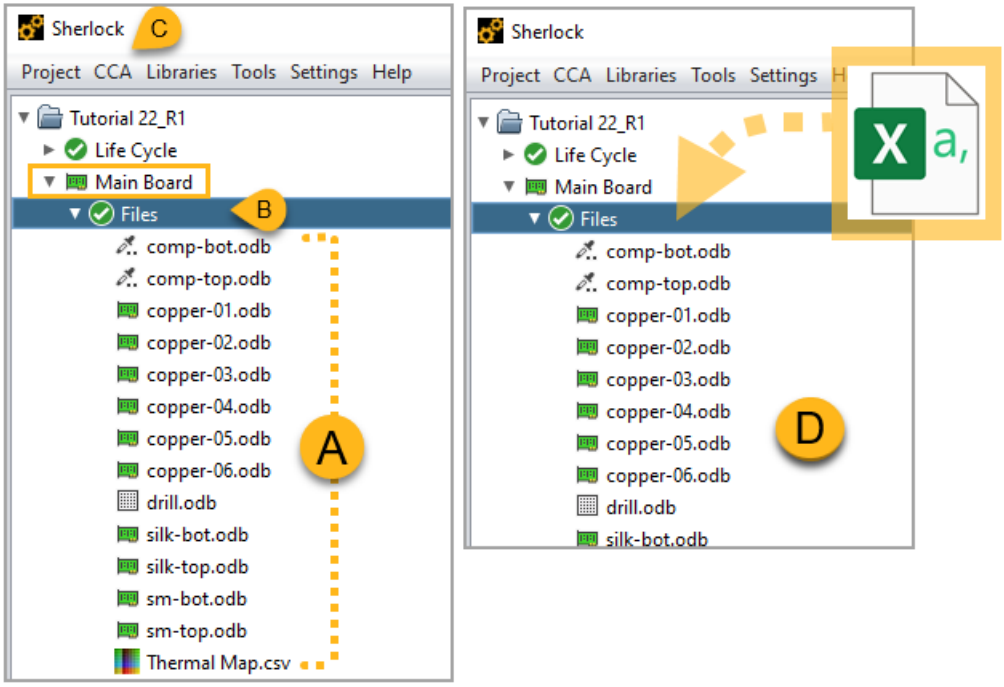
1. Add the files to the **Project Tree**.
2. When prompted by Sherlock, change the file name(s) to a standard format if you wish.
3. For each new file on the **Project Tree**, open the **Edit File Properties** dialog to configure the file's properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step-by-Step Procedure for Adding Circuit Card Files

1. Add the files to the **Project Tree** (A, below). There are several ways to do this:

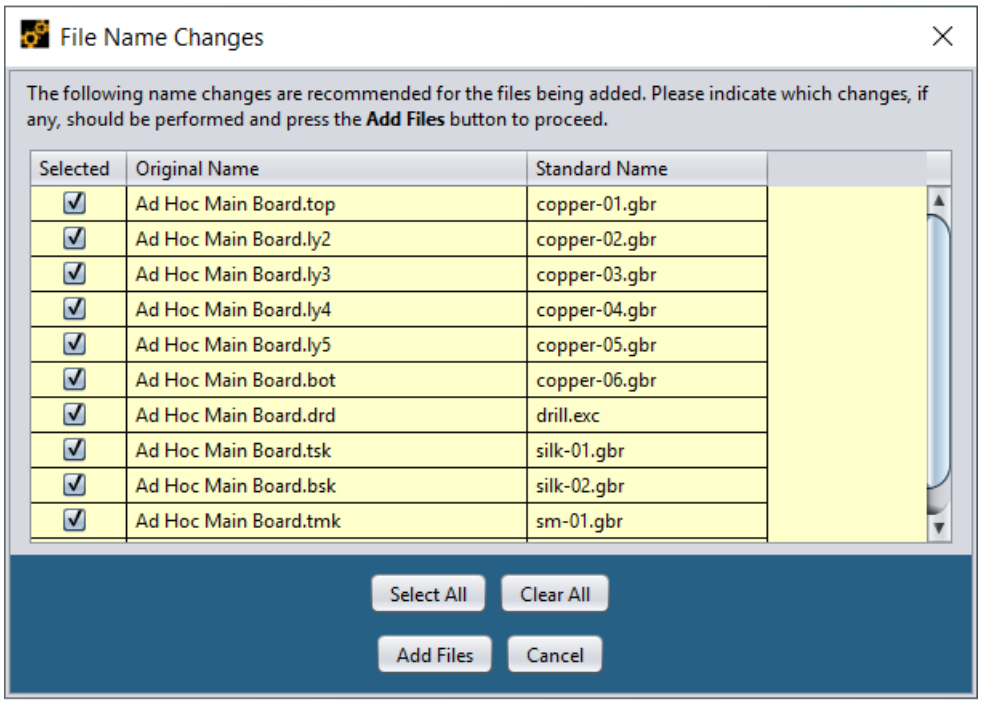
   - Right-click the **Files** folder (B) and select **Add Files**.
   - From Sherlock's **Main Menu**, click **CCA** (C), and then select **Add File(s)**.
   - Or, from Windows **File Explorer**, you can drag files onto the **Files** folder in the **Project Tree** (D). You can also drag and drop a folder or a zipped folder (`.zip`) containing CCA-compatible files.



2. When prompted by Sherlock, change the file name(s) to a standard format if you wish.

   Many CAD packages use cryptic file name prefixes, suffixes, and formats. If Sherlock recognizes the file name format, it recommends a simpler, more easily understood standard name. These suggestions are displayed in the **File Name Changes** dialog as shown below. To accept Sherlock's **Standard Name** suggestion, keep the check mark in the **Selected** column. If you would rather retain the file's original name, remove the check mark. When your selection is complete, click **Add Files**. Sherlock will then add them to the **Project Tree**.

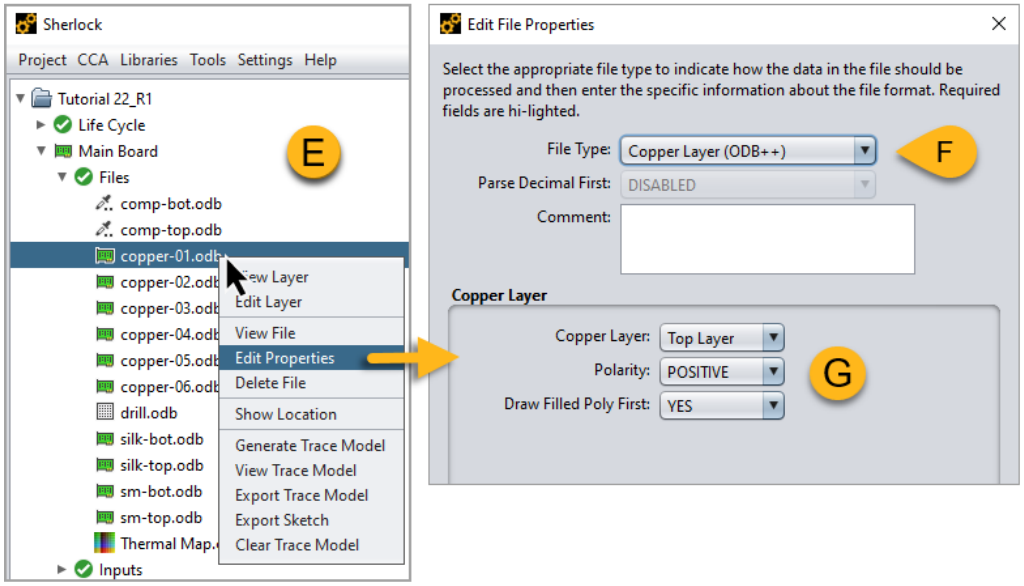
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. For each new file on the **Project Tree**, open the **Edit File Properties** dialog to configure the file's properties.

   After a CCA file is added and it appears in the **Project Tree**, right-click the file (E, below) and select **Edit Properties** to open the **Edit File Properties** dialog. Here you can select the correct file type (if Sherlock hasn't guessed it already) from the dropdown menu (F). Next, you will specify the file properties (G) if required. Initially, most CSV and spreadsheet files will be added as generic **Info** files, requiring you to change the file type.

   For a tutorial of this process, see Lesson 06 – Ad Hoc Project Creation (p. 56).

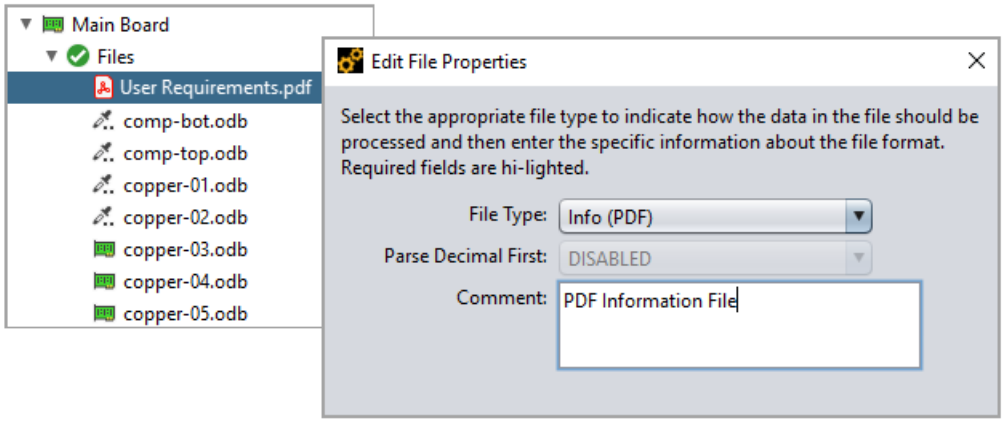


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4.5. Attaching Informational (Info) Files

You may wish to attach a document to the **Project Tree** for informational purposes only, such as a PDF copy of the product requirements or a Word file where you have stored your notes. Sherlock allows you to attach a PDF, Word, text, or an Excel document as an **Info** file to the list of CCA files in your project. Sherlock does *not* extract CCA data from **Info** files. It is simply a way for you to store project-related documentation on the **Project Tree**.

To add an **Info** file, follow the same steps as you would for any CCA file (see Adding Files to a Circuit Card Assembly (p. 100)). Then, when configuring the file properties (as described in step three of the previous section (p. 102)), select the applicable file type: **Info (Excel)**, **Info (PDF)**, **Info (Text)**, or **Info (Word)**.

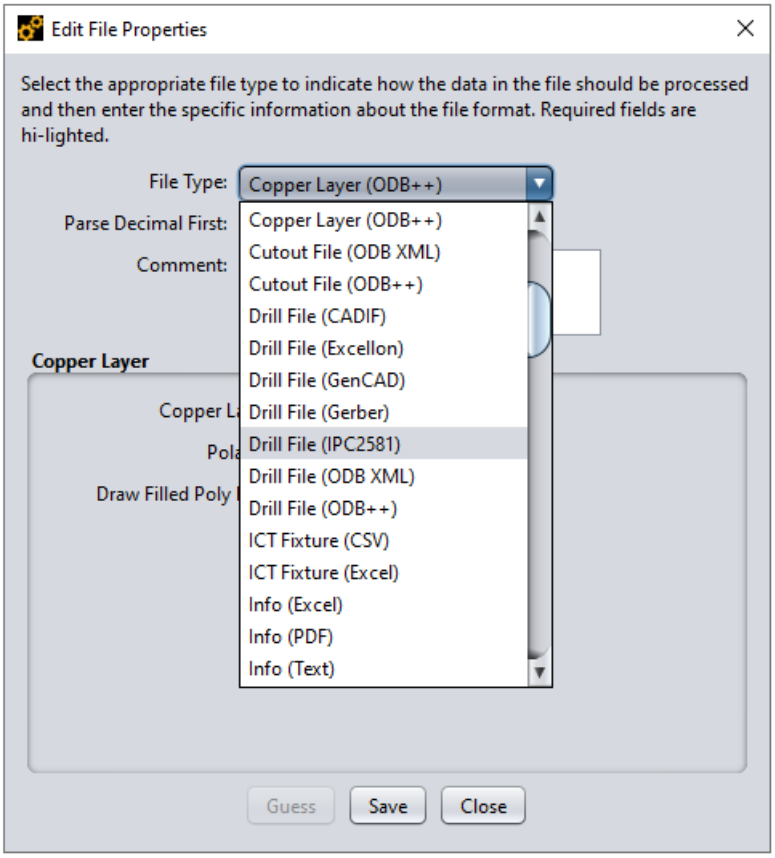


You can open the document from the **Project Tree** by double-clicking its icon, or you can right-click the icon and select **Open** in the context menu.

3.4.6. CCA File Format Specifications

This section contains information on all file-types you can add to your circuit card assembly in Sherlock. With each file type, you will find the file properties (if applicable) that must be configured for Sherlock to interpret the file correctly. To navigate quickly to the file type of interest, use the list of links below, Links to CCA File Types (p. 104).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Links to CCA File Types:

3.4.6.1. CAD File (IPC2581, ODB XML)

3.4.6.2. Copper Layer Files

3.4.6.3. Cutout Files

3.4.6.4. Drill Files

3.4.6.5. ICT Fixture Files (CSV, Excel)

3.4.6.6. Icepak Thermal Map (TMAP)

3.4.6.7. Info (Excel, PDF, Text, Word)

3.4.6.8. Laminate Layer Files

3.4.6.9. Mount Point Files (CSV, Excel)

3.4.6.10. Net List Files

3.4.6.11. Other Layer (Gerber, IPC2581, Image, ODB++)

3.4.6.12. Outline Layer Files

3.4.6.13. Parts List Files

3.4.6.14. Pick and Place Files

3.4.6.15. Silk Screen Files

3.4.6.16. Solder Mask Files

3.4.6.17. Thermal Map Files

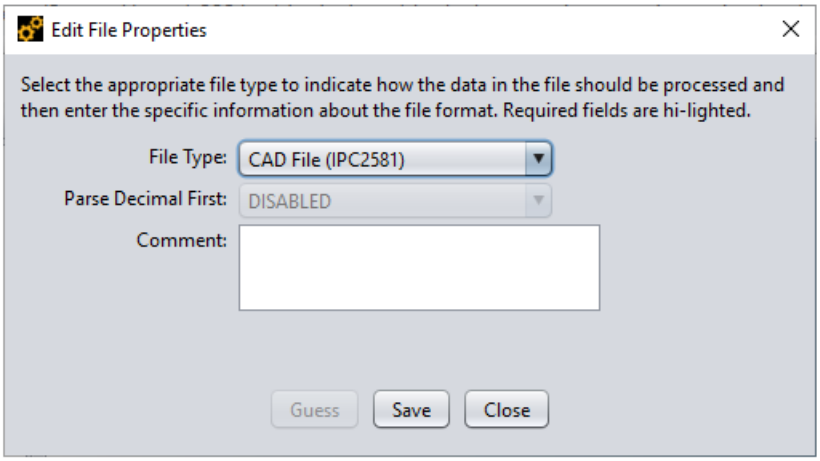
3.4.6.18. Unknown

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4.6.19. Wire Bond (CSV, Excel)

3.4.6.1. CAD File (IPC2581, ODB XML)

When you select these file types, no additional file settings are required.



3.4.6.2. Copper Layer Files

Links to copper layer file types:

3.4.6.2.1. Copper Layer (CADIF, Gerber, ODB XML, ODB++)
3.4.6.2.2. Copper Layer (EDB, IPC2581)
3.4.6.2.3. Copper Layer (Image)
3.4.6.2.4. Copper Layer (Trace)

3.4.6.2.1. Copper Layer (CADIF, Gerber, ODB XML, ODB++)

For these file-types, you must specify the following file properties:

- **Copper Layer**: Select **Top Layer**, **Layer 2**, … or **Bottom Layer**.
- **Polarity**: Select **Positive** or **Negative**.
- **Draw Filled Poly First**: Choose the default setting **YES**. (The **NO** option is for troubleshooting purposes only and is rarely used. It is intended to resolve rendering issues in the **Layer Viewer** involving polygons defined with clear fill areas.)

3.4.6.2.2. Copper Layer (EDB, IPC2581)

For these file-types, you must specify the following file properties:

- **Copper Layer**: Select **Top Layer**, **Layer 2**, … or **Bottom Layer**.
- **Polarity**: Select **Positive** or **Negative**.

3.4.6.2.3. Copper Layer (Image)

For these file-types, you must specify the following file properties:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Copper Layer**: Select **Top Layer**, **Layer 2**, ...or **Bottom Layer**.
- **Polarity**: Select **Positive** or **Negative**.
- **Image Type**: Select **Background** or **Foreground**.
- **Image Color**: Enter the background or foreground color of the image. This helps Sherlock identify objects when parsing the file. Valid values are **BLACK**, **WHITE**, or **CLEAR**.

  For custom color definitions, you can enter RGB color values in one of the following formats: **#RRGGBB** or **#RRGGBBAA** where the value following the # is a hexadecimal number. **RR** is the red component, **GG** is the green component, **BB** is the blue component, and **AA** is the alpha component (transparency). If you are entering a custom color, Sherlock's **Color Settings** window displays colors and their associated hexadecimal numbers (**#RRGGBBAA**). You can use it to look up the correct RGB value. Go to **Main Menu > Settings > Color**.

Tip:

In Sherlock, you can convert any copper layer file to an image file. This allows you to edit the traces using a paint program. For a complete discussion on using image files, see FEA- Image File Layers (p. 514).

Return to main index of CCA file types. (p. 104)

3.4.6.2.4. Copper Layer (Trace)

Sherlock generates these trace files (`.png.traces`) when you choose the **Generate Trace Model** operation. See Trace Model Generation (p. 497). To locate these files, do the following: In the Sherlock **Project Tree**, right-click the copper layer which you converted to a trace model. Select **Show Location**. This opens the folder where the `.png.traces` file is stored. To import the file into your project, refer to Adding Files to a Circuit Card Assembly (p. 100).

For these file-types, you must specify the following file properties:

- **Copper Layer**: Select **Top Layer**, **Layer 2**, ...or **Bottom Layer**.
- **Polarity**: Select **Positive** or **Negative**.

3.4.6.3. Cutout Files

Links to cutout file types:

3.4.6.3.1. Cutout File (ODB XML)

3.4.6.3.2. Cutout File (ODB++)

3.4.6.3.1. Cutout File (ODB XML)

When you select this file type, no additional file settings are required.

3.4.6.3.2. Cutout File (ODB++)

For this file-type, you must specify the following file properties:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Polyline Simplification**: When set to **Enabled**, Sherlock employs polyline simplification using Douglas-Peucker method to reduce the number of points in polygons.
- **Polyline Tolerance**: Enter a value (**in**, **mm**, **m**, **mil**, **micron**, **nm**). This tolerance value is used for polyline simplification. Any points in a curve that vary from a line by this amount will be removed. The units correspond to the units defined in the file.

> **Tip:**
>
> For **Polyline Tolerance**, the recommended starting value is 0.1 mm. Then adjust if needed.

Return to main index of CCA file types. (p. 104)

3.4.6.4. Drill Files

Links to drill files:

3.4.6.4.1. Drill File (CADIF, IPC2581, ODB XML, ODB++)
3.4.6.4.2. Drill File (Excellon)
3.4.6.4.3. Drill File (Gerber)

3.4.6.4.1. Drill File (CADIF, IPC2581, ODB XML, ODB++)

For these file-types, you must specify the following file properties:

- **Start Copper Layer**: Select **Top Layer**, **Layer 2**, ...**Bottom Layer**.
- **End Copper Layer**: Select **Top Layer**, **Layer 2**, ...**Bottom Layer**.
- **Plating Material**: Select **Aluminum** or **Copper**.
- **Plating Thickness**: Enter value in **in**, **mm**, **m**, **mil**, **micron**, or **nm**.
- **Capped**: Select **Yes** or **No**.
- **Thickness**: Enter value in **in**, **mm**, **m**, **mil**, **micron**, or **nm**.
- **Fill Material**: Select **None** or **Copper**.

3.4.6.4.2. Drill File (Excellon)

For this file-type, you must specify the following file properties:

- **Coordinated Format**: Select **00.0000**, **00.00000**...or **000.00**. (See Reviewing Drill Holes (p. 75) in Tutorial: Ad Hoc Project Creation (p. 56).)
- **Coordinate Units**: Select **mm**, **m**, **mil**, **micron**, or **nm**.
- **Coordinate Type**: Select **Absolute** or **Relative**.
- **Zero Format**: Select **Trailing** or **Leading**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Start Copper Layer**: Select **Top Layer**, **Layer 2**, ...or **Bottom Layer**.
- **End Copper Layer**: Select **Top Layer**, **Layer 2**, ...or **Bottom Layer**.
- **Plating Material**: Select **Aluminum** or **Copper**.
- **Plating Thickness**: Enter value in **in**, **mm**, **m**, **mil**, **micron**, or **nm**.
- **Capped**: **Yes** or **No**.
- **Thickness**: Enter value in **in**, **mm**, **m**, **mil**, **micron**, or **nm**.
- **Fill Material**: Select **None** or **Copper**.

3.4.6.4.3. Drill File (Gerber)

File Properties:

- **Hole Type**: Select **Plated**, **Non_Plated**, or **Via**.
- **Start Copper Layer**: Select **Top Layer**, **Layer 2**, ...**Bottom Layer**.
- **End Copper Layer**: Select **Top Layer**, **Layer 2**, ...**Bottom Layer**.
- **Plating Material**: Select **Aluminum** or **Copper**.
- **Plating Thickness**: Enter value in **in**, **mm**, **m**, **mil**, **micron**, **nm**.
- **Capped**: **Yes** or **No**.
- **Thickness**: Enter value in **in**, **mm**, **m**, **mil**, **micron**, or **nm**.
- **Fill Material**: Select **None** or **Copper**.

Return to main index of CCA file types. (p. 104)

3.4.6.5. ICT Fixture Files (CSV, Excel)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

File Properties: (Required properties are in bold) Number of Header Rows, ID Column, **Type Column**, **Shape Column**, Units Column, **X Coordinate Column**, **Y Coordinate Column**, Length Column, Width Column, Diameter Column, Nodes Column, Rotation Column, Height Column, Constraints Column, Board Side Column, Material Column, Polygon Column, and Default Units, and Numeric Format.

When an ID is provided, Sherlock will find the corresponding ICT fixture and update its properties. If an ICT fixture ID is *not* provided, Sherlock will add an ICT fixture with the specified properties. If the specified ICT fixture ID is not found and is not already assigned to another ICT, Sherlock will create a new ICT fixture with that ID.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return to main index of CCA file types. (p. 104)

3.4.6.6. Icepak Thermal Map (TMAP)

Note:

Only Sherlock's Solder Fatigue Analysis is capable of using the `.tmap` file. Other analysis modules do not recognize the file-type.

This file-type is a thermal map exported from Ansys Electronics Desktop (AEDT) - Icepak and can be used as an input for a Solder Fatigue Analysis in Sherlock. See Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis (p. 426).

Return to main index of CCA file types. (p. 104)

3.4.6.7. Info (Excel, PDF, Text, Word)

Initially, most CSV and spreadsheet files will be added as generic **Info** files. Use the **Edit File Properties** dialog to set the correct file type. You can also use the **Info** file type to attach informational documents to your project. See Attaching Informational (Info) Files (p. 103).

Return to main index of CCA file types. (p. 104)

3.4.6.8. Laminate Layer Files

Links to laminate layer files.

3.4.6.8.1. Laminate Layer (CADIF, Gerber, ODB XML, ODB++)

3.4.6.8.2. Laminate Layer (EDB, IPC2581)

3.4.6.8.3. Laminate Layer (Image)

3.4.6.8.4. Laminate Layer (ODB XML)

3.4.6.8.1. Laminate Layer (CADIF, Gerber, ODB XML, ODB++)

File Properties:

- **Laminate Layer**: Select **Layer 1**, **Layer 2**,… or **Layer 5**.
- **Polarity**: Select **Positive** or **Negative**.
- **Draw Filled Poly First**: Choose the default setting **YES**. (The **NO** option is for troubleshooting purposes only and is rarely used. It is intended to resolve rendering issues in the **Layer Viewer** involving polygons defined with clear fill areas.)

3.4.6.8.2. Laminate Layer (EDB, IPC2581)

File Properties:

- **Laminate Layer**: Select **Layer 1**, **Layer 2**,...or **Layer 5**.
- **Polarity**: Select **Positive** or **Negative**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4.6.8.3. Laminate Layer (Image)

File Properties:

- **Laminate Layer**: Select **Layer 1**, **Layer 2**,...or **Layer 5**.
- **Polarity**: Select **Positive** or **Negative**.
- **Image Type**: Select **Background** or **Foreground**.
- **Image Color**: Enter the background or foreground color of the image. This helps Sherlock identify objects when parsing the file. Valid values are **BLACK**, **WHITE**, or **CLEAR**.

  For custom color definitions, you can enter RGB color values in one of the following formats: **#RRGGBB** or **#RRGGBBAA** where the value following the # is a hexadecimal number. **RR** is the red component, **GG** is the green component, **BB** is the blue component, and **AA** is the alpha component (transparency).

  Tip:

  If you are entering a custom color, the Sherlock **Color Settings** window displays colors and their associated hexadecimal numbers (**#RRGGBBAA**). You can use it to look up the correct RGB value. Go to **Main Menu > Settings > Color**.

3.4.6.8.4. Laminate Layer (ODB XML)

File Properties:

- **Laminate Layer**: Select **Layer 1**, **Layer 2**,...or **Layer 5**.
- **Polarity**: Select **Positive** or **Negative**.
- **Draw Filled Poly First**: Select **Yes** or **No**.

Return to main index of CCA file types. (p. 104)

3.4.6.9. Mount Point Files (CSV, Excel)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

File Properties: (Required properties are in bold) Number of Header Rows, ID Column, **Type Column**, **Shape Column**, Units Column, **X Coordinate Column**, **Y Coordinate Column**, Length Column, Width Column, Diameter Column, Nodes Column, Rotation Column, Height Column, Constraints Column, Board Side Column, Material Column, Polygon Column, **Default Units**, Numeric Format.

When an ID is provided, Sherlock will find the mount point corresponding to the ID and update the mount point's properties. If a mount point ID is *not* provided, Sherlock adds a new mount point

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with the specified properties. If the specified mount point ID is not found and is not already assigned to another mount point, then Sherlock creates a new mount point with that ID.

Return to main index of CCA file types. (p. 104)

3.4.6.10. Net List Files

Links to Net List files:

3.4.6.10.1. Net List (CADIF)

3.4.6.10.2. Net List Import (CSV, Excel)

3.4.6.10.3. Net List (IPC, IPC2581)

3.4.6.10.1. Net List (CADIF)

When you select these file types, no additional file settings are required.

3.4.6.10.2. Net List Import (CSV, Excel)

File Properties: (Required properties are in bold) Number of Header Rows, **Net Name Column, Ref Des Column**, **Pin Name Column**, **X Column**, **Y Column**, **Location Units** (in, mm, m, mil, micron, nm), and Numeric Format.

A circuit card Net List defines all electrical connections between component pins as a list of pin and network combinations. Pins that are electrically connected will be matched with the same network name. You can import CSV/XLS files containing the Net List data. Each row must contain at least five columns which include **Net Name**, **Reference Designator**, **Pin Name**, **X coordinate**, and **Y coordinate**. Example:

Table 3.1: Sample Net List Data Showing Required Columns

Net Name	RefDes	Pin Name	X	Y
2512-1	J1	1	0.0375	-1
BGA676	U9	12	0.055	0

3.4.6.10.3. Net List (IPC, IPC2581)

When you select these file types, no additional file settings are required.

Return to main index of CCA file types. (p. 104)

3.4.6.11. Other Layer (Gerber, IPC2581, Image, ODB++)

When you select these file types, no additional file settings are required.

Return to main index of CCA file types. (p. 104)

3.4.6.12. Outline Layer Files

Links to outline layer files:

3.4.6.12.1. Outline Layer (CSV, Excel)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4.6.12.2. Outline Layer (Gerber, IPC2581)

3.4.6.12.1. Outline Layer (CSV, Excel)

File Properties: (Required properties are in bold) Number of Header Rows, **X Location Column**, **Y Location Column**, **Location Units** (in, mm, m, mil, micron, nm).

An outline layer file provides the X and Y coordinates that make up a circuit card assembly outline. The coordinates defined in the file are processed in order to generate a board outline. Since a board outline should be defined as a closed polygon, the first and last coordinate in the file should be the same. However, if they are not, Sherlock will automatically add an additional point back to the starting location if necessary.

An outline layer file must be defined as a CSV or be an Excel spreadsheet (.xls, .xlsm, or .xlsx) file. Each coordinate is defined row by row. Each row must contain two columns which indicate the X and Y coordinates of an outline vertex. Sherlock connects lines between each successive vertex to generate the outline. The first row of the file must be a label such as X and Y to denote the data in the given column.

When setting the file properties for an outline layer file, select the X location column, Y location column, and select the units of measure defined by the coordinates. Example:

Table 3.2: Sample Outline Layer, X and Y Coordinates

X	Y
0	0
5	0
5	5
0	5
0	0

3.4.6.12.2. Outline Layer (Gerber, IPC2581)

When you select these file types, no additional file settings are required.

Return to main index of CCA file types. (p. 104)

3.4.6.13. Parts List Files

Links to parts list files:

3.4.6.13.1. Parts List (CSV, Eagle, Excel)

3.4.6.13.2. Parts List (IPC2581, ODB++)

3.4.6.13.1. Parts List (CSV, Eagle, Excel)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Properties: (Required properties are in bold) **Number of Header Rows**, **Column Delimiter**, Numeric Format, **Ref Des Column**, **Part Description Column**, Part Number Column, Manufacturer Column, Package Mount Column, Package Name Column.

Note:

Regarding BGA Components: When importing an Eagle file, Sherlock calculates the package dimensions from the ball pattern rather than using the dimensions specified for the package. For information on reviewing and editing parts, see Parts List Management Tools (p. 197).

3.4.6.13.2. Parts List (IPC2581, ODB++)

When you select these file types, no additional file settings are required.

Note:

Regarding BGA Components: When importing an ODB++ file, Sherlock calculates the package dimensions from the ball pattern rather than using the dimensions specified for the package. For information on reviewing and editing parts, see Parts List Management Tools (p. 197).

Return to main index of CCA file types. (p. 104)

3.4.6.14. Pick and Place Files

Links to pick and place files:

3.4.6.14.1. Pick & Place (CADIF)

3.4.6.14.2. Pick & Place (CSV)

3.4.6.14.3. Pick & Place (DELIMITED)

3.4.6.14.4. Pick & Place (Eagle)

3.4.6.14.5. Pick & Place (Excel, FIXED)

3.4.6.14.6. Pick & Place (IPC2581)

3.4.6.14.7. Pick & Place (ODB XML)

3.4.6.14.8. Pick & Place (ODB++)

3.4.6.14.1. Pick & Place (CADIF)

When you select this file type, no additional file settings are required.

3.4.6.14.2. Pick & Place (CSV)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Properties: (Required properties are in bold) **Number of Header Rows**, **Reference ID Column**, **X Coordinate Column**, **Y Coordinate Column**, Footprint Column, Rotation Column, Board Side Column, Description Column, **Location Units**, Numeric Format, and Test Point Prefix.

3.4.6.14.3. Pick & Place (DELIMITED)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

File Properties: (Required properties are in bold) **Number of Header Rows**, **Column Delimiter**, **Reference ID Column**, **X Coordinate Column**, **Y Coordinate Column**, Footprint Column, Rotation Column, Board Side Column, Description Column, **Location Units**, Numeric Format, and Test Point Prefix.

3.4.6.14.4. Pick & Place (Eagle)

File Properties:

- **X Offset:** Enter value (**in**, **mm**, **m**, **mil**, **micron**, or **nm**)
- **Y Offset:** Enter value (**in**, **mm**, **m**, **mil**, **micron**, or **nm**)
- **Test Point Prefix**

3.4.6.14.5. Pick & Place (Excel, FIXED)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

File Properties: (Required properties are in bold) **Number of Header Rows**, **Reference ID Column**, **X Coordinate Column**, **Y Coordinate Column**, Footprint Column, Rotation Column, Board Side Column, Description Column, **Location Units**, Numeric Format, and Test Point Prefix.

3.4.6.14.6. Pick & Place (IPC2581)

When you select these file types, no additional file settings are required.

3.4.6.14.7. Pick & Place (ODB XML)

File Properties:

- **SMT Locations**: Select **Center** or **Pin 0**.

3.4.6.14.8. Pick & Place (ODB++)

File Properties:

- **Board Side**: Select **Top** or **Bottom**.
- **Location Units**: Select **in**, **mm**, **mil**, or **micron**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mirror X Values**: Select **Yes** or **No**.
- **SMT Location**: Select **Pin 0** or **Center**.
- **Bottom Rotation View**: Select **Top** or **Bottom**.
- **Test Point Prefix**

Return to main index of CCA file types. (p. 104)

3.4.6.15. Silk Screen Files

Links to silk screen files:

3.4.6.15.1. Silk Screen (CADIF, Gerber, ODB XML, ODB++)
3.4.6.15.2. Silk Screen (IPC2581)

3.4.6.15.1. Silk Screen (CADIF, Gerber, ODB XML, ODB++)

File Properties:

- **Parse Decimal First [Gerber files only]**: Typically, this should be kept at the default value, **DISABLED**.

  The **ENABLED** setting exists to assist in parsing Gerber files that have *not* been formatted correctly. When **ENABLED**, Sherlock parses the Gerber numbers based on the length of the decimal coordinate specifier first and then interprets the remaining digits as the integer portion of the number. This is contrary to standard Gerber file format specifications.

- **Board Side**: Select **Top** or **Bottom**.
- **Draw Filled Poly First**: Select **Yes** or **No**.

3.4.6.15.2. Silk Screen (IPC2581)

File Properties:

- **Board Side**: Select **Top** or **Bottom**.

Return to main index of CCA file types. (p. 104)

3.4.6.16. Solder Mask Files

Links to solder mask files:

3.4.6.16.1. Solder Mask (CADIF, Gerber)
3.4.6.16.2. Solder Mask (IPC2581)
3.4.6.16.3. Solder Mask (ODB XML, ODB++)
3.4.6.16.4. Test Point (CSV, Excel)

3.4.6.16.1. Solder Mask (CADIF, Gerber)

File Properties:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Board Side**: Select **Top** or **Bottom**.
- **Draw Filled Poly First**: Select **Yes** or **No**.

3.4.6.16.2. Solder Mask (IPC2581)

File Properties:

- **Board Side**: Select **Top** or **Bottom**.

3.4.6.16.3. Solder Mask (ODB XML, ODB++)

File Properties:

- **Board Side**: Select **Top** or **Bottom**.
- **Draw Filled Poly First**: Choose the default setting **YES**. (The **NO** option is for troubleshooting purposes only and is rarely used. It is intended to resolve rendering issues in the **Layer Viewer** involving polygons defined with clear fill areas.)

Return to main index of CCA file types. (p. 104)

3.4.6.16.4. Test Point (CSV, Excel)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

File Properties: (Required properties are in bold) **Number of Header Rows**, ID Column, Board Side Column, Units Column, **X Coordinate Column**, **Y Coordinate Column**, Radius Column, Load Type Column, Load Units Column, **Load Column**, **Default Units**, **Default Force Units**, **Default Displacement Units**, and Numeric Format.

When a Test Point ID is provided, Sherlock will find the test point corresponding to the ID and update its properties. If a test point ID is *not* provided, Sherlock adds a new test point with the specified properties. If the specified test point ID is not found and is not already assigned to another test point, then Sherlock creates a new test point with that ID.

Return to main index of CCA file types. (p. 104)

3.4.6.17. Thermal Map Files

See also the following file-type, Icepak Thermal Map (TMAP) (p. 109).

Links to thermal map files:

3.4.6.17.1. Thermal Map (CSV, Excel)

3.4.6.17.2. Thermal Map (Image)

3.4.6.17.1. Thermal Map (CSV, Excel)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sherlock will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

File Properties: Board Side (select **Both**, **Top**, or **Bottom**), Number of Header Rows, Reference ID Column, Temperature Column, and Temperature Units.

Thermal Profile(s): Assign a **Thermal Map** to one or more of the **Thermal Events**. Once assigned, the temperatures defined in the **Thermal Map** will be used for those Events, rather than the ambient values defined in the Thermal Cycle.

Tip:

For a detailed explanation, see the section on assigning Thermal Profiles (p. 402) in *Plated Through Hole (PTH) Fatigue Analysis*.

3.4.6.17.2. Thermal Map (Image)

File Properties:

- **Board Side**: Select **Both**, **Top**, or **Bottom**.
- **Legend Orientation**: Select **Vertical** or **Horizontal**.
- **Min Temperature**: (C, F, or K) Enter the minimum temperature.
- **Max Temperature**: (C, F, or K) Enter the maximum temperature.
- **Thermal Profile(s)**: Assign a **Thermal Map** to one or more of the **Thermal Events**. Once assigned, the temperatures defined in the **Thermal Map** will be used for those Events, rather than the ambient values defined in the Thermal Cycle.

Tip:

For a detailed explanation, see the section on assigning Thermal Profiles (p. 402) in *Plated Through Hole (PTH) Fatigue Analysis*.

Return to main index of CCA file types. (p. 104)

3.4.6.18. Unknown

If Sherlock cannot identify a file, it will default to an **Unknown** file type in the **Edit File Properties** dialog. Use the **File Type** dropdown menu to select the correct file type.

Return to main index of CCA file types. (p. 104)

3.4.6.19. Wire Bond (CSV, Excel)

In order to process the file, Sherlock must correctly identify what kind of data is contained in each column. If you click the **Guess** button at the bottom of the **Edit File Properties** dialog, Sherlock

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

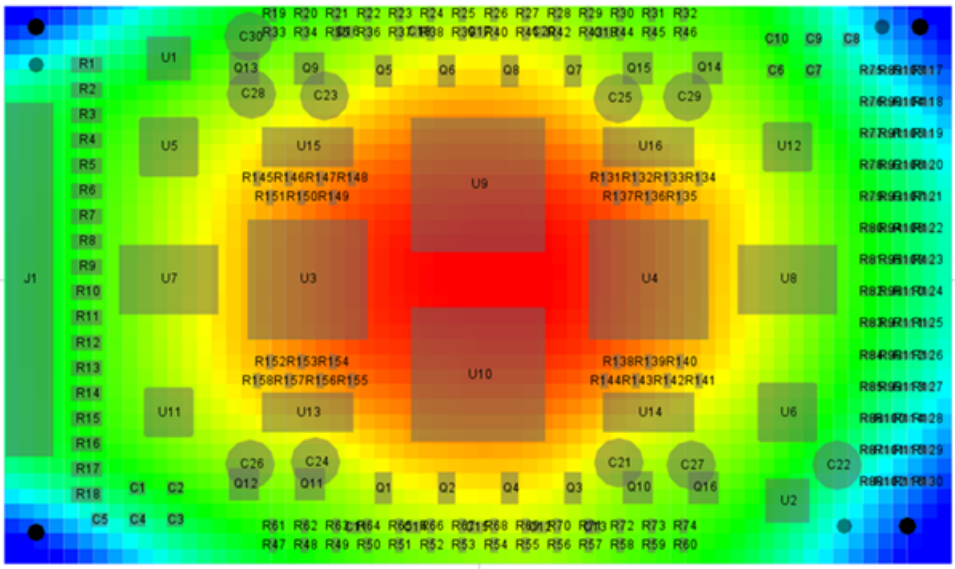
will attempt to identify each column. If required, you can use the dialog to manually select which column contains each property listed.

File Properties: **Number of Header Rows**, ID Column, Board Side Column, Material Column, Units Column, **X1 Coordinate Column**, **Y1 Coordinate Column**, **X2 Coordinate Column**, **Y2 Coordinate Column**, Thickness Column, **Weld Length 1 Column**, **Weld Length 2 Column**, Height Column, Shape Column, Default Units, and Numeric Format.

Return to main index of CCA file types. (p. 104)

3.5. Layer Viewer

See Tutorial: Layer Viewer (p. 43) for a guided tour of the Layer Viewer and its features.



3.6. Importing Projects and ECAD Archives

There are two primary ways to create a project in Sherlock:

- As demonstrated in Tutorial: Ad Hoc Project Creation (p. 56), you can add individual files to your project using the **Add File(s)** feature. The kinds of files Sherlock can process is extensive, including layer files, drill files, data files, Gerber files, pick and place files, bills of material, and so on. You will find a complete list of file-types in the next section Circuit Card Assembly Files (p. 97).
- Or, as described in this section, you can create a project by using the **Import...** feature to import an entire archive or project file from one of the industry-standard file types.

Whether you are in single or multi-project mode, you can import a project at any time by selecting one of the **Import...** options from the **Project** menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

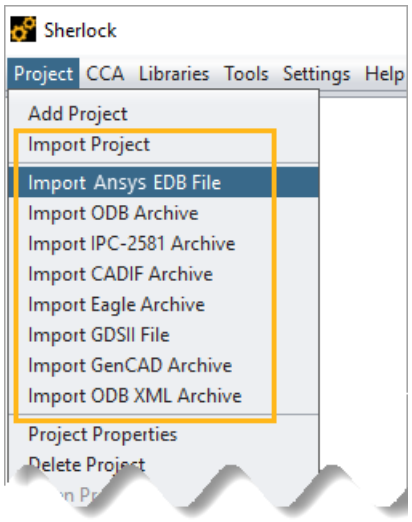


Table 3.3: Import-Compatible Project Files

Ansys EDB File	CADIF Archive	GDSII File
ODB Archive	Eagle Archive	
IPC-2581 Archive	ODB XML Archive	

Topics Covered in This Section:

3.6.1. How to Import a Project into Sherlock
3.6.2. Import-Compatible File Types
3.6.3. IPC-2581 Import Tutorial

3.6.1. How to Import a Project into Sherlock

Topics:

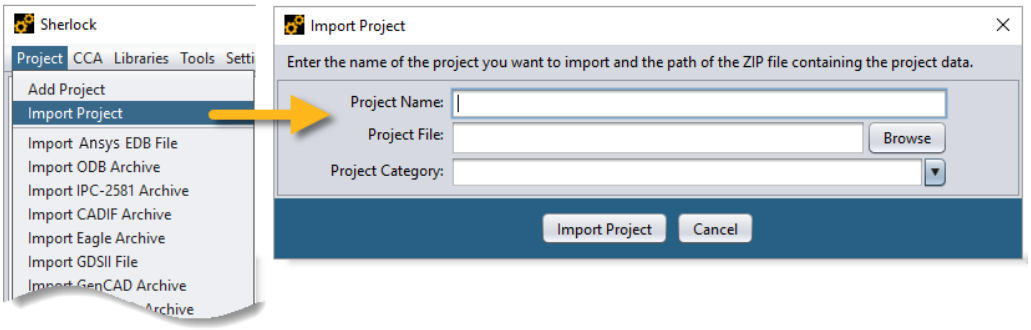
3.6.1.1. Importing Sherlock Projects (.zip)
3.6.1.2. Importing Other File Types

3.6.1.1. Importing Sherlock Projects (.zip)

To import a Sherlock project, select **Import Project** from the **Main Menu**. This will open the **Import Project** dialog. Here you specify a **Project Name** and click the **Browse** button to select the project file. **Project Category** is an optional field. (You may change the **Project Category** later by right-clicking the project folder in the **Project Tree** and selecting **Project Properties**.) Finally, click **Import Project**.

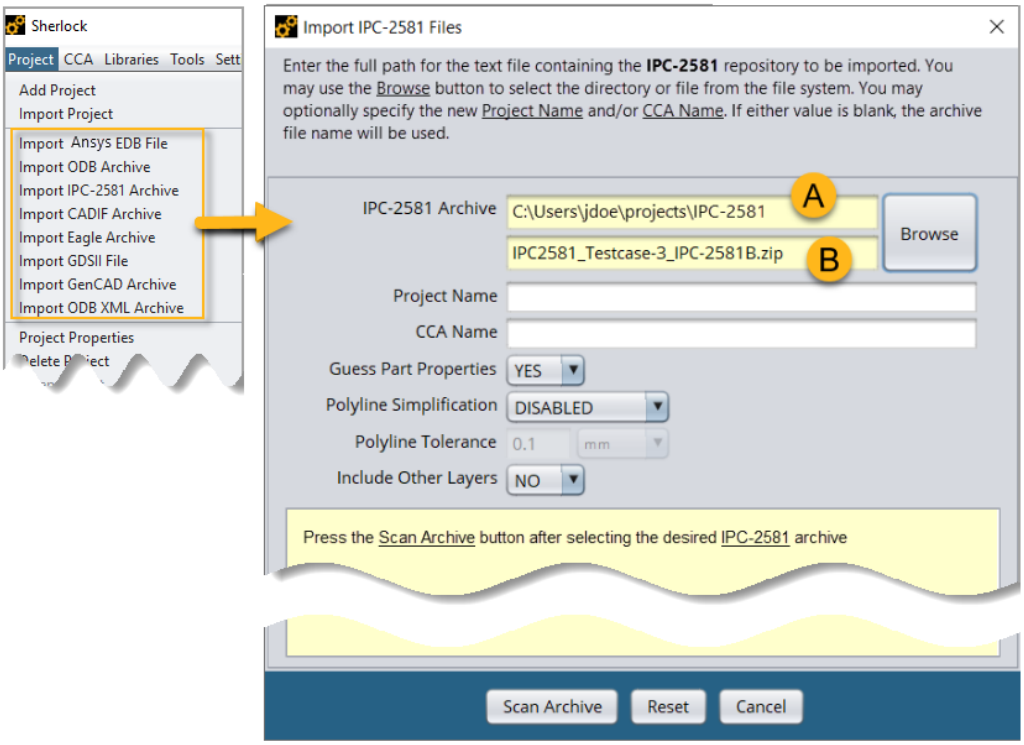
Depending on the size of the project file, the import may take a minute or so to load. During that time, a progress dialog will be displayed. Once the import is complete, the project will appear in Sherlock's **Project Tree**. For additional details on importing a Sherlock project, refer to Importing a Project (p. 8) in the Tutorial: Project Overview (p. 8).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.6.1.2. Importing Other File Types

As mentioned earlier, you can import a variety of industry-standard file types into Sherlock. Simply click **Project** from the **Main Menu** and select one of the **Import...** options. This will open the dialog for the specific file type you chose. The **Import IPC-2581** dialog is shown below as an example of the import interface. Click the **Browse** button to navigate to the file you wish to import. Notice the top field (A, below) displays the file path. The second field (B) displays the file name.



Other options in the **Import Files** dialog:

- **Project Name** and **CCA Name**: (Optional) You can specify a **Project Name** and a **CCA Name** if you wish. If these fields are left blank, Sherlock will use the name of the imported file for the **Project Name** and **CCA Name**.
- **Guess Part Properties**: When set to **YES** (default), Sherlock will attempt to guess certain part properties based on the part descriptions. When set to **NO**, Sherlock will *not* guess part properties. Except for **Package Type**, **Model Part** status, and **Lead Modeling** status, only properties specified in the file will be populated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Polyline Simplification**: (Disabled by default) When ENABLED, Sherlock simplifies the board outline by reducing the number of points which comprise the outline. This can resolve meshing issues that happen when too many vertices define an outline. See **Polyline Tolerance** below.

  Tip:

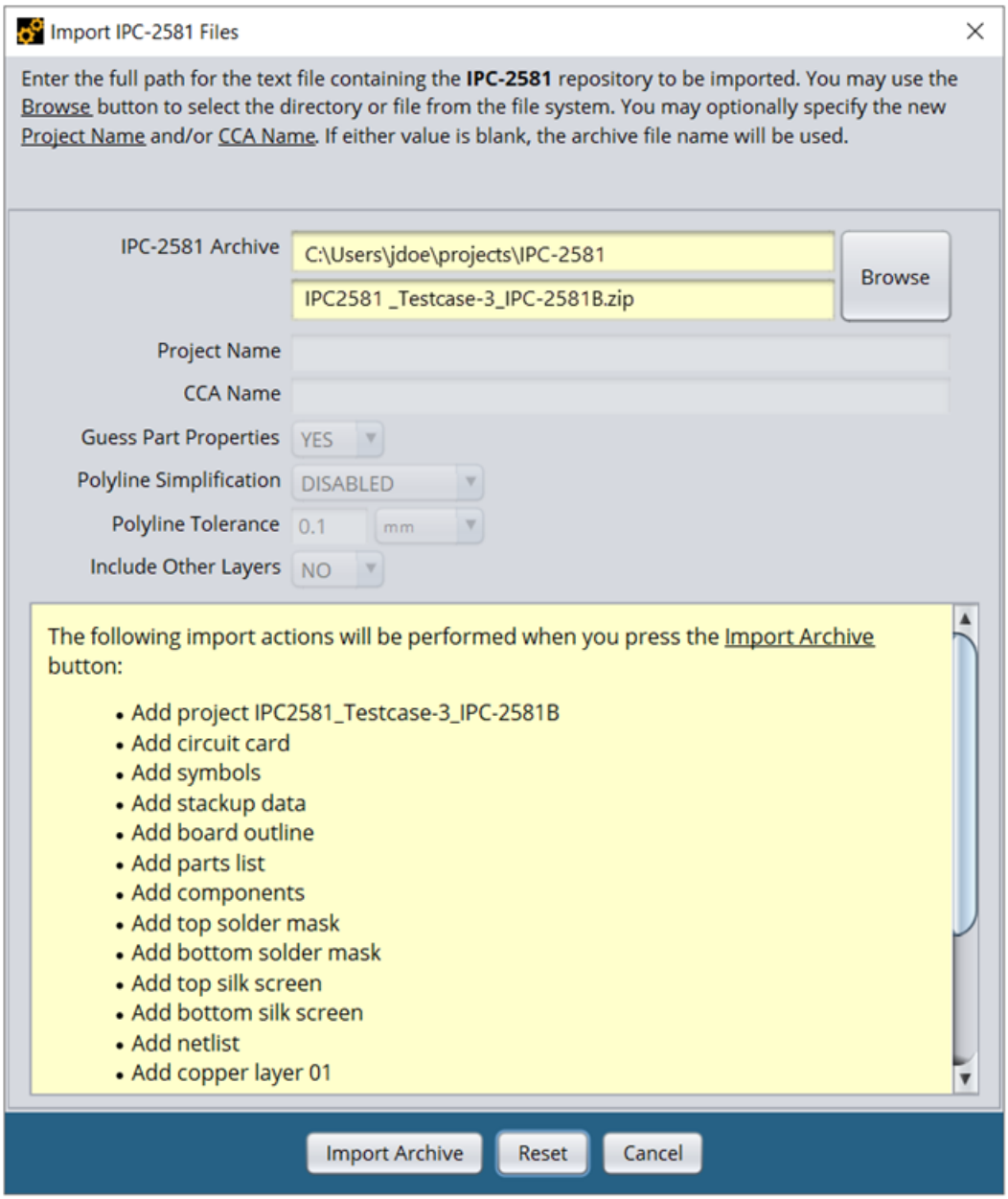
  You can also perform Polyline Simplification *after* importing the file. Use the **Edit Board Outline** function in the **Layer Viewer**. See Board Outline Editor (p. 51) in the *Layer Viewer Tutorial*.

- **Polyline Tolerance**: Sherlock uses this tolerance value for the Polyline Simplification feature. In the polygon outline, any point that is within the specified distance of another will be removed.

- **Include Other Layers**: By default, Sherlock only imports layers it can use. When you set this option to **YES**, Sherlock will import any layer found in the archive. These additional layers will appear in the **Other Layers** folder of the **Layer Viewer**.

Once you have selected the file you wish to import, click the **Scan Archive** button. Sherlock will perform a preliminary scan of the selected file and display the list of import actions it will perform as shown below. To continue with the import, click the **Import Archive** button. If you do not wish to proceed with the import, you can click **Reset**. This will clear the file name and the results of the file scan so you can start over.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each file type has its own import dialog with file-specific parameters. Special considerations for each file-type are contained in the sections that follow. For a detailed explanation of how to import a project, refer to these tutorials:

- Tutorial: ODB++ Project Creation (p. 23) in the *Getting Started* chapter
- The IPC-2581 Import Tutorial (p. 124) below

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.6.2. Import-Compatible File Types

The following is a list of industry standard file types you can import into Sherlock. Each has its own import dialog with file-specific parameters. Special considerations for each file type are noted in the descriptions below.

Tip:

See Importing Other File Types (p. 120) above for explanations of import options that apply to *all* file types.

Ansys EDB File

(.def) Currently, the EDB import supports copper layers, laminates, board outline, and stackup data.

ODB Archive

(.tgz, .tar.gz, .tar) A CAD to CAM data exchange file. Importing and working with an ODB++ archive is covered in detail in the Tutorial: ODB++ Project Creation (p. 23).

Note: When importing a BGA package from an ODB++ archive, Sherlock calculates the package dimensions from the ball pattern rather than using the dimensions specified for the package. See The Importance of Reviewing Part Properties (p. 185).

IPC-2581

(.cvg, .xml, .zip) See the IPC-2581 Import Tutorial (p. 124) below for details.

CADIF Archive

(.paf) CADIF files are generated from Zuken PCB layout software applications.

Eagle Archive

(.zip, .tgz, .tar.gz, .tar) EAGLE (Easily Applicable Graphical Layout Editor) is a scriptable electronic design automation (EDA) application.

Note: When importing a BGA package from an Eagle archive, Sherlock calculates the package dimensions from the ball pattern rather than using the dimensions specified for the package. See The Importance of Reviewing Part Properties (p. 185).

GDSII File

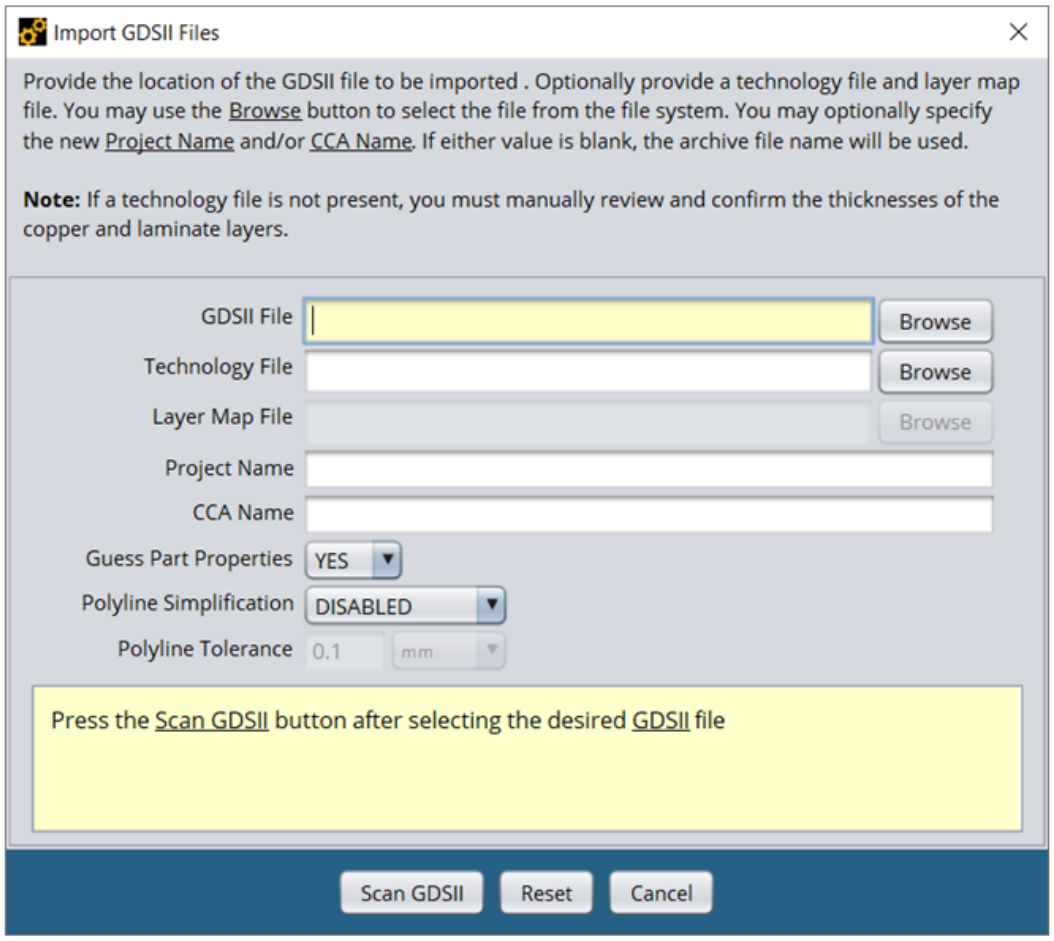
(.gds, .sf, .strm) GDSII is an industry standard database file format used for exchanging integrated circuit layout artwork. In Sherlock, the **Import GDSII Files** dialog, shown below, gives you the option to import a **Technology File** (.xml, .tech, .layermap) and a **Layer Map File** (.map). Click

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the check box to launch the project in ECAD Xplorer. Use the **Project Directory** field to specify the folder where you want to save the imported project.

Note:

If a technology file is not present, you must manually review and confirm the thicknesses of the copper and laminate layers.



ODB XML Archives

(.xml, .xml.gz, .xml.z, .gz, .z) **Note**: When importing a BGA package from an ODB XML archive, Sherlock calculates the package dimensions from the ball pattern rather than using the dimensions specified for the package. See The Importance of Reviewing Part Properties (p. 185).

3.6.3. IPC-2581 Import Tutorial

IPC-2581 is an open global standard for assembly and manufacturing data supported by the IPC-2581 Consortium. More information about IPC-2581 can be found at http://www.ipc2581.com. Sherlock supports IPC-2581 revision A and B of the standard for a **FULL** function mode IPC-2581 file (USERDEF mode in revision B). Some other function mode levels may produce results within Sherlock however these other modes typically do not include all the necessary data for Sherlock to completely define a circuit card. Some of the example files found on the IPC-2581 website are not **FULL** function mode examples.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A list of supported and known unsupported elements from IPC-2581 can be found at the end of this section (p. 129).

For the purposes of this tutorial, Test Case 3 for revision B will be used from the IPC-2581 Consortium website. These sample files can be found at http://www.ipc2581.com/b-test-cases.

Note:

If you download the test cases from the IPC-2581 Consortium website, be aware there is an issue with Test Case 6. The project defines a cutout which surrounds the PCB. This causes Sherlock to eliminate the board during FEA model generation. You should delete the cutout after importing the project.

In this section, the following topics are covered:

3.6.3.1. Importing an IPC-2581 File

3.6.3.2. IPC-2581 Supported Elements

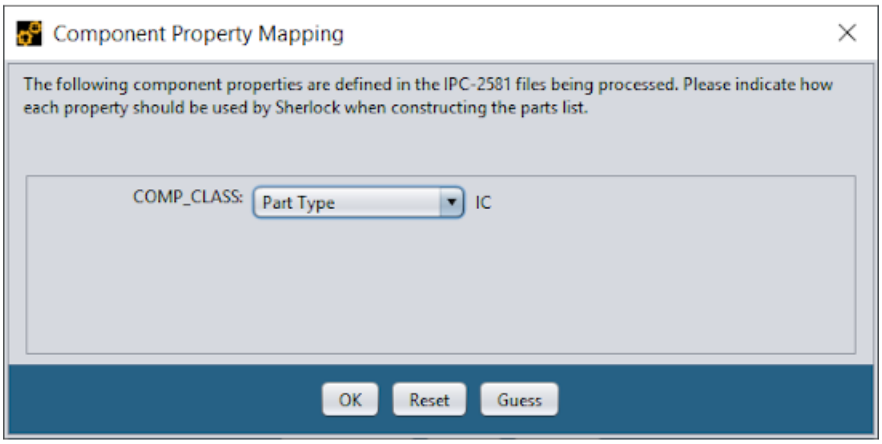
3.6.3.1. Importing an IPC-2581 File

An IPC-2581 file is a text-based XML file containing the complete definition of the printed circuit card, such as the components, layers, BOM, etc. IPC-2581 files prior to revision B of the standard, end with a .cvg extension, and end with a .xml extension with revision B. Sherlock also supports having one of these files enclosed with a .zip file. If there is more than one matching file type found in the zip archive, only the first match will be processed. Note that some of the example files provided by the IPC-2581 Consortium are .zip files. However, these zip files also include ODB++ archives, and other formats and documentation used for comparison and reference to the contained IPC-2581 file. When using these files to import into Sherlock, the desired .cvg or .xml file should be first extracted from the archive to insure the correct IPC-2581 file is imported.

To import an IPC-2581 file, select the **Import IPC-2581 Archive** from the **Project** menu. This will display a dialog to allow the selection of the IPC-2581 file to be imported, along with additional options that allow for manual override of the project name and CCA name. By default, Sherlock will only bring in layers from the IPC-2581 file that Sherlock uses. The option **Include Other Layers** will tell Sherlock to bring in any layer found in the IPC-2581. These additional layers will appear in the **Other Layers** folder of the **Layer Viewer.**

After selecting the desired file to import and optionally changing any other options, click the **Scan Archive** button to begin the import process. The file will first be pre-processed, and a dialog named the **Component Property Mapping** dialog, like the following will be displayed:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Component Property Mapping** dialog is an important, but sometimes overlooked, dialog that allows you to map various component/part properties included in the IPC-2581 file to corresponding Sherlock property fields. Sherlock parses any BomItem description and BomItem textual characteristics found in the IPC-2581 file to produce this dialog. The item **COMP_CLASS** in this example was found and the value **IC** was an example value found in the file. If desired, the indicated Sherlock part property will be assigned the values found for **COMP_CLASS** associated with each part found. If the indicated Sherlock part property is not correct or the indicated IPC-2581 property doesn't apply, change the selection to the correct Sherlock part property or to the value of **Ignore**.

Note:

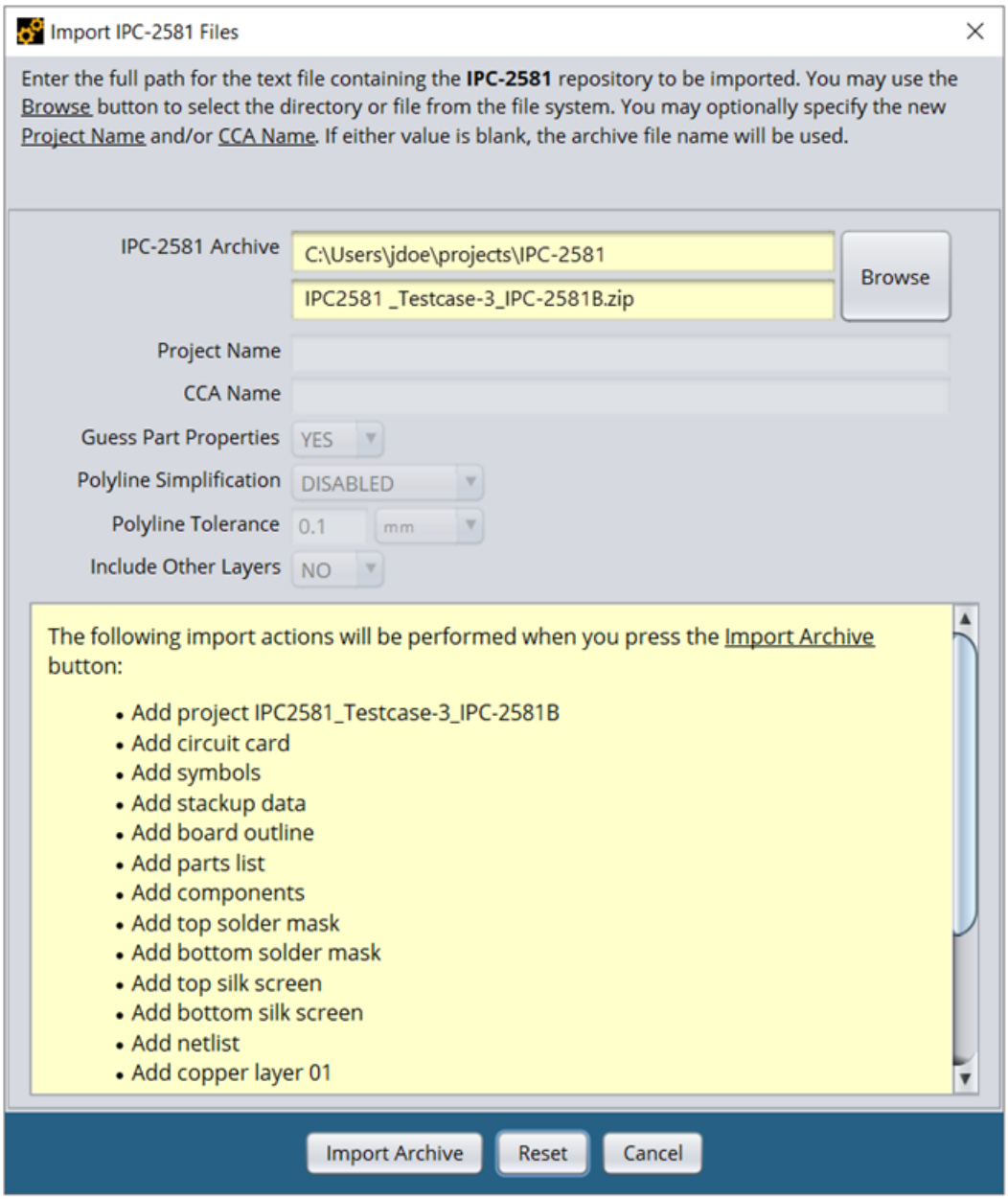
Starting with version 2023 R1, Sherlock's **Component Property Mapping** dialog now lists the **NonstandardAttribute** tag if it is found in the IPC-2581 file.

Note:

In this example, only one property is defined and the names are easily matched with Sherlock property names. In a real-world archive, there can be dozens of property names, in which case you'll need to determine which, if any, should be mapped to Sherlock properties.

After you've reviewed the various properties and have decided that they are mapped appropriately, press the **OK** button to continue the import process.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After the part properties have been mapped the **Import IPC-2581 Files** dialog will display the complete list of data that can be extracted automatically from the archive, as shown here:

In this case, Sherlock has found a complete set of files, including package information, stackup data, board outline, drill holes and all the important layers. Additionally, because the **Include Other Layers** option was selected, it found five additional layers to import which are not used directly by Sherlock.

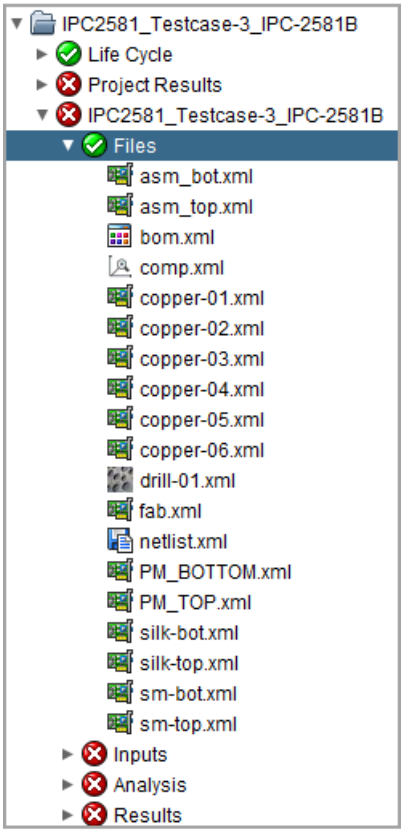
For ease of use and because of the larger size of IPC-2581 files, Sherlock will extract the XML that makes up each of the found items in the list of imported items above. Each will display as a separate XML file within Sherlock and properly associated to each layer within Sherlock.

Press the **Import Archive** button to start the import process, at which point the import dialog will close and the project navigation tree will be updated to include all the files being imported, as shown below. While each file is imported, the icon displayed next to the file will reflect a clock icon.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once complete, the appropriate file icon for the given file will be reflected. If any errors were generated while processing the files, then the file will be highlighted with an orange **"?"** icon.

At this point, the IPC-2581 file has been imported into Sherlock creating a new project containing a new circuit card. Project management can continue by reviewing part properties and various layers as detailed in Lesson 04 – Reviewing Part Properties (p. 30) and Lesson 05 – Layer Viewer (p. 43) of the Sherlock tutorial.

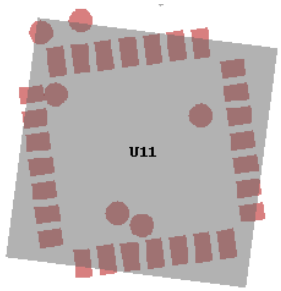


Many of the examples provided by the IPC-2581 Consortium also provide additional import types such as ODB++ for the examples to be used as a comparison. The ODB++ examples may also be loaded into Sherlock, see the ODB++ Project Creation (p. 23) tutorial more information. When doing so you might notice some discrepancies between an ODB++ version of a file and the IPC-2581 revision A and B versions of a file.

One such example appears with the test case 3 file used as an example in this user's guide section. If you view the bottom component and solder mask layers in Sherlock you will notice that component U11 appears to be rotated differently than the soldermask layer, however in the ODB++ version of the same example, the rotation is correct.

IPC-2581defines rotation values as counter-clockwise as viewed from the top of the board. Mirroring is done across the Y-axis and is performed after rotation. If you right-click the **comp.xml** file in the **Files** section of the imported IPC-2581 file, you can select **View File** from the menu to see how this component is specified. Component U11 is found near the bottom of the file. As shown below, the component is rotated to the left or counter-clockwise 82.601 degrees, then mirrored across the Y-axis meaning that all X values are set to -X. Because component U11 is rotated incorrectly, the rotation may be fixed by changing the component rotation or mirror setting.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



<Component refDes="U11" packageRef="PLCC28" layerRef="BOTTOM" part="FILTERED" mountType="SMT" standoff="0.0" height="0.175">

<Xform rotation="82.601" mirror="true"/>

<Location x="1.750" y="1.350"/>

</Component>

For comparison, the same component from the ODB++ archive is defined with a rotation of 277.4 and no mirroring. Rotation values in ODB++ are performed clockwise.

CMP 5 1.75 1.35 277.4 N U11 ??? ;0=1,1=0.175000

3.6.3.2. IPC-2581 Supported Elements

In the IPC-2581 specification, section *4.1 Content: FunctionMode* shows a table of how various functions are supported between the different function modes and levels. Sherlock requires most items from revision A **FULL** mode or revision B **USERDEF** mode:

Table 3.4: IPC-2581 Element Requirements

Name	Sherlock Supported
Hierarchical layer/stack instance files	Yes
Hierarchical conductor routing files	Yes
BOM	Yes
AVL	No
Component Packages	Yes
Land Patterns	No
Device Descriptions	Yes
Component Descriptions	Yes
Soldermask: Solder Paste Legend Layers	Yes
Drill and Routing Layers	Yes
Documentation Layers	Optional
Net List	Yes
Outer Copper Layers	Yes
Inner Layers	Yes
Miscellaneous Image Layers	Optional

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DFX Analysis	No

Sherlock supports most of the IPC-2581 specification, however there are some items which are not supported as Sherlock provides its own implementation for them or they are not required for use by Sherlock. The following tables indicate what is supported. This is not a meant to be a complete list but does cover most of the functionality that Sherlock normally provides.

Table 3.5: IPC-2581 Specification Support

IPC-2581 Item and Section	Status
3.3 Transformation characteristics (Xform)	Supported
3.4 Substitution Groups	
ColorGroup, Fiducial, FirmwareGroup, FontDef	Not supported
Feature, PolyStep, Simple, StandardShape, UserShape	Supported
LineDescGroup	lineProperty not supported; all lines are solid
FillDescGroup	fillProperty of void and solid are supported
StandardPrimitive	All shapes; Moire and Thermal are not fully supported
UserPrimitive	Text attribute not supported
4. Content	
DictionaryStandard	Supported as StandardPrimitive above
DictionaryUser	Supported as UserPrimitive above
DictionaryFont	Not supported
DictionaryLineDesc	Supported as LineDescGroup above
DictionaryFillDesc	Supported as FillDescGroup above
DictionaryColor	Not supported
DictionaryFirmware	Not supported
7. BOM	
BomHeader, RefDes, Textual Characteristics	Supported
8. ECAD	
8.1 CadHeader	Supported
8.2 CadData	
8.2.1 Layer	Supported; DrillTool not supported
8.2.2 Stackup	Supported
8.2.3 Step	

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nonstandard Attribute, Datum, Profile, LogicalNet, PhyNetGroup	Supported
PadStack, PadStackDef, Route, StepRepeat, DfxMeasurementList	Not supported
LayerFeature	Fiducial, SlotCavity, ColorGroup not supported

Other Limitations When Importing IPC-2581 Files

- Test points are created from component **TP** reference designators, not **testPoint** definitions.
- All arcs (other than circles and corners of rounded rectangles) are discretized into small lines when IPC-2581 files are imported into Sherlock.

3.7. Exporting Projects

This section covers the following topics:

3.7.1. Exporting Sherlock Projects (.zip, .dfr-result)
3.7.2. Exporting CCAs to Ansys Icepak / Electronics Desktop (.aedb)
3.7.3. Links to Other Types of Exports

Sherlock has many export features. Below are links to help you find information on specific types of exports. If you do not see what you are looking for, refer to Links to Other Types of Exports (p. 132) at the end of this section.

Export FEA Model (p. 458)

Exporting to Ansys Icepak (p. 809)

Trace Model and Sketch Export (p. 503)

Trace Reinforcement Export (p. 506)

Trace Reinforcement Model Export Automation (*.wbjn) (p. 509)

Exporting to Ansys Workbench (p. 747)

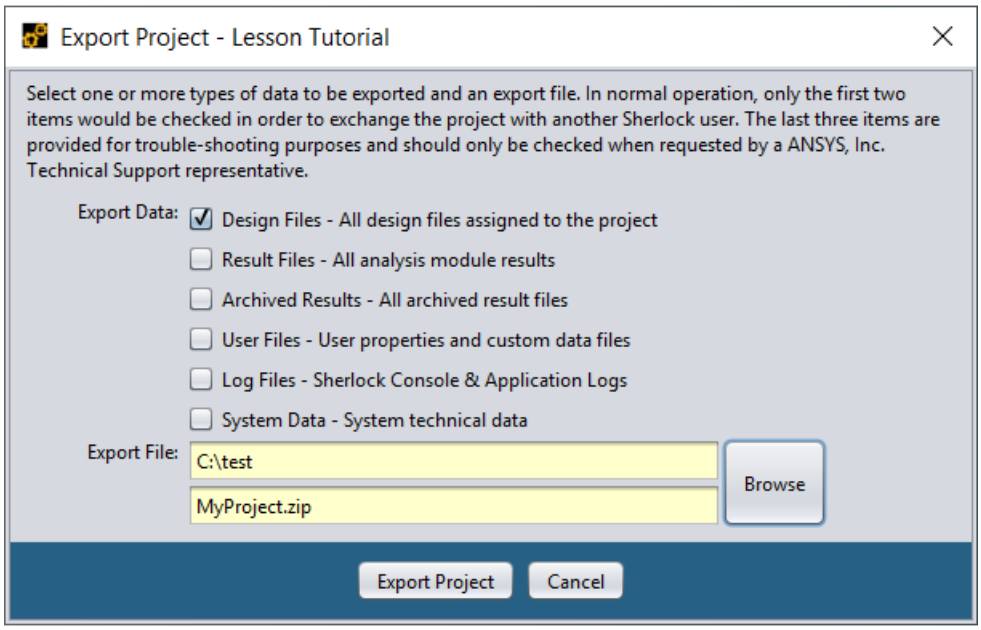
Exporting a CCA from Sherlock to Workbench (p. 749)

Exporting ECAD to Mechanical CAD Automation (p. 750)

3.7.1. Exporting Sherlock Projects (.zip, .dfr-result)

This section describes how to export a project in the standard Sherlock project format. In both single and multi-project mode, you can export a project at any time by selecting the **Project** > **Export Project** menu option from the **Main Menu**. At that point, the **Export Project** dialog will appear, as shown here, allowing you to indicate the data to be exported and the desired export file location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The overall size of the resulting export file depends on the data selections made. The smallest export file comes from saving only the design files, which includes the internal data stores, like the **Parts List** and **Stackup** data. Such an archive could be used to share the project with another Sherlock user, without providing them with specific test results.

Result files and archived result files could result in a large export file, depending on the number and types of tests performed. If you want to exchange results with another user, it is usually best to send them a single archived result file (**.dfr-result** suffix) instead of an exported project file. That way, they can use the **Sherlock Results Viewer** to view the results and all of the input data at the time the results were generated, without providing them with all of the design files as well.

The last three items are provided for trouble-shooting purposes and should only be checked when requested by an Ansys Technical Support representative.

3.7.2. Exporting CCAs to Ansys Icepak / Electronics Desktop (.aedb)

In Sherlock, you can export a circuit card assembly (CCA) file to Ansys Icepak / Electronics Desktop (AEDT). For a full description of the workflow, refer to the section Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis (p. 426).

3.7.3. Links to Other Types of Exports

Sherlock has many export features. Below are links to help you find information on specific types of exports.

- Export Materials (p. 164)
- Export Solders (p. 151)
- Reporting and Data Export (p. 799)
- Exporting Graphical Data (p. 804)
- Exporting Laminates (p. 138)
- Exporting AVL Part Data (p. 273)
- Exporting Part Wizard Patterns (p. 290)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Exporting the Parts List (p. 223)
- Package Export and Import (p. 323)
- Exporting Tabular Data (p. 806)
- Life Cycle Import / Export (p. 358)
- Exporting 3D Images (p. 798)
- Exporting Results from the Results Viewer (p. 790)
- Exporting the Parts List (p. 223)

3.8. Stackups and Copper Layers

3.8.1. Importing Copper Layers

Importing a stackup is explained in the following tutorials:

- *ODB++ Project Creation*: See Importing an ODB++ Archive (p. 24).
- *Ad Hoc Project Creation*: See Importing Copper Layers (p. 65).
- For information on adding CCA files with stackup/copper information to your project, see the section Adding Files to a Circuit Card Assembly (p. 100) and its subsection Copper Layer Files (p. 105).

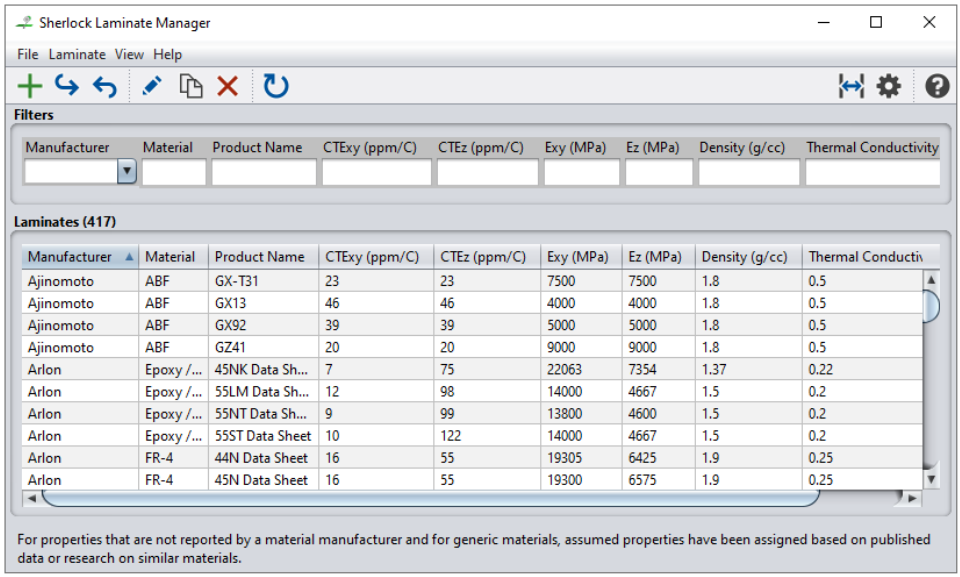
3.8.2. Stackup Tools

- **Stackup Tools:** See Additional Stackup Tools (p. 69) in the tutorial *Ad Hoc Project Creation* for a detailed explanation of Sherlock's stackup tools.
- **Insulated Metal Substrate (IMS) Tools:** Sherlock's tools for generating IMS stackups are explained in the *Ad Hoc Project Creation* tutorial. See Insulated Metal Substrate Stackup (p. 68).

3.9. Laminate Management

The Sherlock **Laminate Manager** allows you to create and modify Sherlock laminate definitions. Properties modified in the **Laminate Manager** are used in the stackup definition of a circuit card assembly. The Manager is accessible from the Sherlock **Libraries** menu, and as a standalone utility, is also accessible from the Sherlock **Start Menu** folder.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



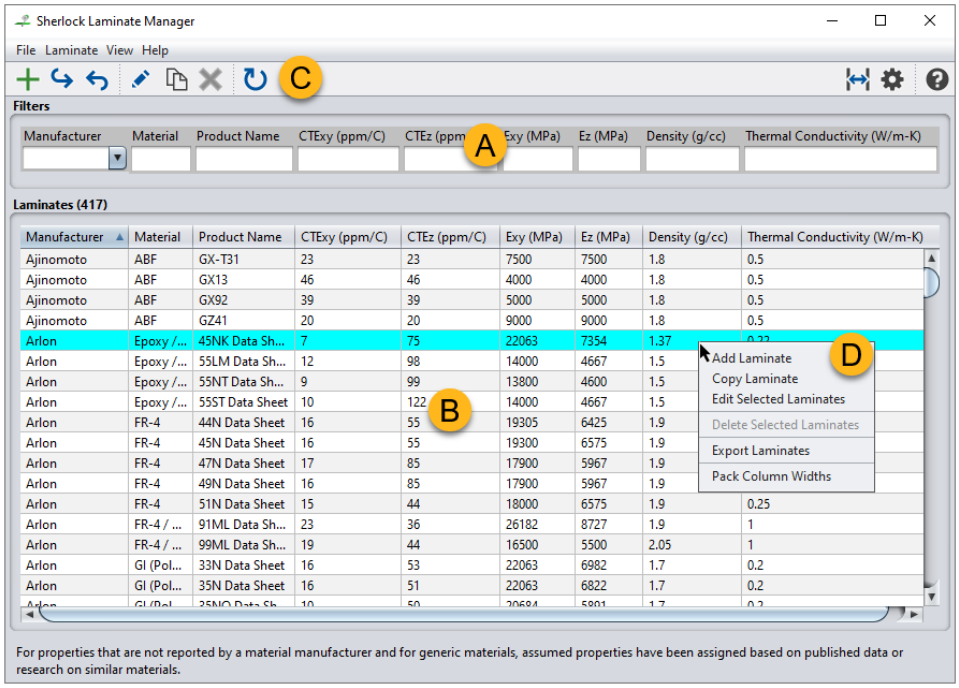
In this section, the following topics are covered:

3.9.1. Laminate Manager
3.9.2. Laminate Manager Icon Ribbon
3.9.3. Filtering Laminates
3.9.4. Selecting Multiple Laminates for Editing, Exporting
3.9.5. Adding and Copying Laminates
3.9.6. Deleting Laminates
3.9.7. Exporting Laminates
3.9.8. Importing Laminates
3.9.9. Laminate Editor
3.9.10. Editing Multiple Laminates Simultaneously

3.9.1. Laminate Manager

To open the **Laminate Manager**, select **Libraries > Management** from the **Main Menu**. The **Laminate Manager** is organized into two sections. The top section (A, below) provides a set of fields that allow you to filter the list of laminates. The bottom table (B) is the list of all laminates or those laminates matching any selected filters. You can access the basic features of the **Laminate Manager** by using the icon ribbon (C). Or you can right-click anywhere on the table and use the context menu (D).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.9.2. Laminate Manager Icon Ribbon

The icon ribbon allows you to perform common functions with a simple button-click. Icons are context sensitive and become active only when they are useable. For example, some icons are grayed out when no laminate is selected.

Laminate Manager Icons



A. **Add New Laminate**: Add a new laminate to the Laminate Library. See Adding and Copying Laminates (p. 137).

B. **Import Laminates**: Import laminates (`.csv` file) into the Laminate Library. See Importing Laminates (p. 140).

C. **Export Laminates**: Export laminates to a `.csv` file. See Exporting Laminates (p. 138).

D. **Edit the Selected Laminate(s)**: Opens the selected laminate(s) in the **Laminate Editor**. See Laminate Editor (p. 141).

E. **Copy the Selected Laminate**: Use this feature to create a new laminate that has properties similar to an existing laminate. Clicking the icon opens the selected laminate in the **Laminate Editor** with the **Product Name** blank. See Adding and Copying Laminates (p. 137).

F. **Delete the Selected Laminate from the Laminate Library**: See Deleting Laminates (p. 138).

G. **Reload the Laminate Library**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



H. **Automatically Resize all Column Widths**: Resizes the column widths to conform to the current table size.

I. **Customize the Laminates Table View**: Launches the **Customize Laminates** window where you can add or remove columns and select what part property each column will display. The interface works the same as that for customizing the **Parts List**. See Customizing the Parts List (p. 191).

J. **Open Sherlock Help**: Click to open the online **Laminate Manager** help.

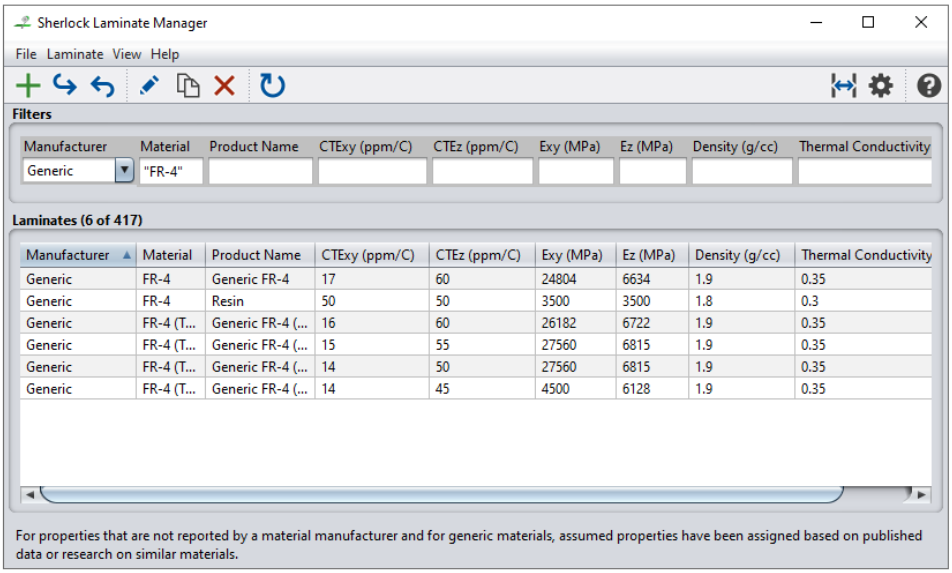
3.9.3. Filtering Laminates

The **Filters** in the **Laminate Manager** allow you to select which laminates are listed in the table. For instance, to filter the laminate definitions to FR-4 material types only, enter **`"FR-4"`** in the **Material** column filter and press **Enter**.

Multiple filters may be applied at the same time. To filter the table even further (continuing the example above), select or enter **`Generic`** for the **Manufacturer** column and press **Enter**. The results are shown below.

Tip:

The main window for the **Laminate Manager** can be customized and filtered in ways similar to that of the Parts List. See Customizing the Parts List (p. 191).



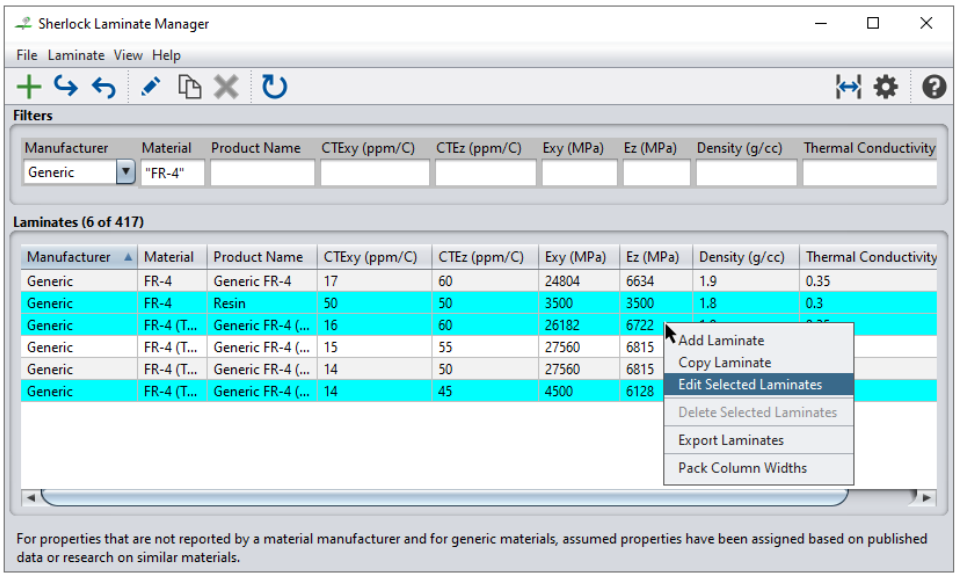
3.9.4. Selecting Multiple Laminates for Editing, Exporting

Some operations, such as editing (p. 141) or exporting (p. 138) laminates, may require you to select multiple laminates (rows) from the Laminates Table. There are several ways to do this:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Left-click any row and drag the mouse up or down to select a range of laminates.
- **SHIFT + Left-Click** to select all rows between the previously selected row and the current row.
- **CTRL + Left-Click** to toggle row selection

Once you have selected the laminates, you can edit or export by clicking the appropriate icon in the icon ribbon or by right-clicking one of the highlighted rows and using the context menu as shown below.



3.9.5. Adding and Copying Laminates

3.9.5.1. Adding a New Laminate

To add a new laminate to the local Laminate Library, do the following from the **Sherlock Laminate Manager**:

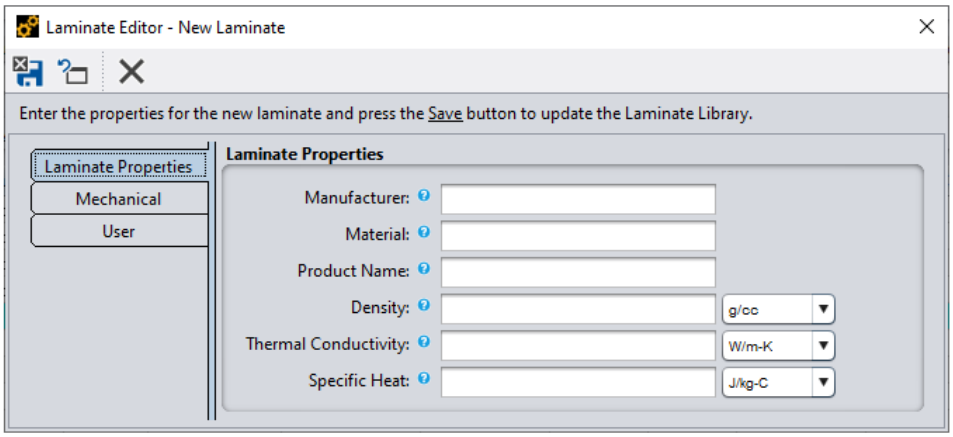
- Click the **Add Laminate** icon .
- Or, from the **Laminate** menu, select **Add Laminate**.
- Or, right-click anywhere on the Laminates Table and select **Add Laminate** from the context menu.

The **Laminate Editor** dialog will appear with all laminate properties blank, ready for you to define a new laminate. When finished, click the **Save** button to add the new laminate to the local Laminate Library.

Tip:

If you are creating a laminate with nearly identical properties to an existing laminate, consider using the **Copy** feature to duplicate and then edit an existing laminate rather than starting with a blank form. See below, Copying a Laminate (p. 138).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.9.5.2. Copying a Laminate

If you are creating a laminate with similar properties to an existing laminate, it might be easier to **Copy** and then edit an existing laminate rather than starting with a blank form.

To **Copy** a laminate, right-click the laminate on the Laminates Table and select **Copy Laminate** from the context menu. Sherlock will create a copy of the laminate, but the **Product Name** will be blank. Provide a **Product Name** for the new laminate and, if needed, edit the properties. Then click the **Save** button to add the new laminate to the local Laminate Library.

3.9.6. Deleting Laminates

Any user-defined laminate that is not a Sherlock system laminate may be deleted. From the **Laminate Manager**,

- Select the laminate to be deleted and click the **Delete Laminate** icon in the icon ribbon.
- Or, right-click the laminate to be deleted and select **Delete Laminate** from the context menu.

If the option to **Delete Laminate** is not available, this means you have selected a system laminate which cannot be deleted. If a system laminate has been modified, however, the **Delete Laminate** option is available. In this case, clicking **Delete Laminate** only removes the user-defined properties and reverts the laminate to system defaults.

3.9.7. Exporting Laminates

Laminate definitions may be exported to CSV or XLS spreadsheet files. The files can also be imported (p. 140) back into the **Laminate Manager** as well. There are several ways to open the **Export Laminate Library** dialog. From the **Laminate Manager**,

- Click the **Export Laminates** icon in the icon ribbon.

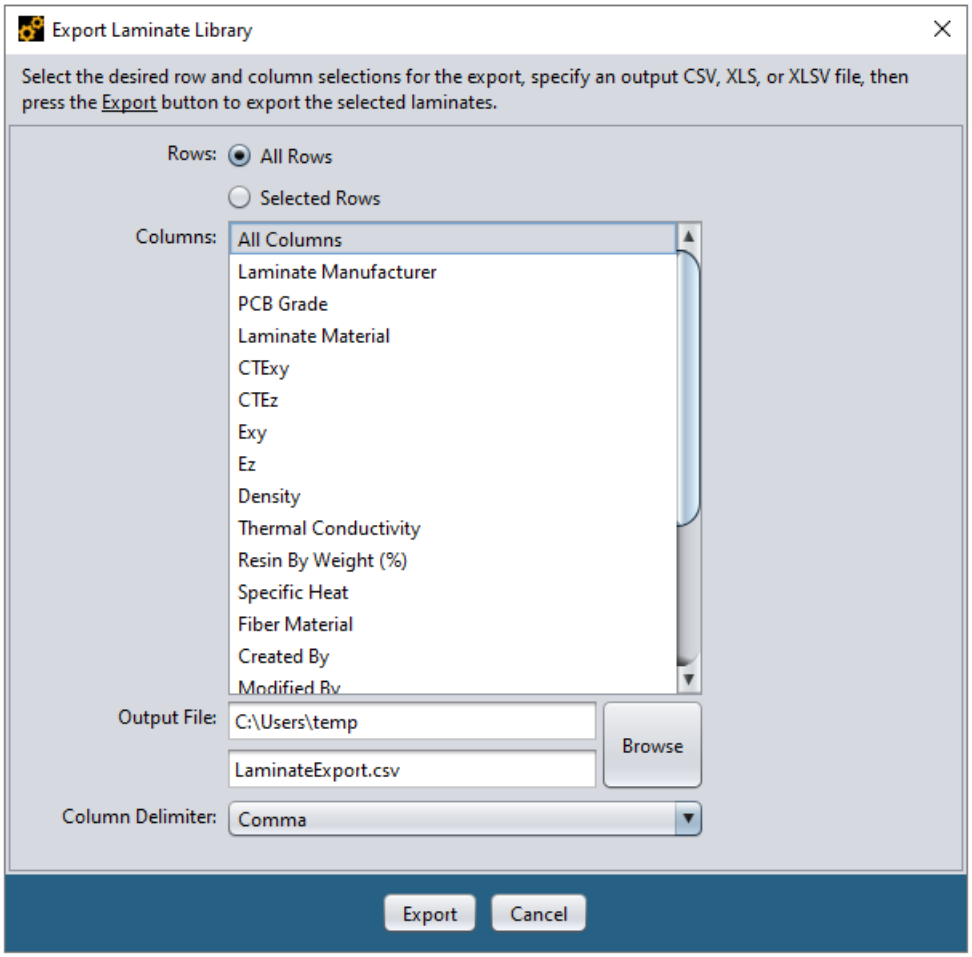
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Or, from the **File** menu, select **Export Laminates**.

Tip:

If you wish to export specific laminates only, you can select the table rows you wish to export (see Selecting Multiple Laminates for Editing, Exporting (p. 136)). Then right-click one of the highlighted rows and select **Export Laminates** from the context menu. Once you are in the **Export Laminate Library** dialog, you must select the option **Selected Rows**.

The **Export Laminate Library** dialog is shown below where you can specify your export preferences:



- **Rows**: By default, Sherlock exports **All Rows**. If you wish to export only the rows you selected on the Laminates Table, choose the option **Selected Rows**.
- **Columns**: Specify whether all available columns are to be exported or only specific columns. Use **SHIFT + Left-Click** and **CTRL + Left-Click** and to make your selection.
- **Output File**: Specify the name and folder destination of the exported file. Supported file extensions are `.csv`, `.xls`, and `.xlsx`. Click the **Browse** button to navigate to the destination folder.
- **Column Delimiter**: Select **Comma**, **Semicolon**, **Tab**, or **Space**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

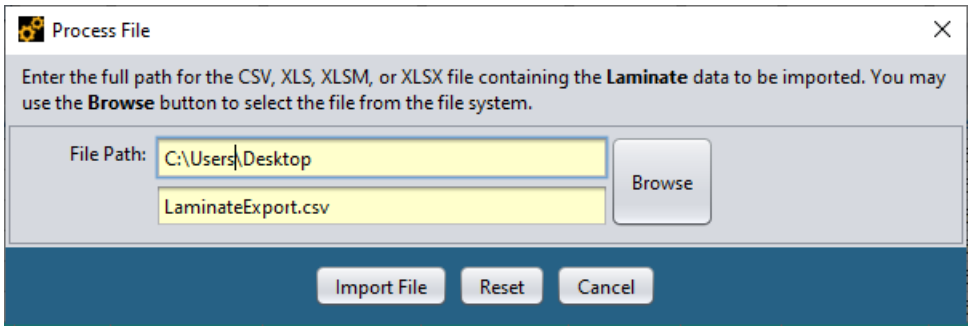
Once you have set your preferences, click the **Export** button. If successful, a message appears indicating the number of rows exported. You may then open the exported file with a spreadsheet program

3.9.8. Importing Laminates

You may also import modified laminate definitions from a spreadsheet or CSV file. To do so,

- Click the **Import Laminates** icon in the icon ribbon.
- Or, from the **File** menu, select **Import Laminates**.

A **Process File** dialog will open. Select the file to be imported. For convenience, the last imported/exported file will be used as the default, but you are free to select any file on your system.

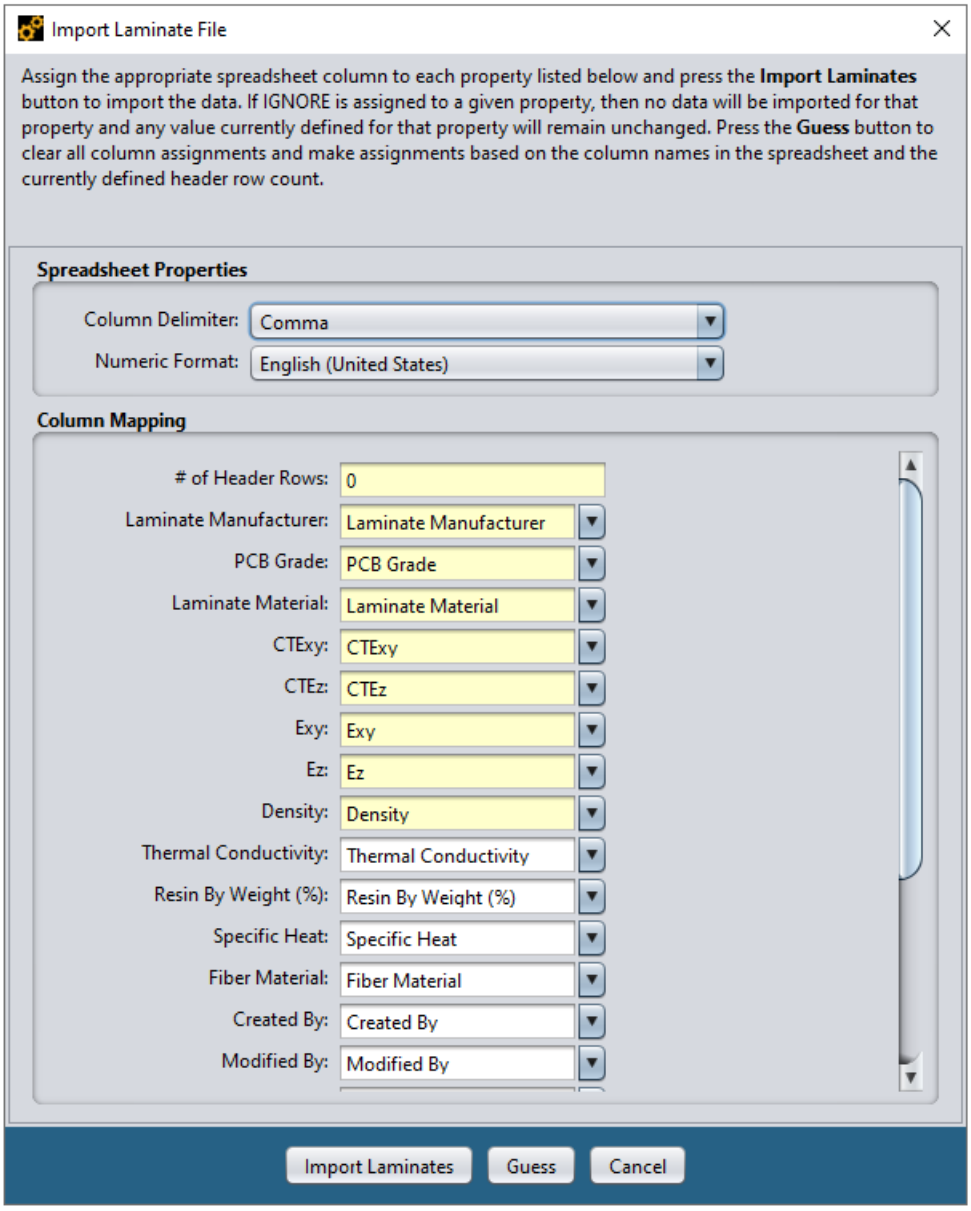


After selecting the file and clicking **Import File**, Sherlock will scan the file to determine the number of header rows and to map the column names. Sherlock also determines which columns should be mapped to Sherlock's laminate properties. When processing is complete, Sherlock opens the **Import Laminate File** dialog shown below where you can verify the results. Sherlock's guesses are usually correct, but it's a good idea to double-check.

Once you have confirmed the column mappings are correct, press the **Import Laminates** button.

When the import process successfully completes, you will receive a confirmation dialog. If any laminate fails to import, then none of the rows in the file will be imported. When this happens, an error dialog appears with information on the reasons for the failure.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



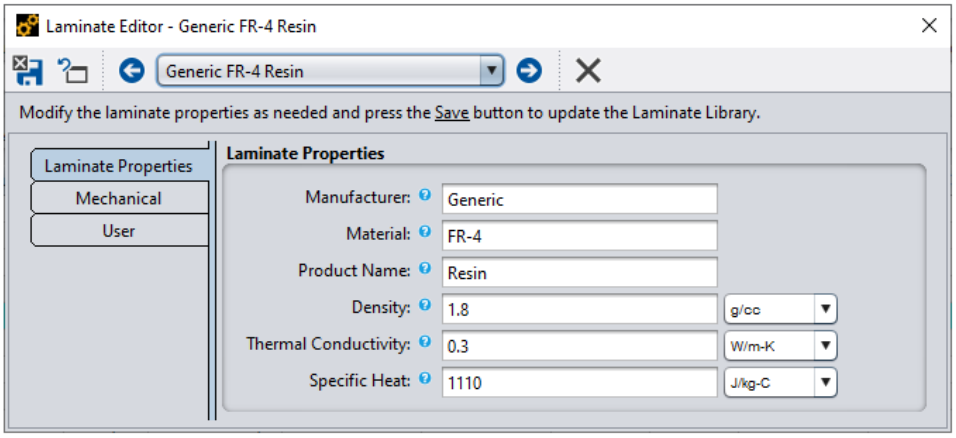
3.9.9. Laminate Editor

The **Laminate Editor** displays all the properties for the selected laminate and allows you to edit their values. You can open a single laminate in the editor or mutliple laminates simultaneously. Laminate properties are organized by three tabs: **Laminate Properties**, **Mechanical**, and **User**. Depending on the laminate selected, certain tabs or properties will be disabled to indicate they are not applicable for the given laminate.

3.9.9.1. Laminate Editor Interface

3.9.9.2. Editing Laminates

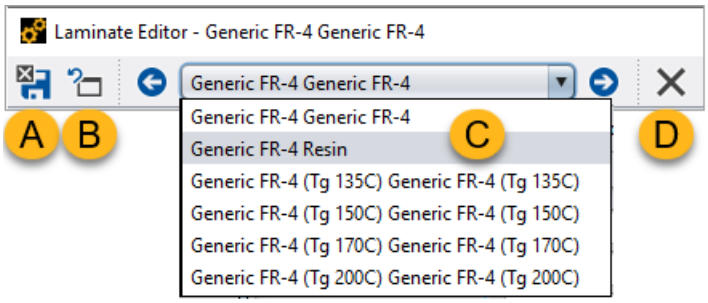
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.9.9.1. Laminate Editor Interface

The Icon Ribbon

The **Laminate Editor**'s icon ribbon provides shortcuts for common functions.



A. **Save Changes and Exit the Editor**

B. **Reset the Form**: Resets all property fields to their last saved values.

C. **Navigation Tools**: You can navigate to another laminate on the list without closing the editor. Simply choose the laminate from the dropdown menu, or use the navigation arrows to select the **Next** or **Previous** laminate on the list.

Note:

Any *unsaved* edits to a laminate are lost when you navigate to another laminate.

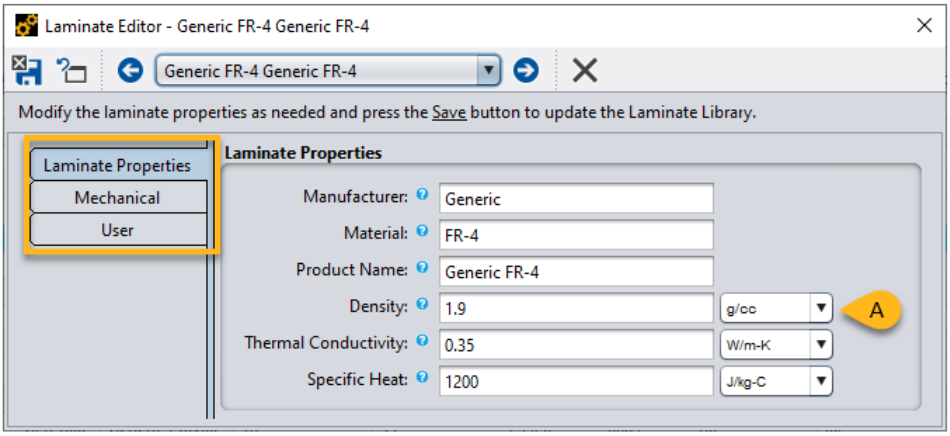
When list filters are active, the navigation tools navigate the *filtered* list only. If no filters are applied, then the entire list of laminates is available.

The navigation tools are *not* available if you are editing *multiple* parts simultaneously.

D. **Exit the Editor**: Close the editor *without* saving changes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Laminate Editor Properties Tabs



- **Laminate Properties** tab
  - **Manufacturer**: Manufacturer of the laminate layer material.
  - **Material**: Grade of material used in the board.
  - **Product Name**: Material used for laminate layer.
  - **Density**: Average density of the circuit card.
  - **Thermal Conductivity**: Thermal conductivity of the laminate.
  - **Specific Heat**: Specific heat of the laminate.

Note:

For properties which require a unit of measure, use the dropdown menu (A, in the image above) to change the units if desired. When a new unit is selected, Sherlock automatically converts the assigned value.

- **Mechanical** tab
  - **Exy**: Elastic modulus, in-plane.
  - **Ez**: Elastic modulus, out-of-plane.
  - **CTExy**: Coefficient of thermal expansion, in-plane
  - **CTEz**: Coefficient of thermal expansion, out-of-plane
- **User** tab
  - **Created By**: The username and timestamp of the person who created the laminate. This is automatically populated by Sherlock. It cannot be edited manually. See note below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Modified By**: The username and timestamp of the person who last modified the laminate. This is automatically populated by Sherlock. It cannot be edited manually. See note below.

> **Note:**
>
> If a laminate does not have any customized values, then the **Created By** and **Modified By** fields will be listed as **Ansys, Inc.**

– **Notes**: In this text-editable field, you can manually add other information you want associated with this specific laminate.

3.9.9.2. Editing Laminates

The **Laminate Editor** displays all the properties for the selected laminate(s) and allows you to modify their values. After making any changes, click the **Save** button to save the changes to the local copy of the Sherlock Laminate Library.

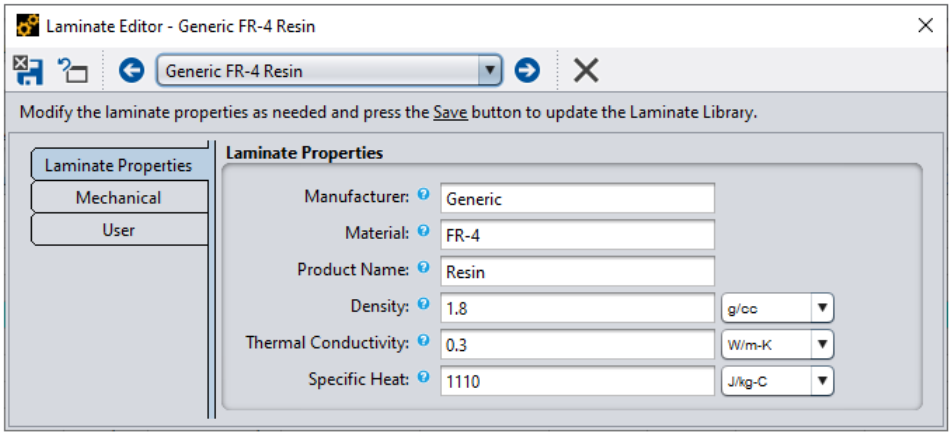
> **Tip:**
>
> See the section on User Data Files (p. 166) for more information about the location and format of the local Laminate Library.

When opening the **Laminate Editor**, you can choose to load a single laminate for editing, or you can load multiple laminates in the editor if you wish to apply the same edits to more than one laminate simultaneously (see Editing Multiple Laminates Simultaneously (p. 145) below).

To open a single laminate in the **Laminate Editor**:

- Double-click the desired laminate (table row) from the Laminates Table.
- Or right-click the laminate and select **Edit Selected Laminates** from the context menu.
- Or select the laminate, and then click the **Edit Laminate** icon  .

If you do *not* want to save any changes or are simply viewing the laminate definition, click the **Close the dialog** icon  . The **Reset** button will revert all fields to their last *saved* values.



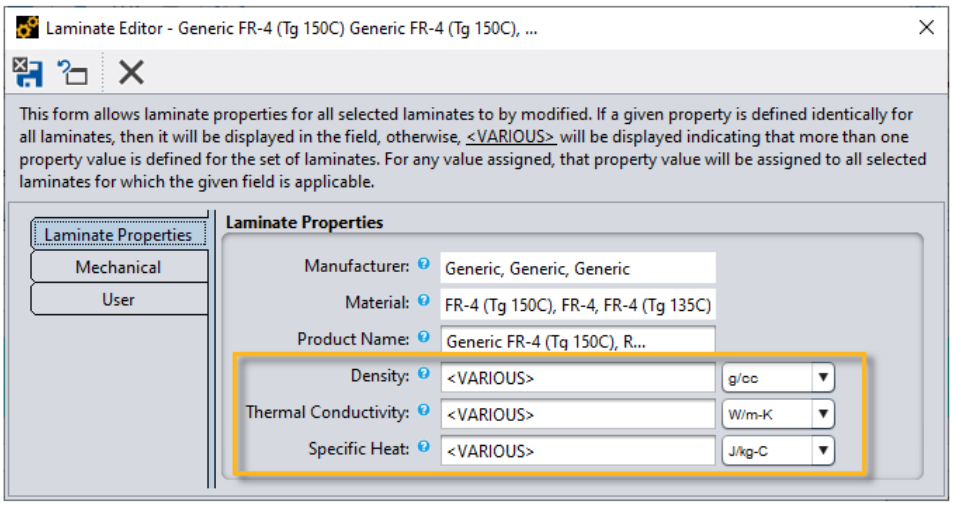
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.9.10. Editing Multiple Laminates Simultaneously

When you have loaded multiple laminates into the editor, some property fields will display the value **<VARIOUS>**. This lets you know the laminates do not *not* have the same value for that property. (In the example below, the laminates being edited have various values for **Density**, **Thermal Conductivity**, and **Specific Heat**.) If you wish, you can replace **<VARIOUS>** with a specific value. When the change is **Saved**, the new value will be applied to all the laminates loaded in the editor.

Remember, when editing multiple laminates at the same time, any change you make to a property is applied to *all* the laminates in the editor when you **Save** the changes. So, for example, if you set the **Density** to 1.9 g/cc and **Save** the change, the **Density** property for all the laminates loaded in the editor becomes 1.9 g/cc. If you make no changes to a property, all the laminates will retain their current value.

If you do *not* want to save any changes, click the **Close the dialog** icon . The **Reset** button will revert all fields to their last *saved* values.



3.10. Solder Management

The **Sherlock Solder Manager** allows for a user to create and modify Sherlock solder definitions. Properties modified in the Solder Manager are used during analysis. The **Solder Manager** is accessible by selecting **Libraries > Solders** from the Sherlock menu and is also a standalone utility accessible from the Sherlock start menu folder.

In this section, the following topics are covered:

3.10.1. Solder Manager Listing

3.10.2. Solder Editor

3.10.3. Temperature-Dependent Material Properties

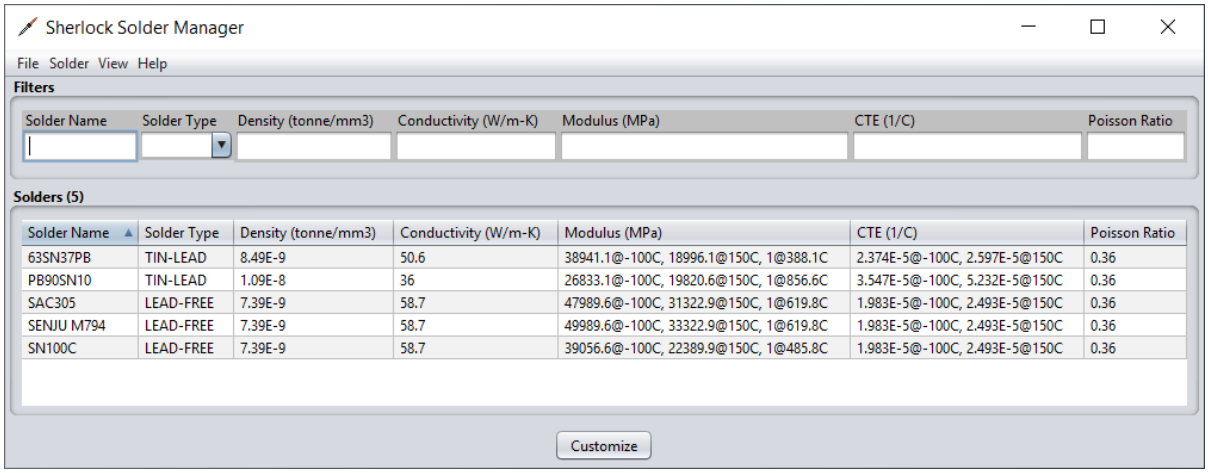
3.10.4. Export Solders

3.10.5. Import Solders

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.10.1. Solder Manager Listing

The main **Solder Manager** window is organized into two sections. The top section provides a set of fields to allow filtering of the solders in the manager. The bottom table is a list of all solders or those solder matching any selected filters from above.



To filter the solder definitions to only the TIN-LEAD solder types, enter or select TIN-LEAD in the **Solder Type** column filter and press Enter. This will immediately filter the solder table to only display solders that at TIN-LEAD. Multiple filters may be applied. The table is filtered when all applied filter definitions match a given solder.

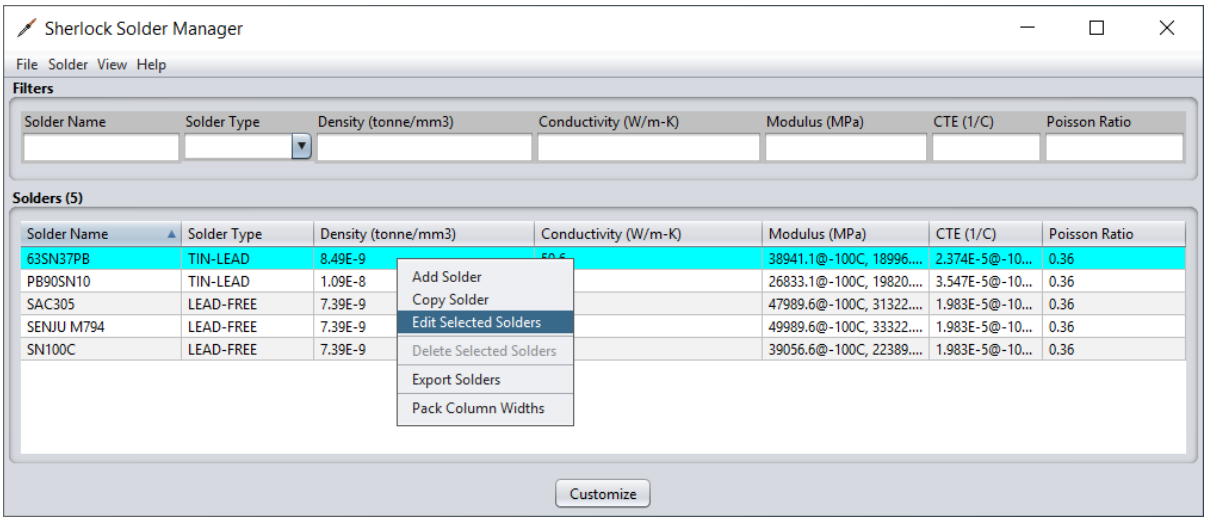
This main window for the **Solder Manager** can be customized and filtered in ways similar to the customization and filtering detailed in the Parts List Management (p. 197) users guide. The actions include, selecting the visible columns and advanced filtering techniques. Refer to the Parts List Management (p. 197) users guide for details on using these features.

3.10.2. Solder Editor

3.10.2.1. Selecting a Solder to Edit

To view or edit a given solder, you may double-click the desired solder from the solder table listing. You may also right-click the solder in the solder table listing and select **Edit Selected Solders** from the context menu.

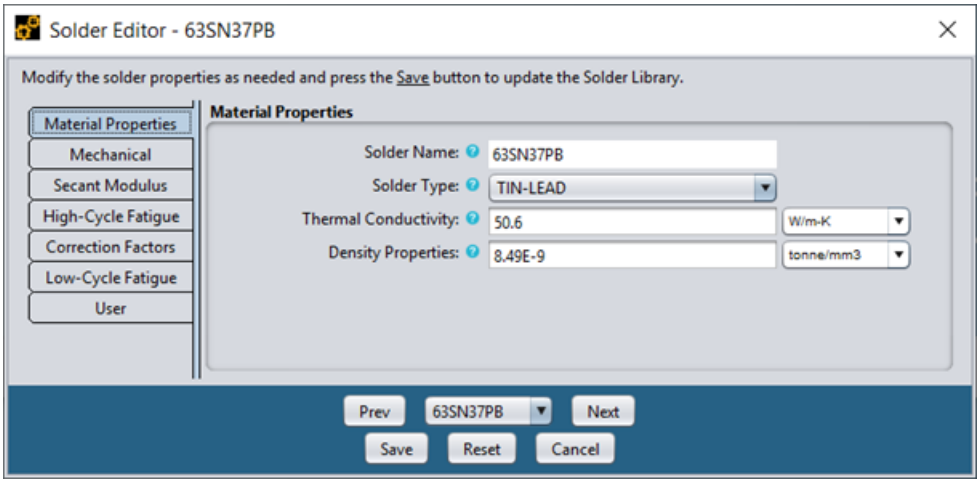
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After selecting a solder to edit, the **Solder Editor** dialog window will be displayed. This window provides a look at all the property values defined for the selected solder and allows for those property values to be modified.

3.10.2.2. Using the Solder Editor

After selecting a solder to edit a new dialog window will appear with a listing of all the available material properties for the selected solder. From the **Solder Editor** a user can view and make changes to any of the material properties. The **Solder Editor Groups** properties into different categories, represented by tabs in the editor.



For properties which indicate a unit of measure, the currently selected unit of measure for that property is displayed next to the field. By selecting a new unit of measure from the list, the value displayed in the form will automatically be converted to the selected units. These includes fields which may have specified multiple temperature-dependent properties.

The **User** tab provides a place to specify notes about a given solder and indicate the creator of the solder and also user that last modified the solder. If a solder does not have any customized values, then the **Created By** and **Modified By** fields will be listed as Ansys, Inc.

After making any desired solder changes, use the **Save** button to save the changes to the local copy of the Sherlock **Solder Library**. See the section User Data Files (p. 166) for more information

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

about the location and format of the local **Solder Library**. If you don't wish to save any changes or are simply viewing the solder definition, select the **Cancel** button when done. The **Reset** button will revert any changes you have made to the definition since opening the dialog to the original settings.

Note:

If you modify any of the solders provided by Sherlock, future Sherlock updates to the solders will not be reflected until the customized solder is removed.

If you wish to view additional solder definitions, you may do so without closing the solder editor dialog window by simply selecting the solder from the selector at the bottom of the window or using the **Prev** and **Next** buttons. The solder selector and the **Prev** and **Next** buttons are both limited to the filtered solders from the solder manager itself. If no filters were applied, then the entire list of solders is available.

3.10.3. Temperature-Dependent Material Properties

Temperature-dependent values can be specified for the following properties:

- Elastic Modulus (MPa)
- Poisson Ratio
- CTE (1/C)
- Density Properties (tonne/mm3)
- Thermal Conductivity (W/m-K)
- Fatigue Exponent, Fatigue Coefficient, FEA Power, Critical Strain, and Shock Strain (used for shock and vibration fatigue cycles to failure calculations)

Temperature-dependent values are defined as a set of temperature-value pairs and are passed directly to the FEA engine for all materials used during a given analysis. FEA engines typically use the given data points to determine the property value for a specific temperature by using linear interpolation between the bounding data points. For all temperatures below the minimum temperature specified, the value associated with the minimum temperature will be used. Similarly, for all temperatures above the maximum temperature specified.

Temperature-value pairs are specified using the following notation:

<Value> @ <Temp> <Units>

where <Value> is the property value, <Temp> is the temperature and <Units> are the temperature units. (NOTE: The value units must be those specified in the list above.) Two or more temperature-value pairs must be comma separated. For example, the following notation:

400@20C, 320.5@212F

specifies a value of 400 at 20 degrees centigrade and a value of 320.5 at 212 degrees Fahrenheit. Temperature units can be freely mixed in a given list of temperature-value pairs.

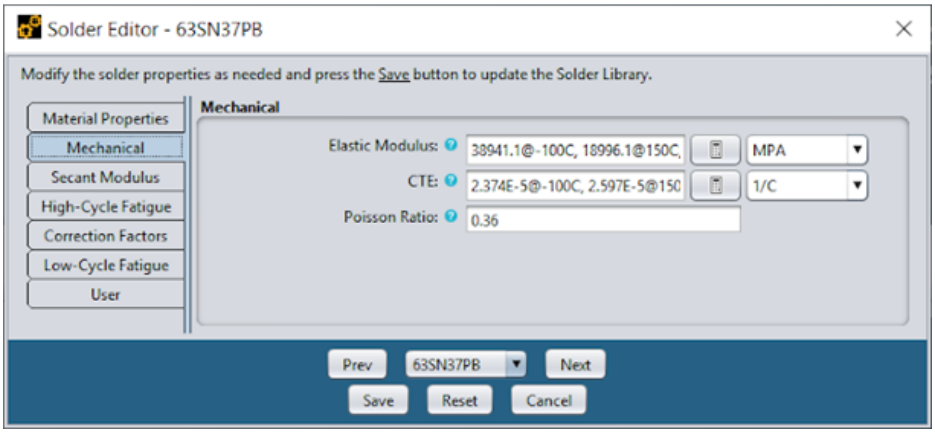
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In situations where Sherlock needs to determine a temperature-dependent material property value when no explicit temperature is defined (such as when mixing material property values in a given PCB layer to determine the average layer property value), a room temperature value of 20C is used for interpolation purposes.

3.10.3.1. CTE and Elasticity Modulus Calculators

To assist with computing temperature dependent values for the **CTE** and **Elastic Modulus** properties, Sherlock provides a temperature value calculator for these properties. This calculator takes in a base CTE or **Elastic Modulus** value, a Tg value, temperature range, and inputs that control the number of temperature values to produce to provide a set of temperature dependent values.

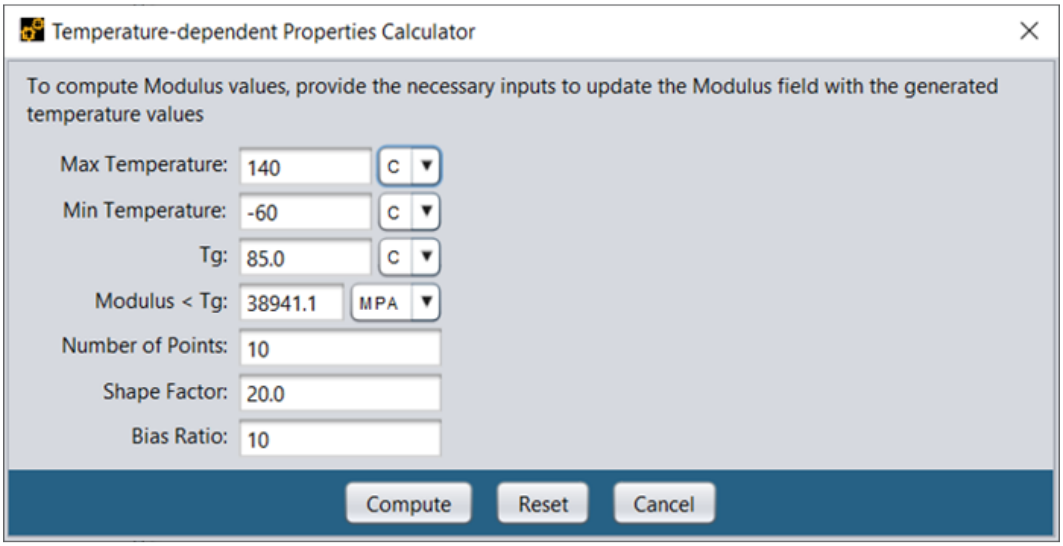
From the **Solder Editor** for the solder 63SN37PB, select the **Mechanical** tab to reveal the current CTE and **Elastic Modulus** values.



The default solder contains several **CTE** and **Elastic Modulus** values. To use the calculator to expand this list, click the calculator icon button next to the desired field.

When you click the calculator for **Elastic Modulus**, the **Temperature-Dependent Properties Calculator** launches. As shown below, the calculator provides a set of default input properties used to compute additional material temperature values for the selected property. The default **Tg** and **Modulus < Tg** (or CTE values) originate from the source material. Note the default values shown below: The **Number of Points** is 10 which means the calculator will generate ten material temperature values for elasticity within the range of -60° and 140° Celsius. Because the calculator generates additional temperature values at the **Max** and **Min Temperatures**, the total number of values generated will be twelve.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Upon clicking the **Compute** button, the values will be computed, and the underlying field updated with new material temperature values. To accept these values, click the **Save** button for the solder and they will be committed.

Note:

The default values of any system provided solder can be restored by using the **Delete Selected Solders** menu item from the Solder Manager listing. Deleting system provided solders simply resets those material property values.

3.10.3.2. Adding and Copying Solders

To add a new solder, select **Add Solder** from the **Solder** menu, or right-click in the solder table listing and select **Add Solder** from the menu. The solder editor dialog will appear with all material properties empty, ready for a new solder to be defined. When you are complete defining a solder, click the **Save** button to add the new solder.

To copy a solder, right-click the solder to be copied from the solder table listing and select **Copy Solder** from the menu. A copy of the solder will be made. Provide a solder name for the copied solder and make any changes desired to the material properties, then click the **Save** button to add the copied solder.

3.10.3.3. Deleting Solders

Any user-defined solder that is not a Sherlock system solder may be deleted. Right-click the solder to be deleted from the solder table listing and select **Delete Solder** from the menu. If the **Delete Solder** item is not available, this indicates that it is not a user-defined solder.

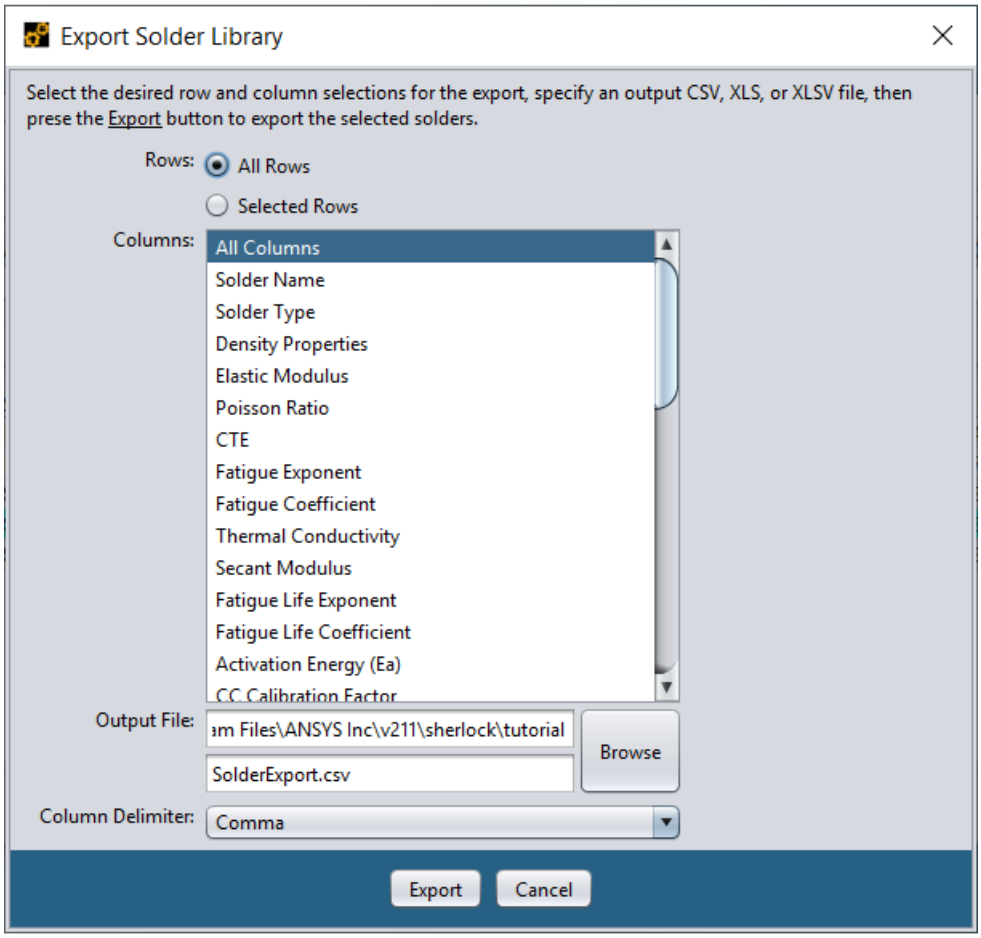
If a system solder has been modified, the **Delete Selected Solders** menu item will be available. However, it will not actually delete the solder. Deleting a system solder that has user-defined properties will simply remove all the user-defined properties and allow the solder to revert to being completely system defined.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.10.4. Export Solders

Solder definitions may be exported to CSV or XLS spreadsheet files to allow external viewing or modifications. Exported files can then be used to import solder definitions back into the solder manager.

To export all **Solder Definitions**, select from the **File** menu **Export Solders** or right-click in the solder table listing and select **Export Solders** from the context menu. If you wish to export only specific solders, select the solders you wish to export from the solder table listing, then right-click the table and select **Export Solders**. In either case, the **Export Solder Library** dialog will be displayed.



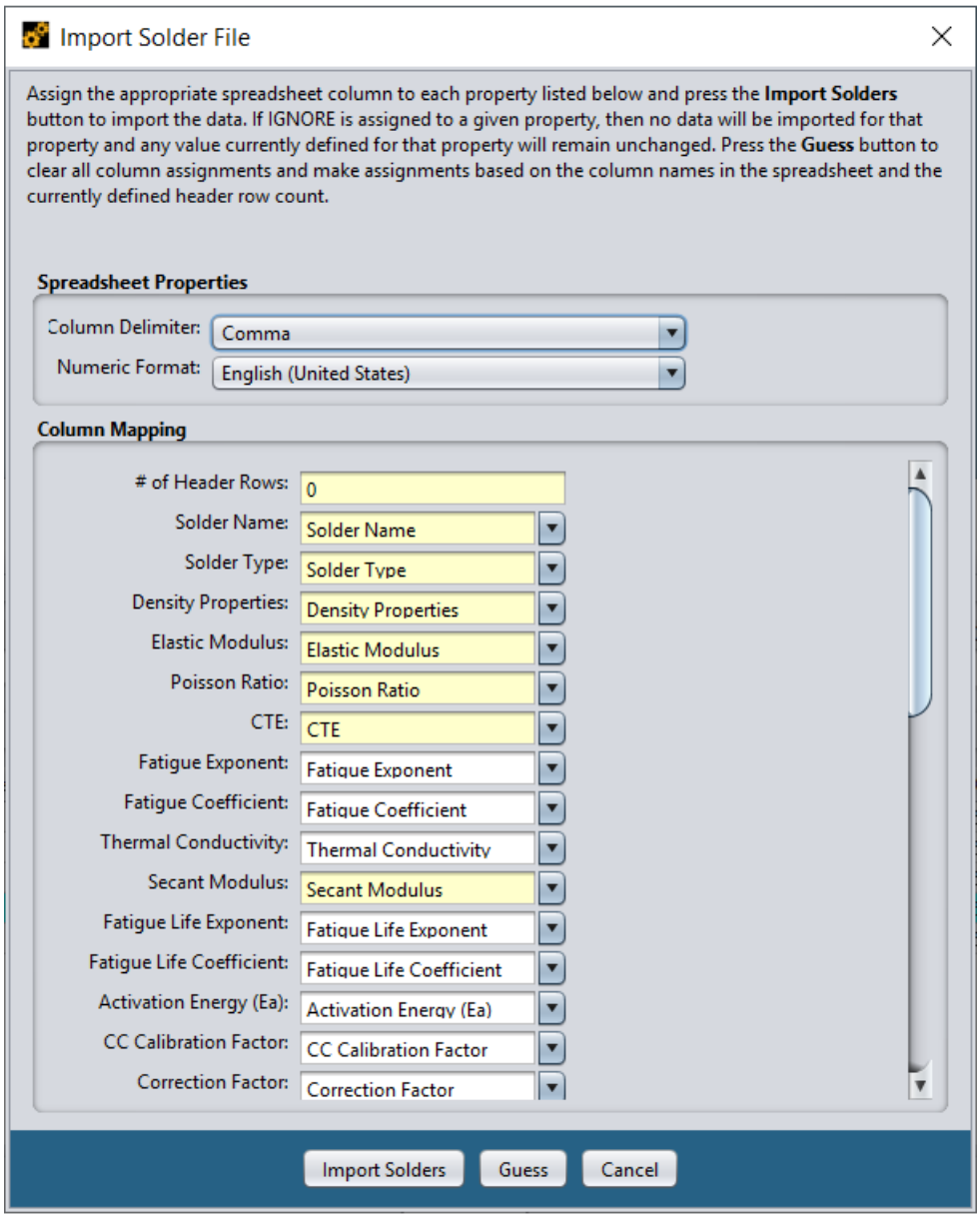
The **Rows** and **Columns** selections allow you to customize which rows and/or columns should be exported. Select **All Rows** if all rows should be exported or select **Selected Rows** if only the rows selected in the solder table listing should be exported. Similarly, select **All Columns** if all columns should be exported or select only specific columns to be exported.

After specifying the output file location, click the **Export** button. Supported file extensions are **.csv, .xls,** and **.xlsx**. If successful, a message will appear indicating the number of rows selected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.10.5. Import Solders

Solder definitions may be imported into the **Solder Manager** from a CSV, XLS or XLSX spreadsheet file by selecting the **File > Import Solders** option from the main menu. At that point, a file dialog will be displayed allowing you to designate the file to be imported.



After selecting the file, the **Import Solder File** dialog will appear, allowing you to verify the column mapping for each file/sheet that is imported. Sherlock scans each file in an attempt to determine the number of header rows, the column names and which columns should be mapped to the various Sherlock solder properties. Such guesses are usually right, but you should check them anyway by viewing the import file using an appropriate spreadsheet application.

When the import form is filled out properly, press the **Import Solders** button to import the data.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the import process has completed, you will see a confirmation dialog. If any solder definitions fail to import, then none of the rows in the file will be imported and an error dialog will be displayed with the information regarding why a particular solder definition failed to be imported.

3.11. Material Management

The Sherlock **Material Manager** allows for a user to create and modify Sherlock material definitions. Properties modified in the **Material Manager** are used during analysis. The **Material Manager** is accessible from **Libraries > Materials** on the Sherlock homepage and as a standalone utility accessible from the Sherlock **Start Menu** folder.

In this section, the following topics are covered:

3.11.1. Material Manager Listing

3.11.2. Material Editor

3.11.3. Temperature-Dependent Material Properties

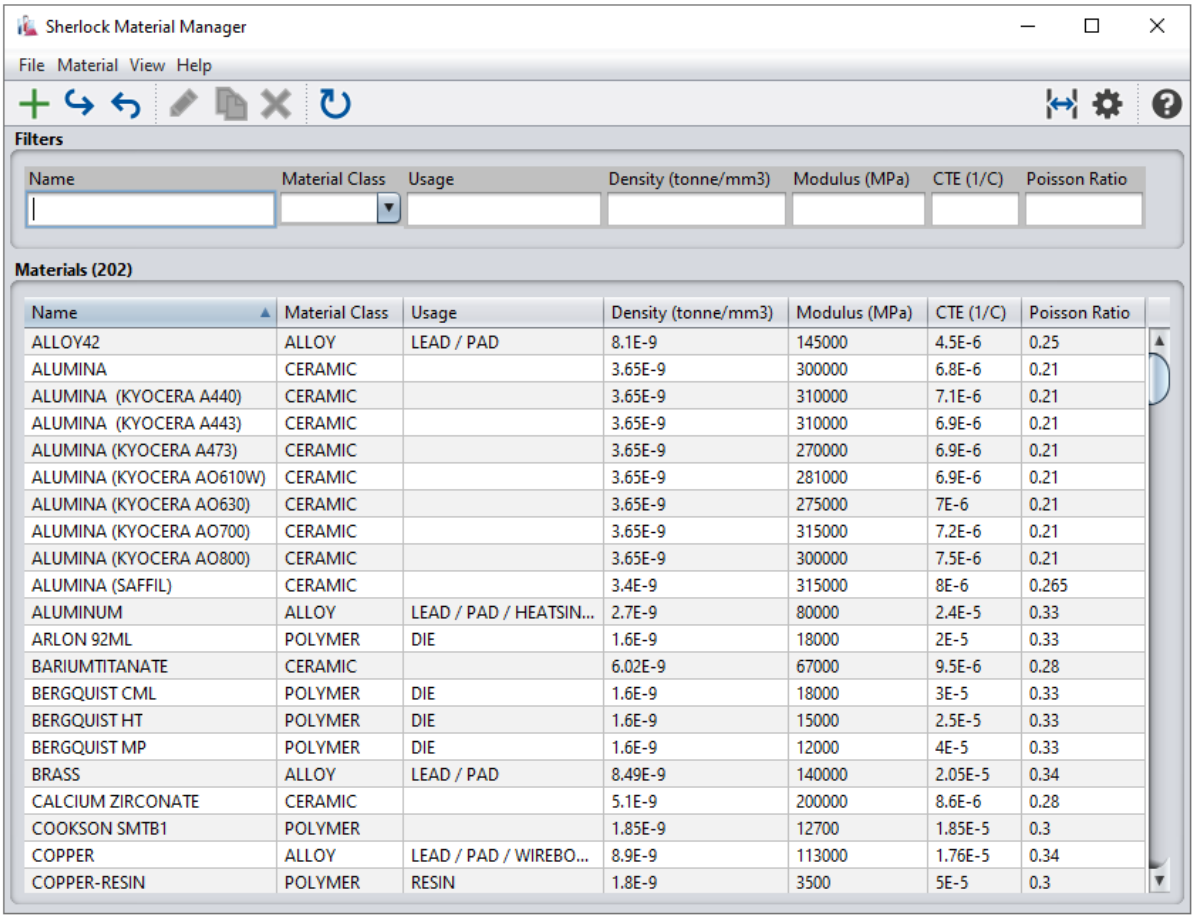
3.11.4. Export Materials

3.11.5. Import Materials

3.11.1. Material Manager Listing

To access the **Material Manager**, click **Main Menu > Libraries > Materials**. The main **Material Manager** window is organized into two sections. The top section provides a set of fields for filtering the list. The bottom table lists all the materials or those materials matching the filtering criteria.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	Material Class	Usage	Density (tonne/mm3)	Modulus (MPa)	CTE (1/C)	Poisson Ratio
ALLOY42	ALLOY	LEAD / PAD	8.1E-9	145000	4.5E-6	0.25
ALUMINA	CERAMIC		3.65E-9	300000	6.8E-6	0.21
ALUMINA (KYOCERA A440)	CERAMIC		3.65E-9	310000	7.1E-6	0.21
ALUMINA (KYOCERA A443)	CERAMIC		3.65E-9	310000	6.9E-6	0.21
ALUMINA (KYOCERA A473)	CERAMIC		3.65E-9	270000	6.9E-6	0.21
ALUMINA (KYOCERA AO610W)	CERAMIC		3.65E-9	281000	6.9E-6	0.21
ALUMINA (KYOCERA AO630)	CERAMIC		3.65E-9	275000	7E-6	0.21
ALUMINA (KYOCERA AO700)	CERAMIC		3.65E-9	315000	7.2E-6	0.21
ALUMINA (KYOCERA AO800)	CERAMIC		3.65E-9	300000	7.5E-6	0.21
ALUMINA (SAFFIL)	CERAMIC		3.4E-9	315000	8E-6	0.265
ALUMINUM	ALLOY	LEAD / PAD / HEATSIN...	2.7E-9	80000	2.4E-5	0.33
ARLON 92ML	POLYMER	DIE	1.6E-9	18000	2E-5	0.33
BARIUMTITANATE	CERAMIC		6.02E-9	67000	9.5E-6	0.28
BERGQUIST CML	POLYMER	DIE	1.6E-9	18000	3E-5	0.33
BERGQUIST HT	POLYMER	DIE	1.6E-9	15000	2.5E-5	0.33
BERGQUIST MP	POLYMER	DIE	1.6E-9	12000	4E-5	0.33
BRASS	ALLOY	LEAD / PAD	8.49E-9	140000	2.05E-5	0.34
CALCIUM ZIRCONATE	CERAMIC		5.1E-9	200000	8.6E-6	0.28
COOKSON SMTB1	POLYMER		1.85E-9	12700	1.85E-5	0.3
COPPER	ALLOY	LEAD / PAD / WIREBO...	8.9E-9	113000	1.76E-5	0.34
COPPER-RESIN	POLYMER	RESIN	1.8E-9	3500	5E-5	0.3

For example, to generate a list that includes only ALUMINA material types, enter ALUMINA in the **Name** column filter and press **Enter**. The table will immediately display only those materials containing ALUMINA. You can apply multiple filters simultaneously.

You can customize and filter the **Material Manager** window just as you can the **Parts List**, including setting column visibility and using advanced filtering techniques. Refer to Parts List Management (p. 197) for details on using these features.

Material Manager Icon Ribbon



A. **Add**: Add a new material to the Material Library. See Adding and Copying Materials (p. 163).

B. **Import**: Import materials into the Material Library: See Import Materials (p. 165).

C. **Export**: Export materials from the Material Library: Export Materials (p. 164).

D. **Edit**: Edit the selected material: Opens the **Material Editor**. See Material Editor (p. 155) below.

E. **Copy**: Copy the selected material to a new material: See Adding and Copying Materials (p. 163).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

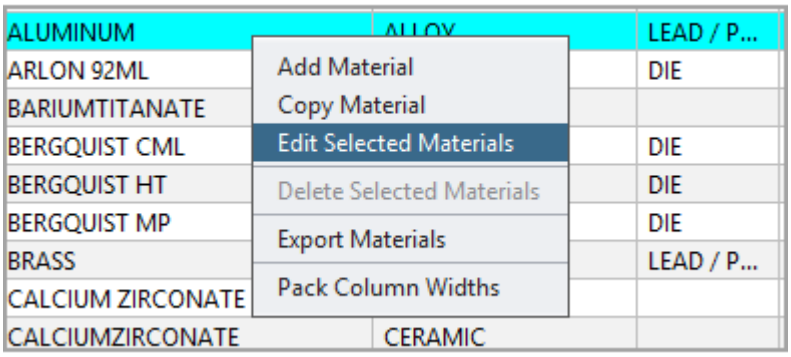
F. **Delete**: Delete the selected material from the Material Library: See Deleting Materials (p. 163).

G. **Reload**: Reload the Material Library.

3.11.2. Material Editor

3.11.2.1. Selecting a Material to Edit

To view or edit a given material, you may double-click the desired material from the materials table. You may also right-click the material entry and select **Edit Selected Materials** from the context menu.



After selecting a material to edit, the **Material Editor** dialog window will be displayed. This window provides a look at all the property values defined for the selected material and allows for those property values to be modified.

3.11.2.2. Using the Material Editor

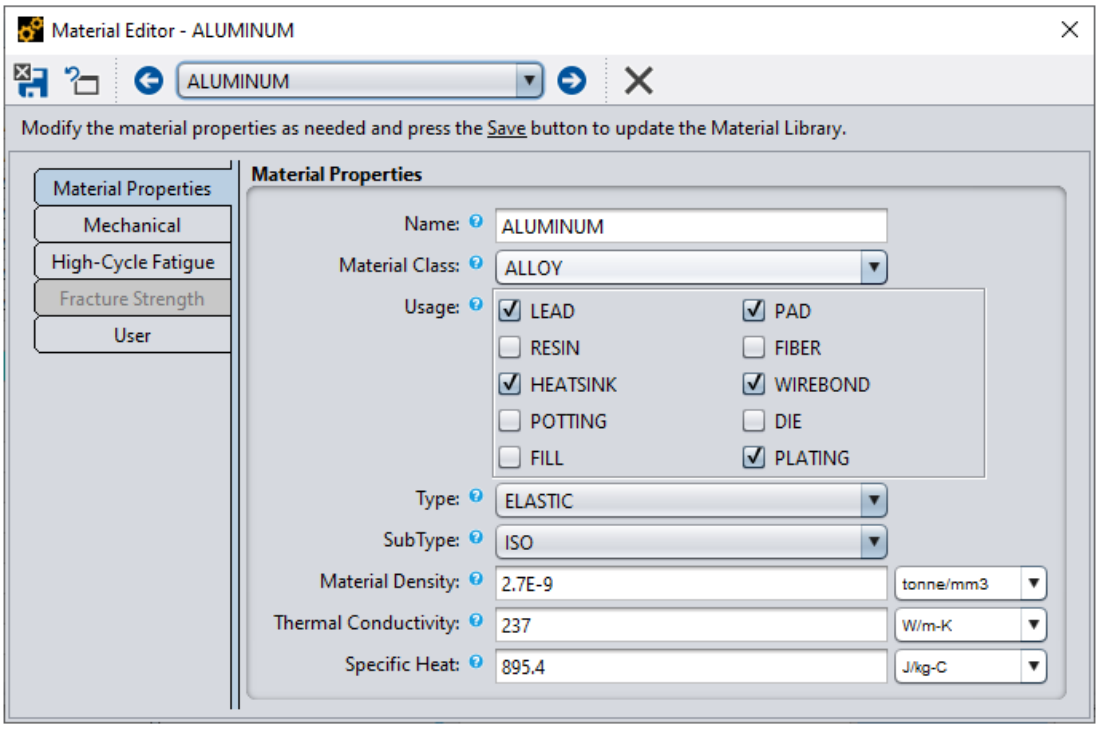
Subsections:

3.11.2.2.1. Material Editor Icon Ribbon

3.11.2.2.2. Material Editor Property Tabs and Fields

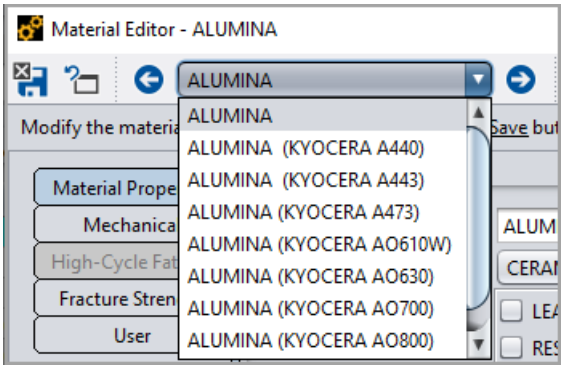
After selecting a material to edit, a new dialog window appears with a listing of all the available properties for the selected material. From the **Material Editor**, you can view and make changes to any of the material properties. The **Material Editor** groups properties into different categories, represented by tabs in the editor. For a complete explanation of the interface, see the sections below, Material Editor Icon Ribbon (p. 157) and Material Editor Property Tabs and Fields (p. 157).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After making any desired material changes, click the **Save** button to save the changes to the local copy of the Sherlock Material Library. See the section User Data Files (p. 166) for more information about the location and format of the local Material Library. If you don't wish to save any changes or are simply viewing the material definition, select the **Cancel** button when done. The **Reset** button will revert any changes you have made to the definition since opening the dialog to the original settings.

If you wish to view additional material definitions you may do so without closing the **Material Editor** dialog window by simply selecting the material from the dropdown menu at the top of the window or using the **Prev** and **Next** icons. The material selector and the **Prev** and **Next** icons are both limited to the filtered materials from the material manager itself. If no filters were applied, then the entire list of materials is available. Following the earlier example where the ALUMINA filter was applied to the **Name** column, only the ALUMINA materials are displayed as shown here.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.11.2.2.1. Material Editor Icon Ribbon



A. **Save and Exit**: Save changes and exit the editor.

B. **Reset**: Reset the form to its last saved values.

C. **Previous**: Show the previous material on the list. If you have applied filters to the **Material Manager** table, only the filtered materials are displayed.

D. **Material Selector**: Use this dropdown menu to select the material you wish to edit. When you select another material, changes you made to the current material will be lost if you haven't saved them. If you have applied filters to the **Material Manager** table, only the filtered materials are listed.

E. **Next**: Show the next material on the list. If you have applied filters to the **Material Manager** table, only the filtered materials are displayed.

F. **Close**: Close the dialog. Changes are *not* saved.

3.11.2.2.2. Material Editor Property Tabs and Fields

Tip:

- Tabs or properties that do not apply to the selected material are disabled.
- For properties requring a unit of measure, select the units from the dropdown list. When you select a new unit of measurement, Sherlock converts the numerical value so the actual value of the property remains the same. This includes fields which may have multiple temperature-dependent properties.

- **Material Properties Tab**
  - **Name**: Name of the material
  - **Material Class**: Supported values are ALLOY, CERAMIC, and POLYMER.
  - **Usage**: Select how the material can be used in Sherlock. For example, if you wish to model component pads with a specific material, you must ensure **PAD** has been selected under **Usage** for that material.
  - **Type**: Supported value is **ELASTIC**.
  - **SubType**: Supported value is **ISO**.
  - **Material Density**: For material density, Sherlock stores scientific notation values to four digits. In some cases, this may result in entered values being rounded slightly when saved.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Thermal Conductivity**: Thermal conductivity of the object being modeled.
  - **Specific Heat**: Specific heat of the object being modeled.
- **Mechanical Tab**
  - **Elastic Modulus**: Elastic modulus for the given material. For instructions on using the Elastic Modulus calculator, see CTE and Elasticity Modulus Calculators (p. 159) below.
  - **Material CTE**: Coefficient of thermal expansion for the given material. For instructions on using the CTE Modulus calculator, see CTE and Elasticity Modulus Calculators (p. 159) below.
  - **Material CTE 2**: Coefficient of thermal expansion in the radial direction used for fiber materials. For instructions on using the CTE Modulus calculator, see CTE and Elasticity Modulus Calculators (p. 159) below.
  - **Poisson Ratio**: Poisson's ratio for the given material.
  - **Tg**: Glass transition temperature (°C) for the given material.
  - **Shape Factor**: Shape factor used for solder joint fatigue analysis.
- **High Cycle Fatigue Tab**
  - **Fatigue Exponent**: Fatigue strength exponent for the given material.
  - **Fatigue Coefficient**: Fatigue strength coefficient for the given material.
- **Fracture Strength Tab**
  - **Fracture Strength**: Fracture strength for the given material.
- **User Tab**: The **User** tab provides a place to record notes about a given material.
  - **Created By**: The username (and timestamp) of the person who created the library item definition. If no customized value exists, then the field will display **Ansys, Inc**.
  - **Modified By**: The username (and timestamp) of the person who last modified the library item definition. If no customized value exists, then the field will display **Ansys, Inc**.
  - **Notes**: Here you can enter your own additional information associated with the library item.

3.11.3. Temperature-Dependent Material Properties

Temperature-dependent values can be specified for the following properties:

- Elastic Modulus (MPa)
- Poisson Ratio
- CTE (1/C): Defines the CTE for all homogeneous material. If the material is a fiber, this property defines the axial CTE of the material.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- CTE 2 (1/C): Defines the CTE along the radial direction of the fiber. The material property is primarily used in laminate property calculations. For all materials other than fiber, this value is ignored.
- Density Properties (tonne/mm3)
- Thermal Conductivity (W/m-K)
- Fatigue Exponent and Fatigue Coefficient (used for shock and vibration fatigue cycles to failure calculations)
- Specific Heat (J/kg-C)

Temperature-dependent values are defined as a set of temperature-value pairs and are passed directly to the FEA engine for all materials used during a given analysis. FEA engines typically use the given data points to determine the property value for a specific temperature by using linear interpolation between the bounding data points. For all temperatures below the minimum temperature specified, the value associated with the minimum temperature will be used. Similarly, for all temperatures above the maximum temperature, the value associated with the maximum temperature will be used.

Temperature-value pairs are specified using the following notation:

<Value> @ <Temp> <Units>

where <Value> is the property value, <Temp> is the temperature and <Units> are the temperature units. (NOTE: The value units must be those specified in the list above.) Two or more temperature-value pairs must be comma separated. For example, the following notation:

400@20C, 320.5@212F

specifies a value of 400 at 20 degrees centigrade and a value of 320.5 at 212 degrees Fahrenheit. Temperature units can be freely mixed in a given list of temperature-value pairs.

In situations where Sherlock needs to determine a temperature-dependent material property value when no explicit temperature is defined (such as when mixing material property values in a given PCB layer to determine the average layer property value), a room temperature value of 20C is used for interpolation purposes.

3.11.3.1. CTE and Elasticity Modulus Calculators

To assist with computing temperature dependent values for the CTE and Elastic Modulus properties, Sherlock provides a temperature value calculator. This calculator takes a base CTE or Elastic Modulus value, a Tg value, and temperature range to produce a set of temperature dependent values.

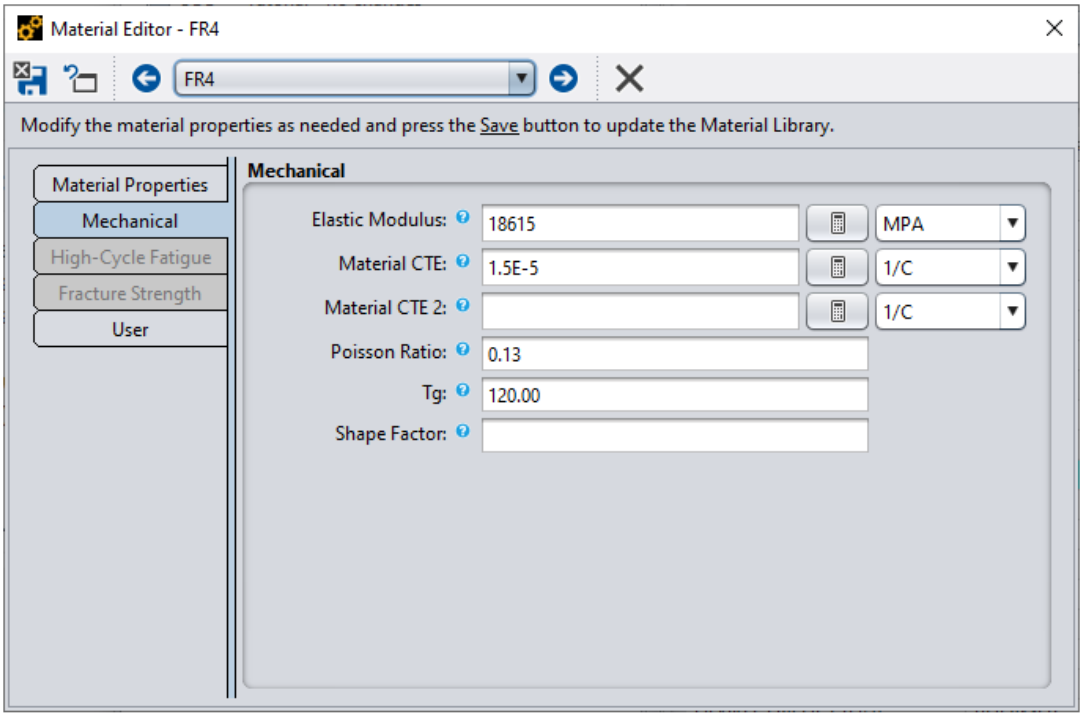
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The calculator bases its calculations on observed behavior in polymers, so you should use the calculator only when dealing with polymers. Generally, the calculator has a 5x decrease in modulus when above Tg, and a 3.5x increase in CTE when above the Tg.

Note:

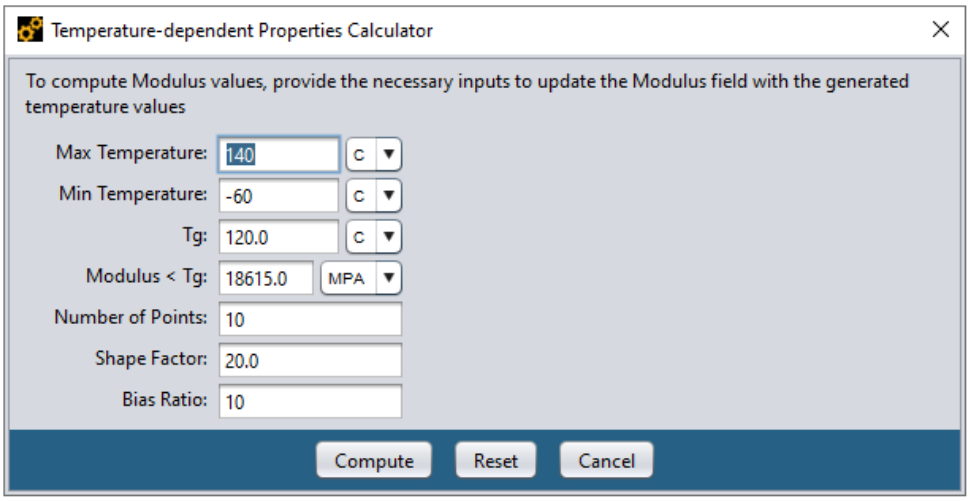
The default values of any system-provided material can be restored by using the **Delete Selected Materials** menu item from the Material Manager listing. Deleting system-provided materials simply resets those values to their defaults.

Here is how you access the calculator: From the **Material Editor** (p. 155) for the material FR4, select the **Mechanical** tab to reveal the current CTE and Elastic Modulus values as shown below.



As you can see, the default material contains a single CTE and Elastic Modulus value. To expand this list, click the calculator icon next to the desired field. This opens the **Temperature-Dependent Properties Calculator** shown below. The calculator provides a set of default input properties used to compute additional material temperature values for the selected property. The default **Tg** and **Modulus < Tg** (or CTE values) originate from the source material. Note the default values shown below: The **Number of Points** is 10 which means the calculator will generate ten material temperature values for elasticity within the range of -60° and 140° Celsius. Because the calculator generates additional temperature values at the **Max** and **Min Temperatures**, the total number of values generated will be twelve.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

To ensure accurate calculations, it is important to clear any parameter you wish to change in the **Temperature-Dependent Properties Calculator** before computing.

When you click the **Compute** button, the values will be computed, and the underlying fields updated with new material temperature values. To accept and commit the new material values, click **Save**.

The input parameters are defined below. The graphs which follow illustrate how the parameters affect the calculations:

Max Temperature: The maximum temperature at which temperature dependent material properties are calculated

Min Temperature: The starting temperature at which temperature dependent material properties are calculated

Tg: Glass transition temperature

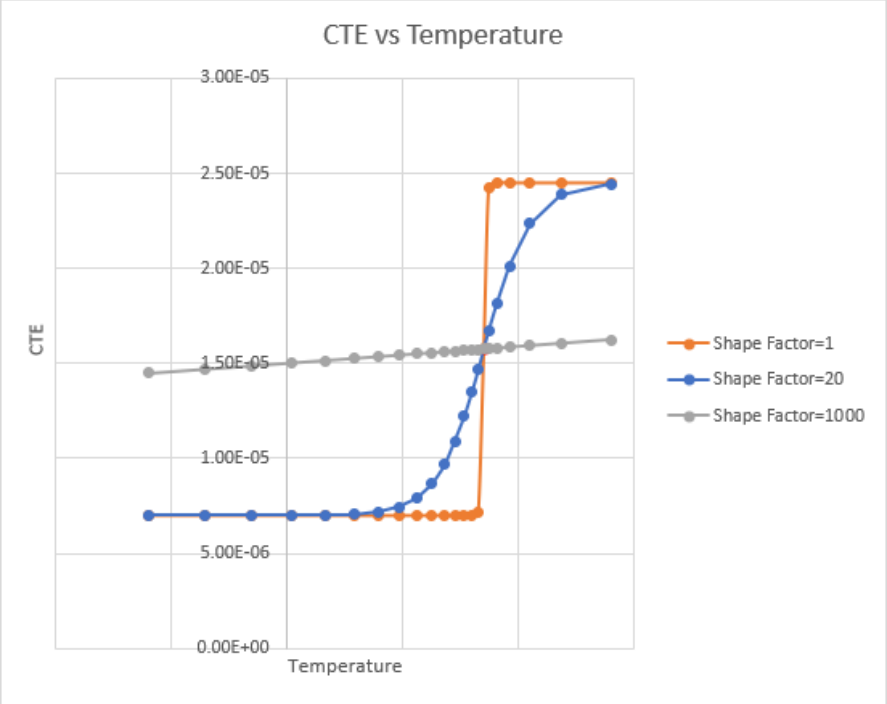
Modulus < Tg: Modulus used to calculate the temperature dependent material properties

Number of Points: The number of points within the temperature range that will be calculated. The recommended value is 20 points.

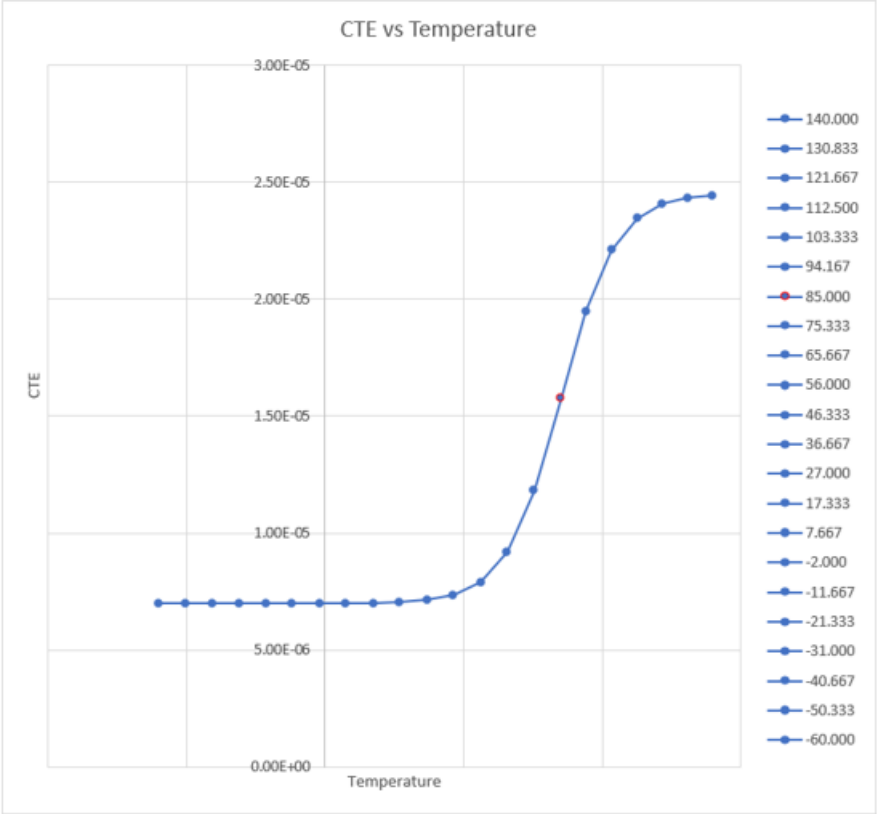
Shape Factor: Controls the transition rate/span of the transition region of the hyperbolic tan curve. The lower the shape factor, the quicker the change of the material property. The shape factor must be greater than 0. Realistically, a shape factor in the range of 5 – 30 is recommended. Sherlock uses a default value of 20.

Bias Ratio: Used to compute the number of points which will cluster around the Tg area by controlling the division or increment size. As you increase the **Bias Ratio**, the points around the Tg grow in density. Sherlock uses a default value of 10, which is recommended.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

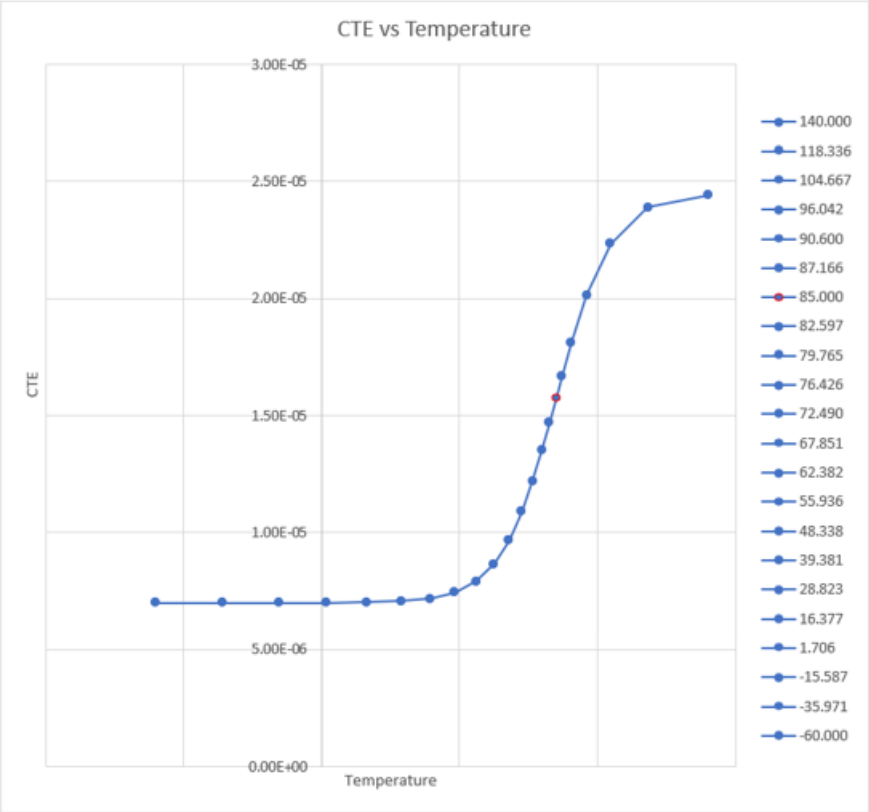


Bias Ratio= 1, Tg=85



Bias Ratio= 10, Tg=85

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.11.3.2. Adding and Copying Materials

To add a new material, select **Add Material** from the **Material** menu, or right-click in the material table listing and select **Add Material** from the menu. The **Material Editor** dialog will appear with all material properties empty, ready for a new material to be defined. When you are complete defining a material, click the **Save** button to add the new material.

To copy a material, right-click the material to be copied from the material table listing and select **Copy Material** from the menu. A copy of the material will be made. Provide a product name for the copied material and make any changes desired to the material properties, then click the **Save** button to add the copied material.

Tip:

Starting with Sherlock 22.2, you can go back and update the material **Name** for materials you have created.

3.11.3.3. Deleting Materials

Any user-defined material that is not a Sherlock system material may be deleted. Right-click the material to be delete from the material table listing and select **Delete Material** from the menu. If the Delete Material item is not available, this indicates that it is not a user-defined material.

If a system material has been modified, the **Delete Selected Materials** menu item will be available. However, it will not actually delete the material. Deleting a system material that has user-defined

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

properties will simply remove all the user-defined properties and allow the material to revert to being completely system defined.

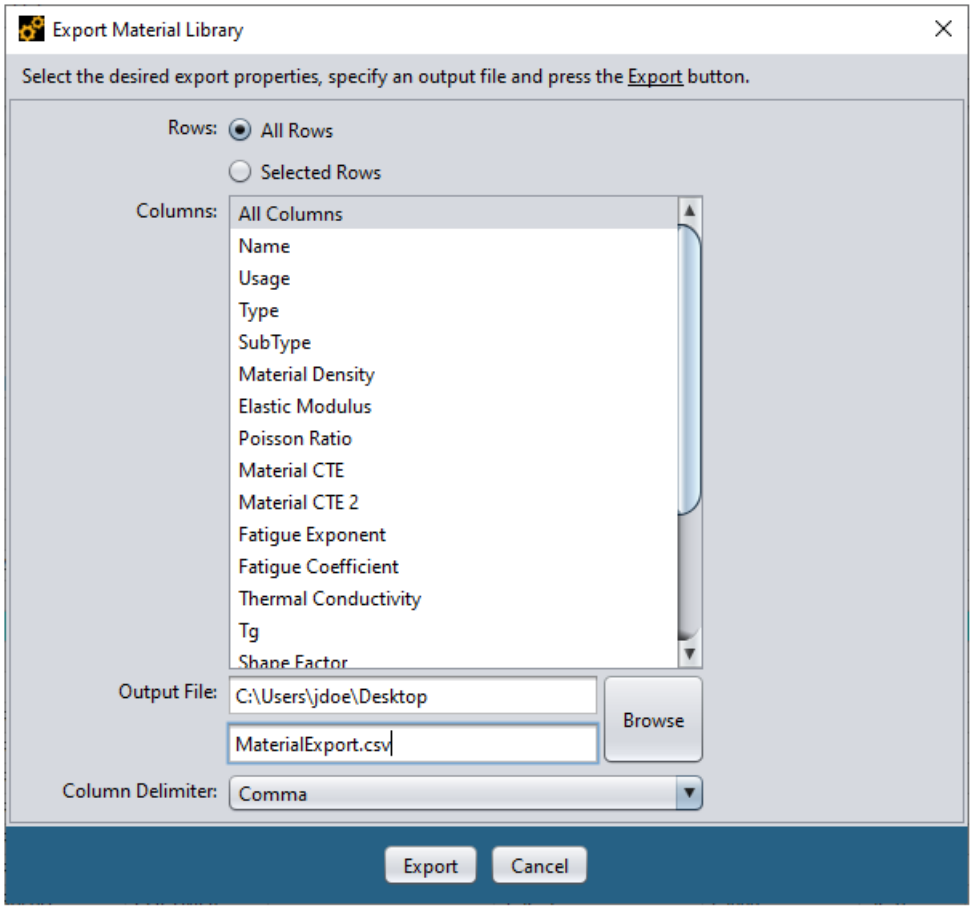
3.11.4. Export Materials

Material definitions may be exported to CSV or XLS spreadsheet files to allow external viewing or modifications. Exported files can then be used to import material definitions back into the material manager as described in the next section Import Materials (p. 165).

To export all material definitions, select from the **File** menu **Export Materials** or right-click in the material table listing and select **Export Material** from the context menu. If you wish to export only specific materials, select the materials you wish to export from the material table listing, then right-click the table and select Export Materials. In either case, the **Export Material Library** dialog, shown below, will be displayed.

The **Rows** and **Columns** selections allow you to customize which rows and/or columns should be exported. Select **All Rows** if all rows should be exported or select **Selected Rows** if only the rows selected in the material table listing should be exported. Similarly, select **All Columns** if all columns should be exported or select only specific columns to be exported.

After specifying the output file location, click the **Export** button. Supported file extensions are **.csv, .xls,** and **.xlsx.** If successful, a message will appear indicating the number of rows selected.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.11.5. Import Materials

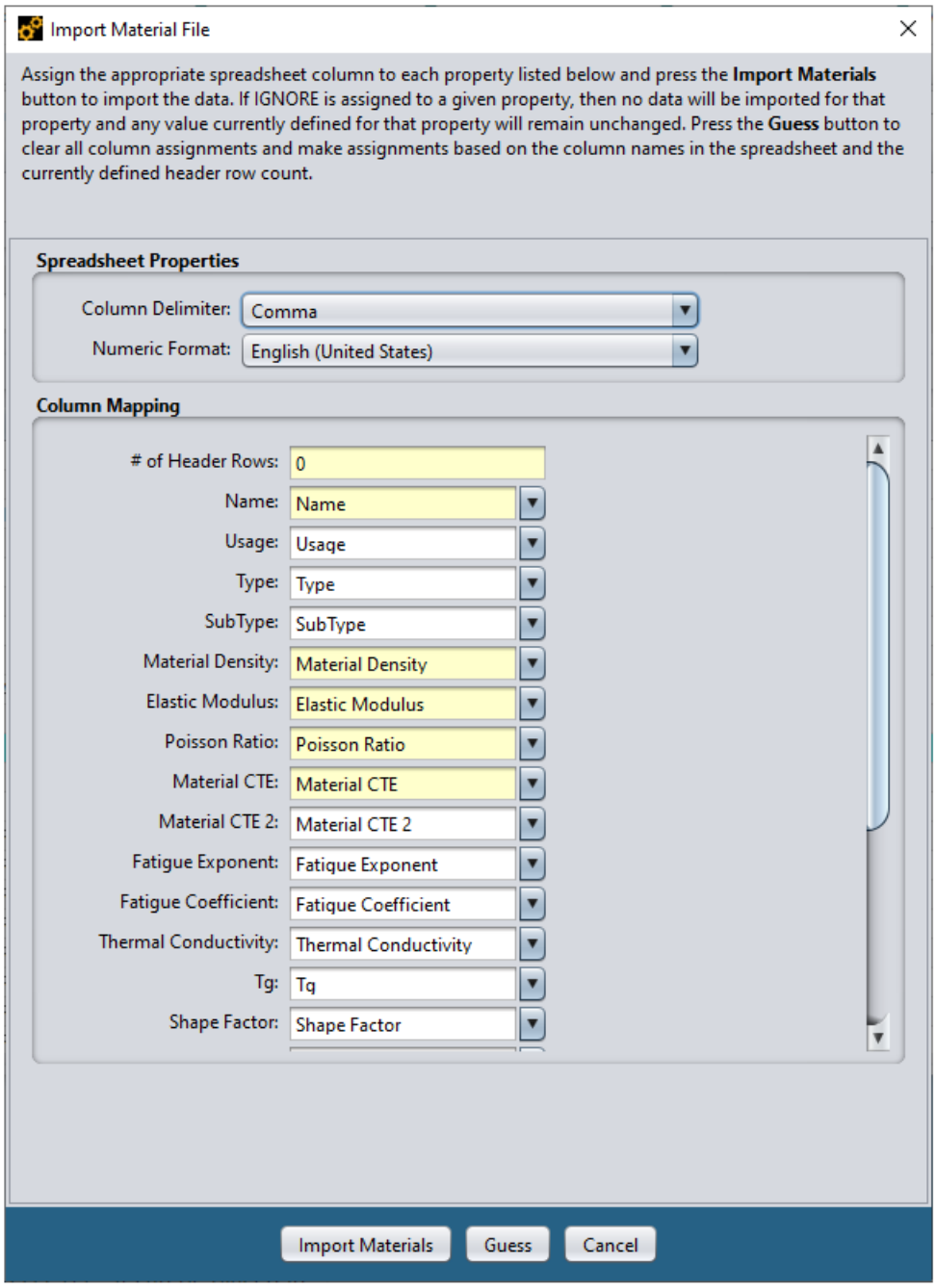
Material definitions may be imported into the **Material Manager** from a CSV, XLS or XLSX spreadsheet file by selecting the **File > Import Materials** option from the main menu. At that point, a file dialog will be displayed allowing you to designate the file to be imported.

After selecting the file, the **Import Material File** dialog (shown below) will appear, allowing you to verify the column mapping for each file/sheet that is imported. Sherlock scans each file to determine the number of header rows, the column names and which columns should be mapped to the various Sherlock material properties. Such guesses are usually right, but you should check them anyway by viewing the import file using an appropriate spreadsheet application.

When the import form is filled out properly, press the **Import Materials** button to import the data.

When the import process has completed, you will see a confirmation dialog. If any material definitions fail to import, then none of the rows in the file will be imported and an error dialog will be displayed with the information regarding why a material definition failed to be imported.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.12. User Data Files and Documents

Ansys Sherlock uses a collection of data files that define various properties used for data gathering and analysis. These data files are usually enough to handle most analysis tasks. Nonetheless, if users need to augment or replace specific properties, they can do so by creating their own CSV data files. Additionally common document and spreadsheet file types may also be included within a Sherlock project as a reference or additional documentation. This guide describes the required contents and location for each of the user-defined CSV data files that are supported.

In this section, the following topics are covered:

3.12.1. User Defined Data Files

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.12.2. User Document Files
3.12.3. User Properties File (user.properties)

3.12.1. User Defined Data Files

Users may create their own CSV data files for each of the following types of data:

Table 3.6: User Defined Data Files

Data Type	Default Filename / Folder
Material Properties	material.csv
Laminate Properties	laminate.csv
Laminate Glass Properties	laminateGlass.csv
Solder Properties	solder.csv
Package Properties	package.csv
Package Images	packageImages
Part Type Properties	partType.csv
PTH Quality Factors	pthQuality.csv
Acceleration Factors	accelFactors.csv
Manufacturer Names	manufacturer.csv

All data files must be stored using the filenames listed above in the Sherlock User Data Folder which is in the following location:

Windows (where USERID is the Windows user ID): `c:/Users/USERID/AppData/Roaming/Sherlock`

Linux (where $HOME is the user's home directory): `$HOME/.sherlock`

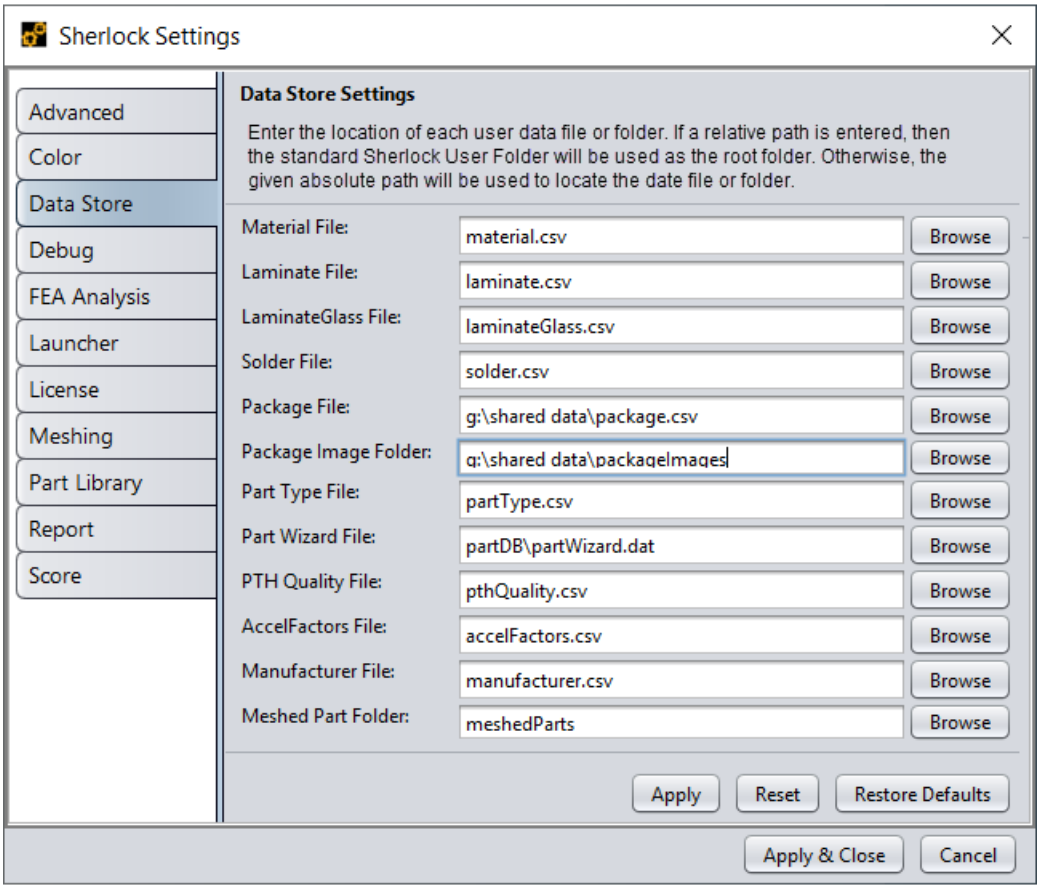
Links to subsections:

3.12.1.1. Shared Data Files
3.12.1.2. Material Properties
3.12.1.3. Laminate Properties
3.12.1.4. Laminate Glass Properties
3.12.1.5. Solder Properties
3.12.1.6. Package Properties
3.12.1.7. Package Images
3.12.1.8. Part Type Properties
3.12.1.9. PTH Quality Factors
3.12.1.10. Die Process Node Acceleration Factors
3.12.1.11. Manufacturer Names

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.12.1.1. Shared Data Files

Sherlock users may specify the exact name and folder location of each data file by using the **Settings > General Settings** option from the main menu, and then selecting **Data Store** from the side tab.



For example, in the settings shown here, the location of the **Material**, **Laminate**, and **Package** properties, as well as the Package Images folder, has been moved to the **g:\shared data** folder.

If that folder is implemented as a network shared folder, then Sherlock users across the network can share those data files. This not only reduces data maintenance tasks, but it also helps to ensure consistency between users in the same organization. For example, by sharing the Package Properties data file, as soon as one Sherlock user in the organization adds a new package, that package data can be used by all other Sherlock users.

For consistency, the default filename is still used in this example, but you are free to use whatever file name works best for your organization. For example, you may want to include a version number for each file to be able to better coordinate changes across all Sherlock users.

The **Data Store Settings** dialog allows you to enter either **relative** or **absolute** folder locations. **Relative** folder locations are always interpreted using the Sherlock **User Data Folder** (described above) as the base folder. For example, the **solder.csv** file specified in the example indicates that the file should be in the Sherlock **User Data Folder** itself.

Note, filenames may be specified in the **Data Store Settings** dialog even if the file doesn't currently exist. On startup, Sherlock checks for the existence of each file and loads them if found.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

We now describe the required and optional contents of each of the data files.

3.12.1.2. Material Properties

The **Material** properties file may be edited directly as noted here; however, it is recommended that any changes to the material properties file be made through the **Material Manager**. See the Material Management (p. 153) user guide section for more information. The **Material** properties file may define the following data columns. Unless noted otherwise each column is required:

Table 3.7: Material Properties File

Column Name	Type	Description	Example
Name	String	Material Name	COPPER
Type	Choice	ELASTIC	ELASTIC
SubType	Choice	ISO	ISO
Elastic Modulus	Numeric	Elastic Modulus (MPa)	113000
Poisson Ratio	Numeric	Poisson's Ratio	0.4
Expansion Type	Choice	NONE, ISO, ORTHO, ANISO	ISO
Expansion Zero	Numeric	Expansion Zero	0
CTE	Numeric	(1/C) Defines the CTE for all homogeneous material. If the material is a fiber, this property defines the axial CTE of the material.	1.8
CTE2	Numeric	CTE (1/C) Defines the CTE along the radial direction of the fiber. The material property is primarily used in resin property calculation. For all materials other than fiber, this value is ignored.	1.8
Density Properties	Numeric	Density (tonne/mm3)	1.70E-009
Usage	Choice	Slash delimited list of LEAD, PAD, RESIN, FIBER, HEATSINK, WIREBOND, POTTING (or empty)	LEAD / PAD
Fatigue Exponent	Numeric	Fatigue strength exponent	-8.6e-2
Fatigue Coefficient	Numeric	Fatigue strength coefficient	3.2e-03
Thermal Conductivity	Numeric	Thermal Conductivity (W/m-K)	400
Tg	Numeric	Glass transition temperature	150
Class (*optional*)	Choice	ALLOY, CERAMIC, POLYMER	ALLOY
Shape Factor	Numeric	Used as input to Material Temperature Values Calculator	20

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fracture Strength (*optional*)	Numeric	Fracture strength in units MPa for CERAMIC materials	345
Created By (*optional*)	String	Indicates who modified and when the material was created	Sherlock @ 2017-12-01
Modified By (*optional*)	String	Indicates by whom and when the material was modified.	Sherlock @2018-01-12
Notes (*optional*)	String	Free-form text of user-supplied notes.	

The **Name** column is used as the unique key to determine if a material should be added to the database or overwritten by the values in the user defined file. Rows that have a blank value for **Name** will be ignored.

If one or more values are defined in the **Usage** column then they must be separated by a slash (/). Materials that are designated as **LEAD** materials will appear in the choice list displayed for component leads. Materials that are designated as **PAD** materials will appear in the choice lists displayed for conductor layers and pads. Materials that are designated as **RESIN** materials will appear in the choice list displayed for resin layers. Materials that are designated as **FIBER** materials will appear in the laminate manager when a glass style is selected.

All other columns defined in the CSV file will be ignored by Sherlock when loading the material data.

3.12.1.2.1. Temperature-Dependent Material Properties

Temperature-dependent values can be specified for the following properties:

- Elastic Modulus (MPa)
- Poisson Ratio
- CTE (1/C): Defines the CTE for all homogeneous material. If the material is a fiber, this property defines the axial CTE of the material.
- CTE 2: (1/C) Defines the CTE along the radial direction of the fiber. The material property is primarily used in resin property calculation. For all materials other than fiber, this value is ignored.
- Density Properties (tonne/mm3)
- Thermal Conductivity (W/m-K)
- Fatigue Exponent and Fatigue Coefficient (used for shock and vibration fatigue cycles to failure calculations)

Temperature-dependent values are defined as a set of temperature-value pairs and are passed directly to the FEA engine for all materials used during a given analysis. FEA engines typically use the given data points to determine the property value for a specific temperature by using linear interpolation between the bounding data points. For all temperatures below the minimum

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

temperature specified, the value associated with the minimum temperature will be used. Similarly, for all temperatures above the maximum temperature specified.

Temperature-value pairs are specified using the following notation:

<Value> @ <Temp> <Units>

where <Value> is the property value, <Temp> is the temperature and <Units> are the temperature units. (NOTE: The value units must be those specified in the list above.) Two or more temperature-value pairs must be comma separated. For example, the following notation:

400@20C, 320.5@212F

specifies a value of 400 at 20 degrees centigrade and a value of 320.5 at 212 degrees Fahrenheit. Temperature units can be freely mixed in a given list of temperature-value pairs.

In situations where Sherlock needs to determine a temperature-dependent material property value when no explicit temperature is defined (such as when mixing material property values in a given PCB layer to determine the average layer property value), a room temperature value of 20C is used for interpolation purposes.

3.12.1.3. Laminate Properties

The Laminate properties file may be edited directly as noted here, however, it is recommended that any changes to the material properties file be made through the **Laminate Manager.** See the Laminate Management (p. 133) user guide section for more information. The **Laminate Properties** file may define the following data columns. Unless noted otherwise each column is required:

Table 3.8: Laminate Properties File

Column Name	Type	Description	Example
Company	String	Manufacturer's name	Ansys, Inc.
Material	String	Material name	FR-4
Product Name	String	Product Name	DfR FR-4 Super
CTExy	Numeric	CTE in XY plane (pre-Tg, max)	15
CTEz	Numeric	CTE in Z direction (pre-Tg)	70
Exy	Numeric	Tensile Modulus (MPa)	27000
Ez	Numeric	Out of Plane Modulus (MPa)	3450
Density	Numeric	Density (g/cc)	1.8
Conductivity	Numeric	Thermal Conductivity (W/m-K)	400
Created By (*optional*)	String	Indicates who modified and when the laminate was created	Sherlock @ 2017-12-01

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modified By (*optional*)	String	Indicates by whom and when the laminate was modified.	Sherlock @2018-01-12
Notes (*optional*)	String	Free-form text of user-supplied notes.	

The **Company**, **Material** and **Product Name** columns are used as the unique key to determine if a laminate should be added to the database or overwritten by the values in the user defined file. Rows that have a blank value for **Company** will be ignored.

Values must be specified in all columns listed above.

All other columns defined in the CSV file will be ignored by Sherlock when loading the laminate data.

3.12.1.3.1. Temperature-Dependent Laminate Properties

Temperature-dependent values can be specified for the following properties:

- CTExy, CTEz (MPa)
- Exy, Ez (ppm/C)
- Density Properties (g/cc)
- Thermal Conductivity (W/m-K)

See the Temperature-Dependent Material Properties (p. 170) section for more information.

3.12.1.4. Laminate Glass Properties

The **Laminate Glass Properties** file must define the following data columns:

Table 3.9: Laminate Glass Properties File

Column Name	Type	Description	Example
Construction	String	Glass style construction description	106
Resin By Weight (%)	Numeric	Percentage of resin by weight (0-1)	0.75
Thickness	Numeric	Thickness of glass style construction	0.002
Thickness Units	String	Units of measure for thickness	in
Resin By Volume (%)	Numeric	Percentage of resin by volume (0-100)	86

The **Construction** column lists the name of the glass style. The combination of **Construction**, **Resin By Weight (%)**, and **Thickness** determines the unique key for a given glass style. Rows that have missing values will be ignored.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The glass styles defined in the file will be displayed in the Stackup editor, allowing users to select the exact type of glass used by a given PCB layer. The resin by weight and thickness are used by Sherlock when determining the overall mechanical properties of each PCB layer.

Values must be specified in all columns listed above.

All other columns defined in the CSV file will be ignored by Sherlock when loading the laminate glass data.

Legacy Laminate Glass Properties

Some Sherlock beta version users may have previously used an older format for the Laminate Glass Style properties. This format is no longer supported, however Sherlock will convert any legacy file to the new format, however the thickness assigned to a given stackup layer may not be as expected.

The new file should have the format above. The following legacy properties should be converted as following:

- The **Glass Style** column should be renamed **Construction**. The best-practice for the format for the data of this column is **1x 106 0.002 in** where **1x** is the number of layers, **106** is the glass style, and **0.002 in** is the thickness of this given construction.
- The **Resin By Weight (%)** column was previously defined percentage values between **0** and **100**. The new format should be between **0** and **1**.
- The **Resign By Volume (%)** column is no longer read by Sherlock and is computed automatically. This column should be removed.

3.12.1.5. Solder Properties

Unlike the other data files, the solder data file is defined as a VERTICAL CSV file, in which each row of the file represents a specific property and each column represents a specific solder type. The first column denotes the property name. For example, the following data file would be acceptable:

Table 3.10: Solder Data File, Example

Solder Name	63Sn37Pb	SAC305
Solder Type	TIN LEAD	LEAD FREE
Elastic Modulus	18996.1@150C	31322.9@150C

The **User Defined Solder Properties** file MUST define the following data rows:

Table 3.11: Solder Data File, Required Data

Row Name	Type	Description	Example
Solder Name	String	Unique solder name	63Sn37Pb
Solder Type	Choice	Tin Lead, Lead Free	Tin Lead

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Solder** file MAY define the following data rows. If any row is not defined, the value listed as the example may be used depending on the property:

Table 3.12: Solder Data File, Optional Data

Row Name	Type	Description	Example
Elastic Modulus	Numeric	Elastic Modulus (MPa)	52755
CTE	Numeric	CTE (1/C)	2.59E-005
PoissonRatio	Numeric	Used as the solder material Poisson ratio	0.36
CTEAtZero	Numeric	Combined with CTESlope and temperature for solder CTE	2.22E-05
CTESlope	Numeric	Combined with CTEAtZero and temperature for solder CTE	8.90E-09
ThermalConductivity	Numeric	Thermal Conductivity (W/m-K)	50.6
Density Properties	Numeric	Density (tonne/mm3)	8.49e-9
Fatigue Exponent	Numeric	Fatigue strength exponent	-8.6e-2
Fatigue Coefficient	Numeric	Fatigue strength coefficient	3.2e-03
CriticalDwellTime	Numeric	Used to determine strain energy	20C
Critical Strain	Numeric	Used when computing cycles to failure during vibration fatigue analysis	3.75E-004
ActivationEnergy	Numeric	Used to determine strain energy	1414
CriticalMaxTemp	Numeric	Used to determine strain energy	368
StrainEnergyCoefficient	Numeric	Used to determine Cycles to Fail	0.0006061
LeadedCalibrationFactor	Numeric	Used by Leaded model to compute Strain Energy	2
WeibullBetaBGA	Numeric	BGA model Solder beta	3
WeibullBetaCC	Numeric	CC model Solder beta	3
WeibullBetaLCCC	Numeric	LCCC model Solder beta	3
WeibullBetaLeaded	Numeric	Leaded model Solder beta	3
WeibullBetaThruHole	Numeric	ThruHole model Solder beta	3
WeibullBetaQFN	Numeric	QFN model Solder beta	3
FEAPower	Numeric	Exponent used when computing cycles to failure for shock and vibration analysis	6.4
Correction Factor	Numeric	List of correction factors for each package used during harmonic and shock analysis. Each tuple is separated by a vertical bar. Within each tuple is the package name with the correction factor separated by a comma.	BGA,2.0|CBEND,1.26|TSOP,1.26

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Random Factor	String	List of correction factors for each package used during random vibration analysis. Each tuple is separated by a vertical bar. Within each tuple is the package name with the random correction factor separated by a comma.	BGA,2.0| CBEND,2.0| TSOP,2.1
ThermalShockFactorBGA	Numeric	List of thermal shock factors for BGA model. Each tuple is separated by a vertical bar. Each tuple is the hurdle, comma-separated with the factor.	15,1.0|30,1.25| 60,1.5
ThermalShockFactorCC	Numeric	List of thermal shock factors for CC model.	15,1.0|30,1.25| 60,1.5
ThermalShockFactorLCCCC	Numeric	List of thermal shock factors for LCCC model.	15,1.0|30,1.25| 60,1.5
ThermalShockFactorLeaded	Numeric	List of thermal shock factors for Leaded model.	15,1.0|30,1.25| 60,1.5
ThermalShockFactorThruHole	Numeric	List of thermal shock factors for ThruHole model.	15,1.0|30,1.25| 60,1.5
ThermalShockFactorQFN	Numeric	List of thermal shock factors for QFN model.	15,1.0|30,1.25| 60,1.5
ThermalShockFactorDefault	Numeric	List of default thermal shock factors.	15,1.0|30,1.25| 60,1.5
ShockStrain	Numeric	Maximum shock strain	7.59E-094
CCCalibration	String	CC solder model calibration factors. Each tuple is separated by a semi-colon (;). Within each tuple is the calibration factor number, the minimum package size to which the factor applies, and the calibration factor value. Users can also specify a starting calibration, an end calibration, and a package size increment for Sherlock to automatically calculate the individual factors between the start and end. Example: **`interpolate(3:3.2:1.0, 3:5.0:1.5, 0.1)`** As shown in the example above, for any package size in between 3.2 mm and 5.0 mm, Sherlock linearly interpolates	1:0.0:0.5; 1:3.2:0.5; 2:0.0:1.0; 2:3.2:1.0; 3:0.0:1.0; 3:5.0:1.5; interpolate(3:3.2:1.0, 3:5.0:1.5, 0.1) **Note:** Starting with Sherlock 2023 R2, the recommended CC Calibration Factors

		the calibration factor 3 between 1.0 and 1.5. **Note:** Prior to 2023 R2, Sherlock made the interpolation at set increments. The 0.1 in the example above reflects this increment. The logic was changed in 2023 R2 so that Sherlock now uses linear interpolation rather than increments. For the purpose of compatibility between versions, you will still see 0.1 in the UI.	have changed. For more information, see Leadless Ceramic Chip Carrier (LCCC) in the *Sherlock Theory Reference*.
Created By	String	Indicates who modified and when the solder was created	Sherlock @ 2017-12-01
Modified By	String	Indicates by whom and when the solder was modified.	Sherlock @2018-01-12
Notes	String	Free-form text of user-supplied notes.	

Some properties may allow temperature-dependent material values. See the Temperature-Dependent Material Properties (p. 170) section for more information.

3.12.1.6. Package Properties

The **Package Properties** file must define the following data columns:

Table 3.13: Package Properties File

Column Name	Type	Description	Example
Package Name	String	Common Package Name	LQFP-64
JEP95	String	JEDEC JEP95 designator	MS-026
Package Units	Choice	in, mm, mil, micron	mm
Package Length	Numeric	Package length (long dimension)	10
Package Width	Numeric	Package width (short dimension)	10
Package Thickness	Numeric	Package thickness	1.6

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lead Count	Integer	Number of pins/balls	64
Lead Geometry	Choice	C_LEAD, GULLWING, J_LEAD, L_LEAD, LEADLESS, STUB, THRUHOLE	GULLWING
packageType	String	OTHER, DIP, PDIP, CDIP, SOIC, PDSO, LSOP, SSOP, TSOP, TSSOP, SOT, SOJ, CBEND, PQFP, CQFP, QFP, QFJ, PLCC, HSOP, MSOP, DIE, LCCC, QFN, SON, USON, CBGA, BGA, MELF, CC, SOD, AVX, DPAK, KEMET, TO, SM, SPAK, DFN, VCHIP, RADIAL, DISC, BOX, AXIAL, CGA, SIP, BTC	CC

The **Package Name** column is used as the unique key to determine if a part type should be added to the database or overwritten by the values in the user defined file. Rows that have a blank value for **Package Name** will be ignored.

If a value is specified in the **JEP95** column then it will be automatically appended to the **Package Name** value to create a unique package identifier. Blank values may be specified in the **JEP95** column if no JEDEC designator is known for a given package.

The following optional column may be defined in the Package properties file:

Table 3.14: Package Properties File, Optional Data

Column Name	Type	Description	Example
Aliases	String	Comma separated list of prefixes	LQFP, TQFP

If an **Aliases** value is defined, then each of the prefixes defined in that list (along with the prefix defined in the **Name** column) will be incorporated into a wild-card pattern used to match package names when importing data. The pattern will automatically include any suffix defined in the **Name** column, so only prefix values should be specified in the Aliases column.

The **Package Properties** file may also define any of the other available package properties. These properties are listed in the **Help > Part Properties** document. Any of the properties listed that is tagged with **[PKG]** are valid package properties that may be defined. Any other properties defined will be ignored by Sherlock when loading the package data.

All of the numeric measurements should be specified using the units indicated by the **Units** column value.

3.12.1.7. Package Images

The **Package Chooser** attempts to display an image for each package name or package type selected. Default images are in the Sherlock Installation directory and are automatically updated whenever Sherlock is upgraded.

Users can override or augment the default images by placing one or more image files in the *packageImages* sub-directory located in the Sherlock **User Data Folder**, by default. Image files stored in that directory must satisfy the following conditions:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Image files must be in JPEG format, with a suffix of **.jpg**. For best viewing, the images should be 160x160 pixels, with a white background if possible. Sherlock will automatically re-size images larger than that size.

2. Image files must be placed in the sub-folder corresponding to the **Package Type** of the package being represented. For example, all images to be displayed for CDIP packages should be in the **CDIP** sub-folder.

3. Image files must be named using either the standard Sherlock **Package Name** or the JEP95 identifier. For example, the image file for the CDIP-08 (MO-036AA) package must use either the **CDIP-08.jpg** or the **MO-036AA**.jpg file name.

A default image file can be defined for each **Package Type** by using the **Package Type** as the file name. For example, the default CDIP image would be stored in the **CDIP** sub-folder using the **CDIP.jpg** file name. Default images are displayed if no package-specific image file is found in either the **Sherlock Installation Directory** or the user defined **packageImages** directory.

3.12.1.8. Part Type Properties

The **Part Type Properties** file must define the following data columns:

Table 3.15: Part Type Properties File

Column Name	Type	Description	Example
designator	String	Part Designator	C
partType	String	Part Type	Capacitor
partMaterial	Choice	Must be a valid Material Name	BARIUMTITANATE

The **designator** column is used as the unique key to determine if a part type should be added to the database or overwritten by the values in the user defined file. Rows that have a blank value for **designator** will be ignored.

The following optional columns may be defined in the **partType.csv** file:

Table 3.16: Part Type Properties File, Optional Data

Column Name	Type	Description	Example
dieEnabled	String	Indicates if DIE tab should be enabled for a given part type in the part editor. Values: on, off	on
frType	String	Indicates the default failure rate type for the given part type. The value must be a valid failure rate type.	CAP, FIXED, ALUMINUM, DRY

All other columns defined in the CSV file will be ignored by Sherlock when loading the part type data.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.12.1.9. PTH Quality Factors

The **PTH Quality File** must define the following data columns:

Table 3.17: PTH Quality File

Column Name	Type	Description	Example
QualityFactor	String	Unique quality factor name	Good
QualityValue	Numeric	Value associated with quality factor between 0.0 and 10.0	6.75
StrainDistFactor	Numeric	Strain distribution factor	1.75
WeibullBeta	Numeric	Weibull beta value used when computing life prediction	4
PredictionProb	Numeric	Prediction probability	0.01

The **PTH Quality File** may optionally define the following additional data columns for each quality factor:

Table 3.18: PTH Quality File, Optional Data

Column Name	Type	Description	Default
CopperYieldStrength	String	Copper yield strength in PSI	25000
CopperCTE	Numeric	CTE of Copper	0.0000185
CopperElasticModulus	Numeric	Elastic modulus of Copper in PSI	12000000
CopperPlasticModulus	Numeric	Plastic modulus of Copper	100000
CopperDuctility	Numeric	Copper ductility	0.3
CopperTensileStrength	Numeric	Tensile strength in PSI	40000

All other columns defined in the CSV file will be ignored by Sherlock when loading the data.

The following are the default **PTH Quality Factors** as defined by Sherlock.

Table 3.19: Default PTH Quality Factors

QualityFactor	QualityValue	StrainDistFactor	WeibullBeta	PredictionProb
Extraordinary	10.0	1.0	6	0.01
Superior	8.75	1.25	5	0.01
Good	6.75	1.75	4	0.01
Marginal	4.75	2.0	3	0.01
Poor	3.5	2.5	2	0.01

3.12.1.10. Die Process Node Acceleration Factors

The acceleration factors data file is defined as a vertical CSV file, in which each row of the file represents a specific property and each column represents a specific process node type. The first column denotes the property name and the remaining columns denote the values for each of the die process nodes indicated by that column. Most of the data in this file is used for computing

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

semiconductor wearout acceleration factors. See the Semiconductor Wearout Analysis (p. 408) user guide section for more information.

The user defined acceleration factors file MUST define the following data rows:

Table 3.20: Acceleration Factors Data File

Row Name	Type	Description	Example
dieProcessSize	Numeric	Die feature size	35
dieProcessType	Choice	PLANAR, FINFET	PLANAR
currentDensityExponentVM	Numeric	EM acceleration factor (n)	2
frequencyExponentEM	Numeric	EM acceleration factor (m)	2
voltageExponentEM	Numeric	EM acceleration factor (γ)	2
activationEnergyEM	Numeric	EM acceleration factor (Ea)	0.6
voltageExponentTDDB	Numeric	TDDB acceleration factor (γ)	0.5
activationEnergyTDDB	Numeric	TDDB acceleration factor (Ea)	0.7
frequencyExponentHCI	Numeric	HCI acceleration factor (n)	1
voltageExponentHCI	Numeric	HCI acceleration factor(γ)	45
activationEnergyHCI	Numeric	HCI acceleration factor (Ea)	-0.15
voltageExponentNBTI	Numeric	NBTI acceleration factor (γ)	6
activationEnergyNBTI	Numeric	NBTI acceleration factor (Ea)	1

3.12.1.11. Manufacturer Names

The user-defined **Manufacturer** file can be used to define one or more manufacturer names and numeric identifiers that are not already recognized by Sherlock. The manufacturer names can then be searched and selected using the **Manufacturer Chooser** dialog when adding or editing a part to ensure that only standard manufacturer names are stored in the **Parts List** and/or **Part Library**.

The **Manufacturer** file must define the following data columns:

Table 3.21: Manufacturer File

Column Name	Type	Description	Example
manufacturerName	String	Unique manufacturer name (case insensitive)	AVX / KYOCERA

If the **manufacturerName** value is not specified in a given CSV row, then the entire row will be ignored when importing data from the **Manufacturer** file.

All manufacturer names are converted to upper case when stored or displayed in Sherlock. The numeric manufacturerID is used when storing parts in large part libraries to reduce space requirements and to speed up queries. As such, once a numeric value is assigned it should never be deleted. Otherwise, Sherlock will not be able to convert the numeric value back to a manufacturer name when retrieving from a Part Library and will not be able to query the **Part Library** using that manufacturer ID.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All other columns defined in the CSV file will be ignored by Sherlock when loading the data.

3.12.2. User Document Files

Along with project specific input files used by Sherlock to model circuit cards, users may wish to also add other files such as PDF's, Word, and Excel to be use for reference or additional documentation. These files can be opened directly from within Sherlock. Supported file types include .pdf, .doc, .docx, .odt, .csv, .ods, .xls, and .xlsx.

To include additional documentation files in Sherlock, select **Add File(s)** from CCA menu item or the context menu from the **Files** folder for the desired project circuit card. The files will be added to the project and listed in the **Files** section for the circuit card. The file should have an icon associated with it that shows it associated with the default program the operating system will use to open the file. Double-clicking any file will open the selected file in that program.

3.12.3. User Properties File (user.properties)

Warning:

- Editing the `user.properties` file is for advanced users only and should be exercised with caution.
- Make sure all instances of the Sherlock application are closed before editing the `user.properties` file. Otherwise, the Sherlock software will overwrite your changes when the aplication closes.

The Sherlock application's `user.properties` file contains variables you can edit with a standard text editor. In Windows, assuming you installed the Sherlock application on the `C:` drive, the location of the file location is shown below. `AppData` might be a hidden file in your Windows system. If you don't see it, turn on **Show hidden and system files** in Windows Settings.

```
C:\Users\<username>\AppData\Roaming\Sherlock\user.properties
```

For Linux users, the file path is listed below. Also, see Standard Folder Locations (p. 2).

```
$HOME/.sherlock/user.properties
```

Variables in the user.properties file:

ceintMovTol

When using Ansys Meshing (p. 85), the Sherlock application uses **CEINTF** for establishing contacts between two surfaces with a default of 0.02 for `MoveTol` which is within the 0.05 range recommended by Ansys. The default value in Ansys Mechanical APDL is 0 which prevents the node from moving when contacts are created. To change the 0.02 default, edit `ceintfMoveTol` in the `user.properties` file. The variable is not listed in `user.properties` by default. You must enter it manually and specify the desired value.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Managing Parts in Sherlock

Note:

Starting with Sherlock version 24 R1, several changes have been made to the package database. See Package Management (p. 315) for details.

Topics Covered in this Chapter

4.1. Parts List Overview
4.2. Parts List Management Tools
4.3. Building Custom Parts
4.4. Part Library
4.5. Part Validation
4.6. Approved Vendor List (AVL)
4.7. Part Wizard
4.8. Sherlock Part Properties
4.9. Package Management
4.10. Tables: Required Part Properties per Analysis Type

4.1. Parts List Overview

This section provides an overview of Sherlock's **Parts List**- it's interface and basic functions. More advanced operations are covered in the subsequent section Parts List Management Tools (p. 197).

Topics Covered in this Section

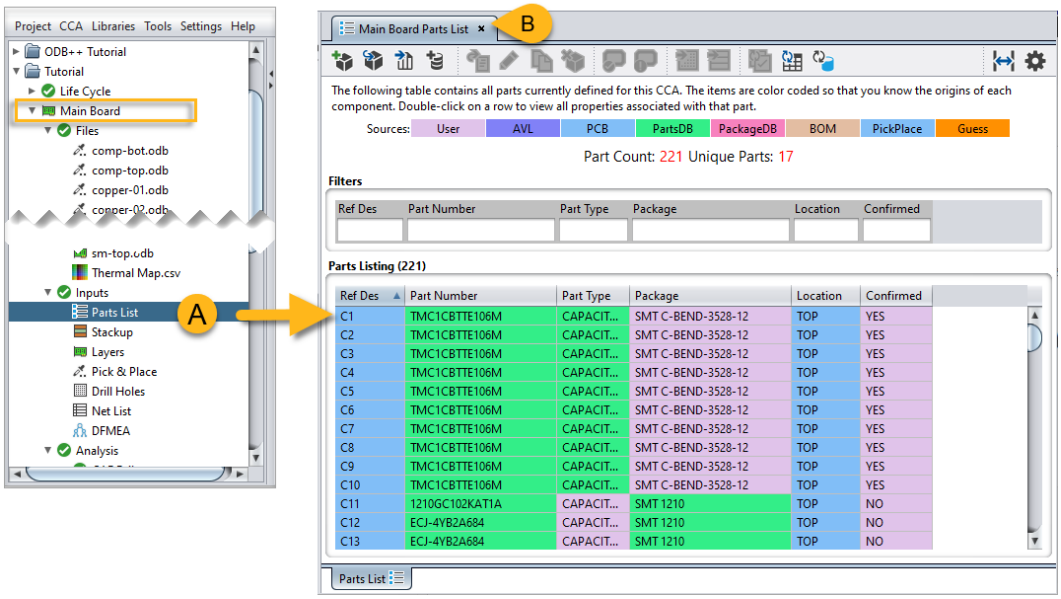
4.1.1. How Sherlock Generates the Parts List
4.1.2. The Importance of Reviewing Part Properties
4.1.3. The Parts List Interface
4.1.4. Customizing the Parts List
4.1.5. Filtering and Selecting Parts
4.1.6. Updating Component Layers

Introduction to the Parts List

In Sherlock, each CCA (circuit card assembly) has its own **Parts List** which is displayed in the **Project Tree** under **Inputs** (A, below). Double-click the **Parts List** icon to open it. In this below, the list is identified as the **Main Board Parts List** (B), named after its parent circuit card assembly, as seen in the **Project Tree**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Regardless of how the data was initially obtained, all the part information for your CCA is contained in the **Parts List**. Sherlock uses the part properties stored in the **Parts List** as inputs to most analysis tasks and for generating the graphical displays associated with the circuit card.



Part properties span a wide range of characteristics, including size, location, orientation, identification, electrical qualities, and more. You can find a complete list of all part properties in the section below, Sherlock Part Properties (p. 294), including property definitions and illustrations. The list is also available within Sherlock in PDF format: From the **Main Menu**, click **Help > Part Properties**.

Note:

In this section, the columns shown in the **Parts List** images might be different than the columns you currently see in your Sherlock application. This is normal. You can add and delete columns to suit your needs. Click the ⚙ icon in the upper right of the table to customize what columns you want Sherlock to display. Refer to Customizing the Parts List (p. 191) for additional information.

4.1.1. How Sherlock Generates the Parts List

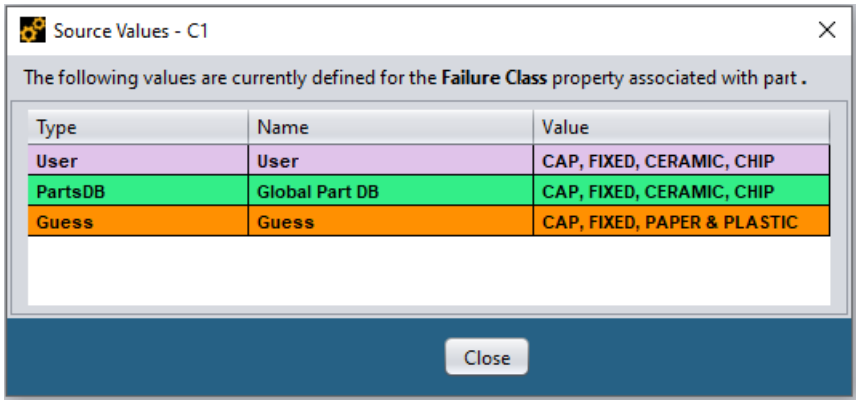
Sherlock tracks well over a hundred properties for each part. Fortunately, when you begin your project, Sherlock streamlines the process of importing all those properties by automatically extracting part information from multiple sources- from your CAD files, parts lists, BOMs, and so on.

Sherlock has built-in heuristics designed to extract as much information as possible from the data sources you provide. By analyzing several key properties of each part (such as package name and package description), Sherlock is able to make educated guesses when determining what values to assign the various part properties.

To ensure consistency and accuracy, Sherlock references a collection of internal databases (as explained in Part Library (p. 230)). This includes Sherlock's nearly exhaustive Global Part Library as well as custom libraries you can create.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the life of your project, Sherlock tracks all the data sources used to define each part property. At any time, you can view the data sources for a part property, select source priorities, and manually modify data as needed. As an example, notice the **Source Values** window below which is accessible from the **Parts List**. For capacitor C1, Sherlock tracks three different data sources for the **Failure Class** property: **User** (user input), **Global Part DB** (Sherlock's global database), and **Guess** (Sherlock's educated guess). Sherlock gives priority to the source listed first (**User**, in this case), although you can change the priority at any time. This feature is further explained in Part Source Priority (p. 207).



4.1.2. The Importance of Reviewing Part Properties

> **Warning:**
>
> Sherlock's import features are powerful, time-saving tools, but most data sources you import into Sherlock will have incorrect or incomplete information. Sherlock will do its best with the data you provide, but it remains your responsibility to review the part properties and correct them as needed to ensure accurate analysis results.
>
> Also, be sure to read the section below, Parts Requiring Special Consideration (p. 186).

Source files are rarely perfect. An engineer might have entered incorrect information in the original CAD file, for example. And if certain properties are not used by a designer or manufacturer, chances are, those properties were not maintained properly. For the above reasons, it is important to review all the components in the **Parts List** to ensure the information is accurate. While you can run analysis tasks in Sherlock at any time, with or without correct part properties, the accuracy of the results depends on the accuracy of the properties themselves.

There is no harm in doing a quick analysis immediately after importing a CAD file, but be aware the results might not be as accurate as possible. Consider working in a cyclic fashion: Review properties, run an analysis, update properties, run another analysis, and repeat the cycle. As you refine the part properties, the analysis results will become more accurate.

> **Tip:**
>
> Sherlock doesn't require an extensive data input phase before starting analysis. Some analysis tasks may not run without key inputs, but often Sherlock will make educated assumptions and select appropriate defaults so that you can get started quickly. As you refine the inputs, the analysis will be more accurate.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.2.1. Parts Requiring Special Consideration

When reviewing parts properties, some parts require special consideration.

- **Through Hole Parts**: Through-hole parts require special scrutiny in the **Parts List** because of the lack of standard package designations, their different shapes, and the different ways that CAD tools designate their location. Whenever you create a Sherlock project, you should always examine each of the through-hole parts.
- **BGA Components**: The following applies to ODB++, Eagle, and ODB XML imports. When importing a BGA package, Sherlock calculates the package dimensions from the ball pattern rather than using the dimensions specified for the package.

4.1.3. The Parts List Interface

Topics Covered:

4.1.3.1. Main Features of the Parts List
4.1.3.2. How to Select Parts
4.1.3.3. The Icon Ribbon
4.1.3.4. Data Source Color Codes

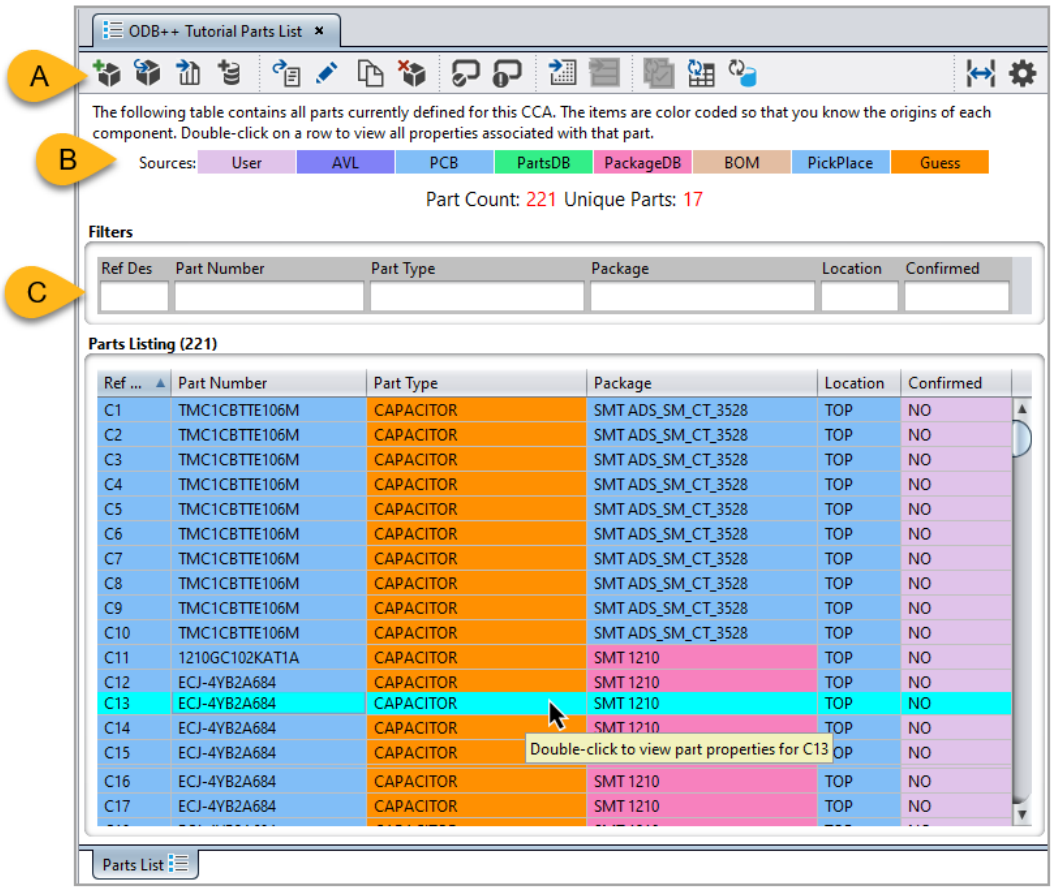
4.1.3.1. Main Features of the Parts List

Each row in the **Parts Table** represents a single part, with several of its properties displayed in the columns. You can adjust the width of the columns by clicking and dragging the column borders. You can also choose the number of columns and which properties they display as explained below in Customizing the Parts List (p. 191).

Some of the basic features of the **Parts List** are shown here:

- The Icon Ribbon (p. 187) (A, in the image below)
- Data Source color codes (p. 190) (B)
- Parts List Filter (p. 196) (C)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Topics in this section:

4.1.3.2. How to Select Parts

Using the **Parts List** features often requires you to select a part or several parts from the list. There are a few ways to make your selection:

- Left-click any row in the **Parts List** to select a single part.
- Left-click any row and drag the mouse up or down to select a range of parts.
- SHIFT + Left-click to select all rows between the previously selected row and the current row.
- CTRL + left-click to toggle row selection
- Double-click any row to view all properties associated with the part

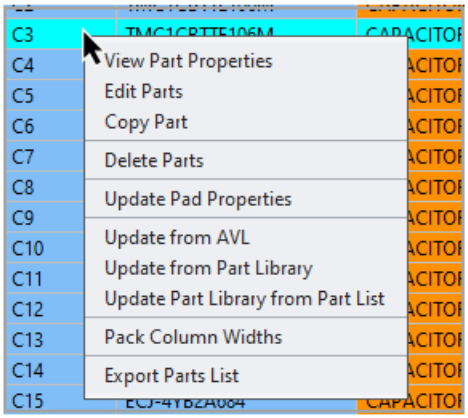
4.1.3.3. The Icon Ribbon

The icons at the top of the **Parts List** allow you to perform common part-management functions with a simple button-click. Icons are context sensitive and become active only when they are useable. For example, most icons are grayed out when no part is selected.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Another way to perform these same operations is to right-click a selected part or parts in the **Parts List**. This opens the context menu shown below.



Parts List Icons:

Below is a brief explanation of each icon in the **Parts List**. Follow the links provided for more detailed information on each feature.



A. **Add a New Part to the Parts List**: Launches the **Add Part** form. The interface is nearly identical to the **Part Editor** (p. 200), except all property fields are blank.

B. **Import Parts to the Parts List**: Launches the **Import Parts List** dialog. Use this feature to add parts which were previously exported from other **Parts Lists** using Sherlock's **Export Selected Parts** feature (item C below). See Importing the Parts List (p. 224).

C. **Export Selected Parts from the Parts List**: Opens the **Export Parts List** dialog where you can export all parts on the list or only the parts you have selected. Once Sherlock has created the exported parts file, you can import the parts into another **Parts List** within Sherlock. See Exporting the Parts List (p. 223).

D. **Add Part from the Part Library**: Opens the **Search Part Library** dialog where you can search for and add parts from any of your part libraries, including Sherlock's Global Parts Library. See Part Library Management (p. 233).



E. **View the Selected Part Properties**: Opens the **Part Properties Viewer** for the selected part(s). See The Part Properties Viewer (p. 197).

F. **Edit the Selected Parts**: Opens the **Part Editor** for the selected part(s). See The Part Editor (p. 200).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

G. **Copy the Selected Part**: Duplicates the selected part and adds it to the **Parts List**. Sherlock automatically a) saves the new part to the list, b) displays it in the **Part Viewer**, and c) assigns the next available reference designator (**Ref Des**) to the new part.

H. **Delete the Selected Parts**: Deletes the select part(s) from the **Parts List**.



I. **Confirm All Part Properties for the Selected Part(s)**: See Confirming Parts (p. 210).

J. **Un-Confirm All Part Properties for the Selected Part(s)**: See Confirming Parts (p. 210).

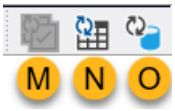


K. **Update Pad Properties for the Selected Parts**: See Updating Pad Properties (p. 423).

L. **Update the Circuit Card Assembly (CCA) Layers**: Updates the CCA to reflect changes you made to the **Parts List**. If there are no layers to be updated, the icon will be inactive. See Updating Component Layers (p. 196).

Tip:

Using the **Update the CCA Layers** icon is an important part of a typical workflow in Sherlock. If you modify a part in the **Parts List**, you must click the **Update the CCA Layers** icon for those changes to be reflected in the component layers and your analysis results.



M. **Update the Parts from the Approved Vendor List**: Refer to the section Approved Vendor List (AVL) (p. 268).

N. **Update the Parts List from the Part Library**: For an explanation of this specific operation, see the section Update Parts List from Part Library (p. 259). For an overview of Sherlock's Part Library feature, see Part Library (p. 230).

O. **Update the Part Library from the Parts List**: For an explanation of this specific operation, see the section Update Part Library from Parts List (p. 257). For an overview of Sherlock's Part Library feature, see Part Library (p. 230).



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P. **Automatically Resize all Column Widths**. Resizes the column widths to conform to the current table size.

Q. **Customize the Parts List View**: Launches the **Customize Parts Listing** window where you can add or remove columns and select what part property each column will display. See Customizing the Parts List (p. 191).

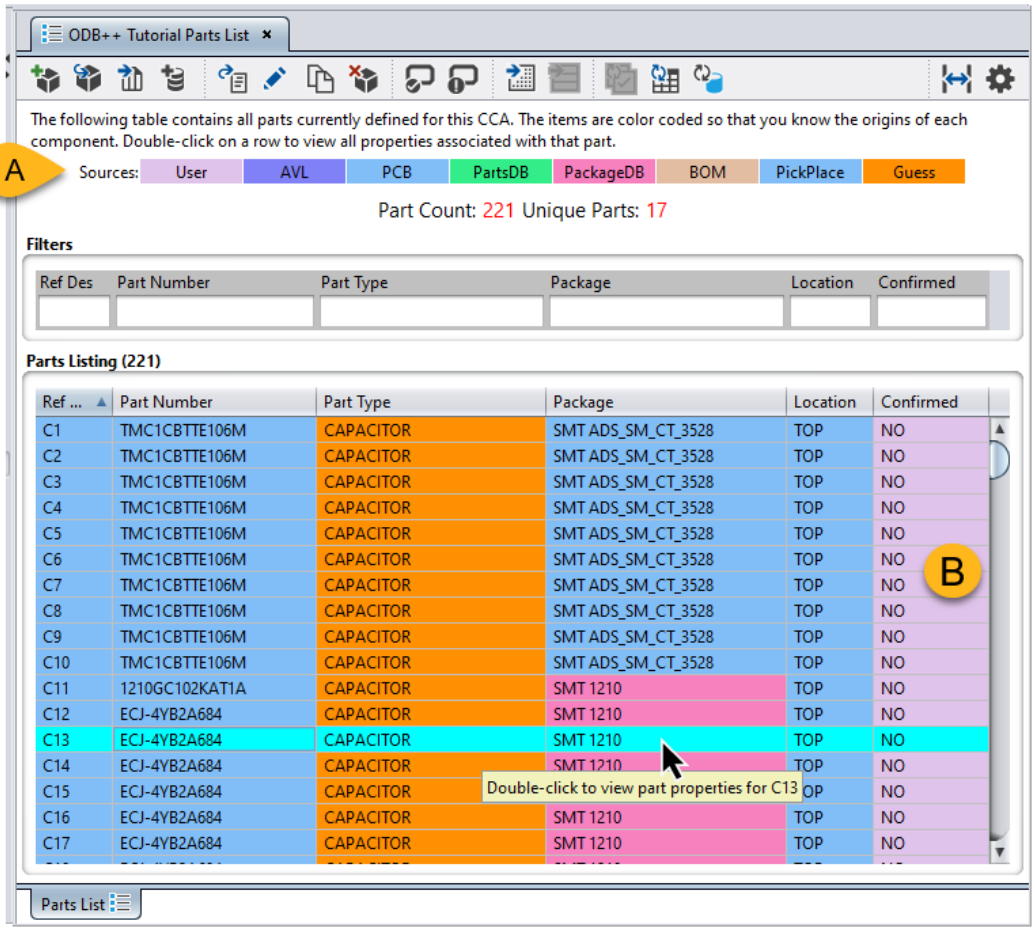
4.1.3.4. Data Source Color Codes

Each high-level part property is color-coded to indicate where the data was obtained (note the **Sources** legend, A below). For example, in the **Parts List** shown below, the **Ref Des** (reference designator), **Part Number**, and **Location** information for each of the capacitors listed were obtained from the **Pick & Place** data extracted from an ODB++ archive, as denoted by the blue cell color. For some parts, the **Package** information was obtained from the Sherlock's Package Database, as denoted by the magenta color. **Part Type** information was guessed by Sherlock as indicated by the orange color. (See the section on Part Source Priority (p. 207).)

At times, the **Location** and **Package** column cells will be color-coded yellow. This indicates the **Location** value and the **Package** value for the part come from two different sources. If **Location** and **Package** come from the *same* source, the cell background displays one of the standard colors shown in the legend (A, below). **Location** is based on the part property X and Y coordinates along with the board-side data. **Package** is based on the package width and length properties.

In this example, none of the parts have been **Confirmed** (B). This means Sherlock can run an analysis, but the analysis will not be considered complete until the properties for all the part have been confirmed. See Confirming Parts (p. 210).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



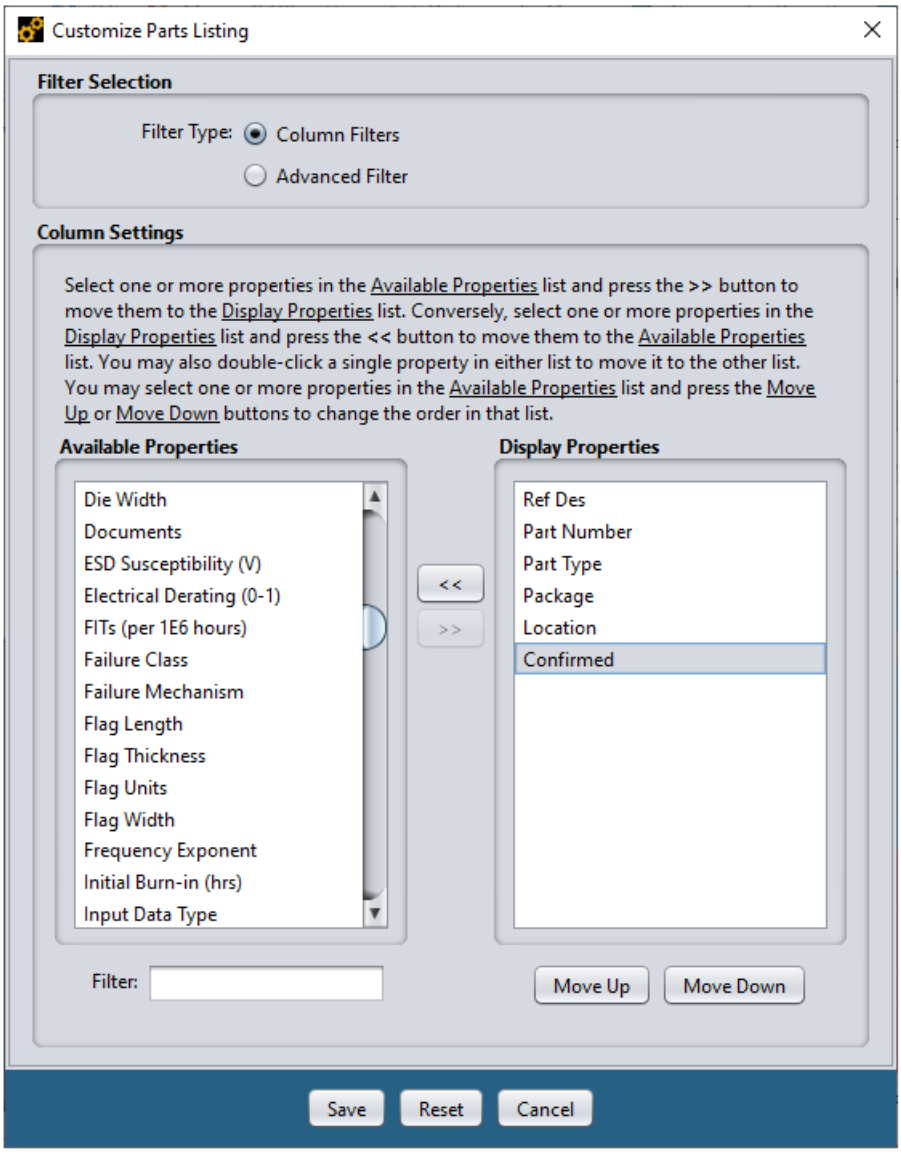
4.1.4. Customizing the Parts List

You may customize what data the **Parts List** displays as well as the width of individual columns as needed. Sherlock automatically saves the table settings and reloads them whenever the **Parts List** is displayed. To begin the customization process, press the **Customize the parts list view** icon  in the icon ribbon of the **Parts List** tab. This launches the **Customize Parts List** dialog shown below.

Use the two lists in the dialog to select the part properties to be displayed in the **Parts List**. The part properties defined in the right-hand list will be displayed as table columns. The list on the left shows all possible properties you can choose from. You can move an one item or multiple items from one list to the other by highlighting your selection and pressing the << or >> button to move the item(s) between the lists.

The **Filter Type** setting allows you to select the type of filtering for the **Parts List**, which is explained in the sections that follow.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Move Up** and **Move Down** buttons below the right-hand list can be used to reorder the properties to be displayed in the table. Simply select one or more properties in the right-hand list and press **Move Up** or **Move Down** to change their relative position in the list.

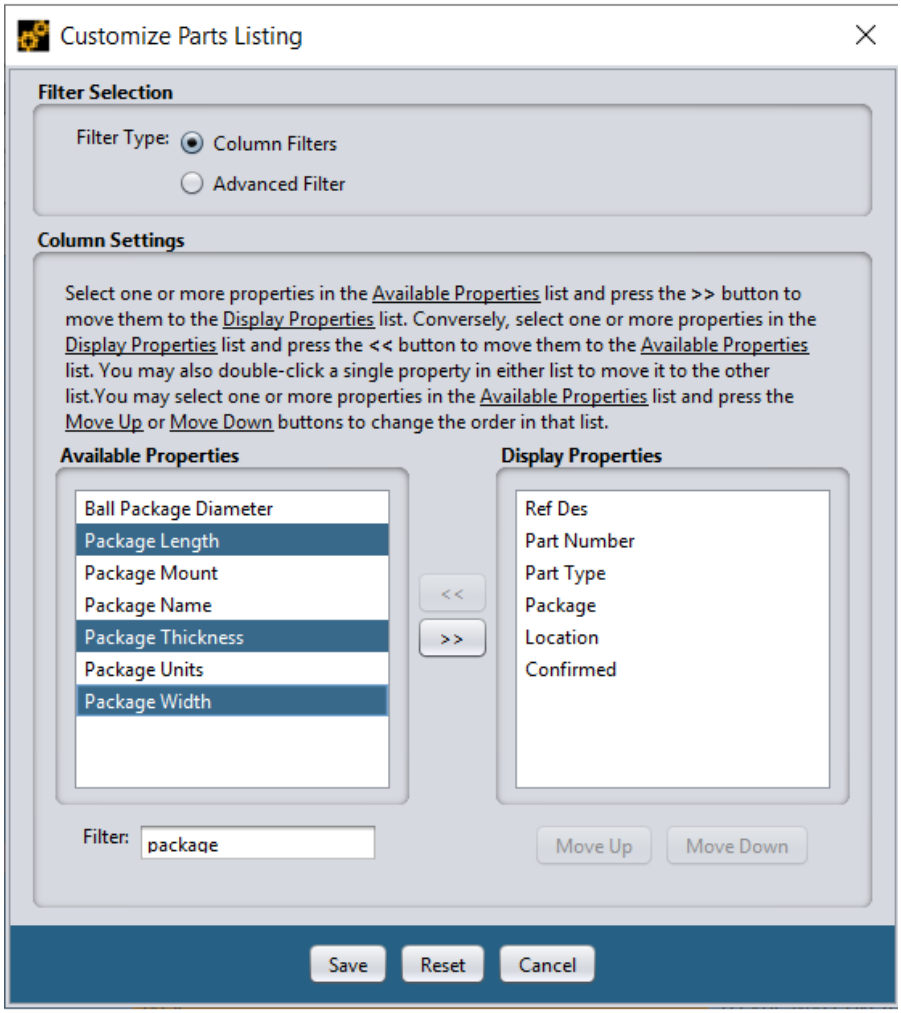
For convenience, the **Filter** field near the bottom can be used to filter the property names. The following series of images shows how the customization works.

Example: Customizing the Parts List

The following example serves as a tutorial on how to customize the **Parts List**.

To add some properties to the **Parts List** table, begin by entering **package** in the **Filter** field to reduce the number of items shown in the **Available Properties** list. (See below.) Then, select the **Package Length**, **Package Thickness** and **Package Width** properties from the left-hand list. Finally, click the **>>** button to move those properties from the left-hand list to the right-hand **Display Properties** list.

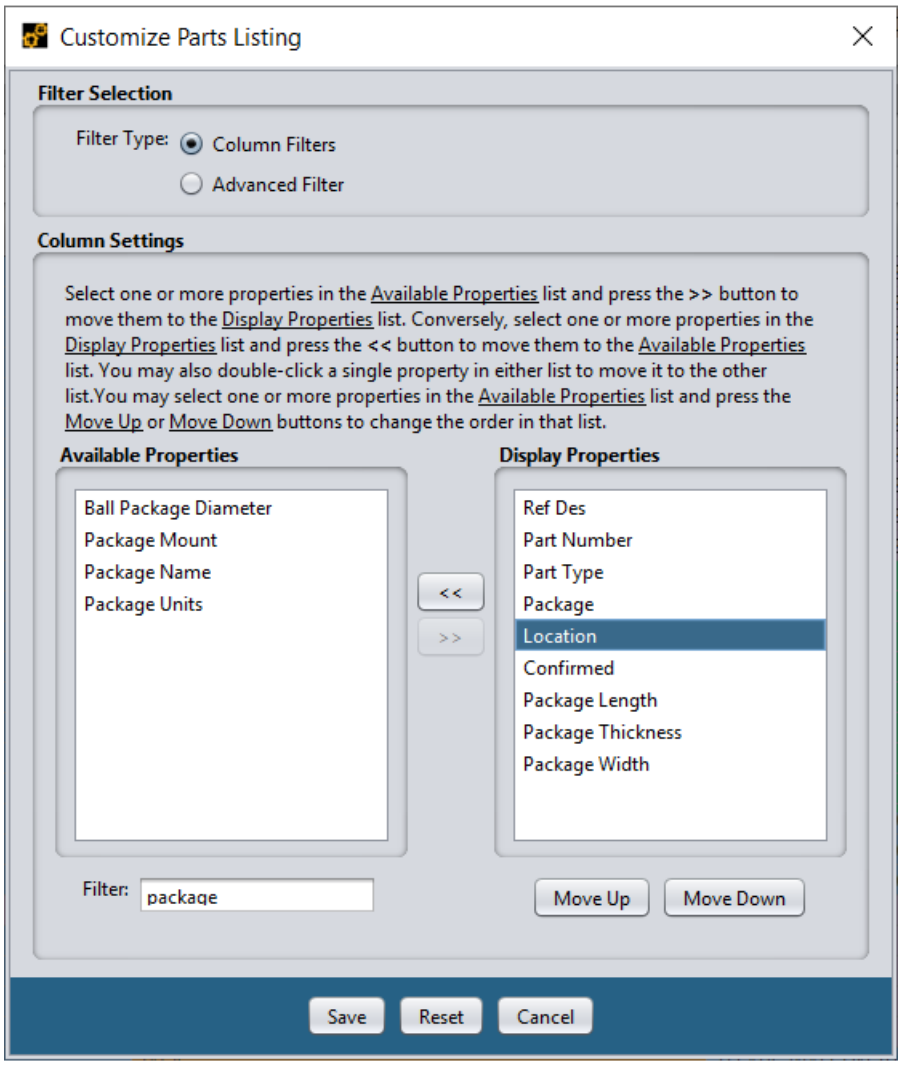
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To remove a property from the Parts List, select the property in the right-hand **Display Properties** list (see below) and press the **<<** button. At that point, the property will be moved from one list to the other.

Note, since the **Filter** is still set to **package**, the **Location** property will not be displayed in the **Available Properties** list until the **Filter** is changed accordingly.

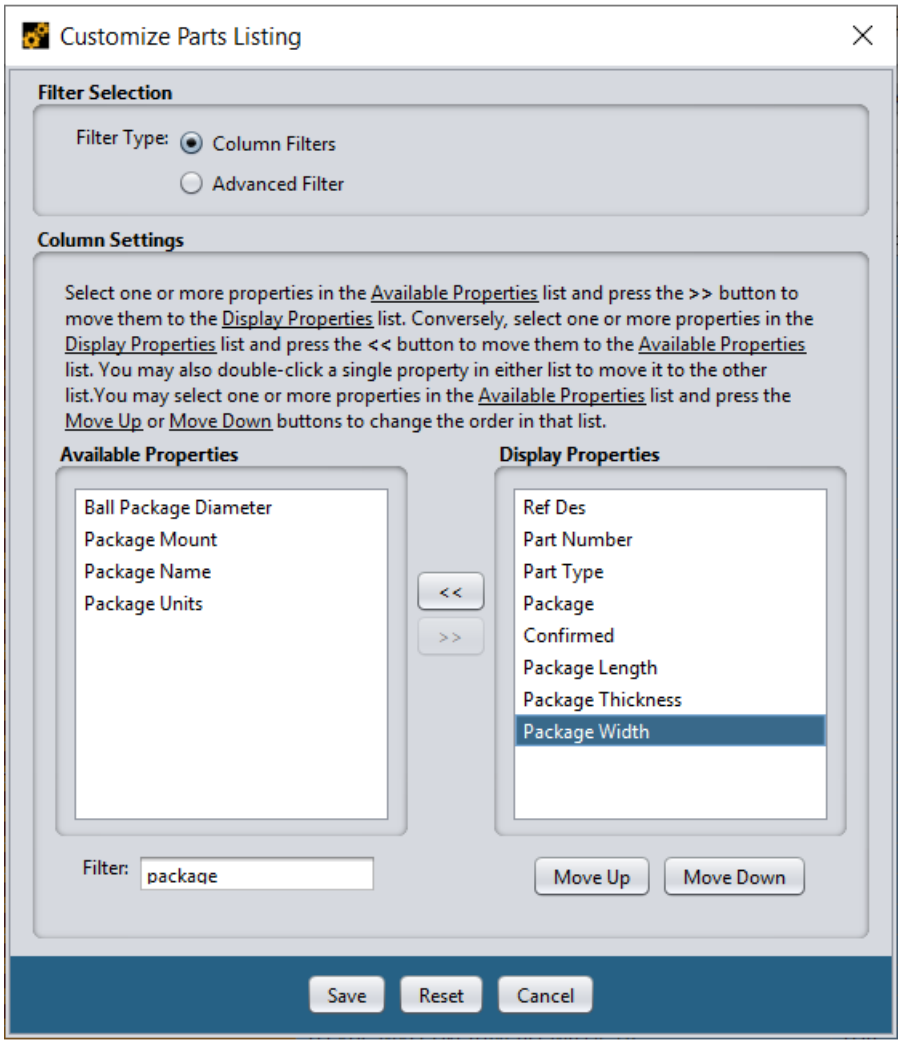
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



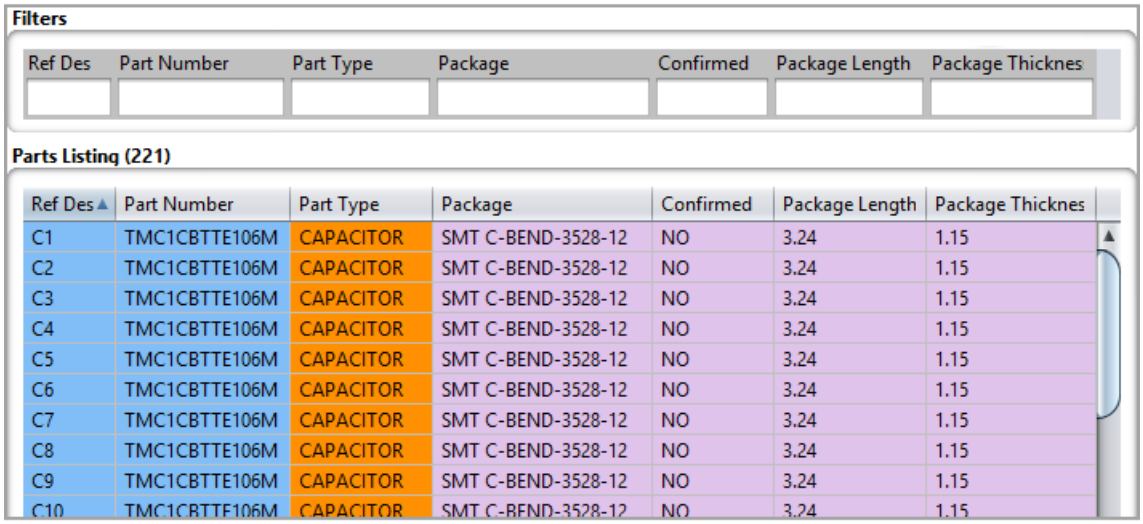
Finally, to change the order of the properties displayed in the **Parts List** select the **Package Width** property (see below) and press the **Move Up** button. At that point, the **Package Width** property will be moved above **Package Thickness**. Press the button again, and the property moves up another position.

If multiple properties are selected, then each of them will be moved up or down by one position whenever the **Move Up** or **Move Down** buttons are pressed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After making changes in the **Customize Parts List** dialog, press the **Save** button to update the **Parts List**. Sherlock will automatically save the **Displayed Properties** and will use it to configure the **Parts List** whenever it is opened. Based on the changes above, the customized **Parts List** is shown here:



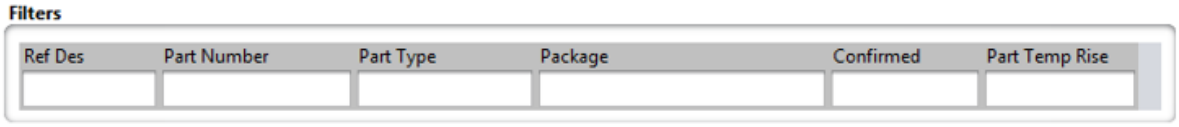
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the number of properties displayed in the **Parts List** is wider than the viewing panel, a horizontal scroll bar will appear, allowing you to scroll left and right. The column filter fields will automatically adjust. You can also select the **Automatically Resize all the Column Widths** button  at the top right of the **Parts List** dialog to get rid of the horizontal scroll bar.

You can also change Individual table column widths by dragging the divider between each column header. Sherlock will save the width of each column and will configure the **Parts List** table accordingly whenever it is displayed.

4.1.5. Filtering and Selecting Parts

Directly above the **Parts List** is a row of **Column Filters** that allows you to reduce the number of parts being displayed. If you enter a value is into one of the filter boxes, then only those parts with a cell entry matching that value will be displayed. For example, if you enter Q into the **Ref Des** filter and press **Enter**, then only parts starting with a reference designator of Q will be displayed.



If you enter values into two or more filter boxes, then only those parts that match *all* of the filter conditions will be displayed. For example, if Q is entered into the **Ref Des** filter and SOT is entered into the **Packaging** filter, then only parts Q1 through Q8 will be displayed.

Sherlock's more advanced filtering capabilities are covered in Filtering the Parts List (Advanced) (p. 211).

Tip:

Sherlock allows you to edit multiple parts at once (see The Multi-Part Editor (p. 203)). Consider using the **Column Filters** when selecting the parts you wish to edit.

4.1.6. Updating Component Layers

Warning:

Always remember: If you modify a part or parts in the **Parts List**, you must click the **Update the CCA Layers** icon for those changes to be reflected in the component layers and in your analysis results.

When you make modifications to the **Parts List**, you must perform an important step before proceeding. To see your part property changes reflected in the layer displays, you must update the component layers. To do this, press the **Update the CCA Layers** button (A, below). After the component layers are updated, the button will be disabled indicating that no part changes are pending.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Also, Sherlock will update the CCA Layers in the following circumstances:

- If you have made any changes to a part or parts, Sherlock will automatically update the component layers if you attempt to close the **Parts List** or the Sherlock application. A progress dialog will be displayed while the process is being performed.
- If you right-click the **Parts List** entry in the **Project Tree** and select the **Update Layers** menu option.

4.2. Parts List Management Tools

This section describes the more advanced features of the **Parts List** and the part management tools available to you in Sherlock. The following topics are covered in this section:

4.2.1. The Part Properties Viewer
4.2.2. The Part Editor
4.2.3. The Multi-Part Editor
4.2.4. The Package Chooser
4.2.5. Part Source Priority
4.2.6. Confirming Parts
4.2.7. Filtering the Parts List (Advanced)
4.2.8. Exporting the Parts List
4.2.9. Importing the Parts List

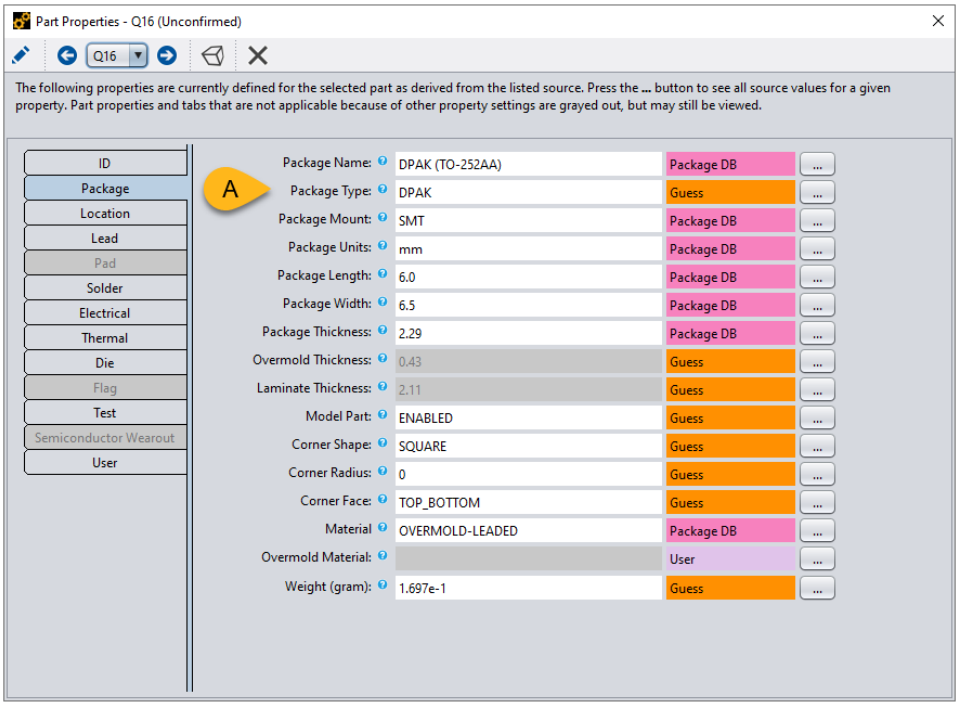
4.2.1. The Part Properties Viewer

The **Part Property Viewer** displays all the properties currently known for the selected part, with properties organized by tabs. The part's **Package Type** (A, below), which is displayed in the **Package** tab, determines which fields and tabs are enabled and which are grayed out.

Topics:

4.2.1.1. Opening the Part Properties Viewer
4.2.1.2. Part Properties Viewer Interface
4.2.1.3. Part Property Viewing/Editing Options

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4.2.1.1. Opening the Part Properties Viewer

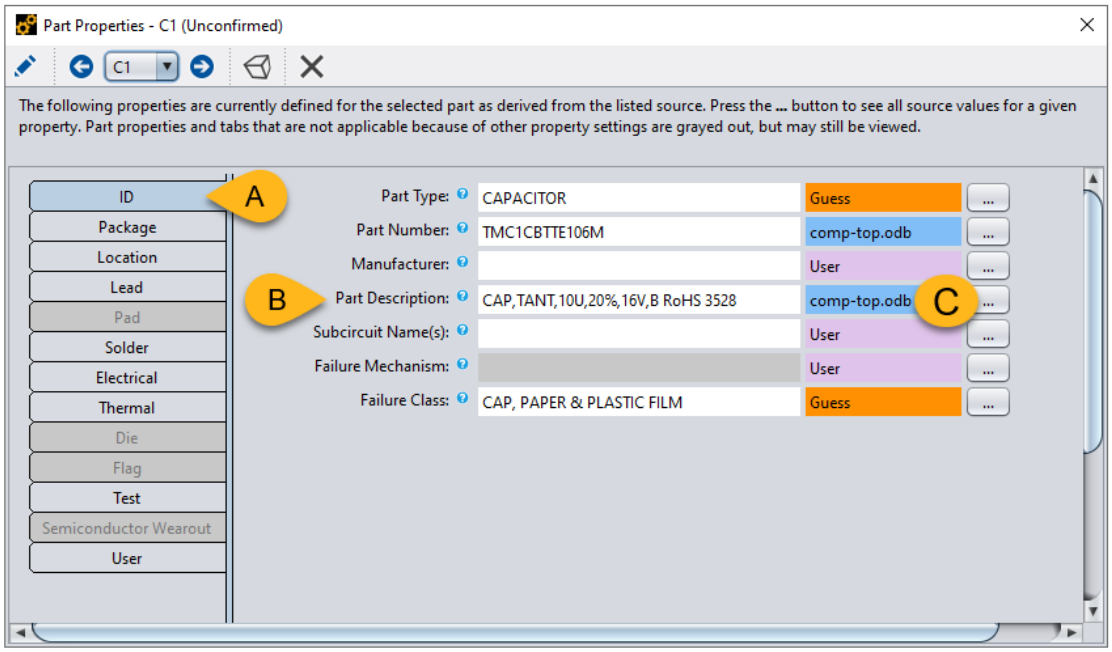
To open the **Part Properties Viewer**, do the following:

- To view the properties for a single part
  - Double-click a row on the **Parts List** to open the **Viewer**.
  - Right-click a row on the **Parts List**, and select **View Part Properties** in the context menu.
- To open the **Viewer** for multiple parts, first select the parts on the **Parts List** you want to view. (Remember, you can use the SHIFT and CTRL key to select multiple parts (p. 187).) Then right-click one of the selected parts and choose **View Part Properties** from the context menu.

4.2.1.2. Part Properties Viewer Interface

The **Part Properties Viewer** is shown below for part **C1** with the **ID** tab (A) selected. The **ID** tab shows the important **Part Description** property (B) which Sherlock used, in this example, to guess the other part properties (as explained in Tutorial: Reviewing Part Properties (p. 30)). The blue box to the right (C) indicates that the description property was obtained from the **comp-top.odb** file extracted from the ODB++ archive, which is denoted as **Pick & Place** data by the blue color (using the same color-coding scheme used in the **Parts List**). See the section Part Source Priority (p. 207) for more on how Sherlock handles part data sources.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Part Properties Viewer Icon Ribbon



A. **Edit the Selected Part**: Opens the **Part Editor**. See The Part Editor (p. 200).

B. **Current Part / Navigate to the Previous or Next Part**: The reference designator (**Ref Des**) of the part currently displayed is shown (Q14 in this example).

   You can use the dropdown menu to select a different part to be displayed. If you opened multiple parts in the **Viewer**, the dropdown list is limited to those parts you selected. Otherwise, all the parts in the **Parts List** are available. You can also navigate to the previous or next part on the list using the arrow icons.

C. **View the 3D Model of the Part**

D. **Close the Part Properties Viewer**

4.2.1.3. Part Property Viewing/Editing Options

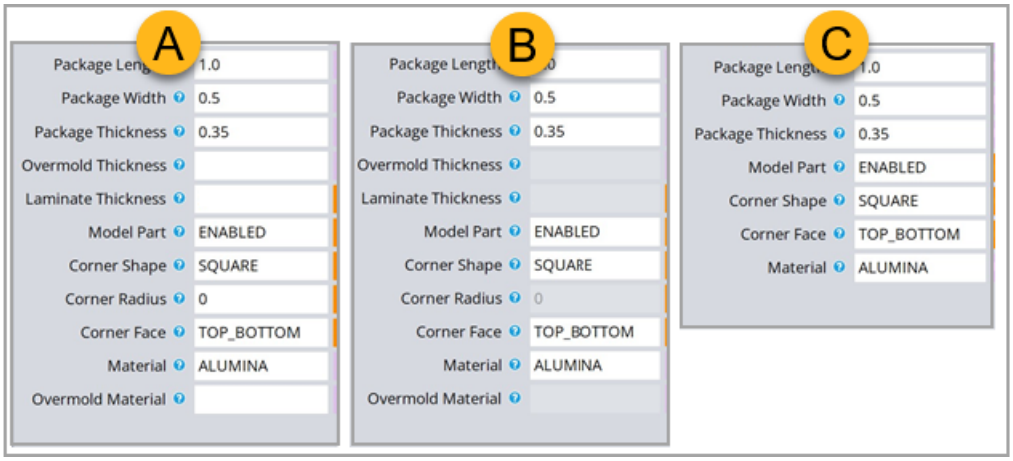
Not all part properties apply to all parts. By default, the **Part Properties Viewer** and **Editor** display all properties whether they are applicable to the selected part or not. You can even edit these unused properties, although it has no effect on how Sherlock handles the part in its analyses. If you prefer, you can adjust how Sherlock displays unused part properties. There are three **Unused Part Properties** options:

- **ENABLED** (default): Unused properties are visible for a given part and are editable. In the images below, package properties are shown for the same part. Although **Overmold Thickness**, **Laminate Thickness**, **Corner Radius**, and **Overmold Material** do not apply to this part, Sherlock displays these properties and allows you to edit them when **ENABLED** is selected. See image A.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **DISABLED**: Unused properties are visible for a given part but may not be edited. Note the grayed out fields in image B.
- **HIDDEN**: Unused properties are not visible at all, as shown in image C.

You can find the **Unused Part Properties** setting in Sherlock's **Advanced Settings** dialog (p. 78): **Main Menu** > **Settings** > **Advanced Settings** > **Unsused Part Properties**.



4.2.2. The Part Editor

The **Part Editor** looks similar to the **Part Property Viewer**, except that it allows you to edit the various property values instead of just viewing them.

Topics:

4.2.2.1. Opening the Part Editor
4.2.2.2. Part Editor Interface
4.2.2.3. The Part Editor and Package Properties
4.2.2.4. Scope of Changes

4.2.2.1. Opening the Part Editor

To open the **Part Editor**, do the following:

- From the **Parts List:**
  - To edit a single part, right-click a row on the **Parts List**, and select **Edit Parts** in the context menu.
  - To edit multiple parts, see the section below, The Multi-Part Editor (p. 203).
- From the **Part Properties Viewer**: Click the **Edit** button  in the icon ribbon.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.2.2. Part Editor Interface

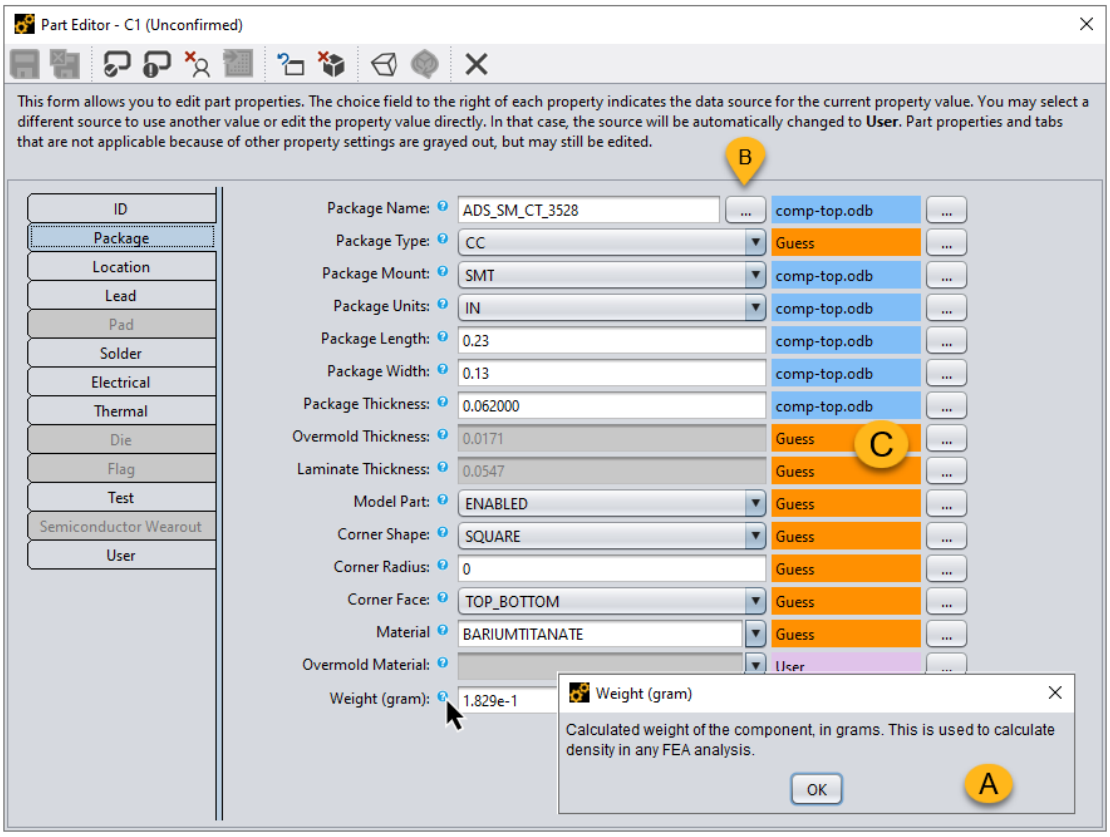
This section explains the **Part Editor** interface. Notice that each part property listed has an information icon (the blue question mark) next to it. Click the icon for an explanation of any of the part properties (A, in the image below).

When editing the **Package Name**, click the ... icon (B, below) to open the **Package Chooser** which simplifies the process of choosing the correct package. (See the section below, The Package Chooser (p. 206).)

For an explanation of the data source color code system (C), see the section Part Source Priority (p. 207).

Tip:

The **Package Name** and **Package Type** properties play an important role in the **Part Editor**. Be sure to read The Part Editor and Package Properties (p. 202) below.



Tip:

The **User** tab contains a text field where you can record your own **Notes**. When you enter a note and click **Save**, Sherlock gives you the choice of a) attaching the note to the specific part you are viewing in the **Part Editor** or b) attaching it to *all* parts with that same **Part Number**. See Scope of Changes (p. 203) below.

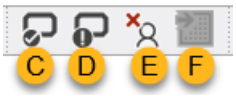
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part Editor Icon Ribbon



A. **Save the Part Editor Changes:** See below, Scope of Changes (p. 203).

B. **Save the Part Editor Changes and Exit the Editor:** See below, Scope of Changes (p. 203).



C. **Confirm all Part Properties**: See Confirming Parts (p. 210).

D. **Un-Confirm all Part Properties**: See Confirming Parts (p. 210).

E. **Clear User Overrides for all Part Properties**. This clears all **User** values for all the properties associated with this part. **Source Values** revert to the next highest priority value available. See Part Source Priority (p. 207).

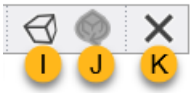
F. **Update Pad Properties for the Selected Parts**: See Updating Pad Properties (p. 423).



G. **Reset the Form to Currently Saved Values**.

H. **Delete the Selected Part**.

> **Warning:**
>
> You cannot undo this action. A deleted part can only be restored manually.



I. **View 3D Model of the Part**.

J. **Refresh the 3D Model**.

K. **Close the Dialog**.

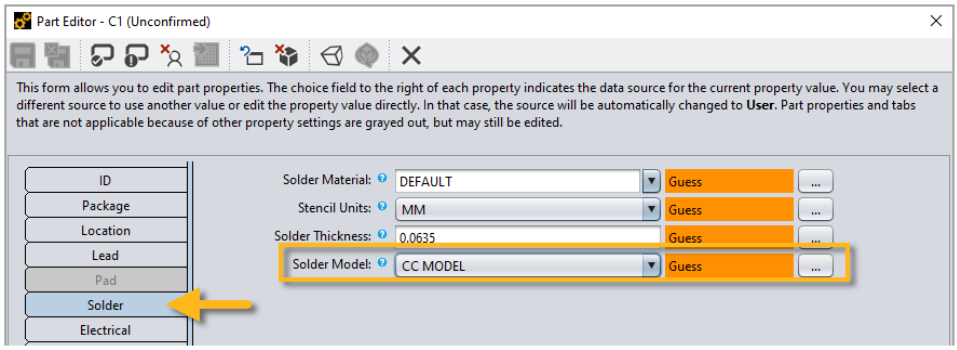
4.2.2.3. The Part Editor and Package Properties

The **Package Name** and **Package Type** properties play an important role in the **Part Editor**. Here are a few things you should know:

- The **Package Type** property determines which fields and tabs are active in the **Part Editor**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

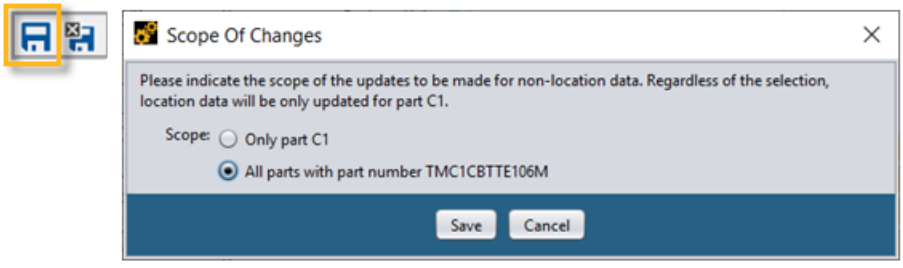
- Sherlock allows you to choose the **Package Type** independently of the **Package Name**. It's important to use this feature carefully. It is possible to end up with a package in your **Parts List** with a *different* **Package Type** property than the same package in your **Package Library**. See Package Management (p. 315) for more on the **Package Library**.
- If you are using the **Part Editor** to add a new part to the **Parts List**, Sherlock will guess the **Package Type** when you first enter the **Package Name**. If you do not agree with Sherlock's choice, you can manually change the **Package Type**.
- Sherlock will guess the **Solder Model** when you change the **Package Type**. Of course, you can manually edit the **Solder Model** type if you wish. You will find the **Solder Model** property on the **Solder** tab as shown below.



4.2.2.4. Scope of Changes

When you **Save** changes in the **Part Editor**, the **Scope of Changes** dialog appears, offering you two options.

- Apply the changes to the selected part only.
- Or, apply the changes to the selected part and all other parts that have the same **Part Number** as the selected part.



4.2.3. The Multi-Part Editor

Sherlock allows you to edit multiple parts for using a slightly different editor called the **Multi-Part Editor**.

Topics:

4.2.3.1. Opening the Multi-Part Editor
4.2.3.2. Multi-Part Editor Interface

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.3.3. Scope of Changes

4.2.3.1. Opening the Multi-Part Editor

To open the **Multi-Part Editor**, do the following:

1. From the **Parts List**, select the parts you wish to edit. There are several ways to select multiple parts:

   - **To select a group of contiguous rows on the Parts List:**

     – Left-click a row and drag the mouse down the list.

     – Left-click the first row of your selection. Then press the SHIFT key while left-clicking the last row of your selection.

   - **To select all the cells displayed on the Parts List**: Left-click any row and and then press CTRL-A on the keyboard.

   Tip:

   To make your part selection easier, use the **Column Filters** (p. 196) at the top of the **Parts List** to reduce the number of parts being displayed. Then select your parts for editing.

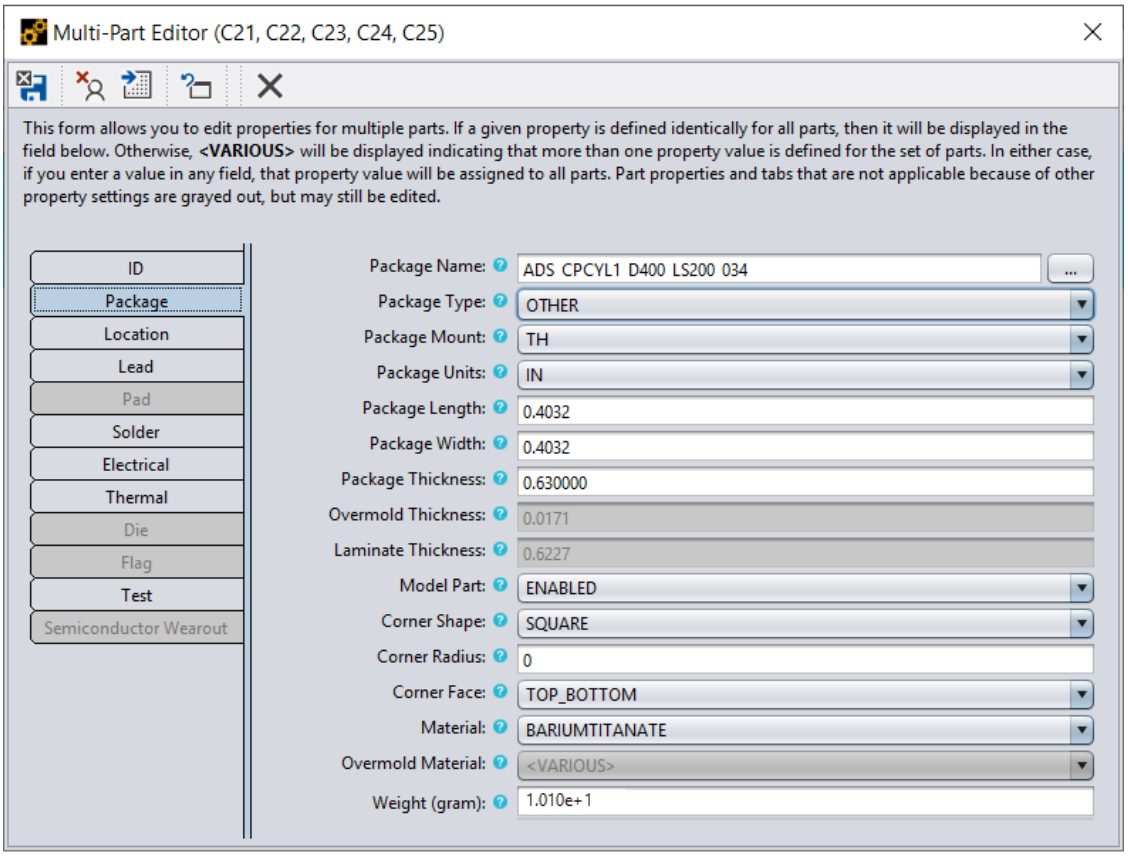
2. Once you have selected the parts for editing, right-click the selection and choose **Edit Parts** from the **Context Menu**.

4.2.3.2. Multi-Part Editor Interface

The **Multi-Part Editor** is similar to the **Parts Editor**, except for two important differences.

- The **Multi-Part Editor** does *not* display the source for each property values, because different parts may have different sources.
- The **Multi-Part Editor** displays <VARIOUS> in some property fields to indicate that the values assigned to the specific parts are either different or not defined at all.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aside from the display differences, the **Multi-Part Editor** is used to update part properties just like the **Parts Editor**.

Multi-Part Editor Icon Ribbon



- **Save Changes and Exit the Part Editor**.
- **Clear User Overrides for all Part Properties**. This clears all **User** values for all the properties associated with the selected parts. **Source Values** revert to the next highest priority value available. See Part Source Priority (p. 207).
- **Update Pad Properites for the Selected Parts**: See Updating Pad Properties (p. 423).
- **Reset the Form to Currently Saved Values**.
- **Close the Multi-Part Editor**: Unsaved changes will be lost.

4.2.3.3. Scope of Changes

When you save your changes in the **Multi-Part Editor**, only the parts you have selected are updated, regardless of their **Part Numbers**. The **Scope of Changes** (p. 203) dialog will *not* appear.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.4. The Package Chooser

When you choose to edit the **Package Name** field in the **Part Editor**, Sherlock's **Package Chooser** makes that task easier. To open the **Package Chooser**, click the ... icon (A, below) next to the **Package Name** field in the **Part Editor**.



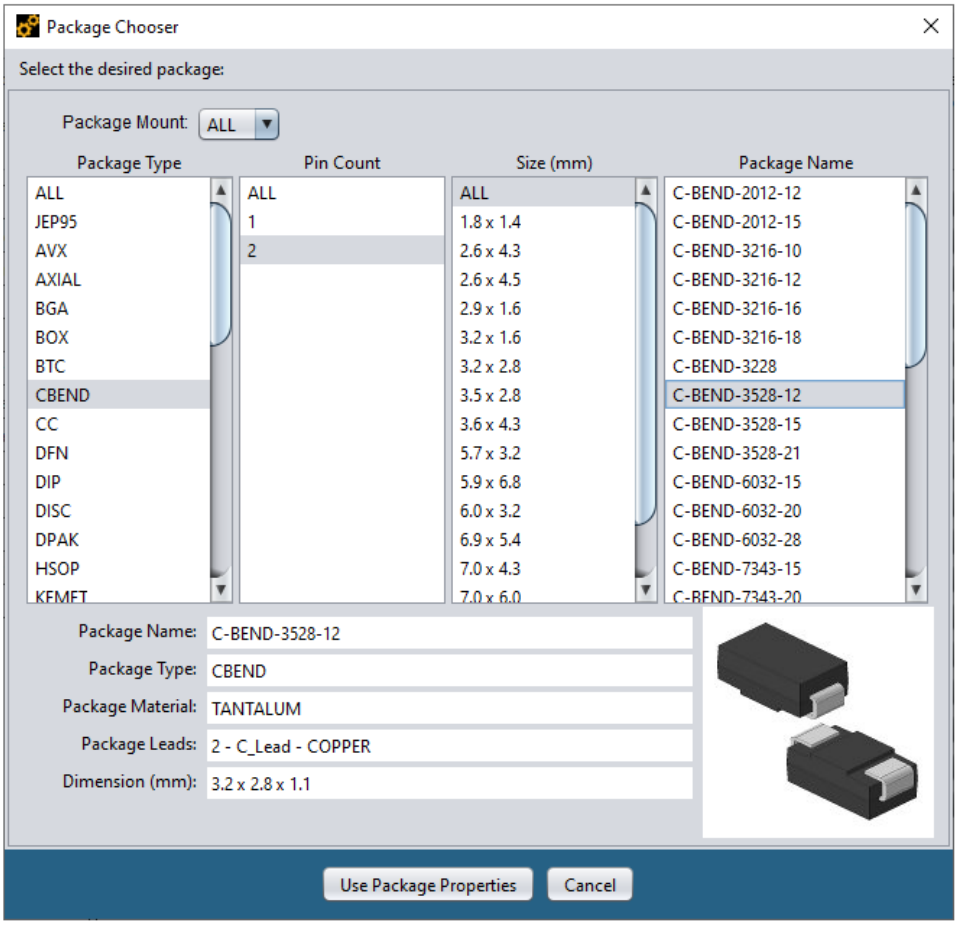
The **Package Chooser** presents a hierarchical list of all standard packages known by Sherlock, organized by **Package Type**, **Pin Count**, and **Size**. As you select the desired property in each column, the **Package Chooser** narrows down the list of **Package Names** to those that fit your criteria. The **Chooser** even provides a graphic representation of the selected package to assist you in your search.

Once you have selected the desired **Package Name** in the right-hand column, click the **Use Package Properties** button to copy all package properties to the **Part Editor**.

Tip:

In addition to the **Package Chooser**, Sherlock provides an additional tool that allows you to create and modify Sherlock package definitions: The **Package Manager**. It is as a standalone application and is accessible from the Sherlock's Main Menu. Select **Libraries > Packages**. See Package Management (p. 315) for more information.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4.2.5. Part Source Priority

The **Parts List** and the **Part Properties Viewer** use a color-coded system to show the source Sherlock is using to obtain the data shown in the property field. For a given part property, Sherlock must often choose from among multiple data sources that have different values for the same property. In such cases, Sherlock uses predefined priorities to determine which value to use. If you wish, you can override Sherlock's choice by manually entering the correct property value or by changing part source priority.

The following is an extended example of how you can investigate and modify Sherlock's source priority for a given part property. In this case, the **Package Name** property for component **C1** is under scrutiny

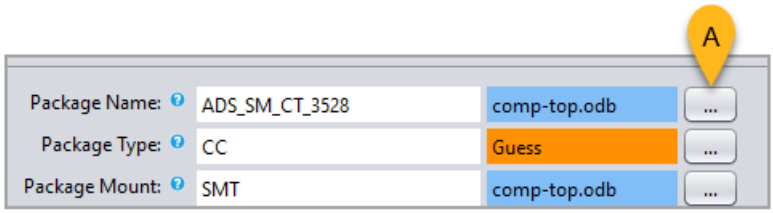
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Examining Source Priority for a Part Property

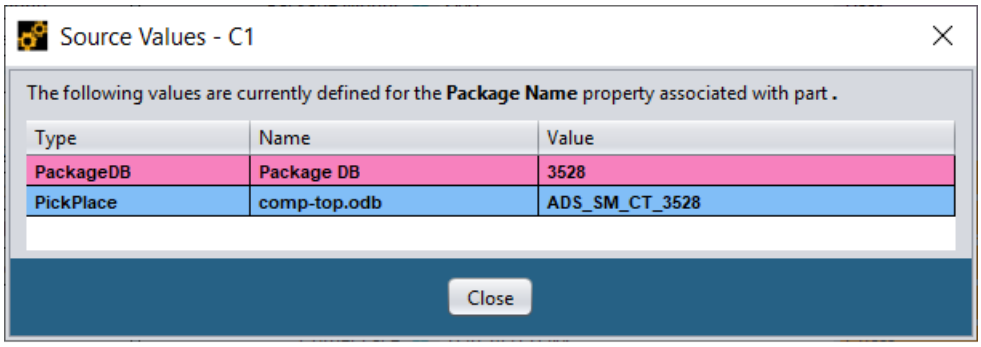
From the **Part Properties Viewer** (p. 197), clicking the ... button located at the far right of the **Package Name** (A) will open the **Source Values** window for **C1**.

Tip:

Hovering over the ... button allows you to view and select Source Values from a context menu without having to open the **Source Values** window.



As shown below, Sherlock indicates there are two data sources for the **Package Name** property. When launched from the **Part Properties Viewer,** the **Source Values** dialog is in read-only mode. It is a reference only and cannot be edited.



If you want to change, add, or remove a **Source Value**, you must launch the **Source Value** dialog from the **Part Editor** by doing the following:

1. Close the **Source Values** window.

2. Click the **Edit the Selected Part** icon located at the top of the **Viewer**. This opens the part in the **Part Editor**.

Changing the Part Source Priority

Once you are in the **Part Editor**, you can modify the part property sources a number of ways.

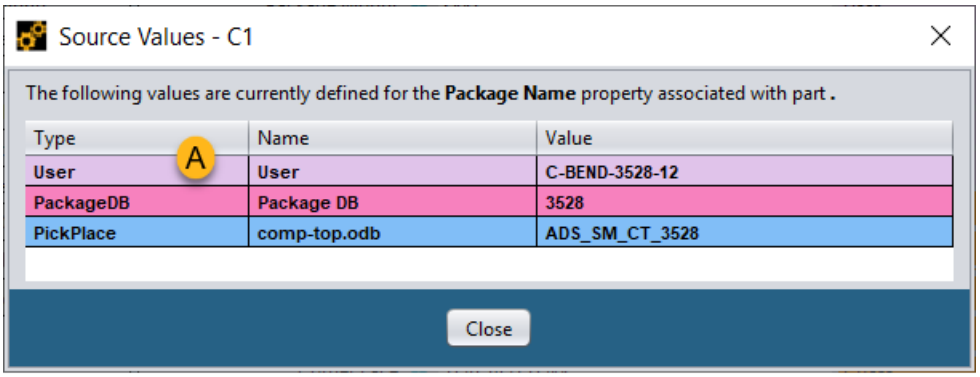
- **Manually modifying a part property creates a new User source:** When you modify any part property using the **Part Editor** (p. 200), Sherlock automatically adds a **User** entry to the **Source Values** table. In the example below, someone has edited the **Package Name** for component **C1**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

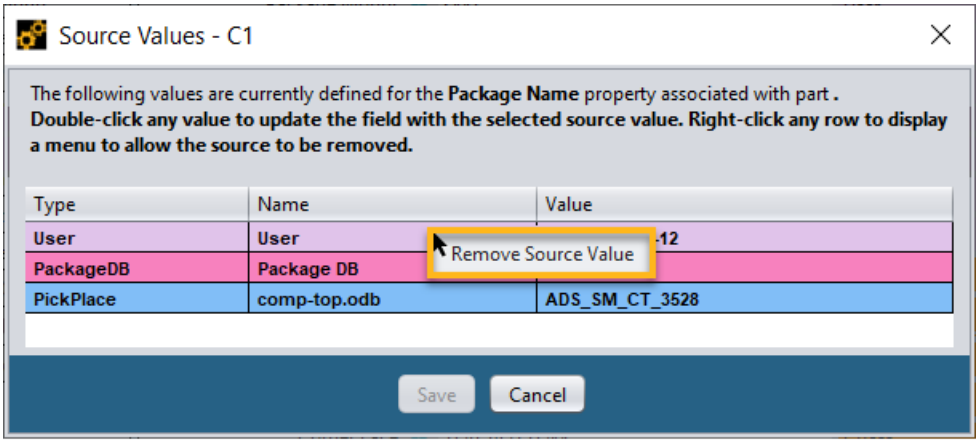
(see A), and Sherlock has recorded this new source. The **User** source is first on the list, which means Sherlock gives it highest priority.

Note:

Sherlock always gives the **User** source highest priorty unless you manually change the source priority as described below.



- **Changing Source Priorities:** To give a source highest priority, double-click the entry in the **Source Values** window. In the image above, for example, double-clicking the last row, **PickPlace**, will move it to the top of the list. Going forward, Sherlock will refer to the **PickPlace** source to get the **Package Name** property for **C1**.
- **Deleting Sources Values**: To delete a source value, right-click the entry in the **Source Values** window. Click the **Save** button to save the changes.



Tip:

You can clear all **User** sources for a given part by using the **Clear User Overrides** (p. 202) button in the **Part Editor**. You can also perform this operation on multiple parts using the **Multi-Part Editor** (p. 203).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

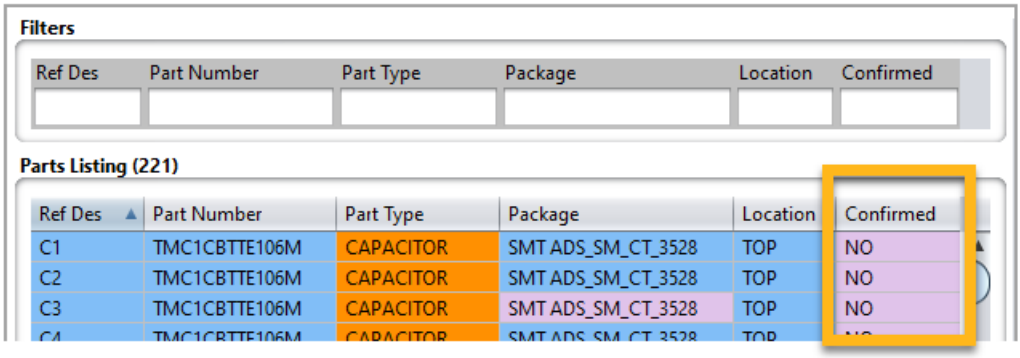
4.2.6. Confirming Parts

In the image below, notice the last column in the **Parts List** labeled **Confirmed**. A **YES** status in the column means you have reviewed the properties for that part and have determined they are correct. A **NO** status means you have not yet reviewed the part's properties, so its properties might *not* be correct.

When a project is initially created and data is loaded from CAD files, Sherlock assigns a **NO** in the **Confirmed** column for all parts. This serves as a reminder to review those parts to make sure their properties are correct.

Tip:

Disciplined use of the confirmation process will ensure more accurate analysis results.

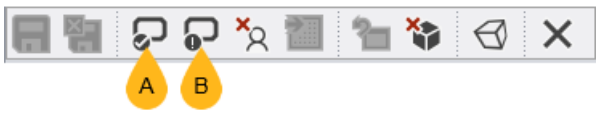


After you have reviewed and (if needed) corrected a part's properties, you should change the part's **Confirmed** status to **YES**. There are a couple ways to do this:

- **From the Parts List:** Select a single part or multiple parts for confirmation. Then click the **Confirm All Part Properties** icon (A, below). Sherlock will prompt you to confirm your decision.
- **From the Part Editor:** Click the **Confirm All Part Properties** icon. You must click the **Save** icon to save the change. Sherlock will prompt you to confirm whether you want to confirm only the selected part or all the parts that share its part number. See Scope of Changes (p. 203).

Note:

You can also un-confirm a part at any time using these same procedures. But use the **Un-Confirm All Part Properties** icon (B, below) instead.



A. **Confirm all part properties**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B. **Unconfirm all part properties**.

4.2.7. Filtering the Parts List (Advanced)

This section introduces you to more advanced filtering techniques when using the **Parts List's Column Filters**. For a basic-level treatment, see Filtering and Selecting Parts (p. 196).

Sherlock provides two types of expression-based filtering for the **Parts List**:

- Column Filters
- Advanced Filter

The type of filter is specified using the **Filter Type** choice list in the **Customize Parts Listing** dialog (described in the previous subsection).

Column Filters allow users to enter individual filter expressions for any of the columns displayed in the **Parts List** table. They are an easy and effective method for filtering the Parts List with a minimal amount of data entry.

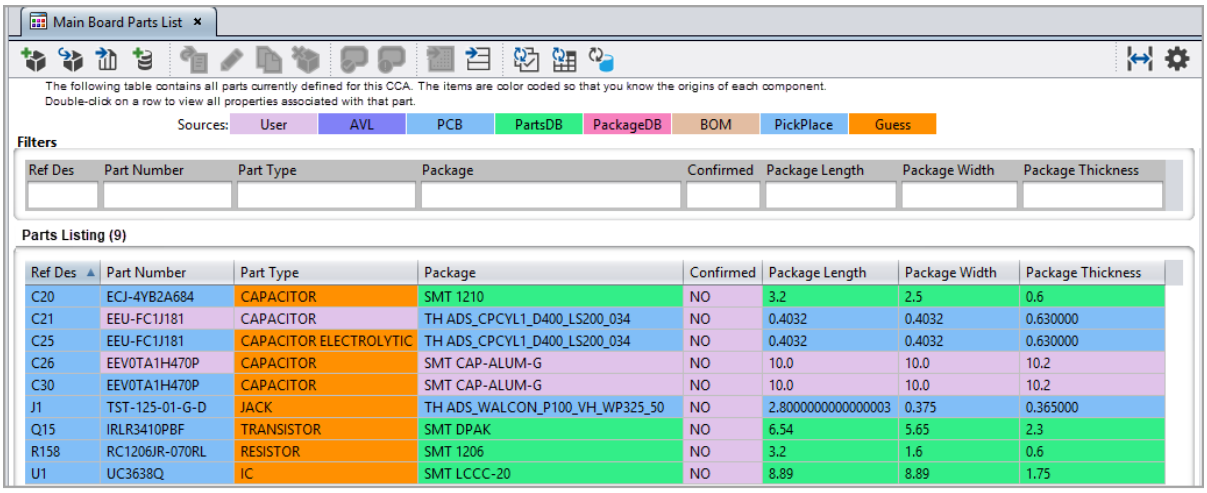
The Advanced Filter allows users to enter simple or complex filter expressions based on one or more part properties. Such filter expressions require more data entry and formatting than those used in Column Filters, but they allow for greater control over which table rows are filtered. Advanced Filter expressions can also reference part properties that are NOT displayed in the Parts List table itself, allowing users to search for parts that have specific properties.

4.2.7.1. Column Filters

Column Filters allow users to enter simple filter expressions that are applied to each of the values in a given table column.

The following **Parts List** will be used as the basis for all the examples in this subsection:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

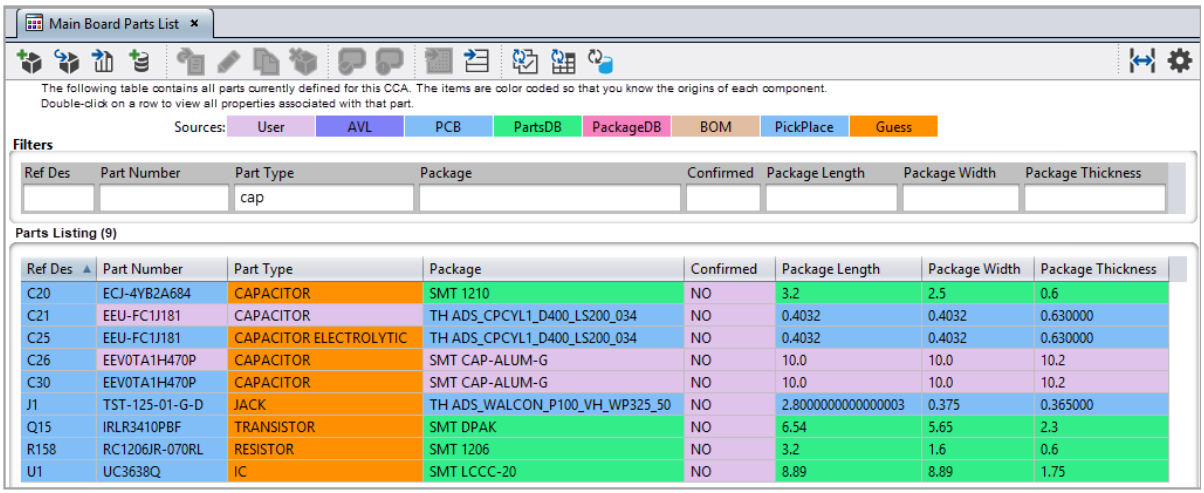


Note:

Column Filters are applied whenever you press the **Tab** or **Enter** key inside of a filter field. The Tab key will move the focus to the next filter field, while the **Enter** key will keep the focus on the same field.

4.2.7.1.1. Column Filter - String Matching

In the simplest form, a string is entered as the filter expression and all rows with cell values that CONTAIN the given string are displayed. For example, as shown below, the **cap** value entered in the **Part Type** filter field results in five of the nine rows being displayed in the table.



Note:

For convenience, filter expressions are case-insensitive unless otherwise noted. Also, strings may be quoted or un-quoted.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition to the default string matching behavior, the following filter expressions may be used for string matching:

Table 4.1: String Matching Operators

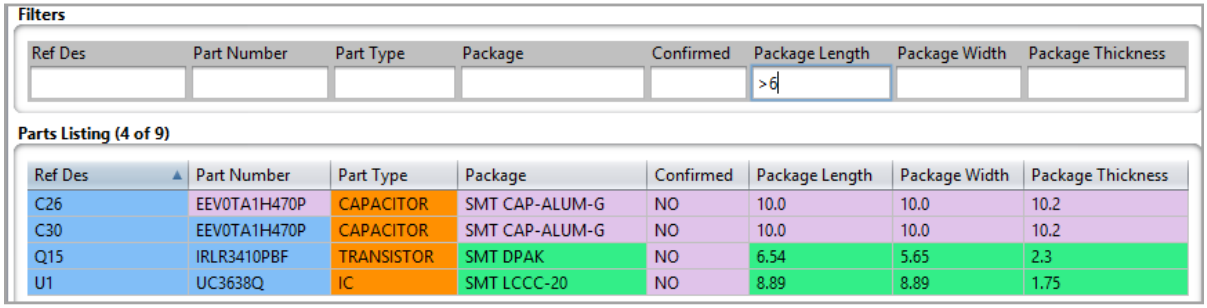
Operator	Description
contains STRING	True if the column value contains STRING
startsWith STRING	True if the column value starts with STRING
endsWith STRING	True if the column value ends with STRING
= STRING	True if the column value equals STRING
$STRING	True if the column value starts with STRING
STRING$	True if the column value ends with STRING
$STRING$	True if the column value equals STRING
=~ "REGEX"	True if the column value satisfies the Java regular expression (case sensitive)
!~ "REGEX"	True if the column value does not satisfy the Java regular expression (case sensitive)

Note:

Regular expressions must always be quoted because they may contain special characters that would otherwise be interpreted as part of the filter expression. See Java documentation for a complete description of regular expressions.

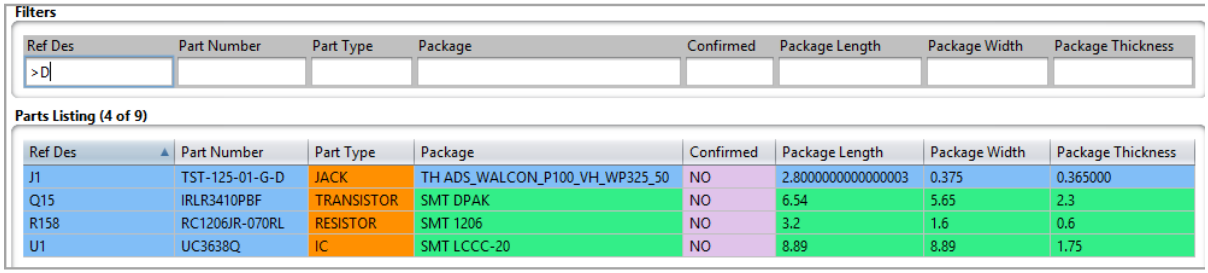
4.2.7.1.2. Column Filtering - Comparison Operators

In addition to the string-matching operators described above, filter expressions can make use of various comparison operators for both string and numeric values. For example, the following filter expression displays all rows that have a **Package Length** value greater than six. In this case, since the **Package Length** values are numeric, a numeric comparison is performed by the **greater than** operator.



In the following example, the **greater than** operator performs an alpha-numeric comparison, resulting in the display of rows where the **Ref Des** value is greater than **D**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

An alpha-numeric comparison differs from a simple string comparison in that it compares numeric substrings as numbers. For example, the value **SMT 0603** is greater than **SMT 100**.

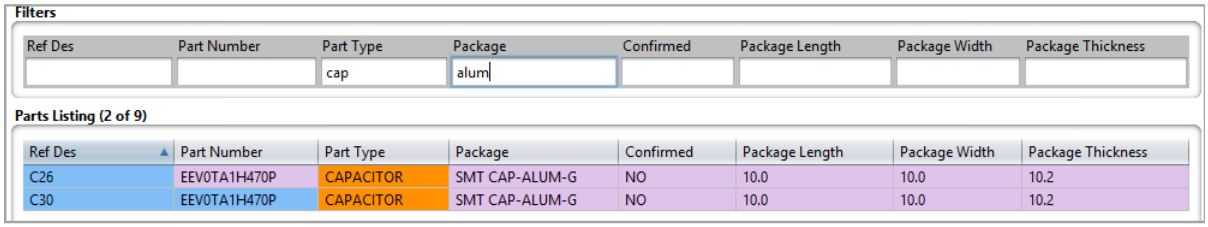
The following comparison operators are supported in column filter expressions for both numeric and string values:

Table 4.2: Column Filtering Comparison Operators

Operator	Description	Operator	Description
= or ==	Equals	!= or <>	Not Equals
<	Less Than	>	Greater Than
<=	Less Than Or Equals	>=	Greater Than Or Equals

4.2.7.1.3. Column Filtering - Multi Column

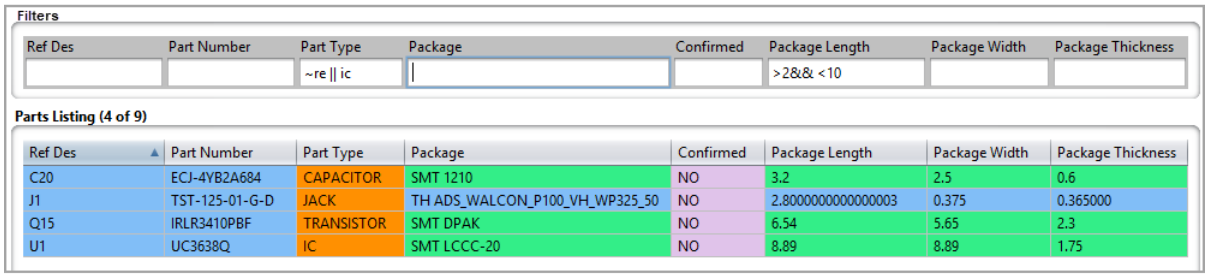
If two or more column filter expressions are entered, then only the rows that match **ALL** of the filter expressions are displayed in the table. For example, as shown below, the three rows displayed each have a **Part Type** value that contains **cap** and a **Package** value that contains **alum**, as specified by the column filters.



4.2.7.1.4. Column Filtering - Boolean Expressions

Boolean expressions can be defined in each column filter expression to facilitate more complex filtering criteria. In the following example, the **Package** filter expression selects values that either do not contain **re** or that contain **ic**, while the **Package Length** filter expression selects values that are greater than two and less than ten.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following boolean operators are supported in column filter expressions:

Table 4.3: Column Filtering Boolean Operators

Operator	Description
| or | |	True if either operand is true
& or &&	True if both operands are true
~ or !	True if operand is false
( )	Sub-expression

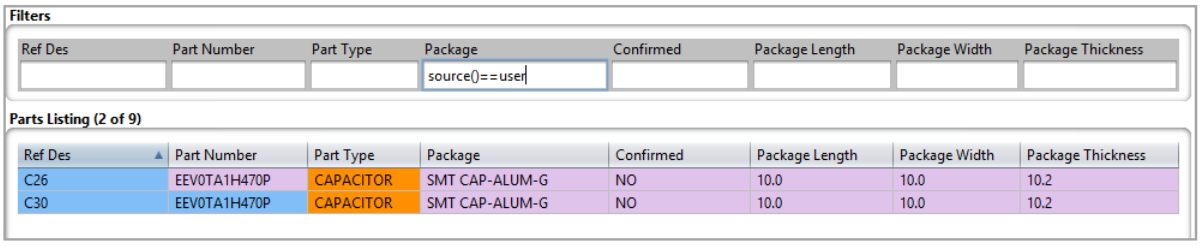
4.2.7.1.5. Column Filtering - Meta-Functions

All of the column filtering operators described above compute a result based on the property values in each row of the table. However, there are times when it is necessary to filter rows based on the metadata associated with a property value. To that end, the following meta-functions are supported in column filter expressions:

Table 4.4: Column Filtering Meta-Functions

Function	Description
defined()	Returns true if the property value is defined
confirmed()	Returns true if the property value has been confirmed
source()	Returns the data source of the property value

The following example shows how the meta-functions can be used to display all rows where the Package source is **User**."



Note:

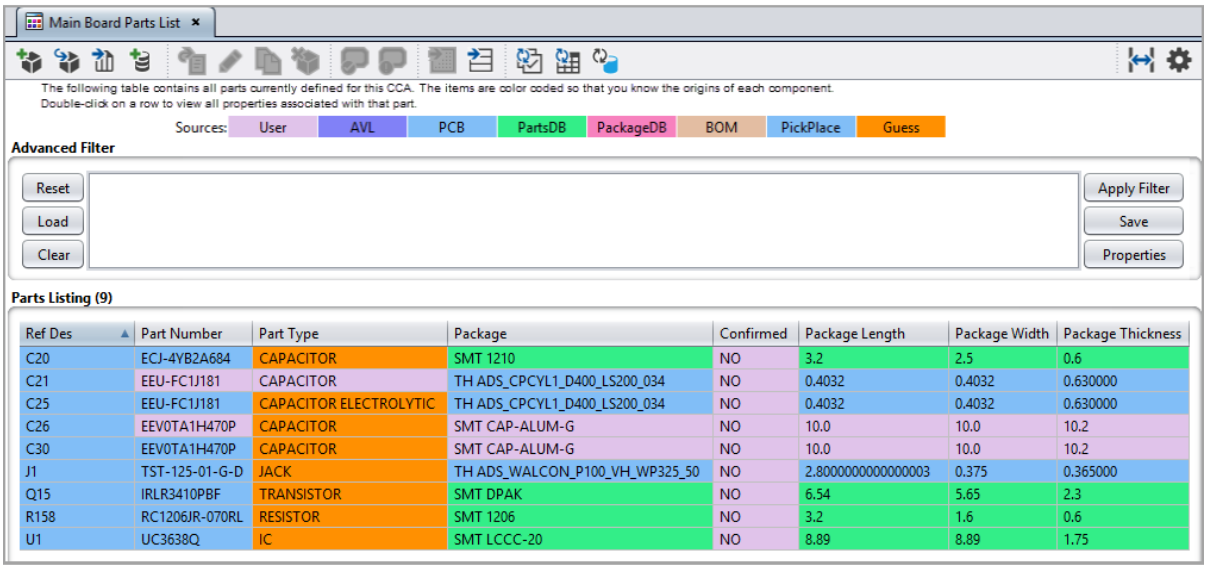
The use of meta-functions in filter expressions can quickly determine which parts in the **Parts List** need to be reviewed or updated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.7.2. Advanced Filter

The **Advanced Filter** allows users to define filter expressions that reference one or more part properties. Unlike Column Filter expressions, an Advanced Filter expression can define arbitrary conditions across multiple part properties and can reference part properties that are not listed in the Parts List table itself.

The following Parts List will be used as the basis for all of the examples in this subsection:



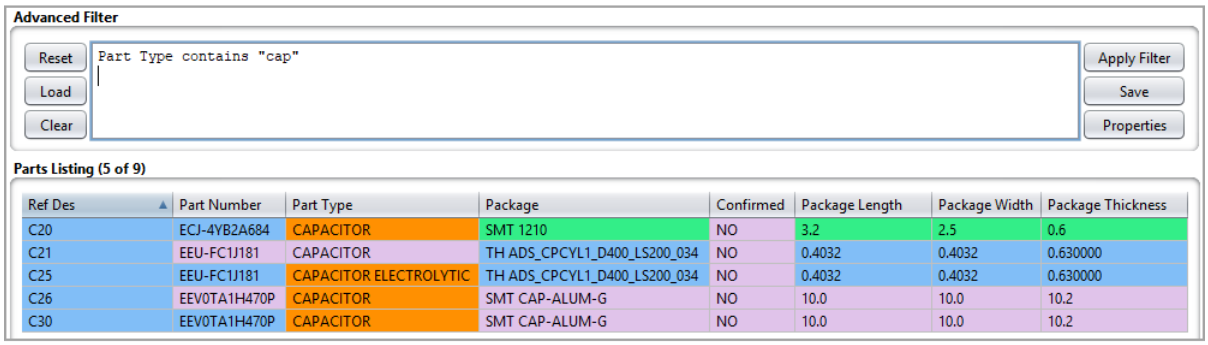
Ref Des	Part Number	Part Type	Package	Confirmed	Package Length	Package Width	Package Thickness
C20	ECJ-4YB2A684	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C21	EEU-FC1J181	CAPACITOR	TH ADS_CPCYL1_D400_LS200_034	NO	0.4032	0.4032	0.630000
C25	EEU-FC1J181	CAPACITOR ELECTROLYTIC	TH ADS_CPCYL1_D400_LS200_034	NO	0.4032	0.4032	0.630000
C26	EEV0TA1H470P	CAPACITOR	SMT CAP-ALUM-G	NO	10.0	10.0	10.2
C30	EEV0TA1H470P	CAPACITOR	SMT CAP-ALUM-G	NO	10.0	10.0	10.2
J1	TST-125-01-G-D	JACK	TH ADS_WALCON_P100_VH_WP325_50	NO	2.8000000000000003	0.375	0.365000
Q15	IRLR3410PBF	TRANSISTOR	SMT DPAK	NO	6.54	5.65	2.3
R158	RC1206JR-070RL	RESISTOR	SMT 1206	NO	3.2	1.6	0.6
U1	UC3638Q	IC	SMT LCCC-20	NO	8.89	8.89	1.75

Note:

The **Advanced Filter** is applied when the **Apply Filter** button is pressed. When editing the filter expression, you can press that button without using the mouse by pressing the **Tab** key, followed by the **Enter** key on the keyboard. For convenience, the focus will automatically return to the text area, allowing you to continue editing the filter expression if necessary.

4.2.7.2.1. Advanced Filter - String Matching

The **Advanced Filter** supports the same string-matching operators that are supported by the **Column Filters.** The only difference is that the part property name must always be specified. For example, the following filter expression displays all rows where the **Part Type** value contains the string **cap**.



Ref Des	Part Number	Part Type	Package	Confirmed	Package Length	Package Width	Package Thickness
C20	ECJ-4YB2A684	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C21	EEU-FC1J181	CAPACITOR	TH ADS_CPCYL1_D400_LS200_034	NO	0.4032	0.4032	0.630000
C25	EEU-FC1J181	CAPACITOR ELECTROLYTIC	TH ADS_CPCYL1_D400_LS200_034	NO	0.4032	0.4032	0.630000
C26	EEV0TA1H470P	CAPACITOR	SMT CAP-ALUM-G	NO	10.0	10.0	10.2
C30	EEV0TA1H470P	CAPACITOR	SMT CAP-ALUM-G	NO	10.0	10.0	10.2

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part properties may be designated in filter expressions using either the property name or label. A complete list of part properties can be found using the **Help > Part Properties** menu option in Sherlock or by pressing the **Properties** button on the right side of the **Advanced Filter** panel (described below).

All strings must be quoted in an **Advanced Filter** expression using either a pair of single quotes or a pair of double quotes.

The following string matching operators are supported in advanced filter expressions:

Table 4.5: String Matching Operators

Operator	Description
PROPERTY contains “STRING”	True if the PROPERTY value contains STRING
PROPERTY startsWith “STRING”	True if the PROPERTY value starts with STRING
PROPERTY endsWith “STRING”	True if the PROPERTY value ends with STRING
PROPERTY == “STRING”	True if the PROPERTY value equals STRING
PROPERTY =~ “REGEX”	True if the PROPERTY value satisfies the Java regular expression (case sensitive)
PROPERTY !~ “REGEX”	True if the PROPERTY value does not satisfy the Java regular expression (case sensitive)

Note:

See Java documentation for a complete description of regular expressions.

4.2.7.2.2. Advanced Filtering - Comparison Operators

The **Advanced Filter** supports the same comparison operators that are supported by the **Column Filters**. The only difference is that the part property name must always be specified. For example, the following filter expression displays rows where the **Package Length** is greater than 10, using numeric comparison.

```
packageLength > 10
```

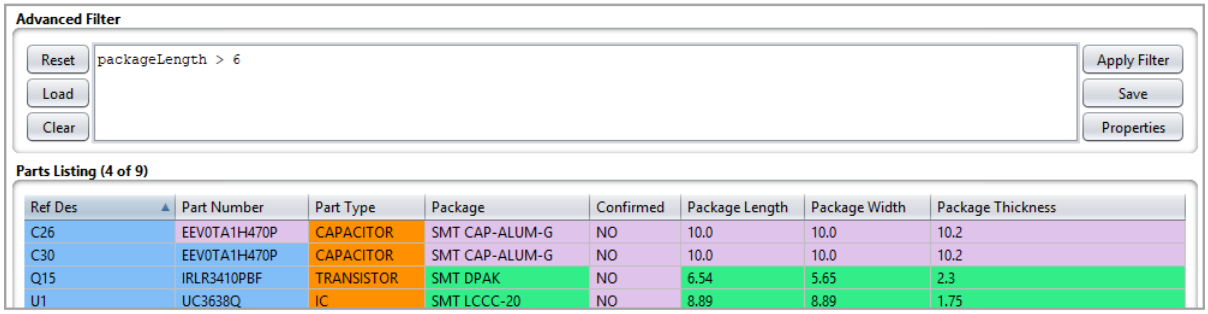
The following comparison operators are supported in advanced filter expressions:

Table 4.6: Comparison Operators

Operator	Description	Operator	Description
= or ==	Equals	!= or <>	Not Equals
<	Less Than	>	Greater Than
<=	Less Than Or Equals	>=	Greater Than Or Equals

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Comparison operators can be used to compare property values to literal values, property values to property values or literal values to literal values. In the following example, all parts with a package length greater than the package width are displayed.



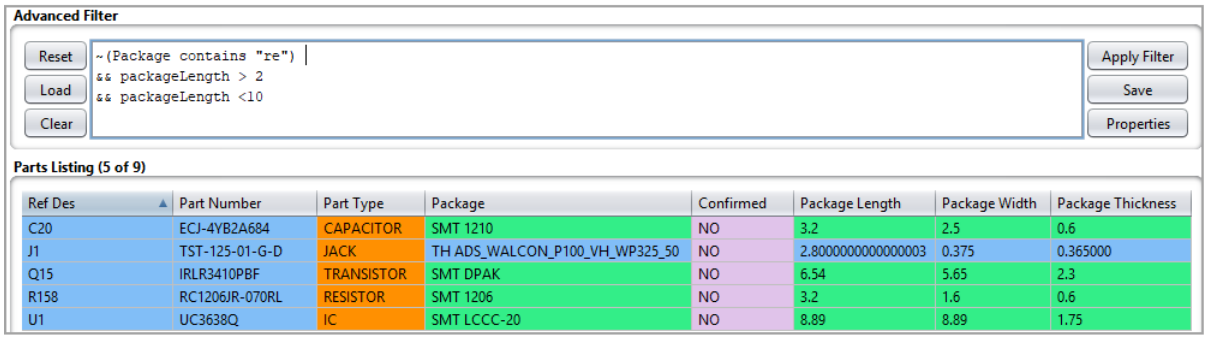
Ref Des	Part Number	Part Type	Package	Confirmed	Package Length	Package Width	Package Thickness
C26	EEV0TA1H470P	CAPACITOR	SMT CAP-ALUM-G	NO	10.0	10.0	10.2
C30	EEV0TA1H470P	CAPACITOR	SMT CAP-ALUM-G	NO	10.0	10.0	10.2
Q15	IRLR3410PBF	TRANSISTOR	SMT DPAK	NO	6.54	5.65	2.3
U1	UC3638Q	IC	SMT LCCC-20	NO	8.89	8.89	1.75

Note:

Comparison operators can be used for either a pair of numeric operands or a pair of string operands but cannot be used to compare a string operand to a numeric operand.

4.2.7.2.3. Advanced Filtering - Boolean Expressions

Boolean expressions can be defined in an advanced filter expression to facilitate more complex filtering criteria. In the following example, the filter selects values where (a) the Package values does not contain **re**, (b) the **Package Length** is greater than 2 and (c) the **Package Length** is less than 10.



Ref Des	Part Number	Part Type	Package	Confirmed	Package Length	Package Width	Package Thickness
C20	ECJ-4YB2A684	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
J1	TST-125-01-G-D	JACK	TH ADS_WALCON_P100_VH_WP325_50	NO	2.8000000000000003	0.375	0.365000
Q15	IRLR3410PBF	TRANSISTOR	SMT DPAK	NO	6.54	5.65	2.3
R158	RC1206JR-070RL	RESISTOR	SMT 1206	NO	3.2	1.6	0.6
U1	UC3638Q	IC	SMT LCCC-20	NO	8.89	8.89	1.75

The following boolean operators are supported in advanced filter expressions:

Table 4.7: Advanced Filtering Boolean Operators

Operator	Description
| or ||	True if either operand is true
& or &&	True if both operands are true
~ or !	True if operand is false
( )	Sub-expression

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.7.2.4. Advanced Filtering - Meta-Functions

All of the filtering operators described above compute a result based on the property values in each row of the table. However, there are times when it is necessary to filter rows based on the metadata associated with a property value. To that end, the following meta-functions are supported in advanced filter expressions:

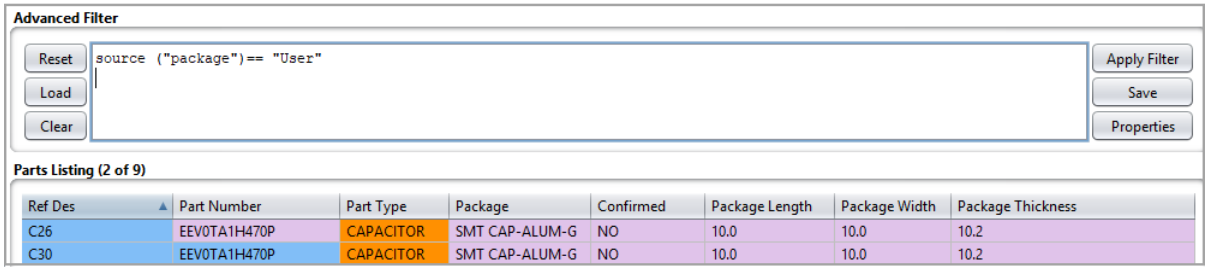
Table 4.8: Advanced Filtering Meta-Functions

Function	Description
defined("PROPERTY")	Returns true if the property value is defined
anyDefined("PROP1","PROP2", ...)	Returns true if any of the property values are defined
allDefined("PROP1","PROP2", ...)	Returns true if all of the property values are defined
confirmed("PROPERTY")	Returns true if the property value is confirmed
anyConfirmed("PROP1","PROP2", ...)	Returns true if any of the property values are confirmed
allConfirmed("PROP1","PROP2", ...)	Returns true if all of the property values are confirmed
source("PROPERTY")	Returns the data source of the property value

Note:

Unlike other operators and functions, the meta-functions require that all property names be specified as quoted strings.

The following example shows how the meta-functions can be used to display all rows where the **Package Source** is **User**.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.7.2.5. Advanced Filtering - Numeric Expressions

The following numeric operators are supported in advanced filter expressions to allow arbitrary calculations to be used as part of the filtering criteria:

Table 4.9: Advanced Filtering- Numeric Expressions

Operator	Description	Precedence
+, -	Addition and Subtraction	1
*, /	Multiplication and Division	2
%	Integer remainder	3
+, -	Unary Plus and Unary Minus	4
^	Power	5

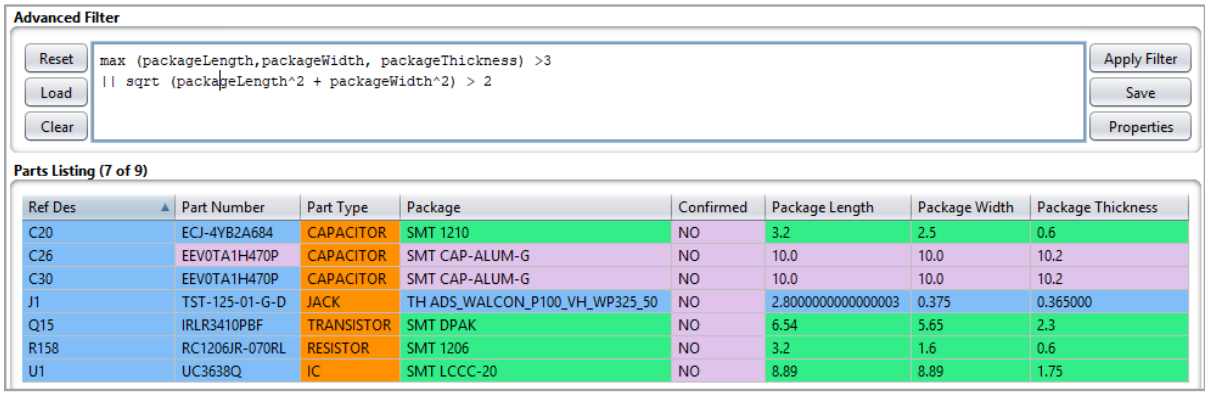
The numeric operators are listed from low precedence to high precedence. Higher precedence operations are performed first in an expression, absent any ordering specified by brackets.

The following numeric functions are supported in advanced filter expressions:

Table 4.10: Advanced Filtering- Numeric Functions

Function	Description
min(VALUE1, VALUE2, ...)	Returns the minimum of all values specified, which may be property values, constants or numeric expressions
max(VALUE1, VALUE2, ...)	Returns the maximum of all values specified, which may be property values, constants or numeric expressions
sqrt(VALUE)	Returns the square root of the given property value, constant or numeric expression

In the following example, the filter selects all rows where either the maximum package dimension (that is, length, width, thickness) is greater than 3 or the diagonal length of the package is greater than 3.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.7.2.6. Advanced Filtering - Filter Management

For convenience, the **Advanced Filter** edit panel maintains a history of the last 20 filter expressions that have been executed. This allows users to develop filter criteria in a piece-wise manner and/or to switch back and forth between a set of filters as needed.

Users may also save/load filters to/from designated text files. This allows users to easily maintain their own set of individualized filters and/or a library of re-usable filters that can be shared across a collection of Sherlock users.

The following buttons and/or keyboard Inputs can be used to manage filters on the **Filter History** list or saved in **user-defined files**:

Table 4.11: Advanced Filtering- Input Shortcuts

Button	Keyboard Input	Description
Apply Filter	CONTROL-F	Apply the currently defined filter and append it to the Filter History list
Reset	CONTROL-R	Reset the filter in the edit panel to that last filter that was executed
N/A	CONTROL-P	Load the previous filter, if any, on the Filter History list
N/A	CONTROL-N	Load the next filter, if any, on the Filter History list
Load	CONTROL-L	Load a saved filter expression from a designated text file
Save	CONTROL-S	Save the current filter expression to a designated text file
Clear	N/A	Clear the edit panel

Note:

The keyboard inputs listed in the table above are case-insensitive.

When a filter expression is saved to a designated text file Sherlock automatically includes a comment header that indicates the Windows username of the user who generated the file, as well as the date and time when the filter was saved, such as shown here:

```
# Sherlock Parts List Filter - Sam Somebody @ 2019-11-06 15:53:22
   Package Length > 10
&& Package Width > 10
```

Users may edit the text files directly to modify filter expressions and/or provide additional comments regarding the filter.

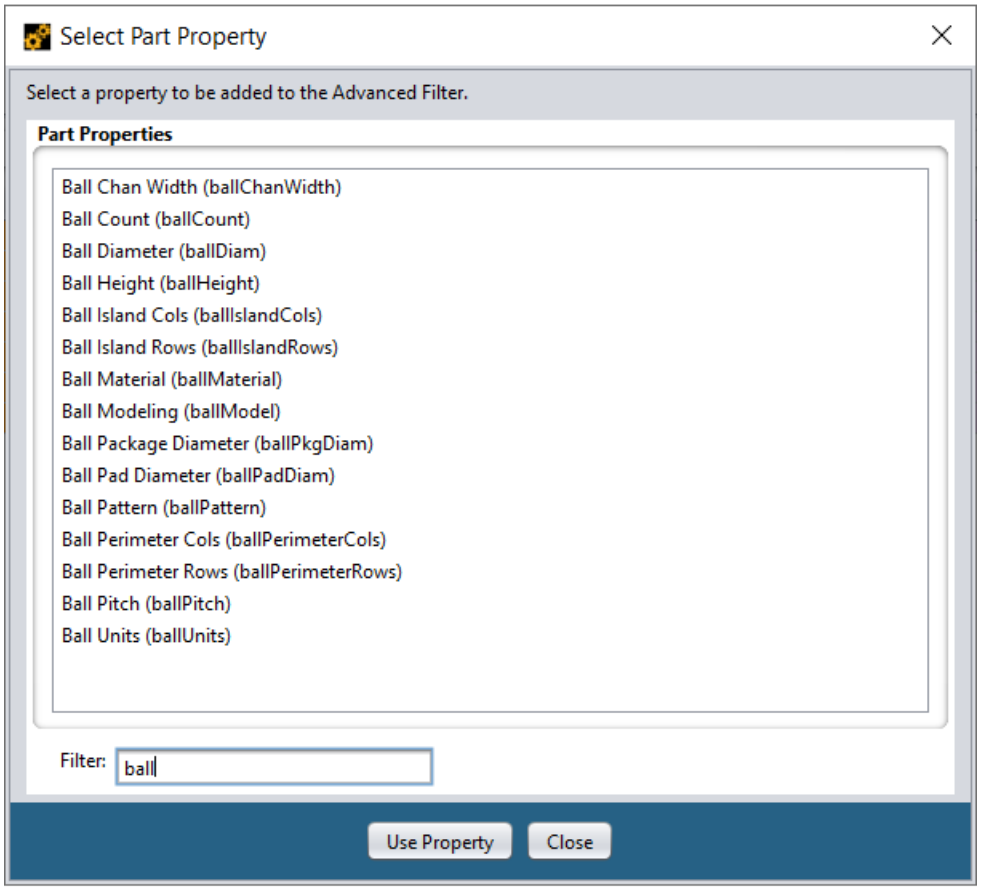
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.7.2.7. Advanced Filtering - Property Chooser

The **Select Part Property** dialog, accessible by pressing the **Properties** button in the Advanced Filter panel, provides a convenient mechanism for finding and entering a valid part property name.



As shown below, the dialog lists all part property labels and names, sorted by label. Users can scroll through the list to select a desired property to be included in the filter expression. When the **User Property** button is pressed, the selected property name will be appended to the current filter expression.



To help search through the list of properties, a filter expression can be entered below the list to display all list items that contain the given string. Such filtering is case-insensitive. Simply clear the filter field to display all part properties. In the example shown here, entering **ball** in the filter field results in the display of only the part properties associated with BGA solder balls.

For convenience, the dialog can be left open while editing the filter expression in the **Advanced Filter** panel so that multiple part properties can be entered as needed. When the dialog is no longer needed, press the **Close** button to hide the dialog until it is needed again.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.8. Exporting the Parts List

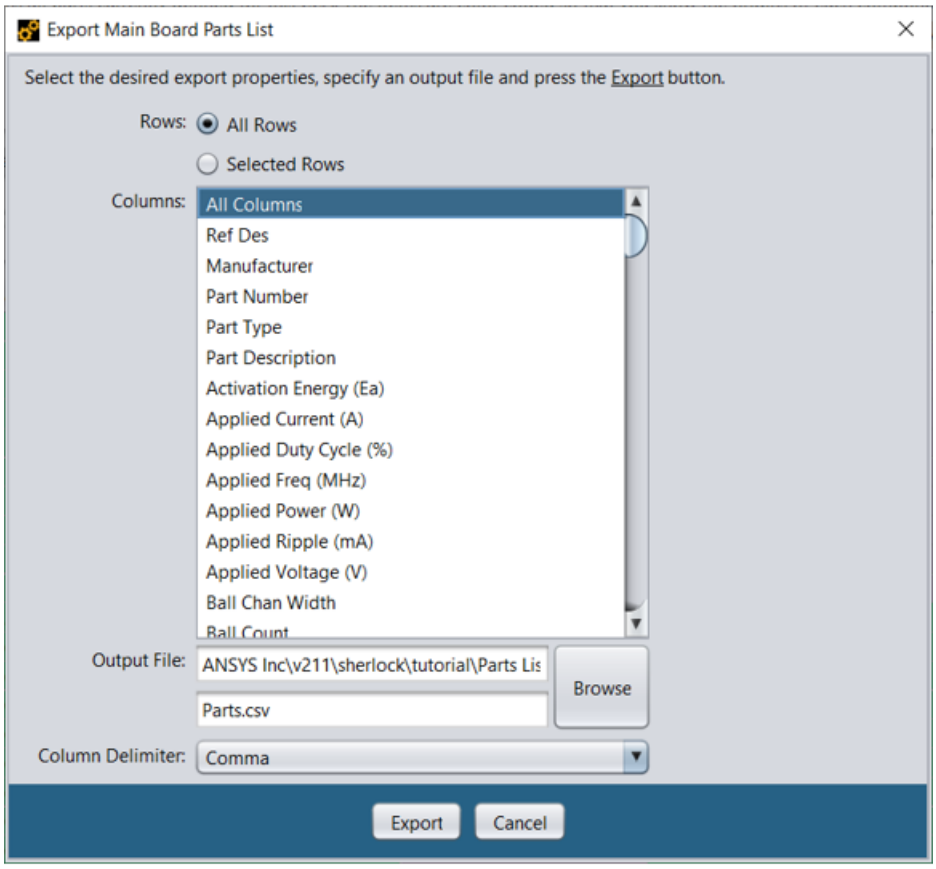
To see how the **Parts List** can be exported, double-click the **Parts List** entry in the Project Tree and select the entire **Parts List** for the **Main Board**. Right-click any selected part and select **Export Parts List** from the context menu or select the **Export Selected Part from the Parts List** button at the top of the Parts List dialog.

 Export selected part from the parts list

For this example, we will export all parts (rows) and all part properties (columns), as shown below. We can also choose the specific rows and/or columns to be exported to the file, depending on our needs.

Use the **Browse** button to locate the folder in which the exported file should be stored, name the file **Main Board Parts List.xlsx**, and press the **Export** button. If the file already exists, Sherlock will prompt you to confirm that the file should be overwritten.

Now that the file is exported we can make adjustments using any spreadsheet editor that can process Microsoft Excel 2010 (XLSX) files.



Note:

Sherlock can import and export Parts List files in the Excel 2010 format (XLSX), Excel 97/2003 format (XLS) and the Comma Separated Values (CSV) format. Sherlock automatically determines the file format based on the specified file name suffix.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.9. Importing the Parts List

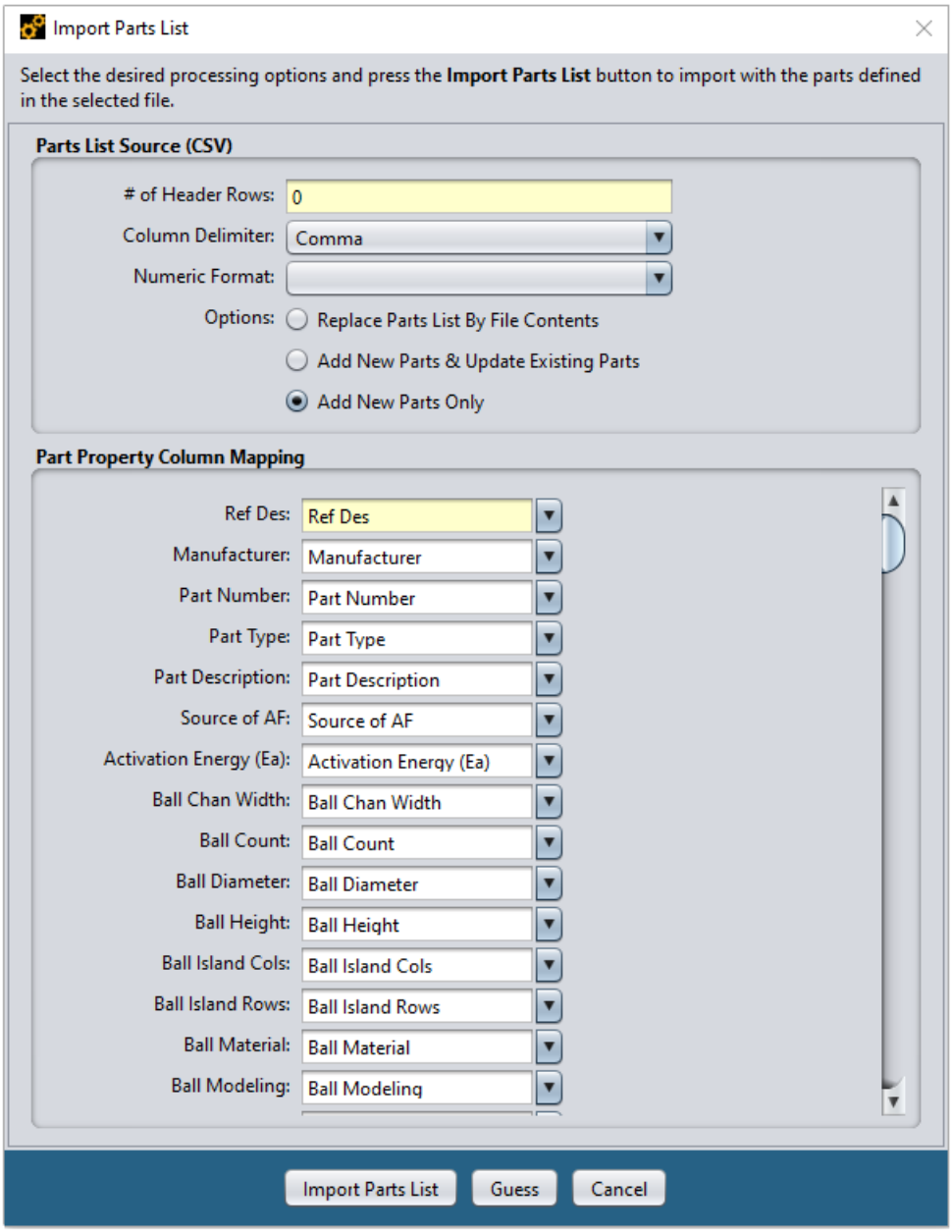
To continue our editing example, make some changes in the spreadsheet that we just exported. For example, as in the Reviewing Part Properties (p. 30)tutorial, we can make changes to the part shape by modifying the **Corner Shape** and **Corner Radius** values. Once the necessary adjustments have been made, the modified Parts List can be re-imported. Right-click the **Parts List** in the Project Tree and select **Import Parts List** from the context menu. At that point, a file dialog will appear, allowing you to locate and select the file to be imported. For convenience, the last imported/exported file will be used as the default, but you are free to select any file on your system.

4.2.9.1. Import Options

After selecting the file, the **Import Parts List** dialog will appear (shown below), allowing you to specify how Sherlock should process the data, with the following options supported:

- Replace Parts List By File Contents
- Add New Parts and Update Existing Parts
- Add New Parts Only

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use the first option if you wish to delete the current **Parts List** and use the imported spreadsheet data instead. The second option will update existing parts and add new parts from the spreadsheet that do not exist in the **Parts List.** If you just want to add missing parts, use the third option.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.9.2. Column Mappings

Next you will need to verify the column mapping for each file/sheet that is imported. Sherlock scans each file in an attempt to determine the number of header rows and the column names. Such guesses are usually right, but you should check them anyway by viewing the CSV file in the content panel or Excel.

In addition to guessing the column names, Sherlock also attempts to guess which columns should be mapped to Sherlock part properties. In this tutorial, the spreadsheet file uses some standard column names, so Sherlock successfully guessed how to map all the columns. In real-world examples, lots of different and unusual column names are used, so you may have to select the desired mapping yourself.

When you have the column mappings correct, press the **Import Parts List** button to import the data.

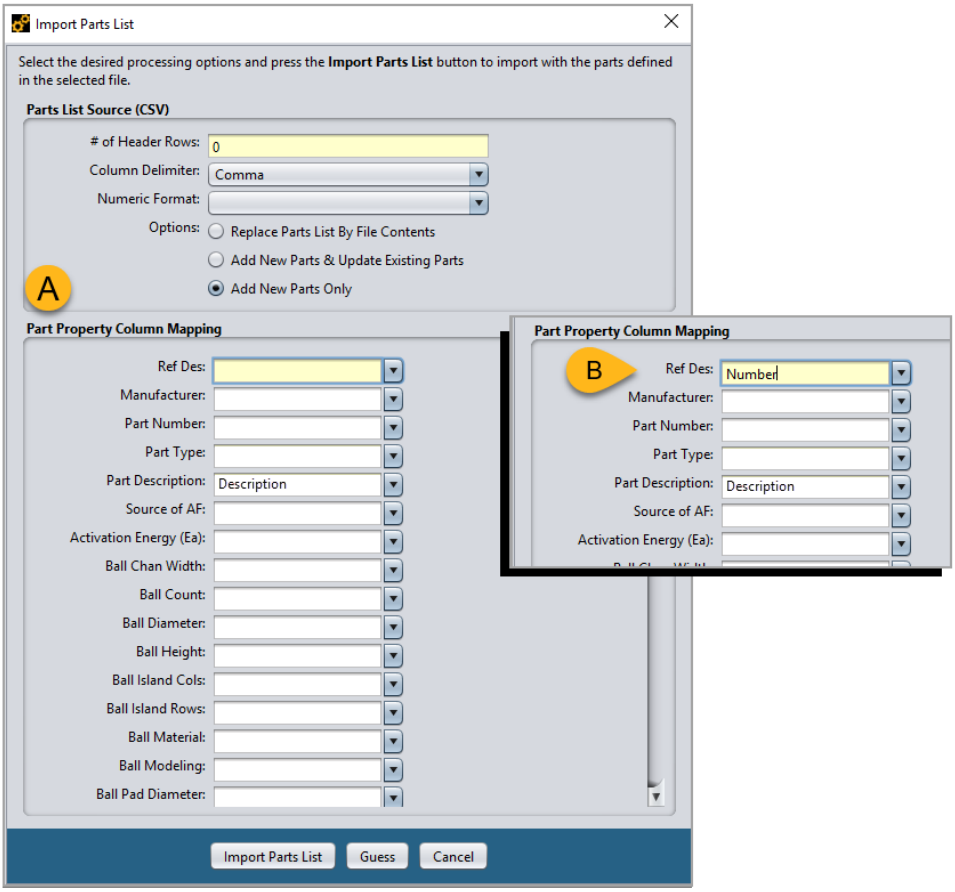
When the import process is complete, you will see a confirmation dialog.

4.2.9.3. Addressing Import Errors

In the event that Sherlock cannot process part of the **Parts List** file, details of the problems will be shown on the **Parts List – Issues** tab. To see how Sherlock handles **Parts List** import problems, let's import the **External Parts List.csv** file located in the **tutorial/Parts List** sub-folder of the Sherlock Installation Directory (usually **c:/Program Files/Ansys Inc/v251/sherlock**).

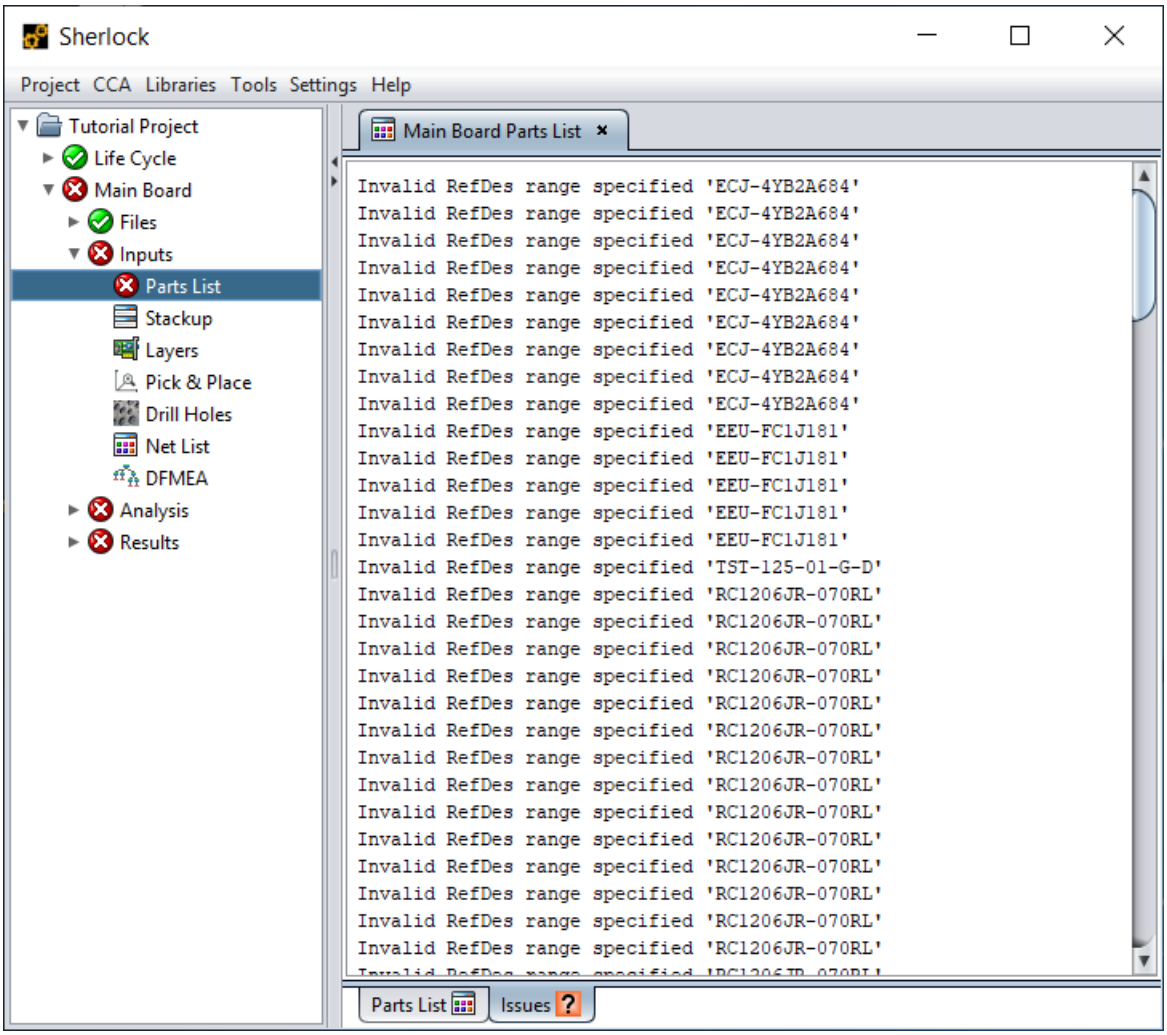
As shown below (A), Sherlock can correctly guess column mappings for the **Part Description** and some other optional properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



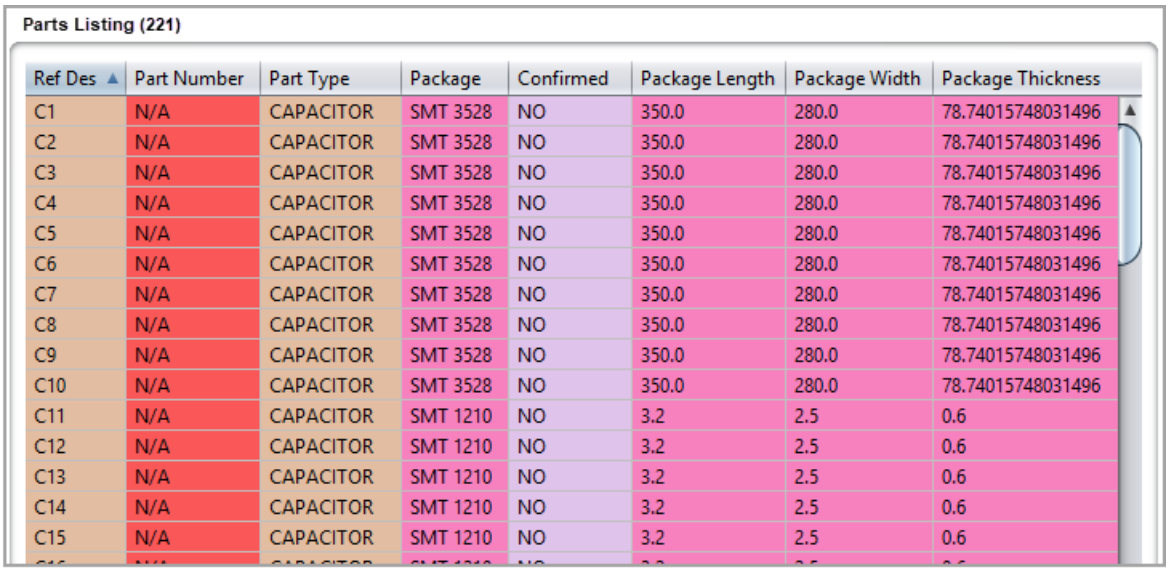
When the **Import Parts List** button is pressed, the import process will generate a list of errors that can be viewed by right-clicking the **Parts List** entry in the **Project Tree** and selecting **Show Error** from the context menu. As shown below, Sherlock attempts to use the **Number** column values as reference designators, but since they are not in the proper format, it generates an error for each row processed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By examining the spreadsheet contents, we can determine that the **Ref Des** property should actually map to **ID** column. So, let's import the file again and map the **ID** column to the **Ref Des** property. This time Sherlock successfully imports the data from the file (in that it didn't generate any error messages), but we still need to confirm that all the data is correct. By examining the **Parts List** content panel:

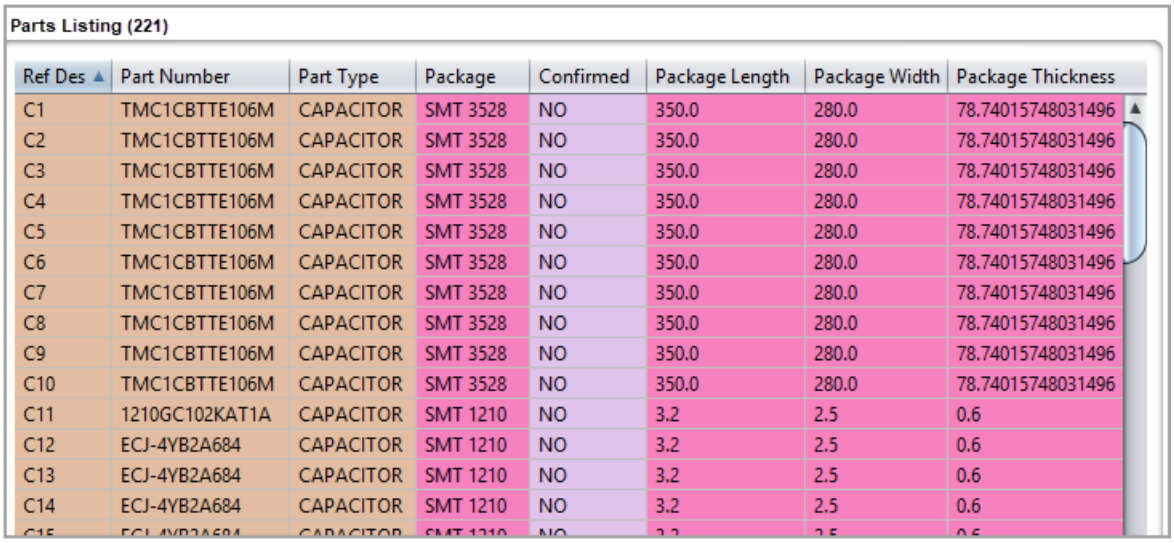
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parts Listing (221)

Ref Des	Part Number	Part Type	Package	Confirmed	Package Length	Package Width	Package Thickness
C1	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C2	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C3	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C4	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C5	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C6	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C7	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C8	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C9	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C10	N/A	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C11	N/A	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C12	N/A	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C13	N/A	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C14	N/A	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C15	N/A	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6

...we see that the **Ref Des, Part Type, Packaging** and **Location** information is correct for each of the imported parts, but that no Part Number data is defined. By examining the spreadsheet again, we see that the **Number** column contains the required data, but it wasn't imported. So, let's import the file one last time, this time mapping the **Number** column to the **Part Number** property. Finally, we see that all the desired data has been imported.



Parts Listing (221)

Ref Des	Part Number	Part Type	Package	Confirmed	Package Length	Package Width	Package Thickness
C1	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C2	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C3	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C4	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C5	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C6	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C7	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C8	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C9	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C10	TMC1CBTTE106M	CAPACITOR	SMT 3528	NO	350.0	280.0	78.74015748031496
C11	1210GC102KAT1A	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C12	ECJ-4YB2A684	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C13	ECJ-4YB2A684	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6
C14	ECJ-4YB2A684	CAPACITOR	SMT 1210	NO	3.2	2.5	0.6

Note:

Whenever part data is imported using the **Import Parts List** feature you will need to re-confirm the part properties after reviewing their correctness.

4.3. Building Custom Parts

Sherlock has a vast and growing library of pre-built electronic parts for populating the circuit boards in your Sherlock project. But for those times you must build your own part, the *Sherlock Parts Build*

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Guide provides the information and tips to make the job easier. The topic is lengthy enough to deserve its own document. Click this link to navigate to the Sherlock Parts Build Guide.

4.4. Part Library

One of the critical tasks performed by any Ansys Sherlock user is that of reviewing and verifying the correctness of the data gathered for each part being analyzed. Without accurate input data, the analysis results generated by Sherlock will not be accurate or useful. The **Parts List** maintained for each circuit card is the primary data store for analysis purposes, containing part-specific data for each part used in a given circuit card. To facilitate part data management, Sherlock allows users to retrieve and store part property values in a part library. The Sherlock parts library includes:

- **Sherlock Part Library** – A database that supports SQL queries and updates. This part library is read only and available to all users.
- **User Created Library** – A customizable database that supports SQL queries and updates. Multiple libraries can be created, and ReadOnly mode can be implemented.

In simplest terms, a **Part library** is a collection of part data that has already been verified by you or another trusted Sherlock user and that can be easily used in one or more Sherlock projects. All part libraries share a common GUI in Sherlock to reduce user confusion, while providing different levels of data sharing amongst Sherlock users. Property values retrieved from any part library can be used to easily update the **Parts List** for a specific circuit card, ensuring data consistency across a collection of circuit cards. Property values defined in a given Parts List can also be easily saved to one or more part libraries, where allowed.

Sherlock's part libraries are implemented with SQLite. A SQLite part library is in the public domain and is a lightweight database that supports SQL queries and updates. If the database file is stored on a Windows shared drive, it can be easily accessed by a group of Sherlock users on the same organizational network. These libraries can be accessed directly by Sherlock or can be accessed using a variety of public-domain desktop and command line tools providing maximum flexibility for both naïve and expert database users.

The remainder of this document describes in detail how the Sherlock **Parts Library** can be used to share part data across multiple circuit cards and across multiple Sherlock users.

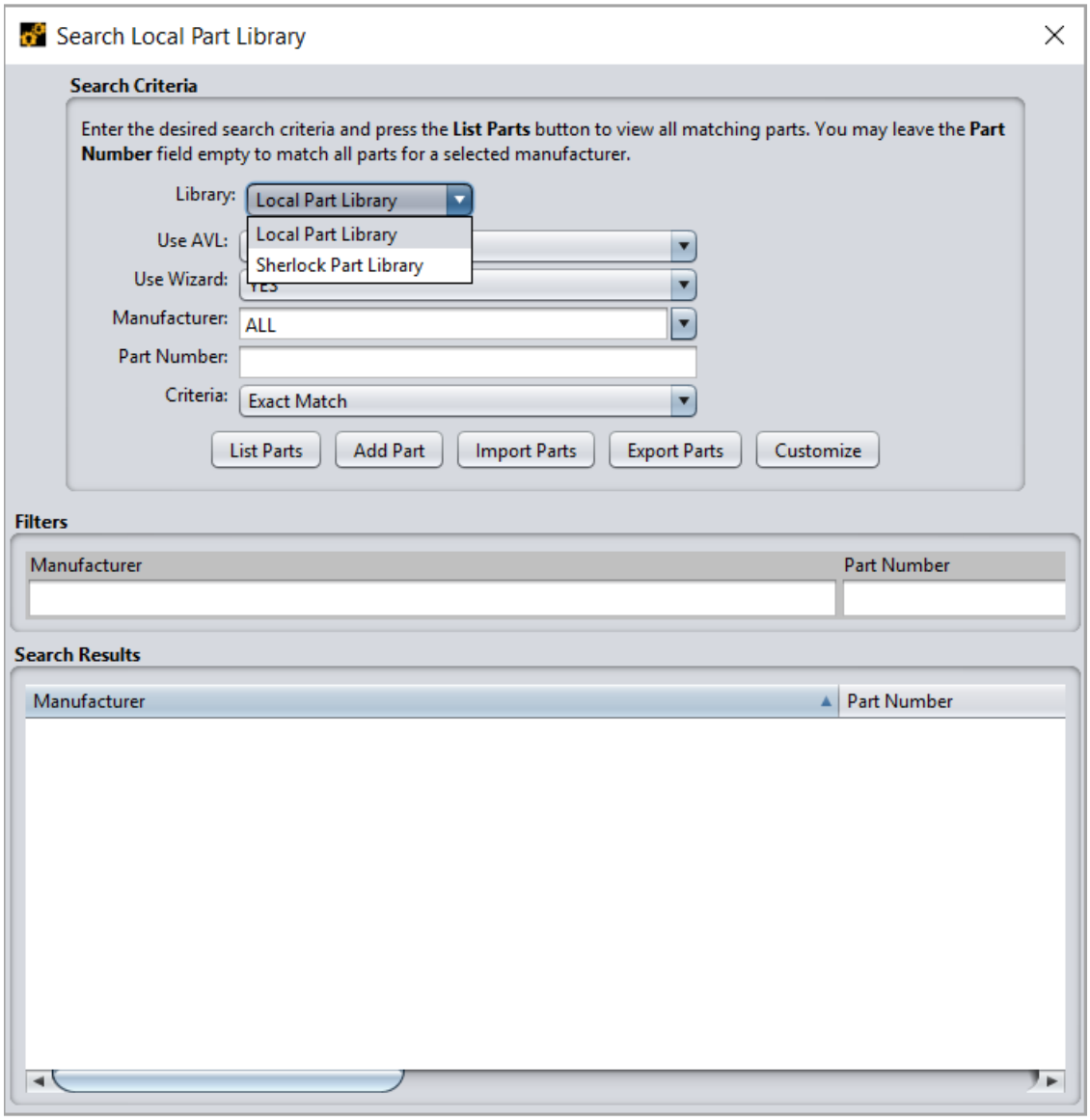
In this section, the following topics are covered:

4.4.1. Parts Library Menu
4.4.2. Creating a Part Library
4.4.3. Part Library Management
4.4.4. Customized Search Results
4.4.5. Filtered Search Results
4.4.6. Update Part Library from Parts List
4.4.7. Update Parts List from Part Library
4.4.8. Import Part Data
4.4.9. Recover Part Library

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.1. Parts Library Menu

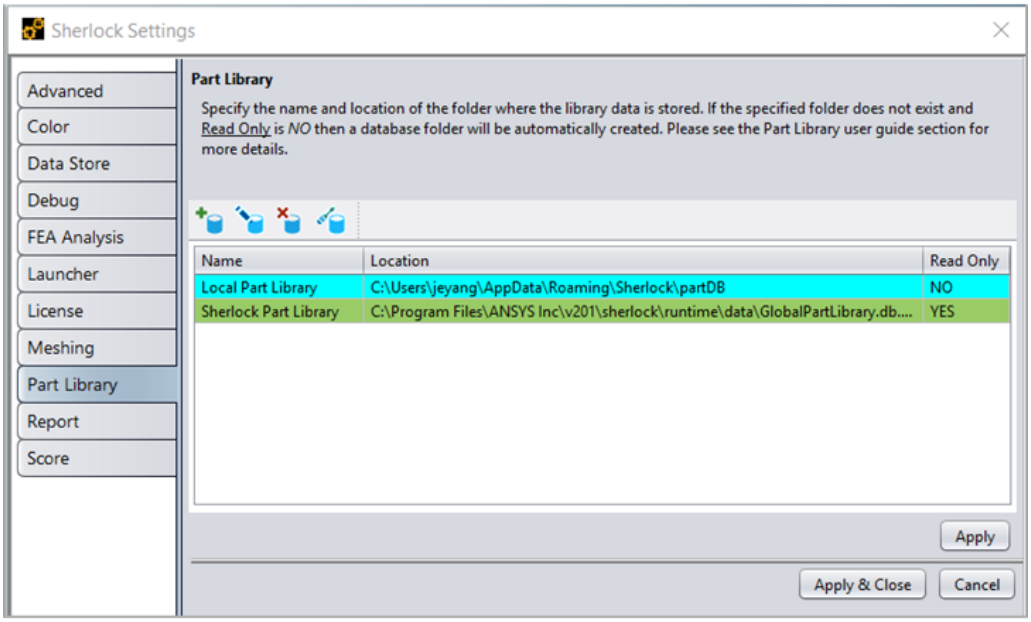
The Parts Library can be accessed and managed by selecting **Libraries** on the Sherlock **Main Menu** and choosing **Parts.** Upon opening the Parts Library, you can select either a user created library, in this case **Local Part Library** or **Sherlock Part Library** from the Library drop-down menu. **Local Part Library** exists only for users who have upgraded from a previous version of Sherlock, and contains data stored in a previous local part library.



4.4.2. Creating a Part Library

To create a part library, select **Settings > Part Library** from the **Main Menu**. This will prompt the **Sherlock Settings** to open with the **Part Library** tab active. The **Part Library** tab displays all of the Part Libraries that currently exist in Sherlock.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this case, the Part Libraries that exist in Sherlock are the **Local Part Library** which is *not* read only, and the **Sherlock Part Library** which *is* read only.

On the **Part Library** page, there are several buttons:

 Add a New Part Library.

 Edit the Selected Part Library.

 Delete the Selected Part library.

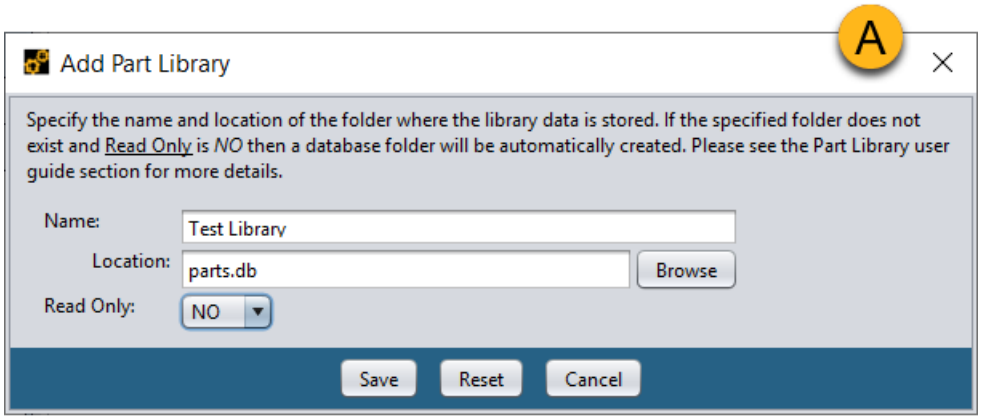
 Recover the Selected Part Library.

You can also right-click a Part Library listed and choose **Edit Selected Part Library**, **Delete Selected Part Library**, or **Recover Selected Part Library**.

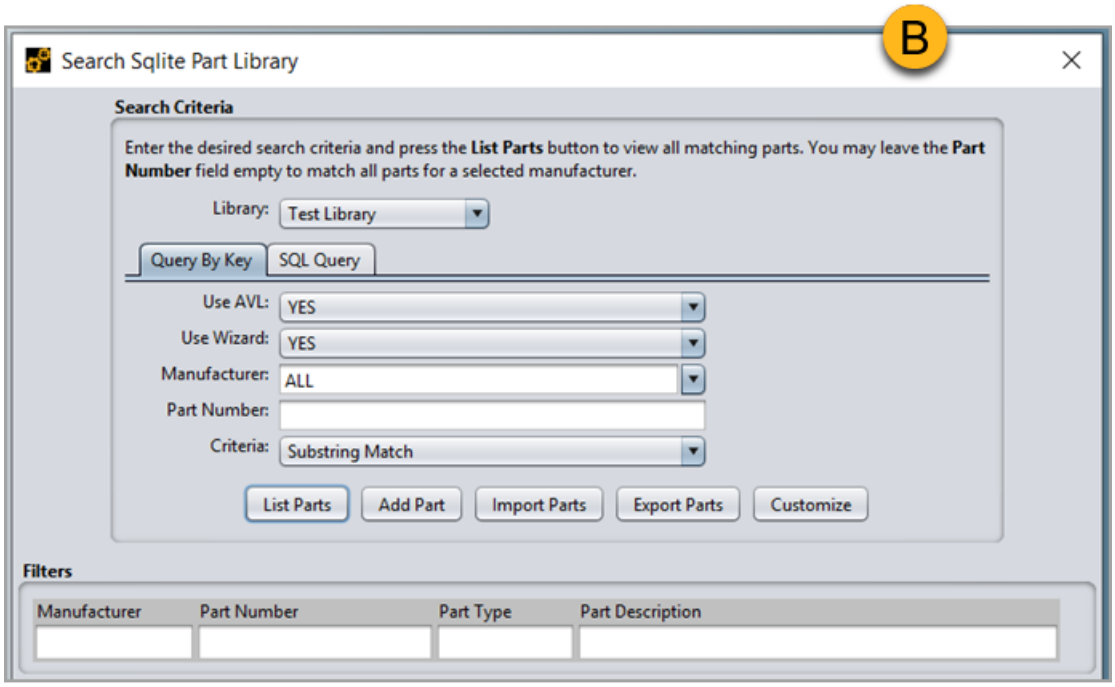
To create a new part library, press the **Add a New Part Library** button. A new pop-up window (image A, below) will appear asking you to enter in a **Name**, **Location**, and specify if the new part library is read only.

Press the **Browse** button to select the location and name of the database file in the file system or enter the data manually. If the database file and/or folder does not exist, you can create a new folder and name a database file which will be created. In this case, the new library is named **Test Library**, and the location of where the library data is stored is specified in the **Location** field. Afterwards, press **Save**, then **Apply & Close**. The new library is now created.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To view the new library, select **Libraries > Parts** from Sherlock's **Main Menu**. You can now toggle the **Library** drop down menu and select the new library (B). In this case, the newly created **Test Library** can be selected.



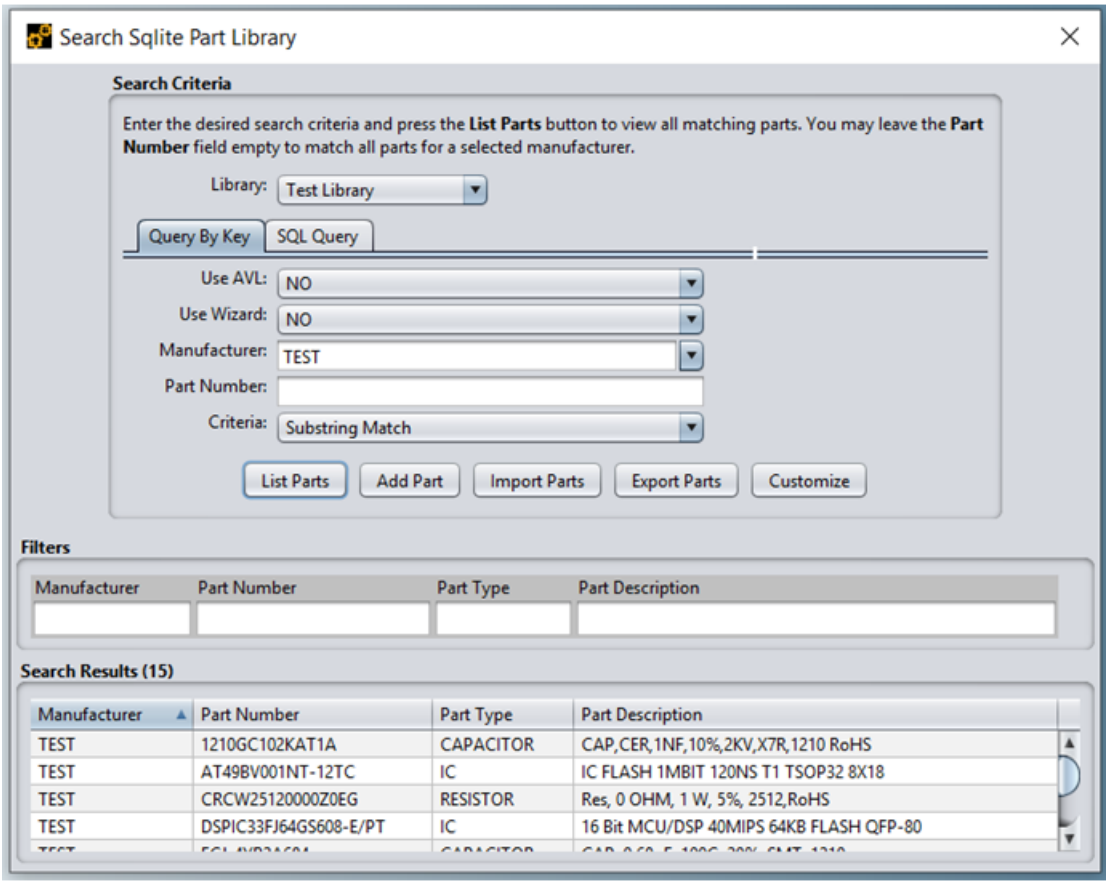
4.4.3. Part Library Management

Since part libraries may contain a large number of parts, a search dialog is provided to select a subset of parts that can then be viewed, edited, exported or deleted. Select **Libraries > Parts** from the **Main Menu** to display the search dialog.

Note:

While the following section uses the Test Library previously created as an example library, the process to update, add, and delete parts is the same for any library that is not read only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



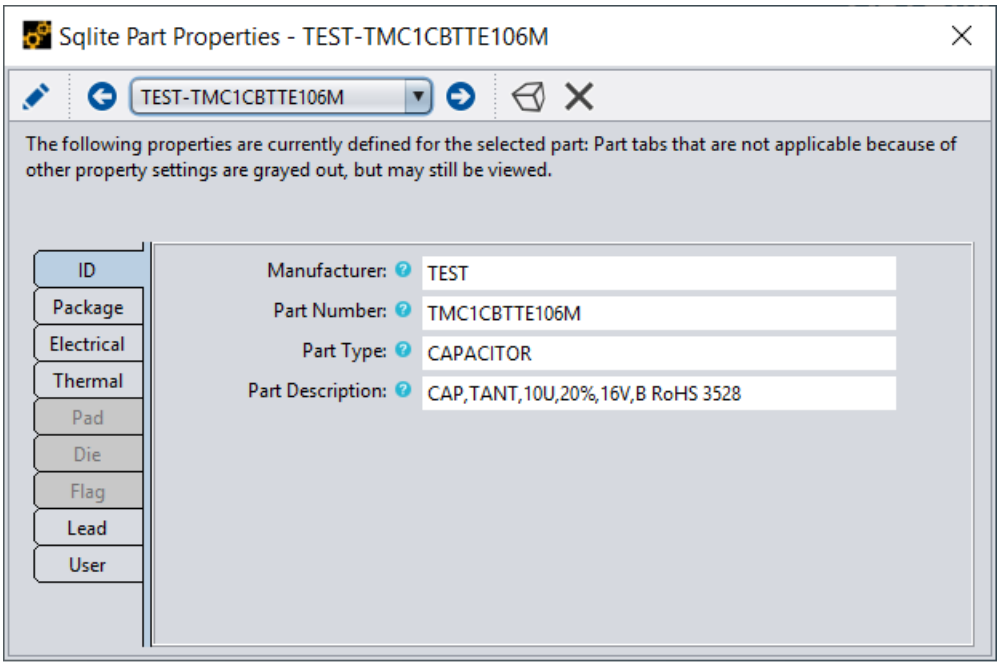
The dialog contains both a **Search Criteria** form and a set of **Filters**. The Search Criteria form is used to search for all parts from a selected manufacturer or search for parts with a specific part number or both. The **Use AVL** and **Use Wizard** choices specify if the **Approved Vendor List** and/or **Part Wizard** should be used when searching for parts. If **Use AVL** is enabled, then all the part numbers listed in the AVL will be searched for the given Part Number string. If **Use Wizard** is enabled and an **Exact Match** criteria is specified, then the Part Wizard will be used to attempt to match the given part number, automatically adding the part data to the Local Part Library if a match occurs.

After entering the desired criteria, press the **List Parts** button to populate the Search Results table. The rows in the Search Results list can be filtered further by entering strings to be matched in one or more of the filter fields. Customization and advanced filtering of the Search Results table is also supported. See the Customized Search Results (p. 239)and Filtered Search Results (p. 243) subsections below for more details and examples.

4.4.3.1. Viewing Part Properties

To view all the properties stored for a given part in the **Part Library**, simply double-click the corresponding row in the Search Results list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



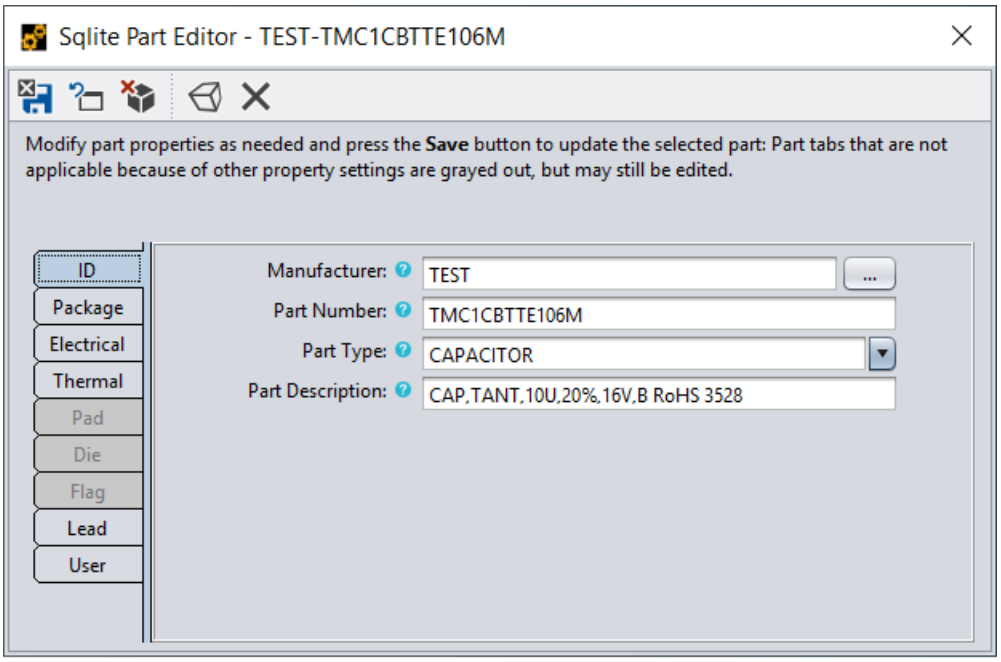
If you select multiple rows in the **Search Results** list, right-click any of the selected rows and select **View Selected Parts** from the context menu. The ribbon at the top of the page will include navigation buttons that allow you to view each of the selected parts. Specifically, the arrows on the ribbon allow you to view previous or next parts. You may also choose to utilize the drop-down menu which allows you to manually select which parts from the selected parts you would like to view. You can also select a specific **Manufacturer-Part Number** from the pull-down list to go to a specific part record.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.3.2. Editing Properties for a Single Part

To edit property values stored in a part library, right-click the desired part in the **Search Results** list and select **Edit Selected Part** from the pull-down menu to display the **Part Editor**. Optionally, you may also double-click to open the part properties, and then select the **pencil icon** to edit.

 Edit the selected part



The **Part Editor** looks similar to the **Part Properties** dialog, except the read-only fields have been replaced by editable fields, allowing you to modify any properties associated with this part. If you are unsure about any part property, click the blue question mark icon next to a field to see a definition of each property. After changing one or more properties, press the **Save** button to update the library. Additionally, you can reset the properties to the last saved properties, or even delete the selected part.

 Save the part editor changes and exit the editor.

 Reset the form to the currently saved values

 Delete the selected part

4.4.3.3. The Part Editor and Package Properties

The **Package Name** and **Package Type** properties play an important role in the **Part Editor**. Here are a few things you should know:

- Sherlock allows you to choose the **Package Type** independently of the **Package Name**. It's important to use this feature carefully. It is possible to end up with a package in your **Parts List**

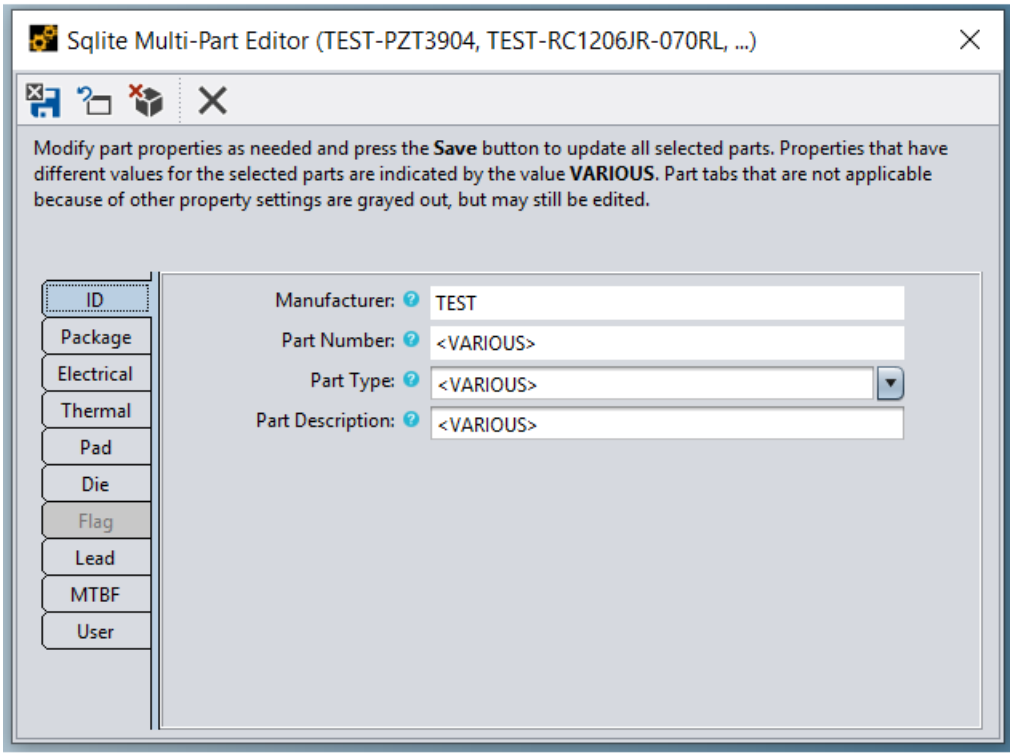
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with a *different* **Package Type** property than the same package in your **Package Library**. See Package Management (p. 315) for more on the **Package Library**.

- When using the **Part Editor** to add a new part to the **Parts List**, Sherlock will guess the **Package Type** when you first enter the **Package Name**. If you do not agree with Sherlock's choice, you can manually change the **Package Type**.
- The **Package Type** property determines which fields and tabs are active in the **Part Editor**.
- If you use the **Package Chooser** to select a package, then Sherlock will update the part's package properties based on the package you selected. See The Package Chooser (p. 206).

4.4.3.4. Editing Properties for Multiple Parts

Similar to the **Parts List**, you can edit properties for multiple parts by selecting them in the **Search Results** list, right-clicking any selected row and choosing **Edit Selected Parts** from the context menu to display the **Multi-Part Editor.**



If a property value is the same for all selected parts, then that value will be displayed in the editable field. Otherwise, <VARIOUS> will be displayed, indicating that there are at least two different values stored for that property among the selected parts. Simply modify one or more property values in any of the tabs and press **Save** to update all the selected parts.

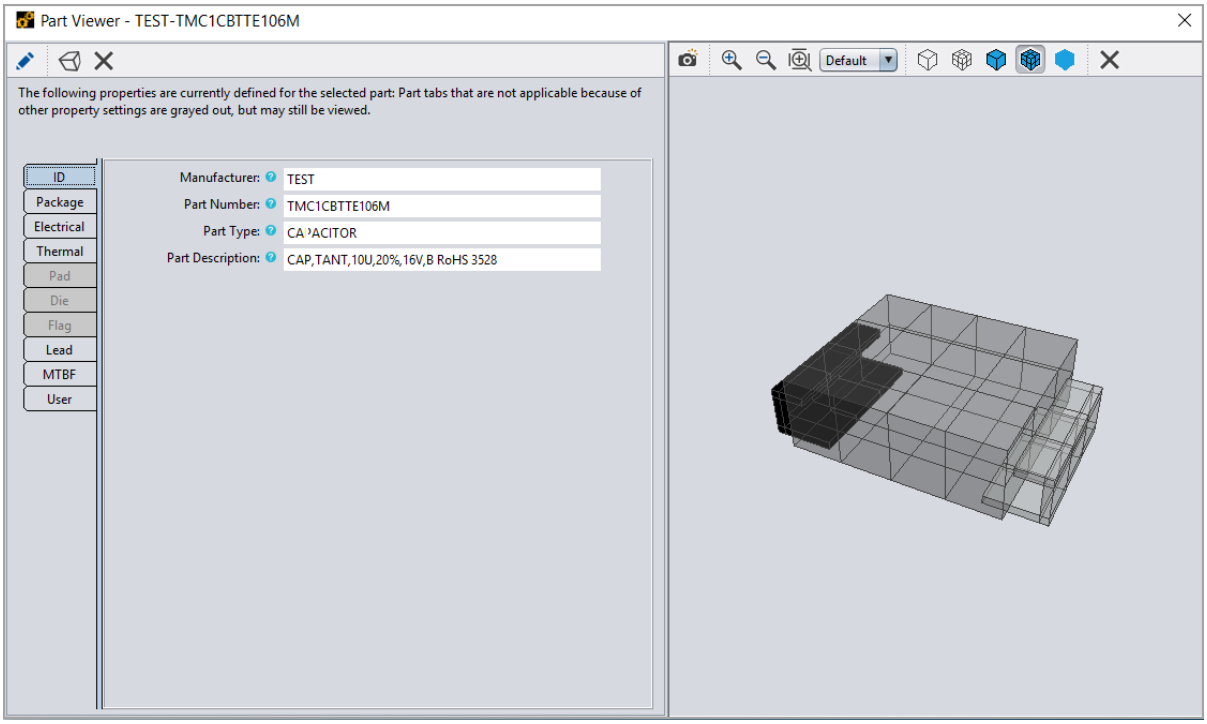
4.4.3.5. Part Visual Previewer

The 3D visual representation of any part can be viewed when view the part properties or in the part editor. At the top of the part viewer ribbon is a button labeled  **View the 3D Model of the Part**." Click this button to view the 3D visual representation of this part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As shown below, the window presented for viewing the 3D visual representation has similar controls to that of the **Sherlock 3D Viewer**. You may pan, rotate and zoom the display using simple mouse controls. Press the **Help** button for a complete list of the mouse controls supported.

The ribbon above the model allows you to select the various viewer options similar to options available in the **Sherlock 3D Viewer** (see Viewing and Managing Analysis Results (p. 785) for more details).



4.4.3.6. Package and Lead Properties

The **Package** tab and **Lead** tab presents the same form as is displayed in the package properties form and lead properties form in the part editor. To edit items in these forms, select the pencil in the upper left-hand corner. To see your changes, re-select **View the 3D Model of the Part**. Part properties do not have to be saved in order to be viewed in the 3D modeler. This allows you to make part modifications from within the previewer tool and optionally save them back to the part editor itself.

Note:

There are two properties which appear in the **Lead** tab of the **Part Properties Window** but are *not* tracked in the Part Library: **Lead Bend Package Radius** and **Lead Bend Radius**. Sherlock uses these two properties only when Advanced Lead Meshing is enabled. See the section Advanced Lead Meshing (p. 566).

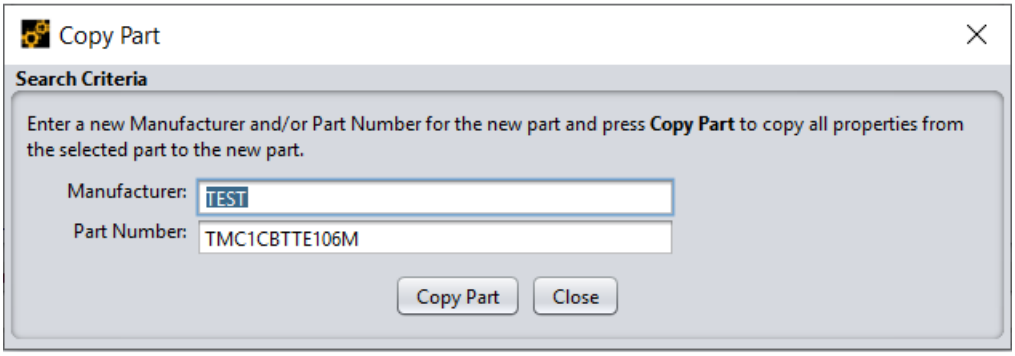
4.4.3.7. Adding, Copying or Deleting Parts

In addition to viewing and editing parts, the search dialog allows you to add, copy or delete parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Press the **Add Part...** button in the Search Criteria form to add a new part. At that point, the Part Editor will be displayed, allowing you to enter specific part properties. The **Manufacturer** and **Part Number** values must be unique in order for the new part to be added to the library.

To copy an existing part and create a new part with the same properties, right-click the part in the Search Results list and choose **Copy Selected Part** from the context menu. At that point, a dialog will appear requiring you to enter the **Manufacturer** and/or **Part Number** values for the new part.



Once again, the **Manufacturer** and **Part Number** values must be unique for the new part to be added. All other part properties will be copied from the selected part and you can modify them using the standard part editing capabilities described above. For convenience, after creating the new part, the **Copy Part** dialog will remain displayed, allowing you to repetitively add more parts using the same part data. This allows for efficient manual adding of parts. When you're done adding all the parts, press the **Close** button to close the **Copy Part** dialog.

To delete one or more parts from the **Local Part Library**, select them in the **Search Results** list, right-click any selected part and choose **Delete Selected Parts** from the context menu. You may also delete parts by pressing the **Delete the Selected Part** button in the **Part Editor** ribbon or the **Delete the Selected Parts** button in the Multi-Part Editor ribbon.

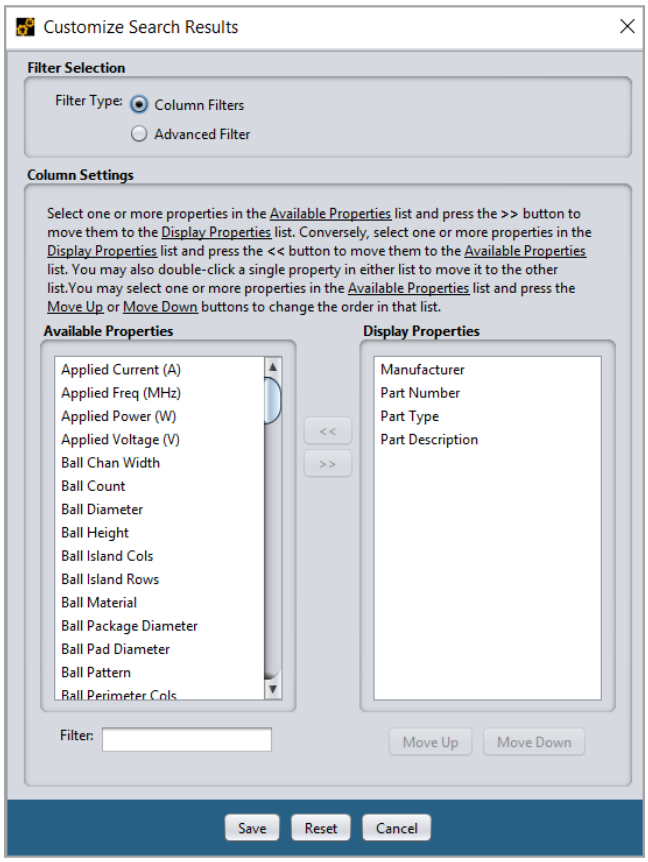
 Delete the selected part

4.4.4. Customized Search Results

Users can customize the list of part properties that are displayed in the search results table to suit their needs. Users can also customize the width of individual columns as needed. Sherlock automatically saves the table settings and reloads them whenever the search dialog is displayed.

To begin the customization process, press the **Customize** button displayed in the lower-right corner of the **Search Criteria** panel. At that point, the **Customize Search Results** dialog will be displayed, as shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Filter Type** setting allows you to select the type of filtering for the table, as discussed in the following subsections.

The two lists in the dialog allow you to select the part properties to be displayed in the table. All the part properties defined in the right-hand list will be displayed as table columns. You can move one or more items from one list to the other by selecting them and pressing either the **<<** or **>>** buttons to move them between the lists.

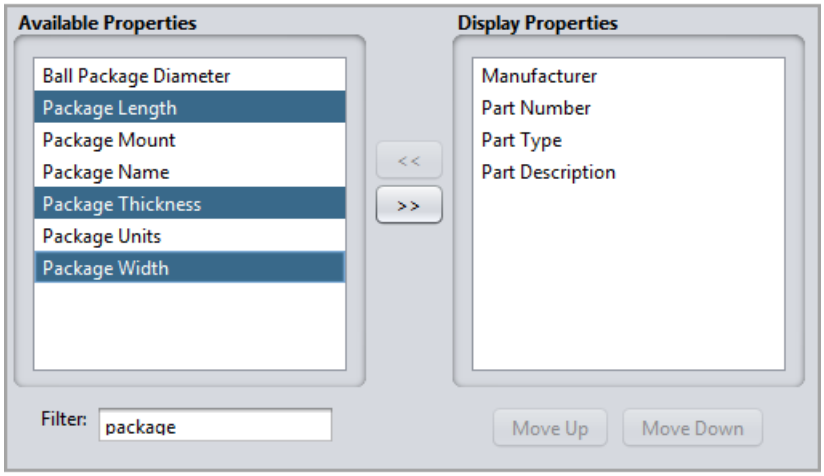
The **Move Up** and **Move Down** buttons below the right-hand list can be used to reorder the properties to be displayed in the table. Simply select one or more properties in the right-hand list and press **Move Up** or **Move Down** to change their relative position in the list.

For convenience, the **Filter** field below the left-hand list can be used to filter the property names shown in the left-hand list.

The following series of images shows how the customization works.

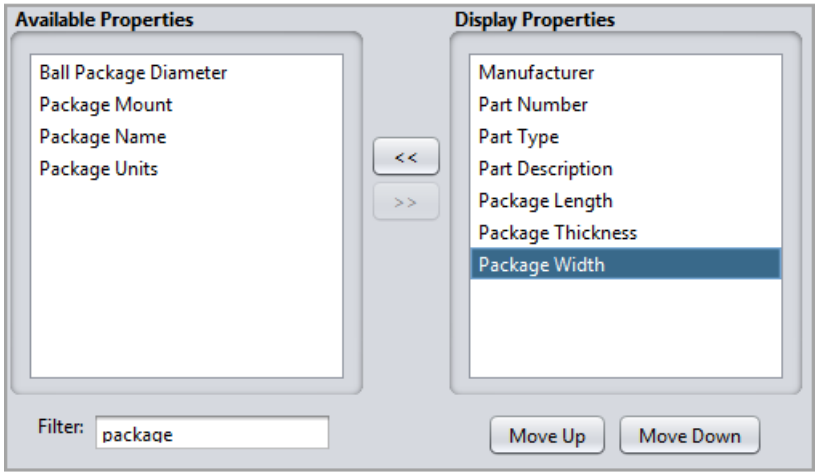
To add some properties to the table, we begin by entering **package** in the **Filter** field to reduce the number of items shown in the **Available Properties** list. Then, we select the **Package Length**, **Package Thickness** and **Package Width** properties from the left-hand list. Finally, we press the **>>** button to move those properties from the left-hand list to the right-hand **Display Properties** list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To remove a property from the table, we select the property in the right-hand **Display Properties** list and press the **<<** button. At that point, the property will be moved from one list to the other.

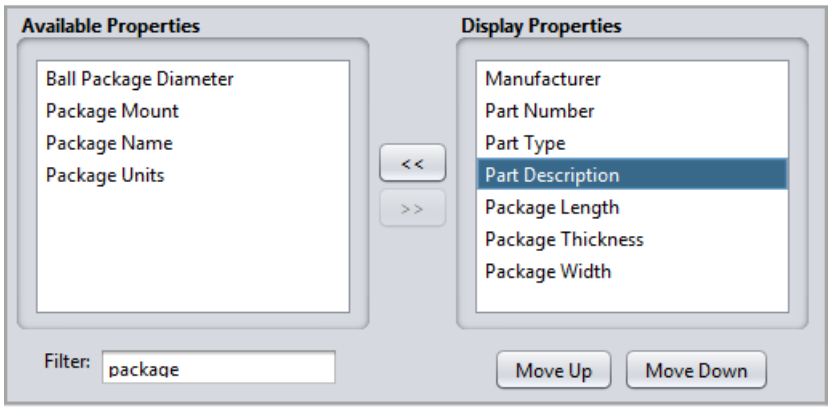
Note, since the **Filter** is still set to **package**, the **Part Description** property will not be displayed in the **Available Properties** list until the **Filter** is changed accordingly.



Finally, to change the order of the properties displayed in the Parts List we select the **Package Width** property and press the **Move Up** button. At that point, the **Package Width** property will be moved above **Package Thickness**. If we press the button again, the property will be moved up another position.

If multiple properties are selected, then each of them will be moved up/down by one position whenever the **Move Up** or **Move Down** buttons are pressed.

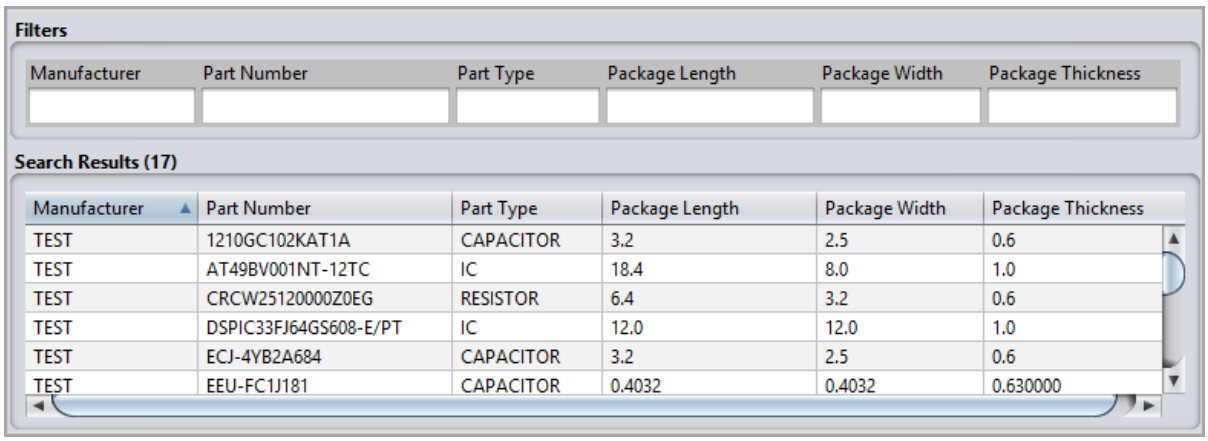
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The **Manufacturer** and **Part Number** properties must always be included in the **Display Properties** list, although they may appear in any order.

After making changes in the **Customize Search Results** dialog, press the **Save** button to update the Search Results table. Sherlock will automatically save the Displayed Properties list and will use it to configure the table whenever it is displayed. Based on the example changes above, the customized table will look like this.



If the number of properties displayed in the table is wider than the viewing panel, a horizontal scrollbar will appear, allowing you to scroll left and right. The column filter fields displayed above the table will automatically scroll to stay aligned with the table columns.

Individual table column widths can be changed by dragging the divider between each column header. Sherlock will automatically change the width of the corresponding column filter field. Sherlock will also save the width of each column and will configure the table accordingly whenever it is displayed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5. Filtered Search Results

Sherlock provides two types of expression-based filtering for the **Search Results** table:

- Column Filters
- Advanced Filter

The type of filter is specified using the **Filter Type** choice list in the **Customize Search Results** dialog (described in the previous subsection).

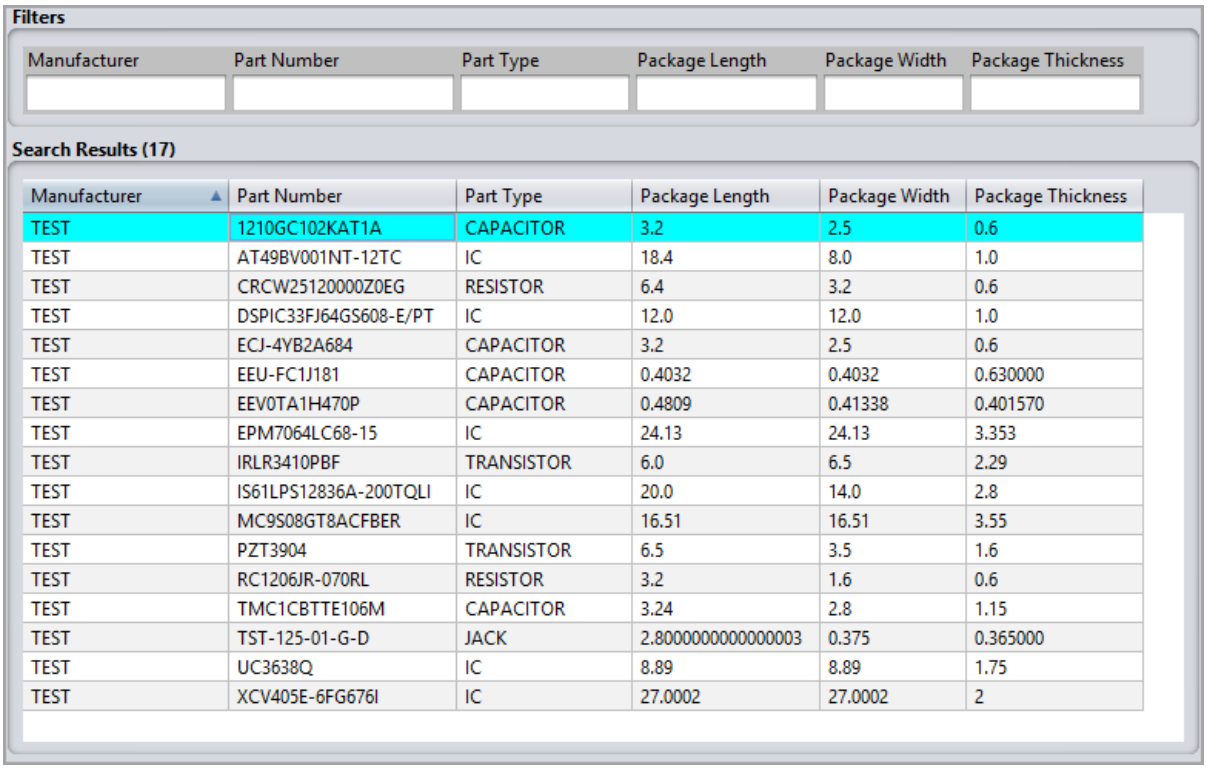
Column Filters allow users to enter individual filter expressions for any of the columns displayed in the table. They are an easy and effective method for filtering the Search Results with a minimal amount of data entry.

The **Advanced Filter** allows users to enter simple or complex filter expressions based on one or more part properties. Such filter expressions require more data entry and formatting than those used in **Column Filters**, but they allow for greater control over which table rows are filtered.

4.4.5.1. Column Filters

Column Filters allow users to enter simple filter expressions that are applied to each of the values in each table column.

The following **Search Results** will be used as the basis for all the examples in this subsection:


Filters

Manufacturer	Part Number	Part Type	Package Length	Package Width	Package Thickness

Search Results (17)

Manufacturer ▲	Part Number	Part Type	Package Length	Package Width	Package Thickness
TEST	1210GC102KAT1A	CAPACITOR	3.2	2.5	0.6
TEST	AT49BV001NT-12TC	IC	18.4	8.0	1.0
TEST	CRCW25120000Z0EG	RESISTOR	6.4	3.2	0.6
TEST	DSPIC33FJ64GS608-E/PT	IC	12.0	12.0	1.0
TEST	ECJ-4YB2A684	CAPACITOR	3.2	2.5	0.6
TEST	EEU-FC1J181	CAPACITOR	0.4032	0.4032	0.630000
TEST	EEV0TA1H470P	CAPACITOR	0.4809	0.41338	0.401570
TEST	EPM7064LC68-15	IC	24.13	24.13	3.353
TEST	IRLR3410PBF	TRANSISTOR	6.0	6.5	2.29
TEST	IS61LPS12836A-200TQLI	IC	20.0	14.0	2.8
TEST	MC9S08GT8ACFBER	IC	16.51	16.51	3.55
TEST	PZT3904	TRANSISTOR	6.5	3.5	1.6
TEST	RC1206JR-070RL	RESISTOR	3.2	1.6	0.6
TEST	TMC1CBTTE106M	CAPACITOR	3.24	2.8	1.15
TEST	TST-125-01-G-D	JACK	2.8000000000000003	0.375	0.365000
TEST	UC3638Q	IC	8.89	8.89	1.75
TEST	XCV405E-6FG676I	IC	27.0002	27.0002	2

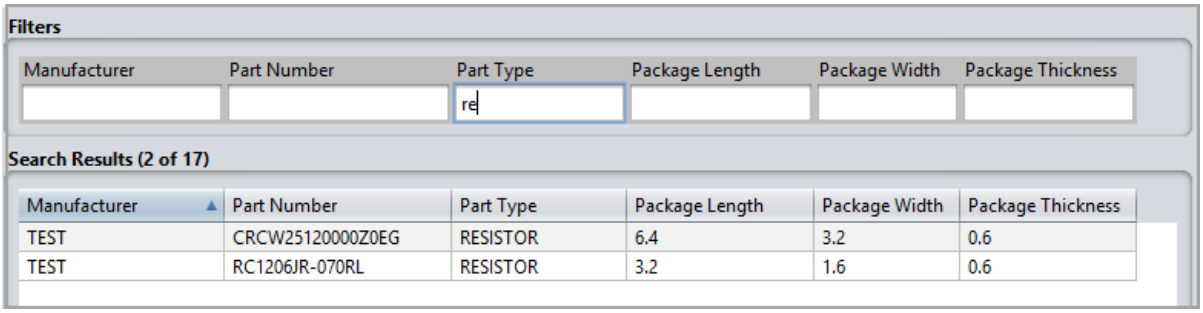
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Column Filters are applied whenever you press the **Tab** or **Enter** key inside of a filter field. The **Tab** key will move the focus to the next filter field, while the **Enter** key will keep the focus on the same field.

4.4.5.1.1. Column Filter - String Matching

In the simplest form, a string is entered as the filter expression and all rows with cell values that contain the given string are displayed. For example, as shown below, the re value entered in the Part Type filter field results in 2 of the 17 rows being displayed in the table.



Note:

For convenience, filter expressions are case-insensitive unless otherwise noted. Also, strings may be quoted or un-quoted.

In addition to the default string matching behavior, the following filter expressions may be used for string matching:

Table 4.12: Column Filter String Matching Operators

Operator	Description
contains STRING	True if the column value contains STRING
startsWith STRING	True if the column value starts with STRING
endsWith STRING	True if the column value ends with STRING
= STRING	True if the column value equals STRING
$STRING	True if the column value starts with STRING
STRING$	True if the column value ends with STRING
$STRING$	True if the column value equals STRING
=~ REGEX	True if the column value satisfies the Java regular expression (case sensitive)
!~ REGEX	True if the column value does not satisfy the Java regular expression (case sensitive)

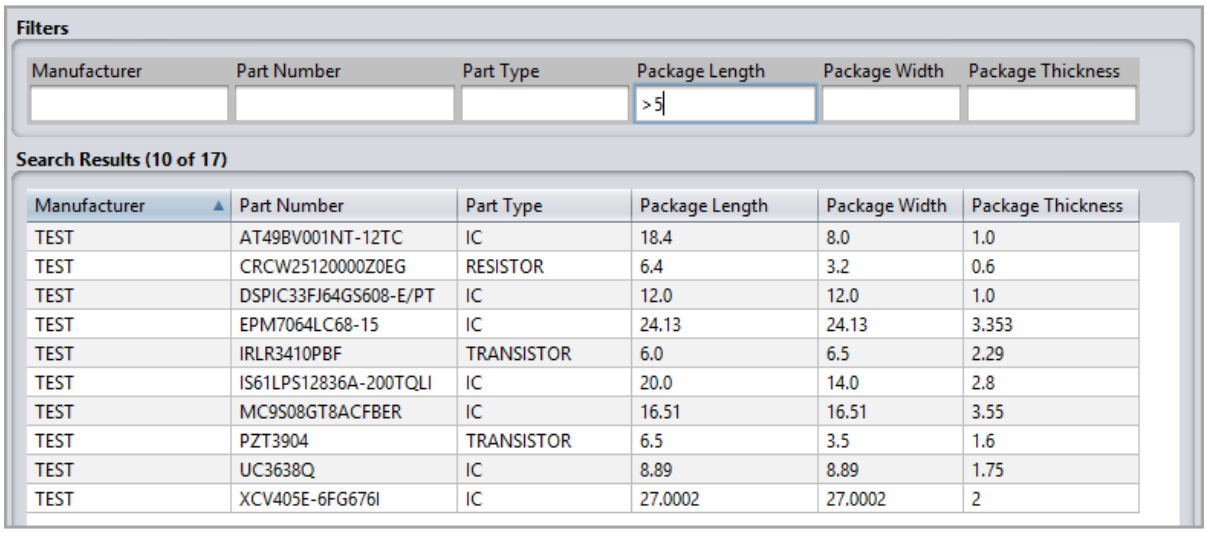
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Regular expressions must always be quoted because they may contain special characters that would otherwise be interpreted as part of the filter expression. See Java documentation for a complete description of regular expressions.

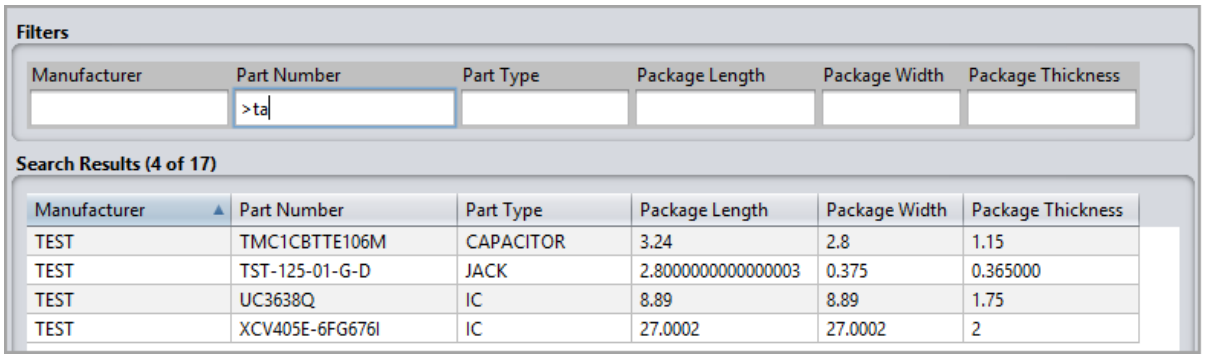
4.4.5.1.2. Column Filtering - Comparison Operators

In addition to the string-matching operators described above, filter expressions can make use of various comparison operators for both string and numeric values. For example, the following filter expression displays all rows that have a **Package Length** value greater than five. In this case, since the **Package Length** values are numeric, a numeric comparison is performed by the **greater than** operator.



Manufacturer	Part Number	Part Type	Package Length	Package Width	Package Thickness
TEST	AT49BV001NT-12TC	IC	18.4	8.0	1.0
TEST	CRCW25120000Z0EG	RESISTOR	6.4	3.2	0.6
TEST	DSPIC33FJ64GS608-E/PT	IC	12.0	12.0	1.0
TEST	EPM7064LC68-15	IC	24.13	24.13	3.353
TEST	IRLR3410PBF	TRANSISTOR	6.0	6.5	2.29
TEST	IS61LPS12836A-200TQLI	IC	20.0	14.0	2.8
TEST	MC9S08GT8ACFBER	IC	16.51	16.51	3.55
TEST	PZT3904	TRANSISTOR	6.5	3.5	1.6
TEST	UC3638Q	IC	8.89	8.89	1.75
TEST	XCV405E-6FG676I	IC	27.0002	27.0002	2

In the following example, the **greater than** operator performs an alpha-numeric comparison, resulting in the display of rows where the Part Number value is greater than **ta**.



Manufacturer	Part Number	Part Type	Package Length	Package Width	Package Thickness
TEST	TMC1CBTTE106M	CAPACITOR	3.24	2.8	1.15
TEST	TST-125-01-G-D	JACK	2.8000000000000003	0.375	0.365000
TEST	UC3638Q	IC	8.89	8.89	1.75
TEST	XCV405E-6FG676I	IC	27.0002	27.0002	2

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

An alpha-numeric comparison differs from a simple string comparison in that it compares numeric substrings as numbers. For example, the value **SMT 0603** is greater than **SMT 100**.

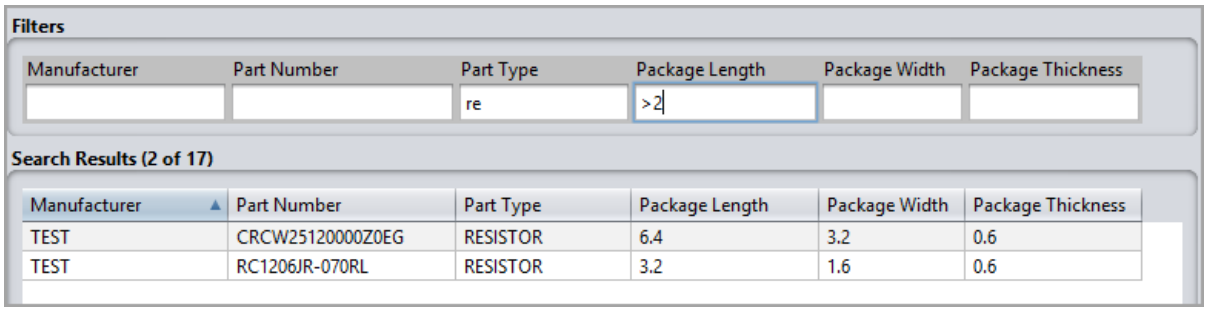
The following comparison operators are supported in column filter expressions for both numeric and string values:

Table 4.13: Column Filter Comparison Operators

Operator	Description	Operator	Description
= or ==	Equals	!= or <>	Not Equals
<	Less Than	>	Greater Than
<=	Less Than Or Equals	>=	Greater Than Or Equals

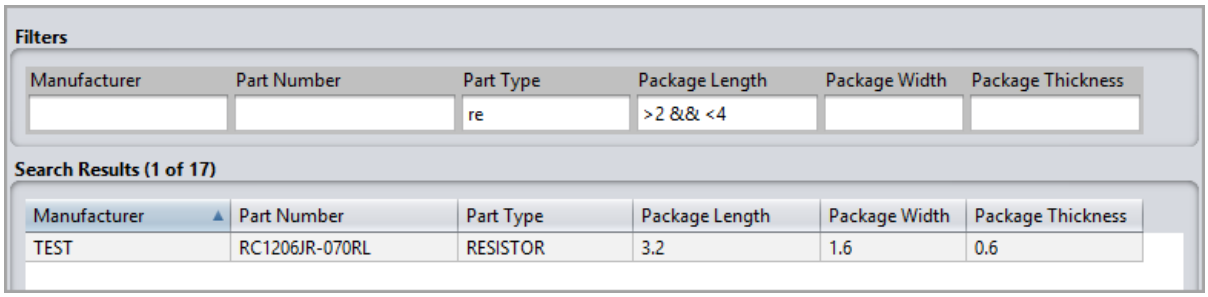
4.4.5.1.3. Column Filtering - Multi Column

If two or more column filter expressions are entered, then only the rows that match ALL of the filter expressions are displayed in the table. For example, as shown below, the two rows displayed each have a **Part Type** value that contains **re** and a **Package Length** value greater than two, as specified by the column filters.



4.4.5.1.4. Column Filtering - Boolean Expressions

Boolean expressions can be defined in each column filter expression to facilitate more complex filtering criteria. In the following example, the **Package Length** filter expression selects values that are greater than two and less than four.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following boolean operators are supported in column filter expressions:

Table 4.14: Column Filter Boolean Operators

Operator	Description
| or | |	True if either operand is true
& or &&	True if both operands are true
~ or !	True if operand is false
( )	Sub-expression

4.4.5.1.5. Column Filtering - Meta Functions

All the column filtering operators described above compute a result based on the property values in each row of the table. However, there are times when it is necessary to filter rows based on the meta data associated with a property value. To that end, the following meta functions are supported in column filter expressions:

Table 4.15: Column Filter Meta Functions

Function	Description
defined()	Returns true if the property value is defined

For example, the meta function can be used to display all rows where the **Package Thickness** is not defined.

Note:

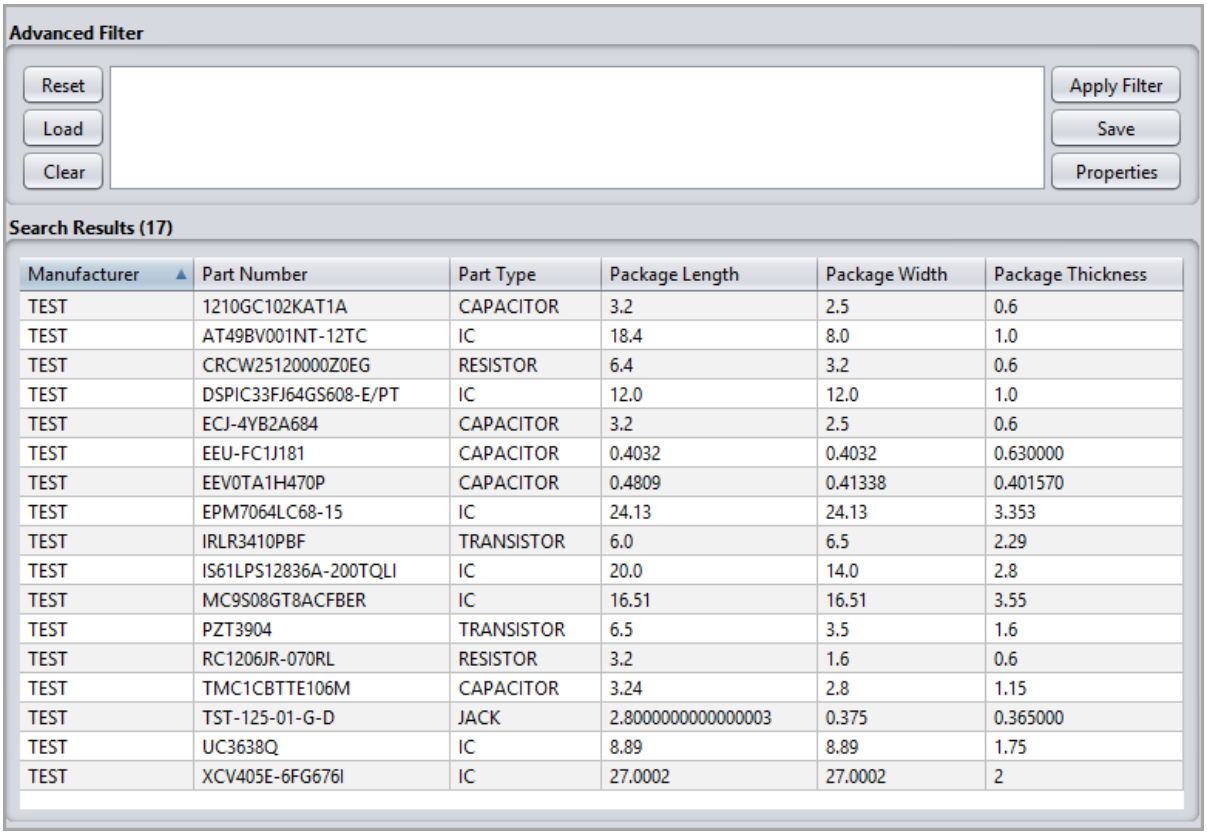
The use of meta functions in filter expressions can quickly determine which parts in the **Parts List** need to be reviewed or updated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.2. Advanced Filter

The **Advanced Filter** allows users to define filter expressions that reference one or more part properties. Unlike **Column Filter** expressions, an **Advanced Filter** expression can define arbitrary conditions across multiple part properties.

The following Search Results will be used as the basis for all the examples in this subsection:



Manufacturer	Part Number	Part Type	Package Length	Package Width	Package Thickness
TEST	1210GC102KAT1A	CAPACITOR	3.2	2.5	0.6
TEST	AT49BV001NT-12TC	IC	18.4	8.0	1.0
TEST	CRCW25120000Z0EG	RESISTOR	6.4	3.2	0.6
TEST	DSPIC33FJ64GS608-E/PT	IC	12.0	12.0	1.0
TEST	ECJ-4YB2A684	CAPACITOR	3.2	2.5	0.6
TEST	EEU-FC1J181	CAPACITOR	0.4032	0.4032	0.630000
TEST	EEV0TA1H470P	CAPACITOR	0.4809	0.41338	0.401570
TEST	EPM7064LC68-15	IC	24.13	24.13	3.353
TEST	IRLR3410PBF	TRANSISTOR	6.0	6.5	2.29
TEST	IS61LPS12836A-200TQLI	IC	20.0	14.0	2.8
TEST	MC9S08GT8ACFBER	IC	16.51	16.51	3.55
TEST	PZT3904	TRANSISTOR	6.5	3.5	1.6
TEST	RC1206JR-070RL	RESISTOR	3.2	1.6	0.6
TEST	TMC1CBTTE106M	CAPACITOR	3.24	2.8	1.15
TEST	TST-125-01-G-D	JACK	2.8000000000000003	0.375	0.365000
TEST	UC3638Q	IC	8.89	8.89	1.75
TEST	XCV405E-6FG676I	IC	27.0002	27.0002	2

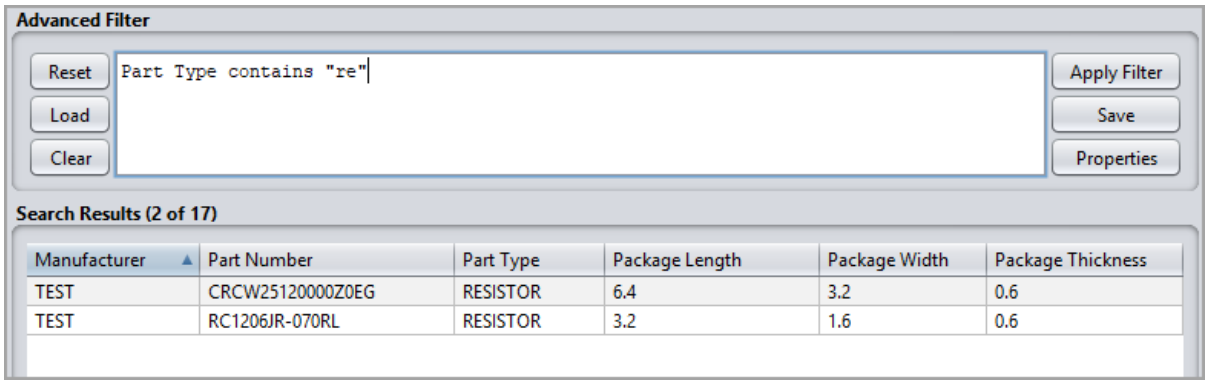
Note:

The **Advanced Filter** is applied when the **Apply Filter** button is pressed. When editing the filter expression, you can press that button without using the mouse by pressing the **Tab** key, followed by the **Enter** key on the keyboard. For convenience, the focus will automatically return to the text area, allowing you to continue editing the filter expression if necessary.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.2.1. Advanced Filter - String Matching

The **Advanced Filter** supports the same string-matching operators that are supported by the **Column Filters.** The only difference is that the part property name must always be specified. For example, the following filter expression displays all rows where the Part Type value contains the string **re**.



Part properties may be designated in filter expressions using either the property name or label. A complete list of part properties can be found using the **Help > Part Properties** menu option in Sherlock or by pressing the **Properties** button on the right side of the **Advanced Filter** panel (described below).

Note:

All strings must be quoted in an **Advanced Filter** expression using either a pair of single quotes or a pair of double quotes.

The following string-matching operators are supported in advanced filter expressions:

Table 4.16: Advanced Filters: String Matching

Operator	Description
PROPERTY contains STRING	True if the PROPERTY value contains STRING
PROPERTY startsWith STRING	True if the PROPERTY value starts with STRING
PROPERTY endsWith STRING	True if the PROPERTY value ends with STRING
PROPERTY == STRING	True if the PROPERTY value equals STRING
PROPERTY =~ REGEX	True if the PROPERTY value satisfies the Java regular expression (case sensitive)
PROPERTY !~ REGEX	True if the PROPERTY value does not satisfy the Java regular expression (case sensitive)

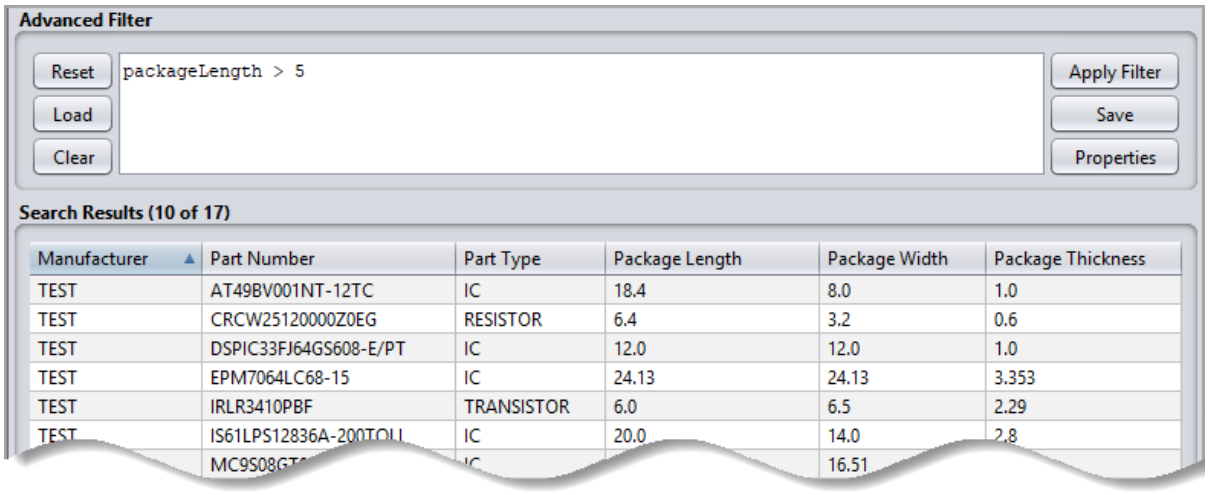
Note:

See Java documentation for a complete description of regular expressions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.2.2. Advanced Filtering - Comparison Operators

The **Advanced Filter** supports the same comparison operators that are supported by the **Column Filters**. The only difference is that the part property name must always be specified. For example, the following filter expression displays rows where the **Package Length** is greater than five, using numeric comparison.



The following comparison operators are supported in **Advanced Filter** expressions:

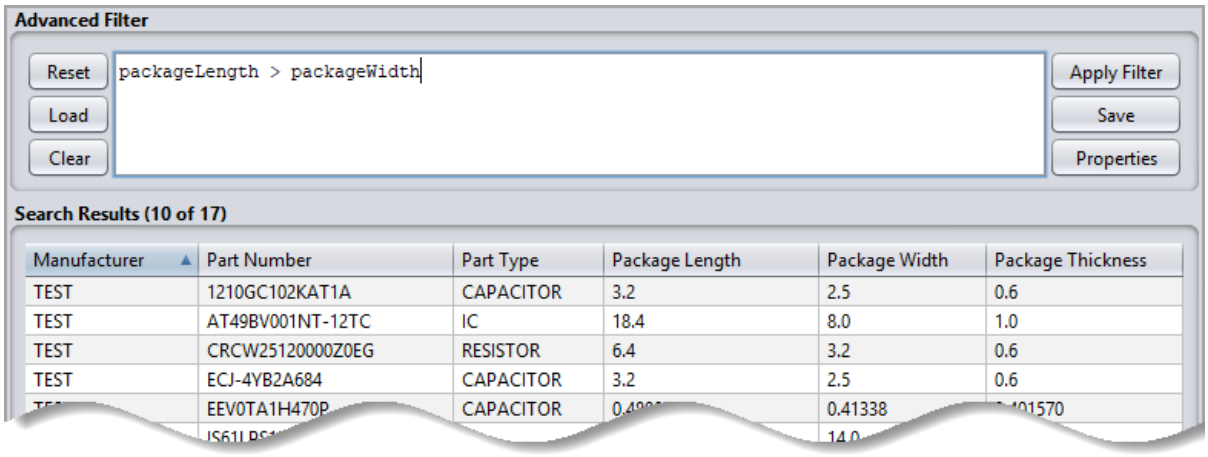
Table 4.17: Advanced Filters: Comparison Operators

Operator	Description	Operator	Description
= or ==	Equals	!= or <>	Not Equals
<	Less Than	>	Greater Than
<=	Less Than Or Equals	>=	Greater Than Or Equals

Note:

Comparison operators can be used to compare property values to literal values, property values to property values or literal values to literal values. In the following example, all parts with a package length greater than the package width are displayed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

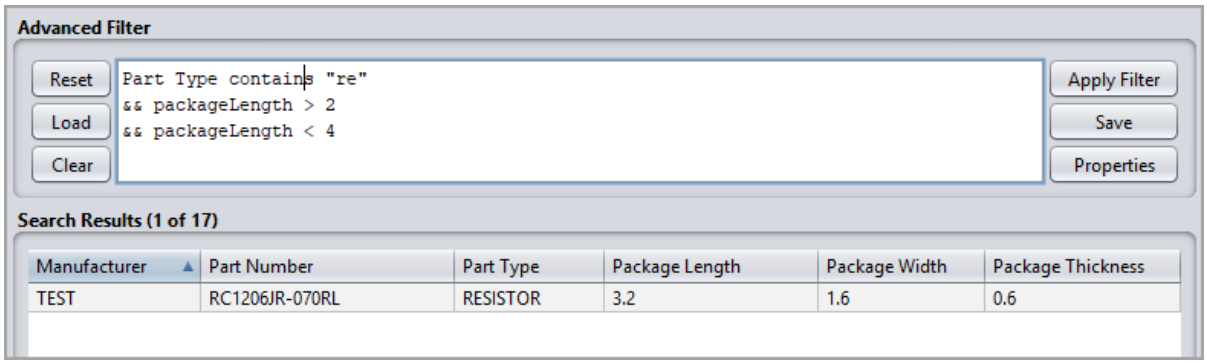


Comparison operators can be used for either a pair of numeric operands or a pair of string operands but cannot be used to compare a string operand to a numeric operand.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.2.3. Advanced Filtering - Boolean Expressions

Boolean expressions can be defined in an advanced filter expression to facilitate more complex filtering criteria. In the following example, the filter selects values where (a) the Part Type value contains the string re, (b) the Package Length is greater than 4 and (c) the Package Length is less than 10.



The following boolean operators are supported in **Advanced Filter** expressions:

Table 4.18: Advanced Filters: Boolean Operators

Operator	Description
| or ||	True if either operand is true
& or &&	True if both operands are true
~ or !	True if operand is false
( )	Sub-expression

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.2.4. Advanced Filters - Meta Functions

All the filtering operators described above compute a result based on the property values in each row of the table. However, there are times when it is necessary to filter rows based on the meta data associated with a property value. To that end, the following meta functions are supported in **Advanced Filter** expressions:

Table 4.19: Advanced Filters: Meta Functions

Function	Description
defined(PROPERTY)	Returns true if the property value is defined
anyDefined(PROP1, PROP2, ...)	Returns true if any of the property values are defined
allDefined(PROP1, PROP2, ...)	Returns true if all of the property values are defined

Note:

Unlike other operators and functions, the meta functions require that all property names be specified as quoted strings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.3. Advanced Filtering - Numeric Expressions

The following numeric operators are supported in **advanced filter** expressions to allow arbitrary calculations to be used as part of the filtering criteria:

Table 4.20: Advanced Filters: Numeric Operators

Operator	Description	Precedence
+, -	Addition and Subtraction	1
*, /	Multiplication and Division	2
%	Integer remainder	3
+, -	Unary Plus and Unary Minus	4
^	Power	5

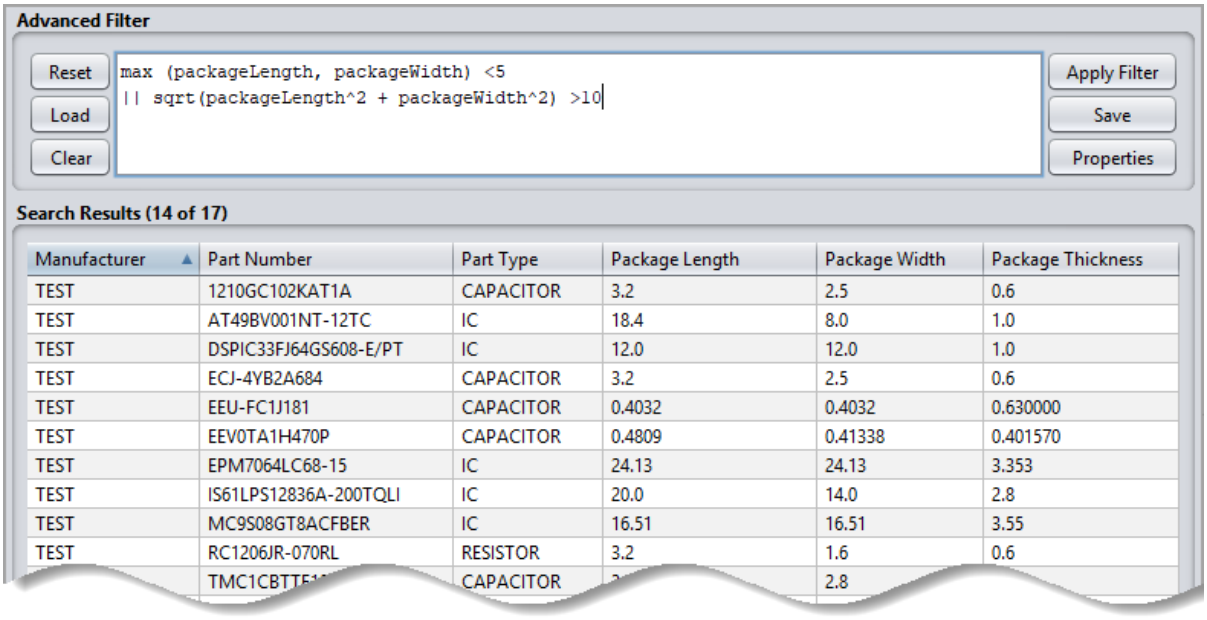
The numeric operators are listed from low precedence to high precedence. Higher precedence operations are performed first in an expression, absent any ordering specified by brackets.

The following numeric functions are supported in **Advanced Filter**expressions:

Table 4.21: Advanced Filters: Numeric Functions

Function	Description
min(VALUE1, VALUE2, ...)	Returns the minimum of all values specified, which may be property values, constants or numeric expressions
max(VALUE1, VALUE2, ...)	Returns the maximum of all values specified, which may be property values, constants or numeric expressions
sqrt(VALUE)	Returns the square root of the given property value, constant or numeric expression

In the following example, the filter selects all rows where either the maximum package dimension (that is, length or width) is less than 5 or the diagonal length of the package is greater than 10.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.3.1. Advanced Filtering - Filter Management

For convenience, the **Advanced Filter** edit panel maintains a history of the last twenty filter expressions that have been executed. This allows users to develop filter criteria in a piece-wise manner and/or to switch back and forth between a set of filters as needed.

Users may also save/load filters to/from designated text files. This allows users to easily maintain their own set of individualized filters and/or a library of re-usable filters that can be shared across a collection of Sherlock users.

The following buttons and/or keyboard Inputs can be used to manage filters on the **Filter History** list or saved in user-defined files:

Table 4.22: Advanced Filters: Filter Management

Button	Keyboard Input	Description
Apply Filter	CONTROL-F	Apply the currently defined filter and append it to the Filter History list
Reset	CONTROL-R	Reset the filter in the edit panel to that last filter that was executed
N/A	CONTROL-P	Load the previous filter, if any, on the Filter History list
N/A	CONTROL-N	Load the next filter, if any, on the Filter History list
Load	CONTROL-L	Load a saved filter expression from a designated text file
Save	CONTROL-S	Save the current filter expression to a designated text file
Clear	N/A	Clear the edit panel

The keyboard inputs listed in the table above are case-insensitive.

When a filter expression is saved to a designated text file Sherlock automatically includes a comment header that indicates the Windows username of the user who generated the file, as well as the date and time when the filter was saved, such as shown here:

```
# Sherlock PartLibrary Filter - Sam Somebody @ 2019-10-28 15:53:22
Package Length > 10
&& Package Width > 10
```

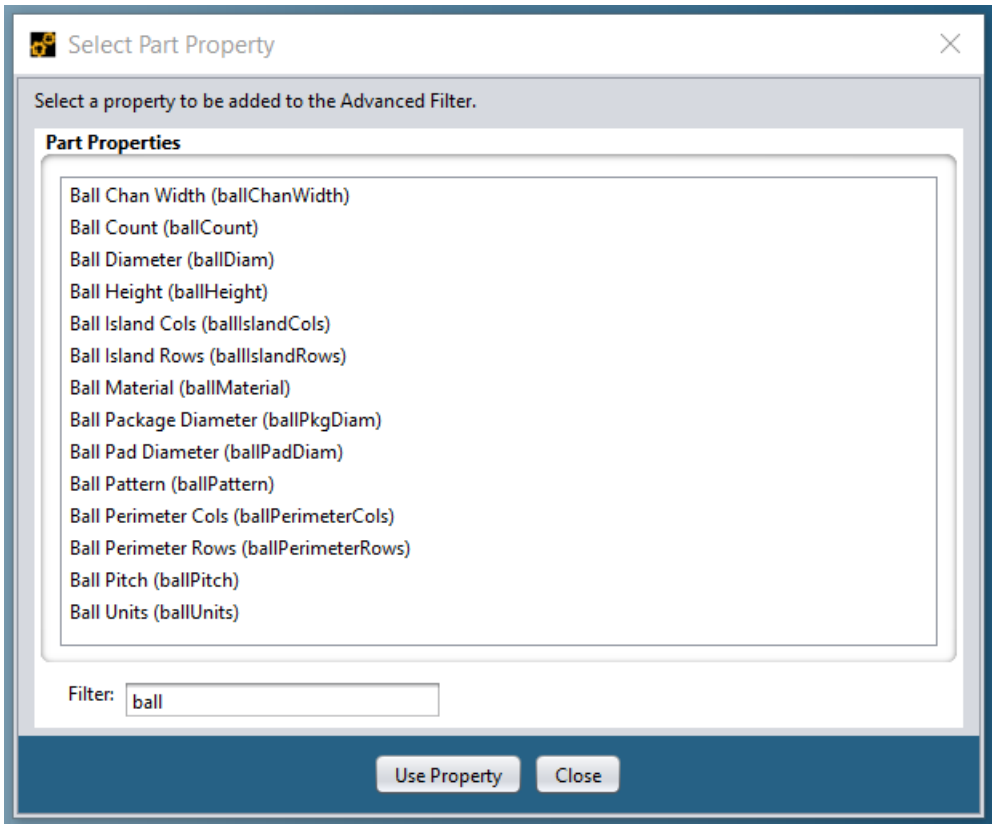
Users may edit the text files directly to modify filter expressions and/or provide additional comments regarding the filter.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.5.3.2. Advanced Filtering - Property Chooser

The **Select Part Property** dialog, accessible by pressing the **Properties** button in the **Advanced Filter** panel, provides a convenient mechanism for finding and entering a valid part property name.

As shown here, the dialog lists all part property labels and names, sorted by label. Users can scroll through the list to select a desired property to be included in the filter expression. When the **User Property** button is pressed, the selected property name will be appended to the current filter expression.



To help search through the list of properties, a filter expression can be entered below the list to display all list items that contain the given string. Such filtering is case-insensitive. Simply clear the filter field to display all part properties. In the example shown here, entering **ball** in the filter field results in the display of only the part properties associated with BGA solder balls.

For convenience, the dialog can be left open while editing the filter expression in the Advanced Filter panel so that multiple part properties can be entered as needed. When the dialog is no longer needed, press the **Close** button to hide the dialog until it is needed again.

Note:

If the SQL Query attempts to retrieve more than 1,000 parts from the database, then a message will be displayed and only the first 1,000 parts will be displayed in the Search Results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.6. Update Part Library from Parts List

To update the **Local Part Library** directly from the **Parts List**, right-click the **Parts List** icon in the **Project Tree** and select **Update Part Library from Parts List** from the context menu. Or, from the **Parts List**, you may click on the icon ribbon.

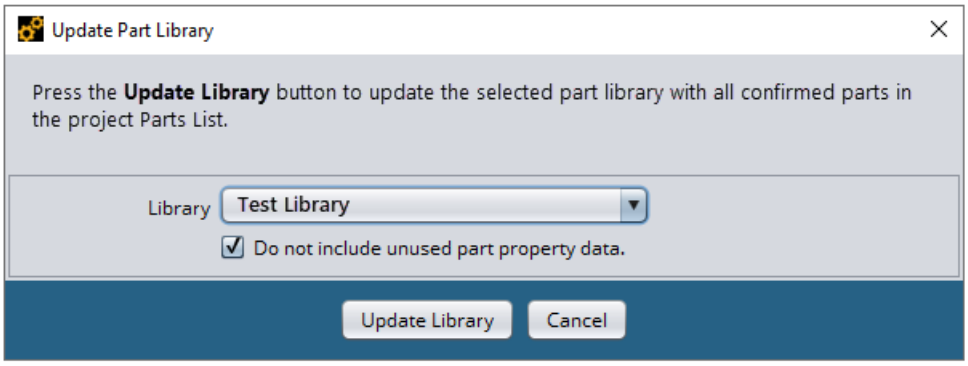
In either case, the dialog shown below appears:

- **Library**: From the dropdown menu, select the local library you wish to update.
- **Do not include unused part property data**: This option only appears if **Unused Part Properties** is set to HIDDEN in **Advanced Settings**. (See Part Property Viewing/Editing Options (p. 199) for information on the HIDDEN setting.)

  When the **Do not include...** option is enabled, Sherlock exports only the *required* properties to the local Parts Library—that is, those properties that are not hidden or grayed out in the **Parts Editor/Viewer**.

  If the **Do not include...** option is *not* enabled, then all relevant required and non-required properties are exported.

> **Note:**
>
> If a part was previously exported to the local Parts Library with both required and non-required properties, then the **Do not include...** check box will have no effect on further export updates. Both required and non-required properties will always be added. The **Do not include...** option only works if the initial export contains only required properties and the **Do not include...** option is enabled on all subsequent updates.



> **Warning:**
>
> Any *false* data added to the **Part Library** could propagate into the Parts Lists used by other circuit cards, projects or users. Always double-check your inputs before proceeding.

Click the **Update Library** button to update the selected Part Library. At that point, Sherlock examines all parts in the **Parts List** looking for parts with confirmed **Manufacturer** and **Part Number** fields. For all such parts found, Sherlock will then look at the confirmed part properties for each of the parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the confirmed part property values are the same, then the part will be used to update the **Local Part Library.** If two parts with the same manufacturer and part number have confirmed part properties with different values, an error will be generated, and the library will not be updated.

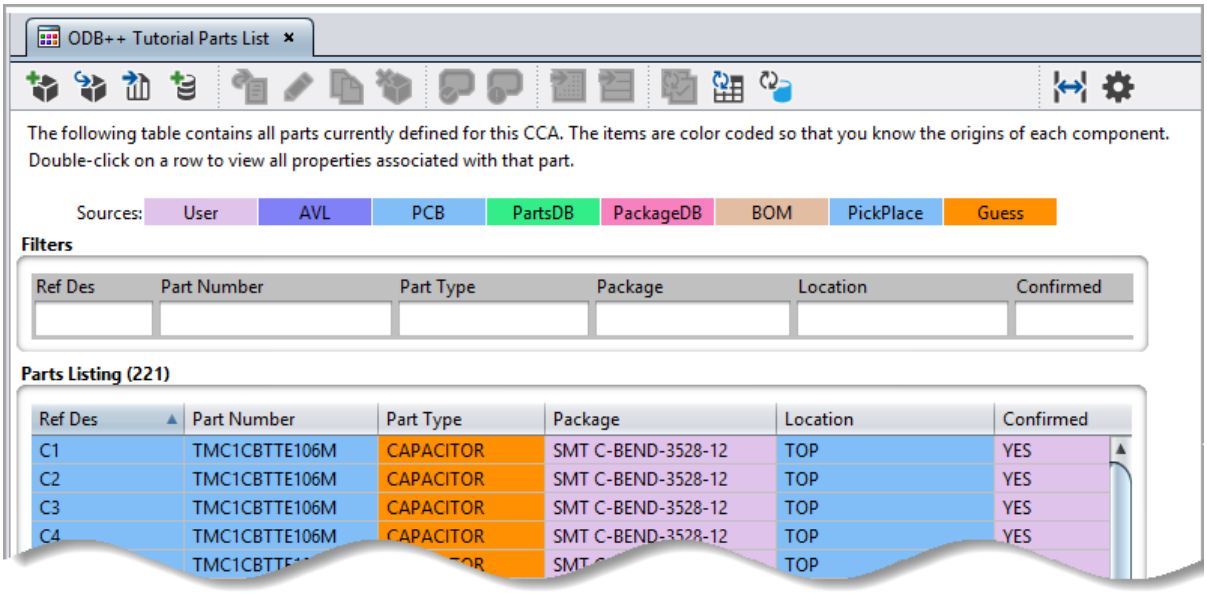
After a successful update, Sherlock will display a message indicating the number of parts actually updated in the library.

4.4.6.1. Part Library Interface with Parts List

For convenience, you can update the Parts List from the Part Library or update the Part Library from the Parts List using the context menu associated with the **Parts List** item on the **Project Tree**. On the ribbon located at the top of the Parts List, the last two icons also denote these options.


Update the Parts List from the Part Library


Update the Part Library from the Parts List



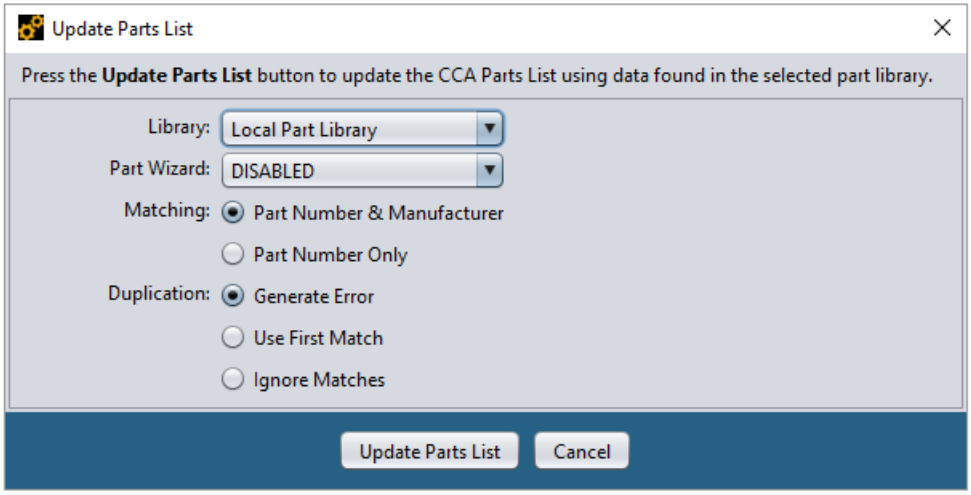
When selecting **Update the Part Library from the Parts List**, specify the **Library** you wish to update. Then press **Update Library**. When the update is complete, Sherlock reports the number of parts added to the Part Library.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.7. Update Parts List from Part Library

The real power of the **Part Library** can be seen when it is used to update the **Parts List** for a given circuit card. With just a few mouse clicks, you can update literally thousands of part properties using the part property values stored in the **Part Library**. This feature significantly reduces the time needed to review properties for a new circuit card, assuming that most, if not all, of the parts used in that circuit card were used in other circuit cards processed by Sherlock. Since **Part Library** data can be shared by multiple Sherlock users in an organization (details will be discussed shortly), over time the organization can develop a truly comprehensive database of part data.

Updates to the **Parts List** use an approach similar to that used when extracting data from the **Parts List.** Specifically, Sherlock examines every part in the **Parts List,** looking for a "matching" part in the **Part Database** (the exact meaning of matching will be discussed shortly). To begin the process, select **Parts List** from the **Project Menu** and right-click **Update Parts List** from Part Library to display the Update Parts List dialog.



- **Library**: Select which Part Library Sherlock will use to update your CCA Parts List.
- **Part Wizard**: (Default = DISABLED.) If ENABLED, Sherlock will use the Part Wizard to assist in finding matches. That is, if a conventional search does find a match for a specific part, Sherlock will then employ the Part Wizard's pattern matching in its search.
- **Matching**: Define how the Parts List should be matched with parts in the Part Database.
  - **Part Number & Manufacturer**: Both Part Number *and* Manufacturer must match.
  - **Part Number Only**: Match by part number only. This setting is useful when your Parts List does not include Manufacturing values. This may result in duplicate matches. See the **Duplication** setting below.
- **Duplication**: Determines how Sherlock should handle finding duplicate matches in the Part Database.
  - **Generate Error**: If selected, a duplicate match generates an error message. Sherlock will perform no part updates.
  - **Use First Match**: Sherlock uses the first match found in the Part Database.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Ignore Match**: Sherlock will not update the part but will continue processing other parts.

When the **Update Parts List** button is pressed, Sherlock will scan all parts in the **Parts List**, looking for matching parts in the **Part Library**. If no errors are generated, Sherlock will display a status message after the update process has completed, indicating the number of parts that were actually updated.

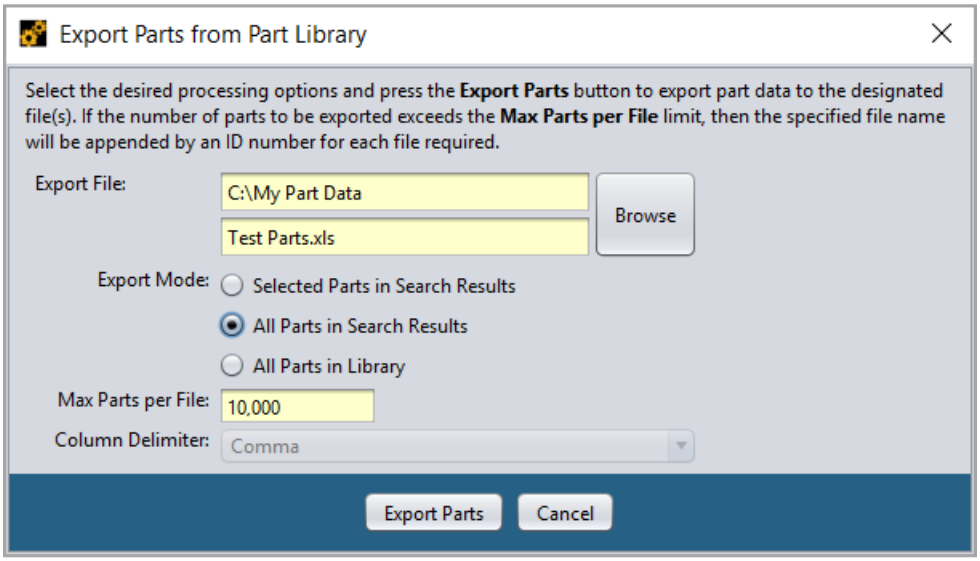
Note:

Even if Sherlock finds matching parts in the **Local Part Library**, it's possible that no **Parts List** updates will be made if all the **Parts List** property values are already the same as those defined in the library.

4.4.7.1. Export Part Data

Sherlock automatically shares the **Local Part Library** across all projects and circuit cards processed by a given user on a given platform. If you want to share the **Local Part Library** across multiple Sherlock users or you want to keep an archive copy of the **Local Part Library**, you can export the data for one or more parts via the search dialog.

To export **Local Part Library** data, select **Libraries > Parts** from the main menu and press the **Export Parts...** button in the Search Criteria panel to display the **Export Parts** from **Part Library** dialog, as shown here.



The **Export File** fields allow you to browse the file system to select an appropriate folder and file to which the data should be exported. Part data may be exported as a CSV, XLS, XLSX, or XML file.

The **Export Mode** choice indicates which parts should be exported to the file. In this example, all of the parts listed in the **Search Results** table in the **Search Part Library** dialog will be exported. The **All Parts in Library** option allows you to export all of the parts currently defined in the **Part Library** to one or more export files. The exact number of export files created depends on the value specified in the **Max Parts per File** and the total number of parts to be exported. If multiple files are required, Sherlock will automatically append an index number to the file name. For example,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

if two export files are required in the example above then they will be named **Test Parts-1.xls** and **Test Parts-2.xls**.

Note:

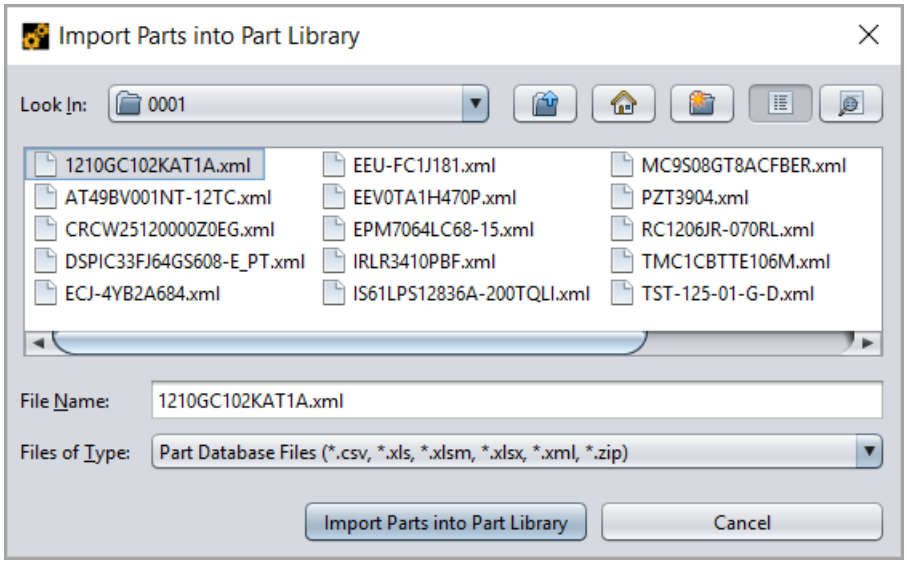
If you want to copy the entire **Local Part Library**, you may also copy the entire **PartDB** directory in the Sherlock User Directory. See the **Recover Part Library** section for more details.

4.4.8. Import Part Data

Data exported from one Sherlock Part Library can be imported into any Sherlock Part Library to:

- Facilitate sharing of data between Sherlock users
- Use different part data to compare with previous results, *or*
- Revert back to a previous version of the **Part Library**.

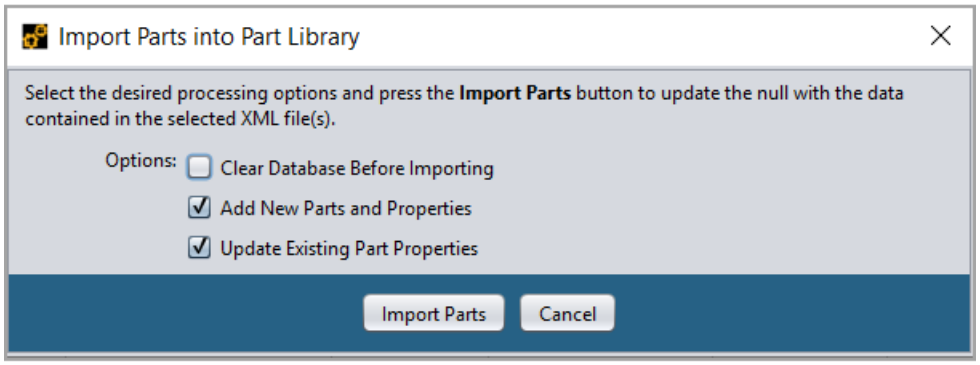
To import data into the Local Part Library, select **Libraries > Part Parts** from the main menu and then press the **Import Parts...** button to display a dialog that allows you to select the part database file to be imported.



After selecting the desired file, press the **Import Parts into Part Library** button to proceed to the next dialog.

The second dialog displayed will vary depending on the type of import file selected. The dialog allows you to select the desired processing options.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The options allow you to clear the existing **Part Library** before loading the data from the XML file, add all new parts and properties found in the XML file and/or update part properties that exist in both the current Part Library and the XML file.

The default options shown here are usually appropriate for common purposes.

When importing a spreadsheet file, in addition to the options shown here, options will exist to select the name of the sheet from the file and the column mapping options. If Sherlock is not able to properly guess the column names, manually choose the columns from the file to associate to the missing part attribute columns.

After the file is processed and the library is updated Sherlock will display a message indicating the number of parts updated or indicating that no updates were made.

Note:

If you open any part library which has missing columns and the part library isn't read-only, the columns will be added automatically, and all rows will have null values. Extra columns are ignored.

4.4.9. Recover Part Library

For users who are upgrading from previous versions of Sherlock, the previous **Local Part Library** is maintained by Sherlock as a collection of folders and XML files found in the **PartDB** folder in the Sherlock User Directory. Advanced users were free to examine and/or modify the XML files directly, copy them for safe-keeping or exchange them with other Sherlock users. Such an approach has been taken to allow for a free exchange of part data files amongst reliability professionals.

To facilitate fast retrieval of part data from the previous **Local Part Library**, in addition to the collection of XML files Sherlock maintains a binary data file containing search indexes. In previous operation, Sherlock keeps the binary data file consistent with the XML files being managed. However, if users modify the XML files themselves, for any reason, the binary data file must be refreshed or recovered.

Part Library recovery is performed automatically by Sherlock on start-up if the binary data file is missing or if there is a problem reading the binary data file. If you are upgrading from a previous version of Sherlock, you can also recover the library index file manually by selecting from the main menu. At that point, a confirmation dialog will appear to ensure that you really want to recover the **Part Library** index file from the XML files. This function is only enabled for the local part library and shared part library of previous Sherlock versions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.5. Part Validation

The **Part Validation** analysis module allows you to validate the part properties used in a selected circuit card assembly (CCA) against the part properties currently maintained in the Part Database and/or the Approved Vendor List (AVL) (p. 268). Such validation ensures that part numbers, descriptions, and other part properties are consistent across all CCA's maintained by your organization. In addition to running Part Validation as specified here, it can be automatically run as needed prior to any other component-related analysis using the current settings.

Note:

The examples shown in this document are based on the Tutorial Project provided as part of the Sherlock Tutorial package as discussed in the Project Overview (p. 8) lesson.

In this section, the following topics are covered:

4.5.1. Input Data

4.5.2. Analysis Options

4.5.3. Analysis Results

4.5.4. Component Analysis Part Validation

4.5.1. Input Data

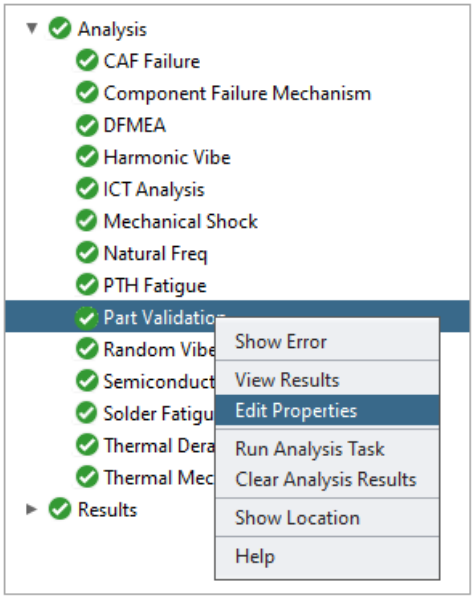
The **Part Validation Analysis Module** makes use of the following input data:

- Parts List
- Part Library
- Approved Vendor List

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

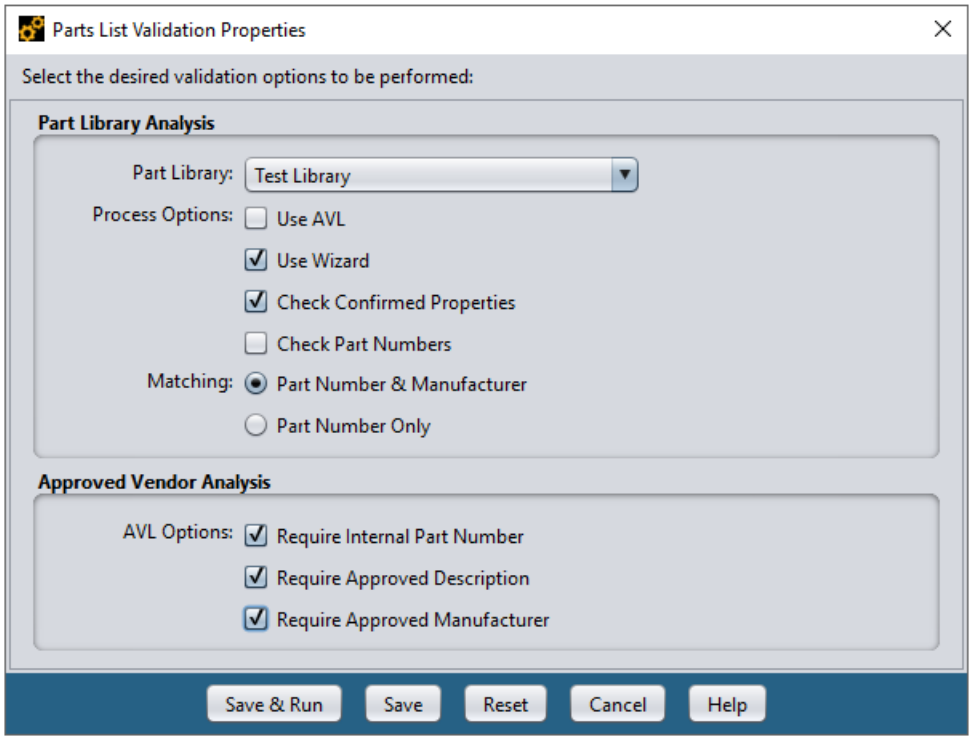
To open the dialog, right-click **Part Validation** under **Analysis** in the **Project Tree** and select **Edit Properties**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4.5.2. Analysis Options

The **Part Validation** analysis module allows you to choose several processing options which are explained below.



- **Part Library Analysis**
  - **Part Library**: Use the dropdown menu to select the part library you wish to validate.
  - **Process Options**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **Use AVL**: (Default: Disabled) When you enable this option, Sherlock will use any matching AVL part numbers when searching the part library.

→ **Use Wizard**: (Default: Disabled) If this option is enabled and Sherlock cannot find a match (based on the **Matching** options below), Part Wizard patterns are checked. If the Part Wizard finds a match, it automatically adds part data (from the Part Database) to the Local Part Library.

→ **Check Confirmed Properties**: This ensures that all confirmed part property values in the Parts List are the same as the corresponding part property values in the Part Database. This option is useful for ensuring that the Parts List is correct.

→ **Check Part Numbers**: This ensures that all confirmed Part Numbers stored in the Parts List are also defined in the Part Library. This option is useful for finding parts that were not previously defined in the Part Database.

– **Matching**:

A given part in the Parts List is matched with a part in the Part Database based on the **Matching** criteria selected below.

→ **Part Number & Manufacturer**: If selected, both the part number and the manufacturer are compared to the Part Database to determine a match.

→ **Part Number Only**: If selected, only the part number is used to determine a match with parts in the Part Database. This allows you to validate parts in the Parts List when no Part Manufacturer has been specified (which is common when importing data from various design files).

- **Approved Vendor Analysis**

When validating part properties against the Approved Vendor List (AVL) (p. 268), Sherlock users can choose the options below. (The AVL is only searched if you select one or more of the options below or if you select **Use AVL** above.)

Tip:

The options below validate only select part properties against the Approved Vendor List (AVL) (p. 268). For a comprehensive validation that checks all part properties, select **Use AVL** under **Process Options** (p. 264).

– **Require Internal Part Number**: When selected, an error will be generated if the Part Number property value stored in the **Parts List** does not match an approved (that is, internal) part number in the AVL.

– **Require Approved Description**: When selected, an error will be generated if the Part Description property value stored in the **Parts List** does not match the Description defined in the AVL for the matching part.

– **Require Approved Manufacturer**: When selected, an error will be generated if the Manufacturer property value stored in the **Parts List** does not match one of the approved vendors in the AVL for the matching part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

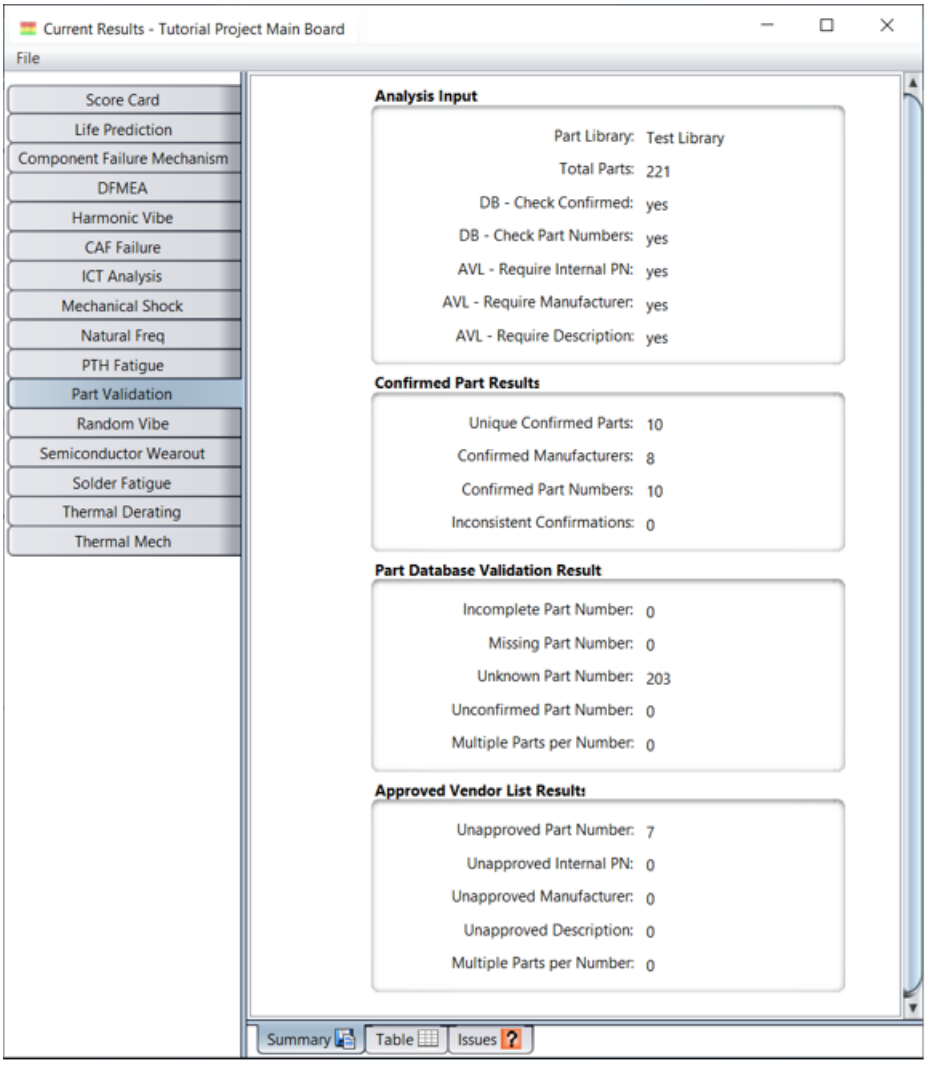
4.5.3. Analysis Results

The following results are produced by the Part Validation analysis module:

- Summary Results
- Table of individual Parts List Results
- List of issues found

4.5.3.1. Summary Results

The **Summary** tab shows the various processing options selected, the number of parts analyzed, and numeric summaries of the various issues found. When used over time, the **Summary Results** (shown below) provide an overall indication of the consistency of a selected CCA with regard to **Parts List** data.



The **Confirmed Part Results** summary shows the number of parts in the **Parts List** that have been **Confirmed** (i.e. verified by the user). This is critical information with regard to **Part Database** validation in that only confirmed parts are checked against the **Part Database**.

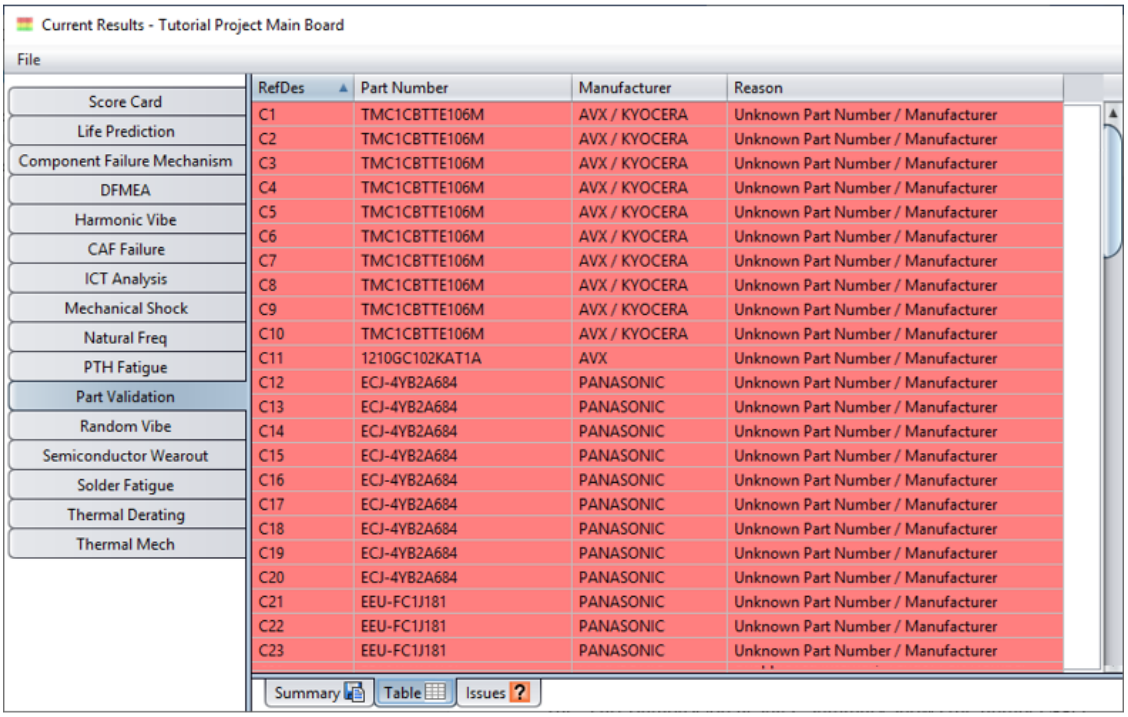
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Part Database Validation Results** summary shows the number of parts that triggered each of the validation errors possible when comparing against the **Part Database**.

The **Approved Vendor List Results** summary shows the number of parts that triggered each of the validation errors possible when comparing against the AVL.

4.5.3.2. Table of Individual Part List Results

The **Table** tab generated by the **Part Validation Analysis Module** shows the individual errors for each of the parts found in the **Parts List**. If one or more errors were generated for a given part, then each error will be displayed in the table, color-coded by severity, as shown here:



In this example, we can see that two errors were generated for part **C27**. The first error (**Incomplete Part Number / Manufacturer**) was generated because that part is missing a **Manufacturer** property value. The second error (**Unapproved Internal / Vendor Part Number**) was generated because the **Part Number** stored in the **Parts List** does not match any INTERNAL Part Number stored in the AVL and we choose to require internal part numbers in the input dialog above.

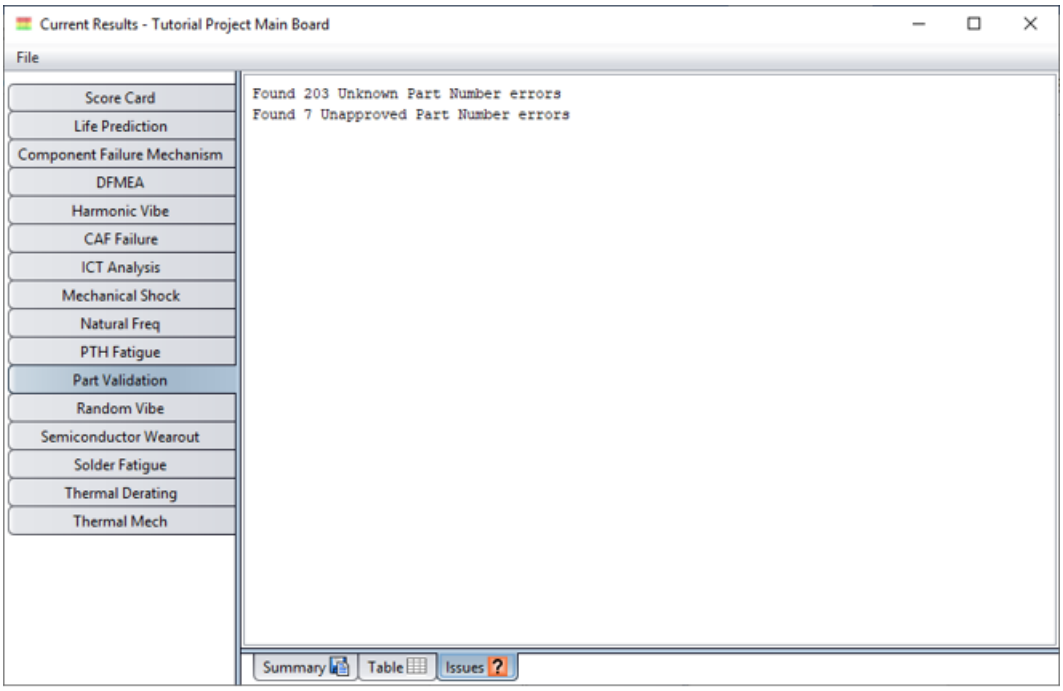
Note:

As with all Sherlock analysis results tables, you can double-click any of the rows to view the part properties for the selected part or right-click any row to export all or part of the results table to a spreadsheet file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.5.3.3. List of Issues Found

The **Issues** tab shows a summary list of the issues found during processing, possibly including detailed error descriptions for specific parts. You can easily cut-and-paste the contents to another document.



4.5.4. Component Analysis Part Validation

Any of the analysis tasks in Sherlock that relate to components will generate an analysis warning if part validation is not up to date and passing prior to executing those other analysis tasks. The settings dialog for each of the component-related analysis tasks has an option for **Part Validation** to be enabled or disabled prior to executing the given analysis task if required. If the Part Validation analysis has already executed and is up to date, Sherlock will not execute it again. **Part Validation** will only execute when the given analysis setting is enabled and **Part Validation** isn't currently in a passing state. When Sherlock executes all available analysis tasks as once, as directed by the user, it will run the **Part Validation** task first and the other tasks will hold until it is complete.

4.6. Approved Vendor List (AVL)

To assist with project management, Ansys Sherlock allows users to create and maintain an **Approved Vendor List (AVL)** that can be shared across all projects to translate between internal organizational part numbers and one or more approved vendor part numbers. Users can then search for parts based on any of the part numbers defined in the AVL, update a selected CCA Parts List using the information from the **AVL** and/or verify that all part numbers referenced by a selected CCA are approved. As with other Sherlock data structures, the **AVL** can be edited manually using the Sherlock Graphical User Inter-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

face, can be exported to a spreadsheet for external or batch editing and imported from a spreadsheet. This document describes how to create, maintain and use an **Approved Vendor List** in Sherlock.

Note:

The examples shown in this document are based on the Tutorial Project provided as part of the Sherlock Tutorial package discussed in the Project Overview (p. 8) lesson.

In this section, the following topics are covered:

4.6.1. AVL Menu Options

4.6.2. Maintaining the AVL

4.6.3. Using The AVL

4.6.1. AVL Menu Options

The **Approved Vendor List** is created, maintained, and utilized via the following menu items located in the **Libraries > Approved Vendor List (AVL)** main menu option.

For your convenience, you may update a parts list from the **AVL** by right-clicking **Parts List** and selecting **Update Parts List** from **AVL**. The AVL is also integrated into the **Part Library** search process and the **Part Validation** analysis module, as will be discussed below.

4.6.2. Maintaining the AVL

All **AVL** maintenance processing is performed via the **Libraries > Approved Vendor List (AVL)** main menu option. The AVL itself is shared across all Sherlock projects and is empty after Sherlock is initially installed. Over time, you may manually add, edit, and remove AVLentries as needed. You may also import / export all or part of the AVL to facilitate sharing with other Sherlock users or for organizational configuration control.

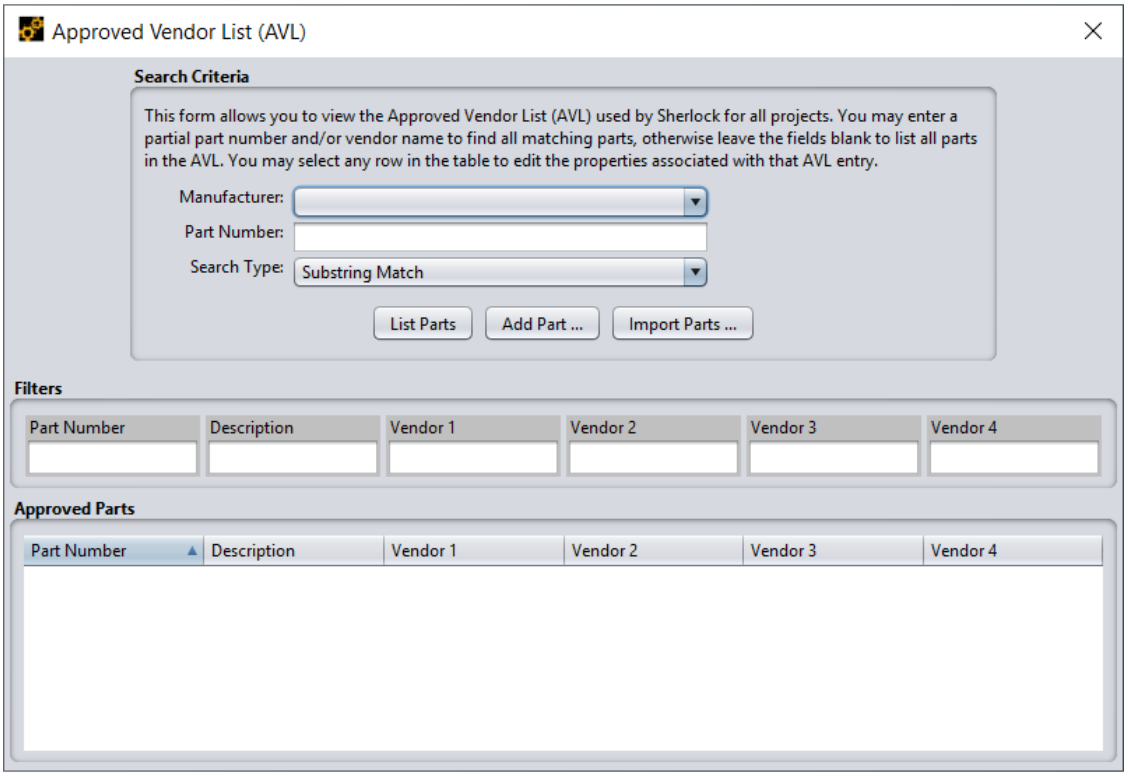
4.6.2.1. Importing AVL Data

The most common way to create an **AVL** in Sherlock is to import the AVL data directly from a spreadsheet file maintained by your design or reliability organization. Sherlock accepts the following spreadsheet file formats based on the file name suffix (case insensitive):

- Comma Separated Values (CSV)
- Excel 97/2003 (XLS) *and*
- Excel 2010 (XLSX)

To see how the import process works, let's import the **Tutorial Project AVL.xls** file located in the **tutorial** sub-folder of the Sherlock Installation Directory (usually **c:/Program Files/Ansys Inc/v251/sherlock**). Select the **Libraries > Approved Vendor List (AVL)** main menu option to display the following edit dialog:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

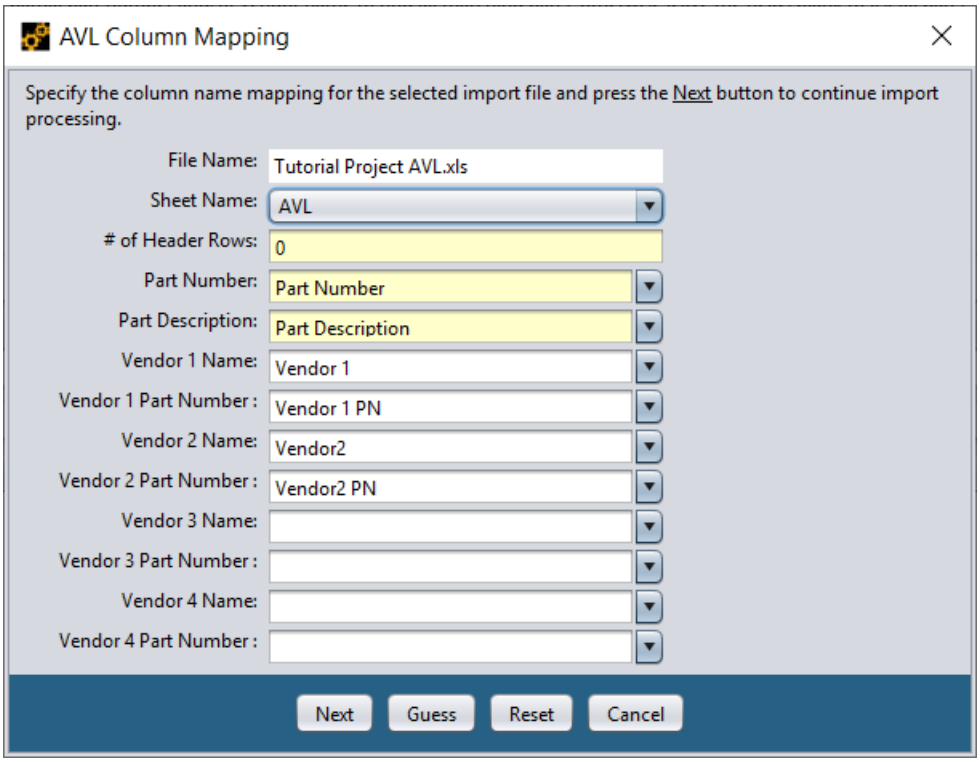


Shortly we will see how this dialog can be used to maintain the **AVL**, but for now we need only press the **Import Parts...** button to display the Import **AVL File(s)** dialog that allows us to import the AVL spreadsheet file.

Locate and select the spreadsheet file referenced above on your file system and press the **Import AVL File(s)** button to start the import process. You may also double-click the desired file in the file browser dialog as a convenience. Sherlock will open the selected file and determine the relevant columns available.

The **AVL Column Mapping** dialog will then be displayed to show what Sherlock found in the file. Specifically, it will display the sheet names found in the file (for XLS and XLSX files), the number of header rows found before the column header row and a mapping of column headers to **AVL** properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



It is important to note that Sherlock makes educated guesses about the file format to automatically determine the desired mapping. In this tutorial example, Sherlock guesses everything correctly. However, depending on the complexity of the spreadsheet being imported, you may need to manually update the values in the **AVL Column Mapping** dialog. The most common such adjustments are selecting the proper sheet from a multi-sheet file, specifying the actual number of header rows and/or selecting the desired columns.

Note:

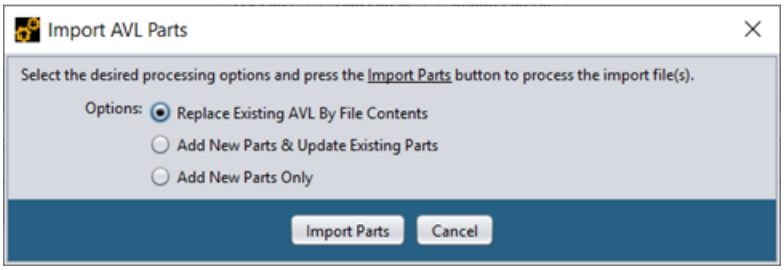
After manually specifying the Sheet Name and/or Header Row count you can press the **Guess** button to have Sherlock guess the column mappings again.

Note:

Sherlock automatically re-uses the column mapping used during the last AVL Import process. If you are processing a new AVL file, you can press the **Guess** button to clear the old mapping and re-guess based on the new file.

After entering the proper column mapping, press the **Next** button to proceed to the next import step - selecting the desired processing options.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the selected AVL spreadsheet file contains the entire AVL used by your organization, then you can safely select the **Replace Existing AVL By File Contents** processing option. As the name implies, that processing option will replace all existing AVL data by the data found in the file. This ensures that the AVL data used by Sherlock is identical to that specified in the spreadsheet file.

If the selected AVL spreadsheet file contains a subset of your organization's AVL, select either of the remaining options to append and/or update the Sherlock AVL using the data found in the selected spreadsheet file. These processing options allow you to incrementally update the Sherlock AVL when new parts are added, or existing parts updated.

In either case, after selecting the desired processing option, press the Import Parts button to process all rows in the selected spreadsheet file and update the Sherlock AVL. After the import process has completed, the **Approved Parts** list will be automatically refreshed to show the AVL parts that match the given search and filter criteria. Since we didn't enter any search or filter criteria, all the AVL parts will be displayed as shown here.

Approved Parts (16)

Part Number	Description	Vendor 1	Vendor 2	Vendor 3	Vendor 4
DFR-CAP1	CAP,TANT,10U,20%,16V,B R...	DFR SOLUTIONS/TM...			
DFR-CAP2	CAP,CER,1NF,10%,2KV,X7R,...	AVX/1210GC102KAT1A	AVX/1210GC102KAT1AJ		
DFR-CAP3	CAP, 0.68uF, 100C, 20%, SM...	PANASONIC/ECJ-4Y...			
DFR-CAP4	Cap, 180uF, 63V, Elec FC, Ra...	DFR SOLUTIONS/EEU...			
DFR-CPLD	IC MAX 7000 CPLD 64 QFJ-68	ALTERA/EPM7064LC...			
DFR-FLASH	IC FLASH 1MBIT 120NS T1 T...	ATMEL/AT49BV001N...			
DFR-FPGA	IC FPGA 1.8V 676 I/O BGA	XILINX/XCV405E-6FG...			
DFR-HEADER1	Header, 2X25, Shded, ST .10...	SAMTEC/TST-125-01...			
DFR-MCU-8K	IC MCU 8K FLASH 1K RAM ...	FREESCALE SEMICON...			
DFR-MCU-64K	16 Bit MCU/DSP 40MIPS 64...	MICROCHIP TECHN...			
DFR-MOSFET	MOSFET,PWR,100V,17A,.105...	INTERNATIONAL RE...			
DFR-PWM	IC ADV PWM MOTOR CTRL...	TI/UC3638Q			
DFR-SRAM	SRAM 4Mb 128Kx36 200Mhz...	ISSI/IS61LPS12836A-2...			
DFR-TRANSISTOR	XSTR,GEN PURP,NPN,SOT-...	FAIRCHILD SEMICON...			
TEST-RES-1/8W	Res,0 OHM,1/8W, 5%,1206,...	DFR SOLUTIONS/RC1...			
TEST-RES-1W	Res, 0 OHM, 1 W, 5%, 2512,...	DFR SOLUTIONS/CR...			

The **Approved Parts** list shows the approved part number and description for each part, as well as the various vendor/part combinations. In our example, most of the parts only have a single vendor part associated with it, but up to four vendor/part combinations are supported.

4.6.2.2. Importing Multiple Files

The **Import AVL File(s)** dialog described above can be used to import one or more spreadsheet files at the same time. This allows users to maintain their organizational AVL across different spreadsheet files (perhaps by vendor or functional group) and still be imported by Sherlock without too much fuss. To import multiple spreadsheet files, you can select multiple spreadsheet files by

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressing and holding the Ctrl key when selecting the files in the dialog or any of the standard Windows GUI mechanisms for selecting multiple files. You may also select a folder instead of an individual spreadsheet file. In that case, all spreadsheet files contained in that folder (and all sub-folders) will be imported by Sherlock.

When multiple files are imported, Sherlock will display a separate **AVL Column Mapping** dialog for each of the files. After specifying the column mappings for each file (which may be the same), the **Import AVL Parts** dialog will be displayed to allow you to select the desired processing options for all the files being imported.

4.6.2.3. Editing AVL Part Data

The **Approved Parts** result list displayed in the **Approved Vendor List (AVL)** dialog allows you to edit any AVL entry simply by double-clicking any row. You may also right-click an entry in the **Approved Parts** list and select **Edit Selected Part** from the context menu. At that point the **Edit AVL** Part dialog will appear, allowing you to edit the approved part number (aka. Internal Part Number), the approved description and up to four vendor name/part number pairs.

Note:

If you change the **AVL Part Number** you must enter a new part number that is not already defined in the AVL.

4.6.2.4. Deleting AVL Part Data

To delete one or more AVL entries simply select one or more rows in the **Approved Parts** list, right-click any of the selected parts and select **Delete Selected Parts** from the context menu. You will be prompted to confirm the deletion.

Note:

There is no undo-action provided to restore deleted AVL entries. As such, you may want to periodically export the AVL data (as explained in the next sub-section) for safe keeping outside of Sherlock.

4.6.2.5. Exporting AVL Part Data

AVL data can be exported to a spreadsheet file at any time simply by selecting one or more rows in the **Approved Parts** list, right-clicking any of the selected entries and selecting **Export Selected Parts** from the context menu.

An easy way to export the entire AVL is:

- Clear all Search Criteria and Filter fields in the **Approved Vendor List (AVL)** dialog
- Press the **List Parts** buttons to display all AVL parts
- Select any row in the **Approved Parts** list
- Press **Ctrl+A** to select all rows in the table

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click any row and select **Export Selected Parts** from the context menu.

In either case, when the **Export AVL Parts** dialog is displayed, locate the folder in which you want to store the spreadsheet and specify the desired file name. If you specify an existing file name, Sherlock will prompt you to ensure that the file should be overwritten.

When the **Export AVL Parts** button is pressed, Sherlock will export the data for all the selected rows to the designated spreadsheet file.

Note:

Sherlock automatically determines the export file format based on the specified file name suffix, which must be either .XLS, .XLSX or .CSV (case-insensitive).

4.6.3. Using The AVL

Sherlock makes use of the **Approved Vendors List** in the following ways:

- Searching the Part Library
- Updating the Parts List *and*
- Validating the Parts List

We now discuss examples for each use of the AVL.

Note:

The examples shown in the section assume that the Parts List and Part Library have been updated using the examples provided in the Part Library (p. 230) user guide. If you haven't already worked through those examples, you may not have the necessary Parts List or Part Library data entries to work through these examples.

4.6.3.1. Searching the Part Library

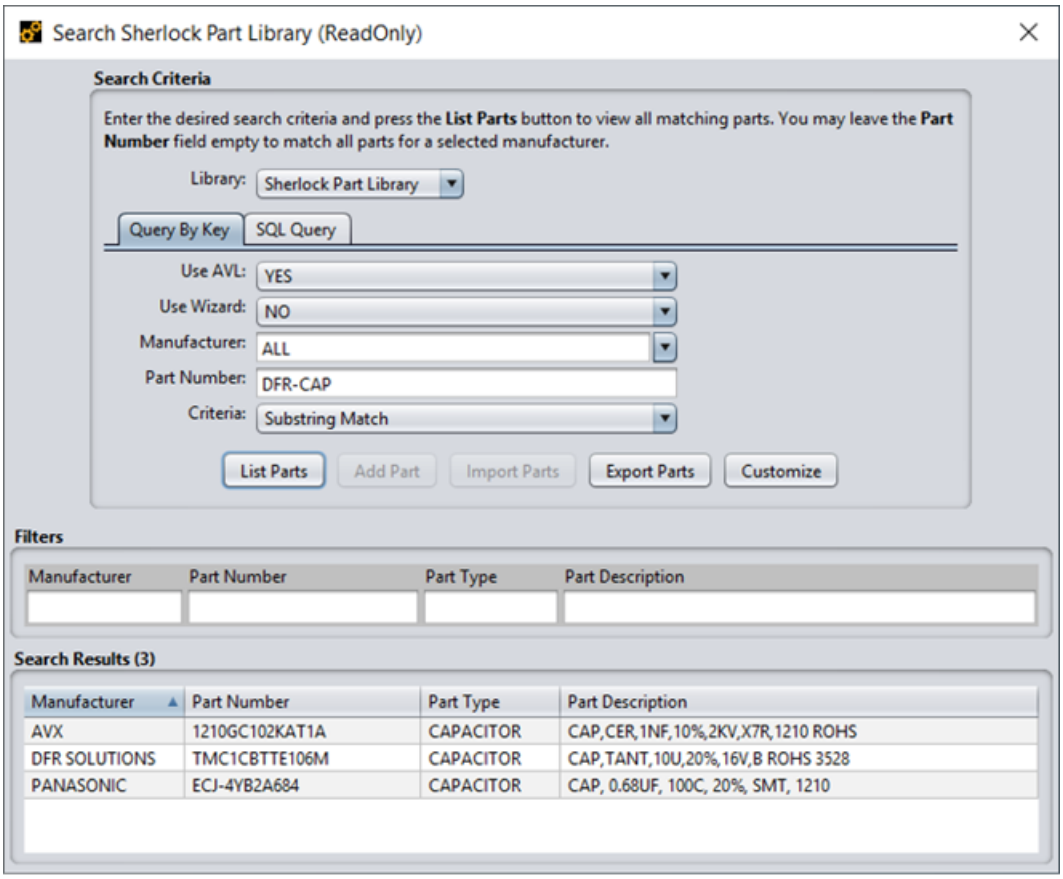
Part library searches automatically make use of the AVL to cross-reference approved (aka. internal) part numbers and manufacturer part numbers. This allows Sherlock users to store either part number in the **Parts List** and/or **Part Library**, while still being able to locate parts using the other part numbers.

To see how this works, we'll use the **Search Part Library** dialog (shown below) to search for parts using the approved part numbers, even though only vendor part numbers are used in the **Parts List** and **Part Library.**

Select **Libraries > Parts** from the main menu, fill out the search criteria as shown here and press the **List Parts** button to list all part numbers prefixed by DFR-CAP.

The results table show multiple matching parts, even though neither part number starts with DFR-CAP. These results were found because the DFR-CAP1 entry in the AVL recently imported reference the TMC1CBTTE106M vendor part number, and that part number is stored in the **Part Library.** Similar results would occur if an approved part number was stored in the **Part Library** and we were searching for a vendor part number cross-referenced to that part in the AVL.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

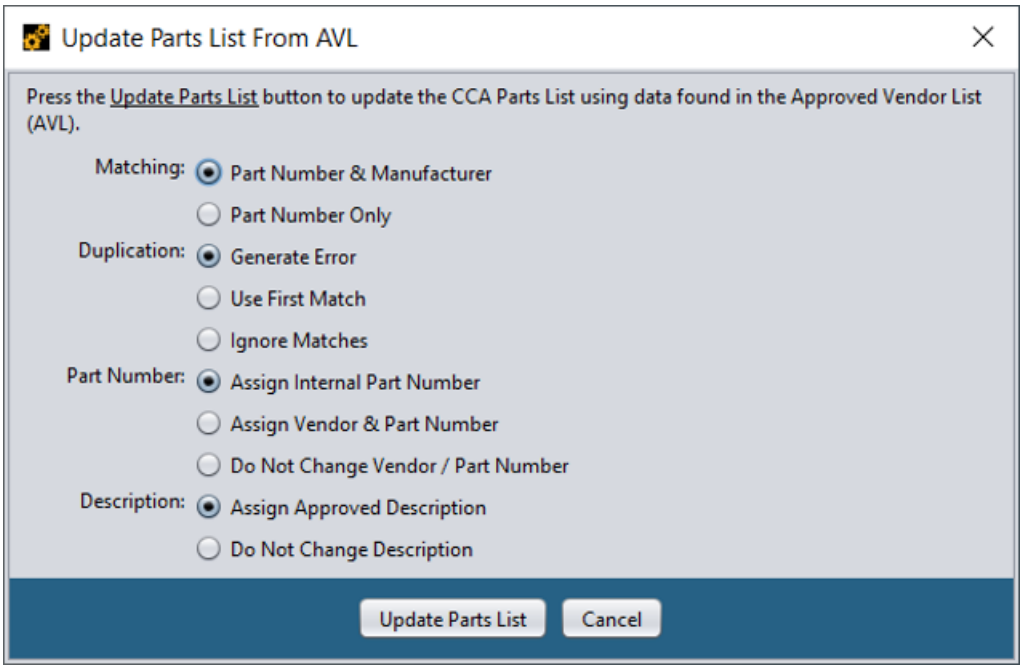


4.6.3.2. Updating the Parts List

In many cases, the various design files used by Sherlock to gather part data use inconsistent part numbers, sometimes using internal part numbers in one file and vendor part numbers in other files. Similarly, the part descriptions gathered from various design files may be inconsistent, misleading or down-right wrong. To help clean up such problems, Sherlock allows you to update the part number, manufacturer and/or description properties for any part in the Parts List that has a corresponding AVL entry. To see how this works, right-click the **Parts List** entry in the **Project Tree** and select the **Update Part List from the Approved Vendor List** option (A, below) from the ribbon.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dialog shown above allows you to indicate which parts in the **Parts List** should be updated, how to handle duplication problems and how to update the part properties using the AVL data.

4.6.3.3. Matching Criteria

When processing each part in the Parts List, Sherlock uses the selected **Matching** criteria to determine if the given part matches an entry in the AVL. If **Part Number & Manufacturer** are selected, then a match will occur only if both the manufacturer and part numbers listed in the **Parts List** entry exactly match one of the vendor/part number pairs defined in the AVL. If **Part Number Only** is selected, then a match will occur if the part number in the **Parts List** entry matches either the approved part number in the AVL or any of the vendor part numbers in the AVL.

4.6.3.4. Duplication Handling

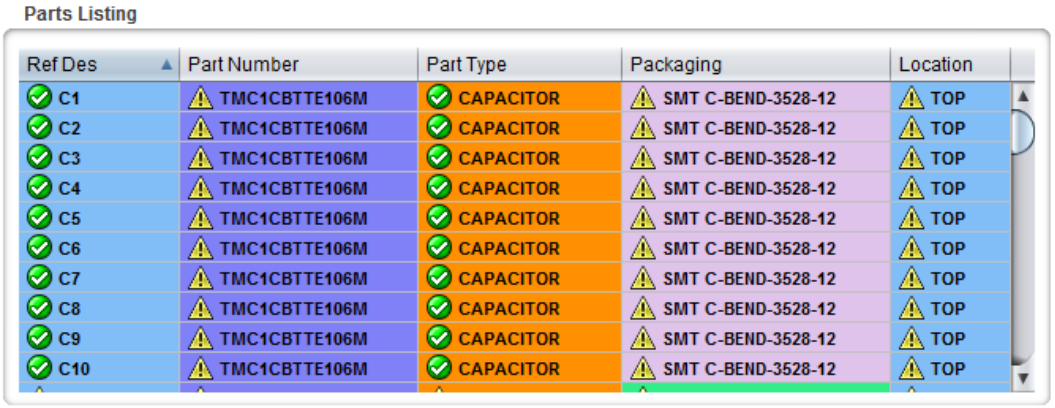
The **Duplication** mode indicates how Sherlock should react if more than one match occurs for a given part. If **Generate Error** is selected, then Sherlock will print an error to the **Sherlock Console** and will NOT update the given part in the Parts List. If **Use First Match** is selected, then Sherlock will select one of the matching AVL entries (essentially at random) and will use that data from that AVL entry to update the given part. Finally, if **Ignore Matches** is selected, then Sherlock will simply skip that part without reporting any error.

4.6.3.5. Part Property Updates

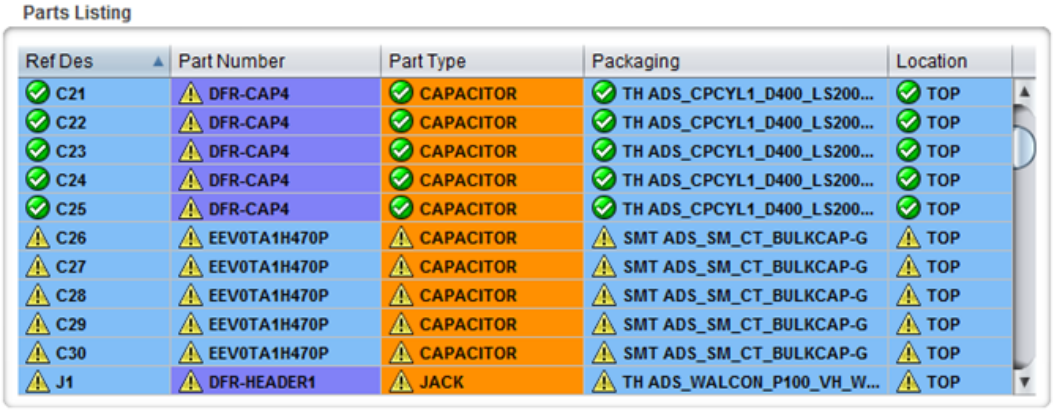
The **Part Number** and **Description** options indicate which data properties should be updated in the **Parts List** for a part that matches an AVL entry. If **Assign Internal Part Number** is selected, Sherlock will store the approved part number from the AVL entry in the **Part Number** field in the **Parts List**. If **Assign Vendor & Part Number** is selected, Sherlock will store the vendor name and part number from the AVL entry in the **Manufacturer** and **Part Number** fields in the **Parts List**. Similarly, if **Assign Approved Description** is selected, then the approved description from the AVL entry will be assigned to the **Description** field in the **Parts List.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Press the **Update Parts List** button to update the **Parts List** using the current AVL entries. After the processing has completed the Parts List will show that the **Part Number** information has been updated for most parts, as indicated by the purple highlight color.



If you double-click any of those entries, you'll see that the approved vendor/part number and description have been assigned. To store the approved part number in the **Parts List** entries instead of the vendor part numbers, use the **Update Parts List from AVL** menu option again, but select the Assign **Internal Part Number** option. After the **Parts List** is updated, the **Parts List** display shows that internal part numbers have now been assigned to most parts.



Note:

If you select both **Do No Change Vendor / Part Number** and **Do Not Change Description,** then Sherlock will delete all current AVL property settings for each matching part in the **Part List**, allowing other property value sources, such as those defined in the **Part Library**, to be used.

4.6.3.6. Validating the Parts List

One of the most common uses of the AVL is that of validating the **Parts List.** Specifically, the part number, manufacturer and/or description of each part in the **Parts List** is compared with the current AVL entries and all inconsistencies are reported. AVL validation is integrated into the **Part Validation** analysis module, which also validates the **Parts List** against entries in the **Part Library.** Together,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

these Sherlock data stores can be used to ensure that any CCA uses parts consistently across your organization. See the Part Validation (p. 263) tutorial for more details.

4.7. Part Wizard

The part libraries provided by Sherlock consist of individual part records gathered over time from a variety of sources, including the **Parts List**, manual data entry and perhaps even the automated import of external database data. As individual records, they can be updated and customized over time to suit your specific needs. However, there are literally millions of parts for which key properties (for example, part type, package type, manufacturer, etc) can be determined simply by recognizing and parsing the manufacturer's part number. For example, the part number **08055A101JAT2A** encodes information about the part in the part number itself and additional information that can be gathered from data sheets based on elements of the part number. Some of the information that can be obtained from this part number includes:

Table 4.23: Part Information for 08055A101JAT2A

Property	Value
Manufacturer	AVX / KYOCERA
Chip Size	“0805”
Dielectric	“C0G”
Capacitance	1e-4 pF
Tolerance	+/- 5%
Voltage	25V
Marking	No Mark

Although the number of encoded properties probably isn't sufficient for all Sherlock analysis needs, they are definitely a good start and should be captured whenever possible. To facilitate such automatic property capturing, Sherlock provides a **Part Wizard** that is integrated with the following part libraries:

- Sherlock Part Library
- User Defined Library

The **Part Wizard** maintains a collection of PATTERNS that are used to both recognize specific part number schemes used by various manufacturers (and user organizations) and to extract useful data from those part numbers.

In this section, the following topics are covered:

4.7.1. Part Number Recognition
4.7.2. Viewing Part Wizard Patterns
4.7.3. Editing Part Wizard Patterns
4.7.4. Editing Part Wizard Default Properties
4.7.5. Editing Part Wizard Pattern Fields
4.7.6. Editing Part Wizard Field Codes
4.7.7. Managing Part Wizard Patterns
4.7.8. Bulk Pattern Testing

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.9. Bulk Test Results
4.7.10. Bulk Pattern Matching

4.7.1. Part Number Recognition

In normal usage, the **Part Wizard** works behind the scenes to magically create part library entries whenever possible. Specifically, whenever Sherlock searches any of the supported part libraries (either because of a manual search request or to update the **Parts List**, for example) it first looks for an existing library entry that matches the given manufacturer and part number. If an entry is found, then that entry is used without further processing. If, however, no part was found and an exact part number match was requested, then the **Part Wizard** will be invoked automatically to examine the part number. If the **Part Wizard** recognizes the part number, then it will automatically extract whatever data is available (more on that later), create a new part record in the part library, and then return a reference to that part record as the search result.

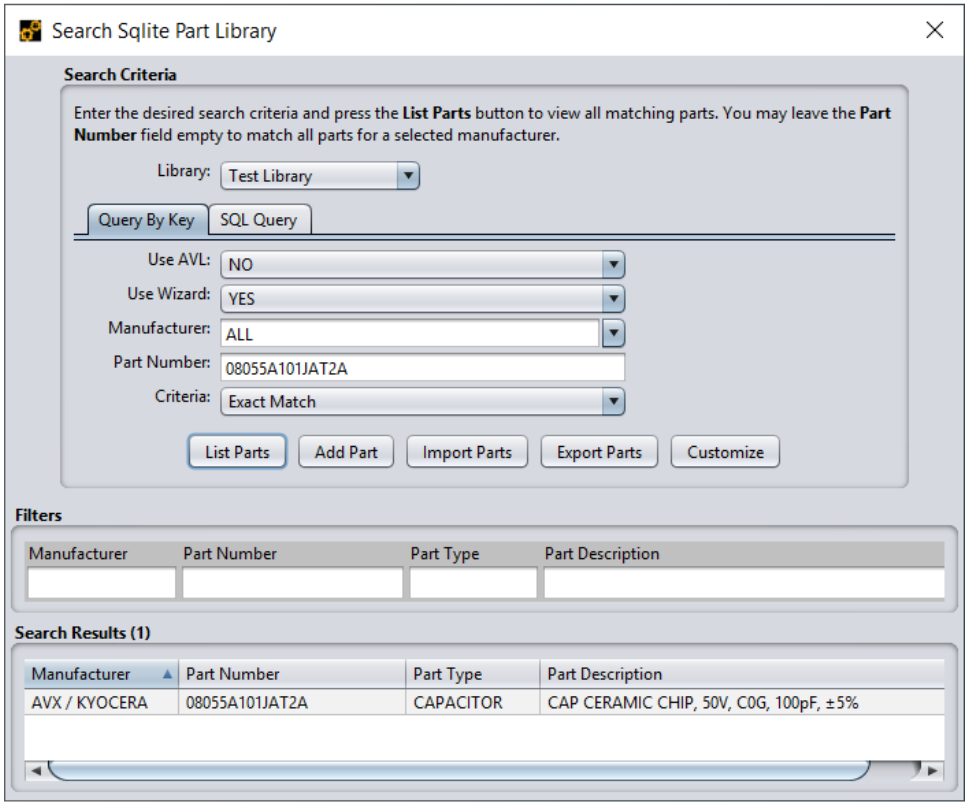
To see the **Part Wizard** in action, select **Parts > Part Library** from the Sherlock **Main Menu**, enter the following search criteria:

Manufacturer: ALL

Part Number: 08055A101JAT2A

Criteria: Exact Match

And press the **List Parts** button. Since no such record currently exists in the part library, the **Part Wizard** will be invoked automatically to recognize the part number and extract the data, generating a single search result.

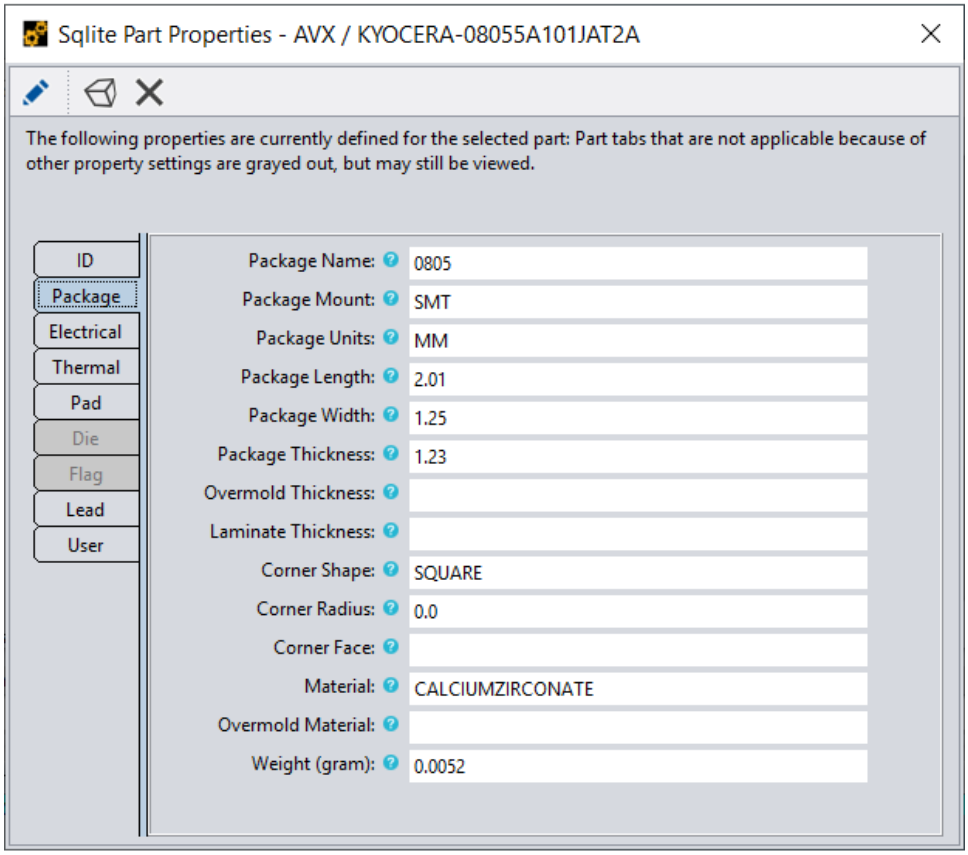


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Part Wizard** automatically determined the **Manufacturer** and **Part Type** and created a description that contains various other part properties.

If you double-click the search result to display all of the part properties, you'll see that the **Part Wizard** also determined many of the package properties associated with the part.

Since the **Part Wizard** automatically creates a new record for the part, you can edit the part properties at any time to fill out more properties or customize the part to your needs (such as changing the description). Moreover, since a part record has been created it will be returned for all subsequent searches involving the associated part number.



Note:

Part Wizard patterns are maintained in the Sherlock User Folder, allowing each user to have their own set of patterns if desired. The standard patterns defined by Ansys and all custom-defined patterns are used when accessing a **Part Library**.

Note:

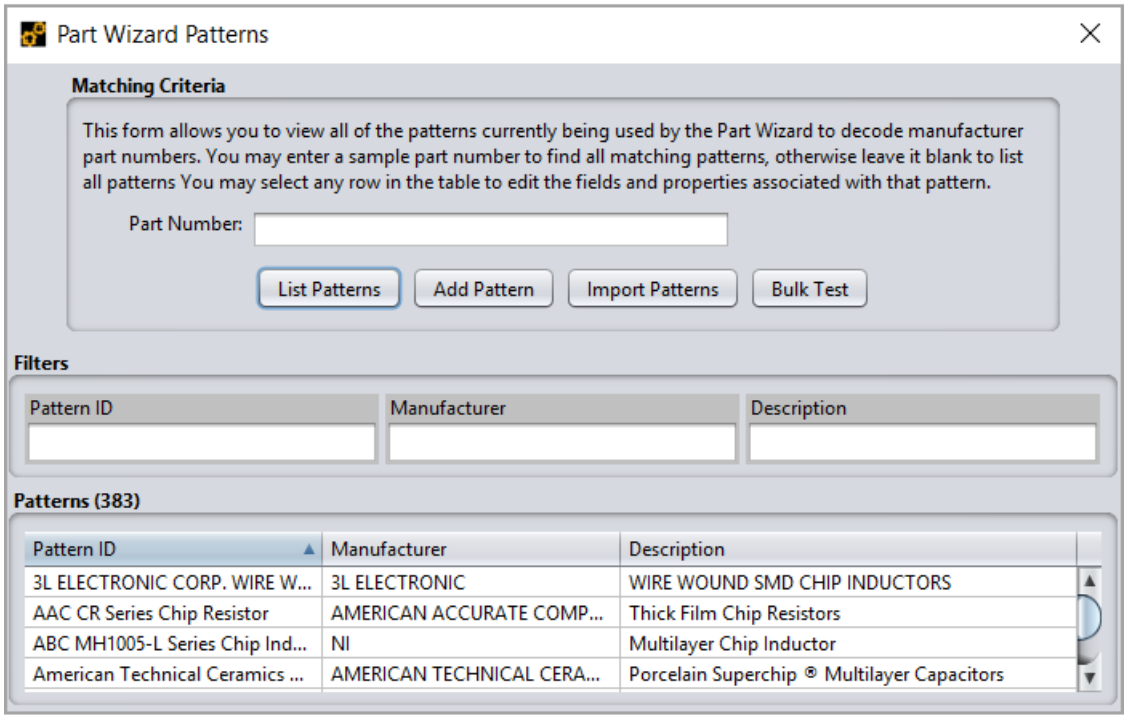
For consistency, the Sherlock Server makes use of only the standard patterns defined by Ansys when users access the **Global Part Library.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.2. Viewing Part Wizard Patterns

The **Part Wizard** uses a collection of PATTERNS to recognize part number schemes and to extract available data from the part number itself. Sherlock provides a default set of patterns that are loaded automatically the first time that Sherlock is executed after being installed and/or upgraded. You are free to add, modify or delete patterns to suit your specific requirements. You may also export/import one or more patterns to/from a ZIP file, share them with other Sherlock users and/or archive them outside of Sherlock for safe-keeping or configuration control purposes. The **Part Wizard** is designed to be an extensible repository that can be used across your entire organization.

To view all the patterns currently defined for your **Part Wizard**, select **Parts > Part Wizard Patterns** from the Sherlock main menu and press the **List Patterns** button.



When the **Part Number** field is empty, the **List Patterns** button lists all patterns currently defined. If a part number is specified, then the **List Patterns** button will display all patterns that match the given part number.

Note:

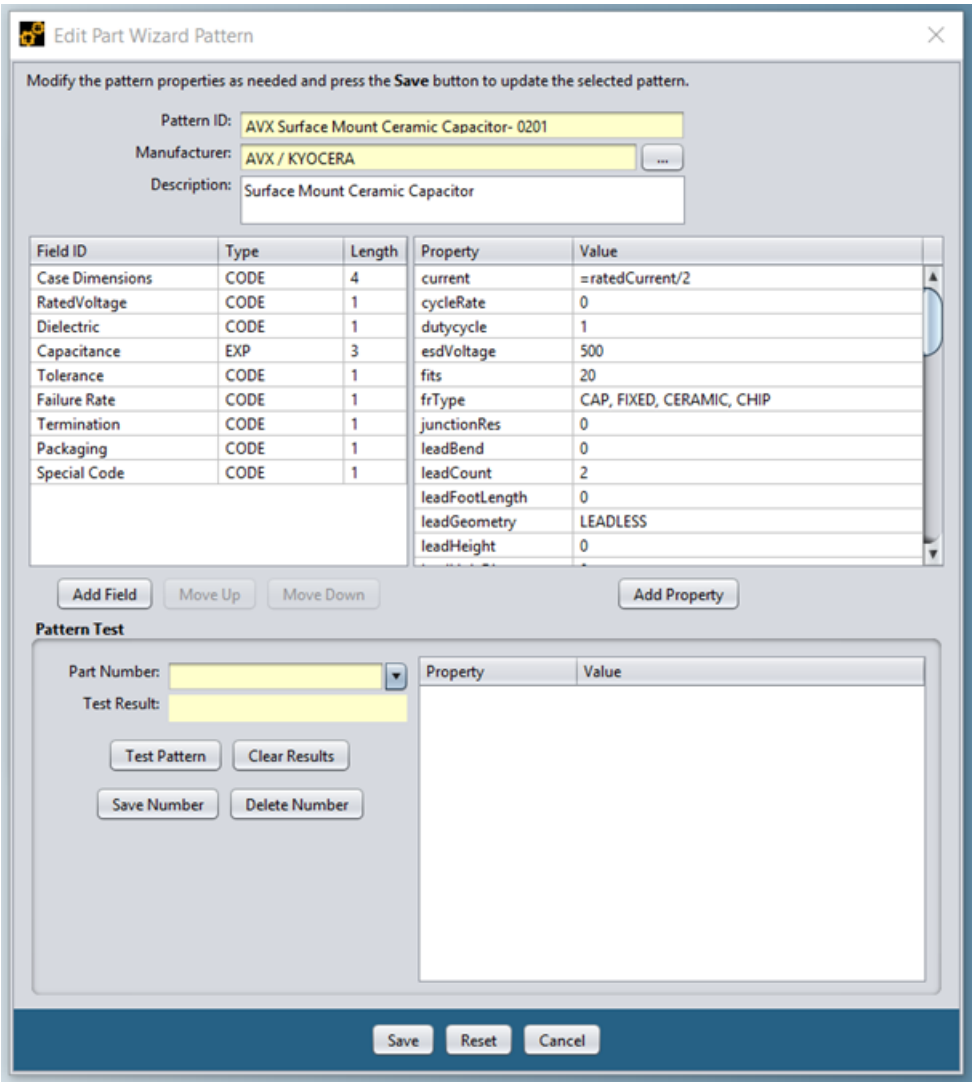
In general, patterns should be designed so that only one pattern matches a given part number. In some cases, however, manufacturer and/or organizational numbering schemes may return multiple matching patterns for the same part number.

As with other search results, you may enter values in one or more of the filter fields to show only those patterns matching the filter conditions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.3. Editing Part Wizard Patterns

You may edit the properties associated with a given pattern by either double-clicking the entry in the **Patterns** list or right-clicking the pattern list entry and selecting **Edit Selected Pattern** from the context menu.



The **Part Wizard** Pattern dialog shows the unique **Pattern ID** assigned to the pattern, as well as the **Manufacturer** associated with the pattern. A description field is provided for configuration management purposes and is not otherwise used by Sherlock.

Each pattern consists of one or more **Pattern Matching Fields** that indicate the type (format) and length of the data values encoded in the part number. You may add and re-order the fields using the buttons below the field list. You may also edit, delete and re-order a specific field by right-clicking the field in the list and selecting the appropriate context menu item.

The list of properties shown on the right-side of the dialog define the default part properties associated with any part number that matches this pattern. That is, when the **Part Wizard** creates a part record for a part number that matches the current pattern, all the properties listed on the right will be used as the default part properties. Specific part properties extracted by any of the fields (more on this shortly) will be used to append the default property values.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The **partType** property MUST be defined by ALL patterns. Usually, the **partType** value is the same for all parts matched by the pattern and is specified as a default property value. However, the **partType** value can also be defined based on one of the pattern fields if required.

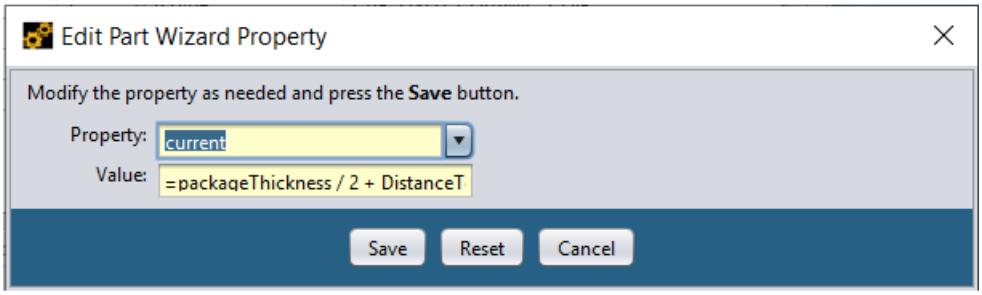
The **Pattern Test** form at the bottom of the dialog provides a convenient method for testing the pattern definition as it is being created or modified. To use it, simply enter a test string in the **Part Number** field and press the **Test Pattern** button. If the pattern matches the test string, all the property values extracted from the test string will be listed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

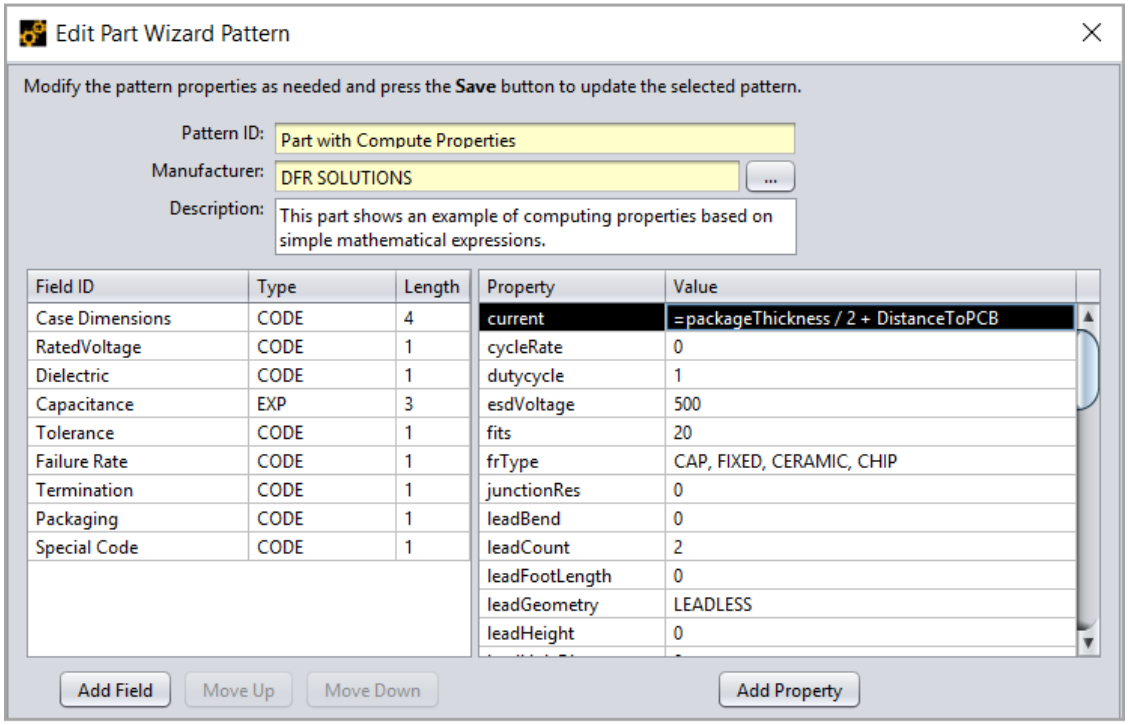
4.7.4. Editing Part Wizard Default Properties

Each **Part Wizard** pattern may also define a list of part properties to receive hard-coded values. In the above example, the package mount, part description, and part type are assigned. As previously noted, since the part description is assigned as a default part property and assigned to many of the fields in the wizard pattern, the final part description will be the concatenation of the default value plus all the defined fields that assign a value to this field.

In addition to assigning hard-coded values, part properties can be assigned values based on simple mathematical expressions and string Boolean expressions. A hypothetical example expression may define the part property **leadHeight** as shown below. To define an expression instead of a hard-coded value, simply start the value of the part property with an equal sign followed by the expression.



Variables may be the names of any valid part property names or may reference any of the **Field ID** names for the pattern itself. When referencing **Field ID** names, if the name contains spaces, the variable name is the same as that **ID** without the spaces. For example, a field **ID** value of **Distance To PCB** would be referenced in an expression as **DistanceToPCB**. See the examples below.



When expressions are encountered by the **Wizard** matching engine, they are evaluated last so that all assigned part properties are in place in order, to compute the expression.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expressions are evaluated using the precedence rules found in Java. Parentheses can be used to control the order of precedence. Conditional expressions can be nested with parentheses.

4.7.4.1. Supported Operators

The following is the list of supported operators that may be used to define part property values:

Table 4.24: Parts Wizard- Supported Operators

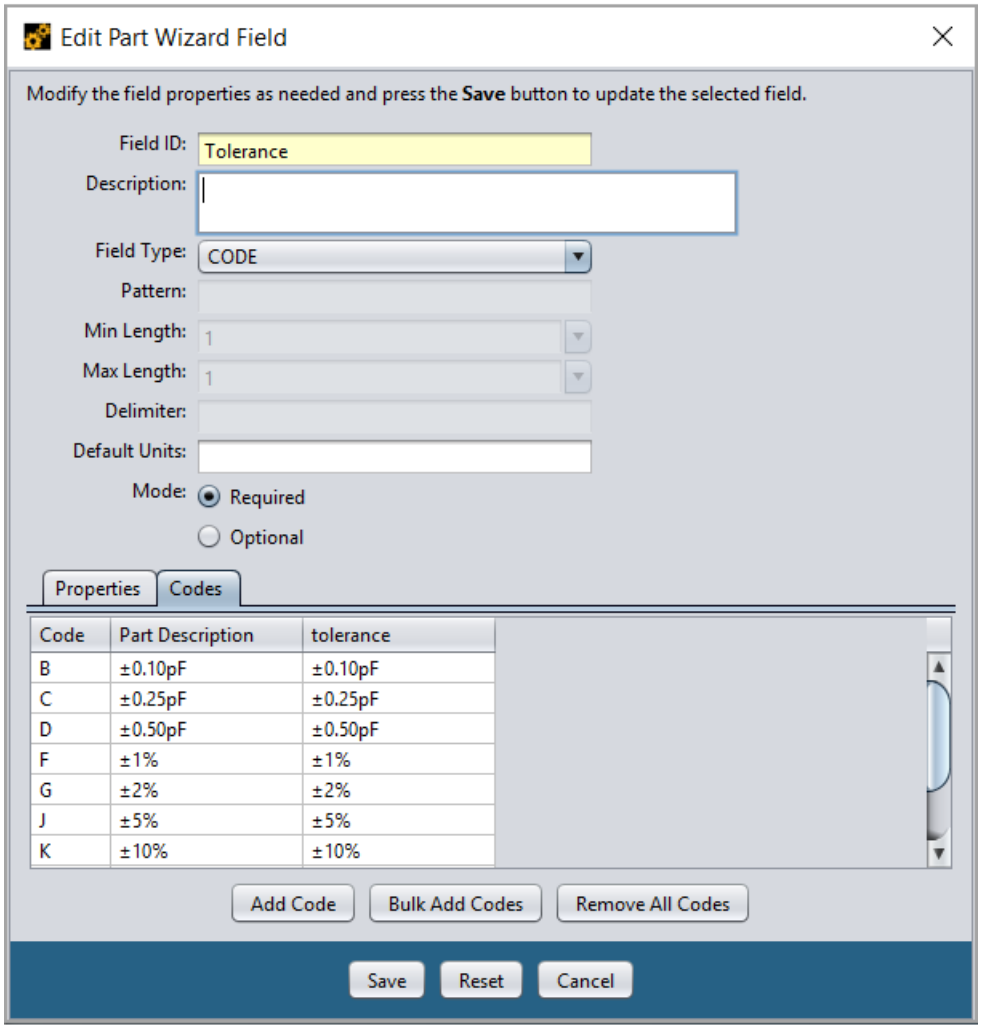
Basic Arithmetic Operators	Description
+	Addition: *2+2*
-	Subtraction: *5-3*
*	Multiplication: *2*2*
/	Division: *9/3*
%	Remainder: *10%3*
abs	Absolute value: *abs -23*
pow	Power: *8 pow 2*
int	Round to integer: *int 8.4*
sqrt	Square root of number: *sqrt 81*
Conditional Expressions	
Condition ? If-true : if false	Ternary: *x > y ? x : y*
Comparison Operators for Conditional Expressions	
<	Less than
<=	Less than or equals
==	Equals
>	Greater than
>=	Greater than or equals
!=	Not equals
<>	Not equals
Boolean Operators for Conditional Expressions	
&&	And: *x > y && x != 4 ? x : y*
||	Or: *x <= y || x == 4 ? y : x*
String Operators for Conditional Expressions	
equals, equalsIgnoreCase	var.equals(“abc”) ? 1 : 2
notEquals, notEqualsIgnoreCase	var.notEquals(“abc”) ? 1 : 2
contains, containsIgnoreCase	var.contains(“xyz”) ? 1 : 2
notContains, notContainsIgnoreCase	var.notContains(“xyz”) ? 1 : 2

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.5. Editing Part Wizard Pattern Fields

To edit a specific pattern matching field, double-click the field listed in the **Part Wizard Pattern** dialog to display the **Edit Part Wizard Field** dialog.

In the example below, we see that the **Tolerance** field is defined as a **CODE** field, which must consist of one of the codes listed in the **Codes** tab. If one of the codes is found in the part number at the location associated with this field (as determined by the lengths of the previous fields), then this field will be deemed a match and the value associated with that code will be extracted to zero or more part properties defined in the **Properties** tab. In this case, the **partDescription** property is assigned to this field, so the extracted value will be appended to the part's description property.



If the same part property is assigned to more than one field, then the value extracted for each field will be appended to that part property, separated by commas.

The **Pattern**, **Min Length**, and **Max Length** fields are enabled depending on the type of field being defined (more on that shortly).

The **Units** field allows you to specify a string to be appended to the extracted value, usually to indicate the units associated with the field value.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Mode** field indicates whether this field *must* be found in the **Part Number** (that is, it's required) or *may* be found in the **Part Number** (that is, it's optional).

Note:

Unfortunately, manufacturers don't always document which fields are optional and part numbers don't always contain all the required fields, so you may have to create alternate patterns to properly recognize variations encountered with real-world part numbers.

The following types of pattern matching fields are supported by the **Part Wizard**:

Table 4.25: Supported Pattern Matching Fields

Field Type	Description
ALPHA	Field contains a sequence of alphabetic characters (a-z, A-Z) with a specified minimum and maximum length
ALPHANUMERIC	Field contains a sequence of alpha-numeric characters (a-z, A-Z, 0-9) with a specified minimum and maximum length
CODE	Field contains one of the codes listed in the Codes tab
EXP3	Field contains a 3-digit exponential value consisting of 2 significant digits followed by the number of zeros (for example, 103 = 10e3) or two digits separated by an alphabetic character (for example, 3V3 = 3.3). When the character is a K, the value is multiplied by 1,000. When the character is a M, the value is multiplied by 1,000,000. By default the alphabetic character can be any letter, but can also be defined as a specific character in the Delimiter field.
EXP	Field contains a 3 or more-digit exponential value consisting of 2 or more significant digits followed by the number of zeros or some number of digits with an alphabetic decimal separator. The alphabetic decimal separator may appear anywhere in the sequence whereas the EXP3 version requires the separator to be immediately before the final single decimal digit (for example, 3R14 = 3.14). When the character is a K, the value is multiplied by 1,000. When the character is a M, the value is multiplied by 1,000,000. By default the alphabetic character can be any letter, but can also be defined as a specific character in the Delimiter field.
SCALED	Numeric value that ends with a K or an M character which forces the value to be multiplied by 1,000 or 1,000,000.
NUMERIC	Field contains a sequence of digits with a specified minimum and maximum length
REGEX	Field contains a sequence of characters defined by the Java regular expression defined in the Pattern field.
IPC_TPAD	Used by IPC-7351B to indicate a value which includes both the pin quantity and the existence of a thermal pad. When odd values are found, a value of one is subtracted from the value indicated. (e.g. 29 = 28, 14=14)
DECIMAL1	Used to indicate a numeric value one place past the decimal point, including trailing zeros. (e.g. 32 = 3.2)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DECIMAL2	Used to indicate a numeric value two places past the decimal point, including trailing zeros. (e.g. 200 = 2.00, 52 = 0.52)

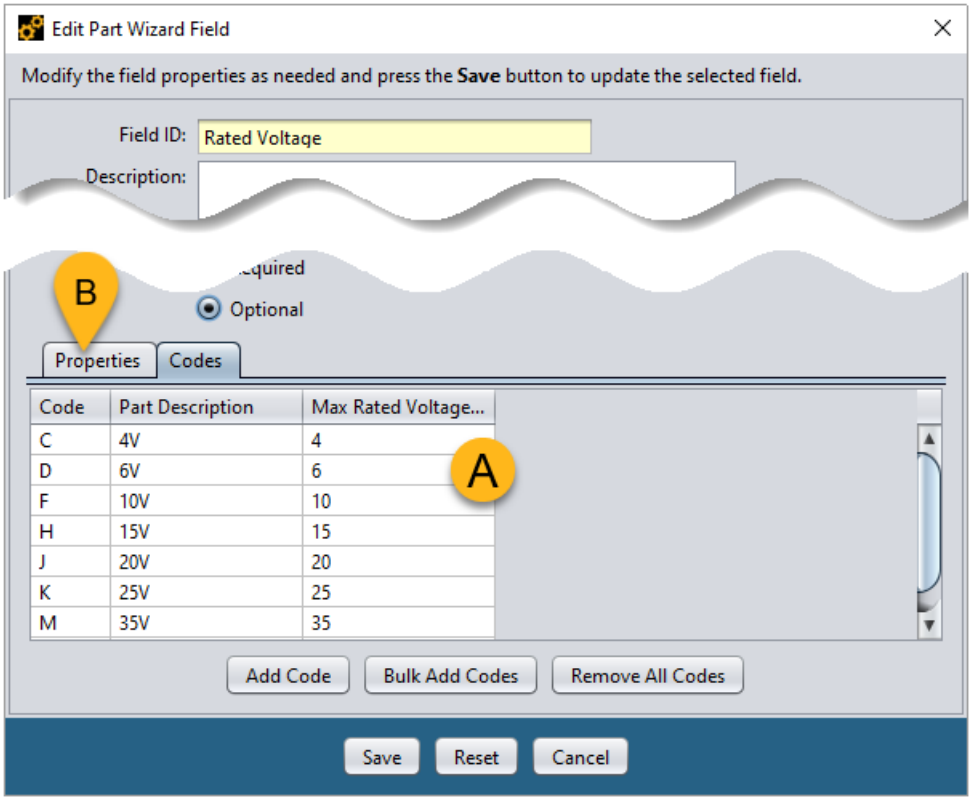
Note:

After saving changes to any pattern matching field you should always re-test the pattern using the **Test Pattern** form provided in the pattern edit dialog.

4.7.6. Editing Part Wizard Field Codes

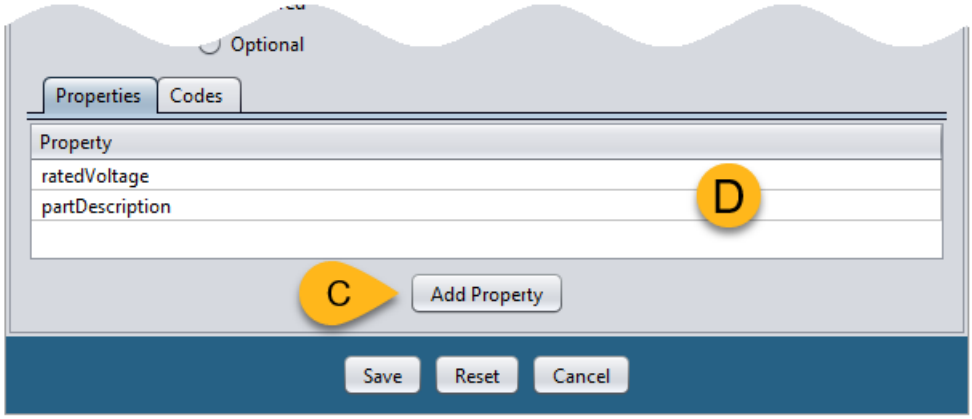
A code field type is a list of codes (see A, below) which is compared to the provided part number and is then used to assign part properties based on the value of the code. In the example below, a code field type for **Rated Voltage** is shown. It has been assigned two part properties: **Part Description** and **Max Rated Voltage**. For each code, a list of values for each of the part properties is specified.

If no part properties are associated to a code, then the field is simply used for pattern matching to help identify a part number.



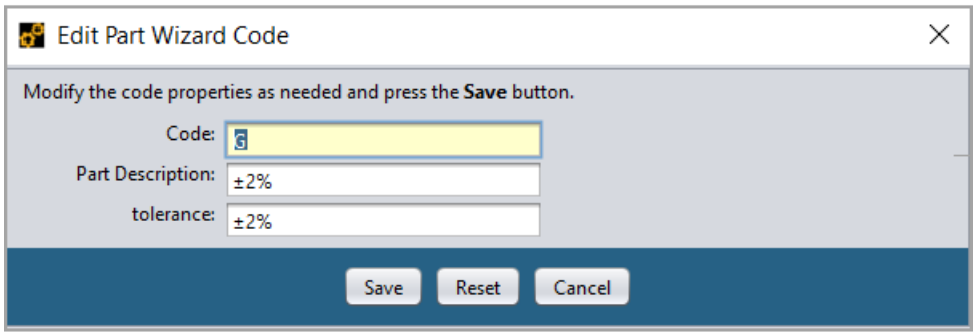
To edit the properties associated to a code field, click the **Properties** tab (B, above). Part properties can then be added by clicking the **Add Property** button (C, below). Or, in the Properties table (D), double-click the row you wish to edit, or right-click the row and select **Edit Property** from the context menu. To delete a row, right-click the row, then select **Delete Property**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The first part property added to a code will automatically be associated to the default values previously defined for the list of codes in the field. When deleting the final property from a code, the values of that final part property will become the default values for the code.

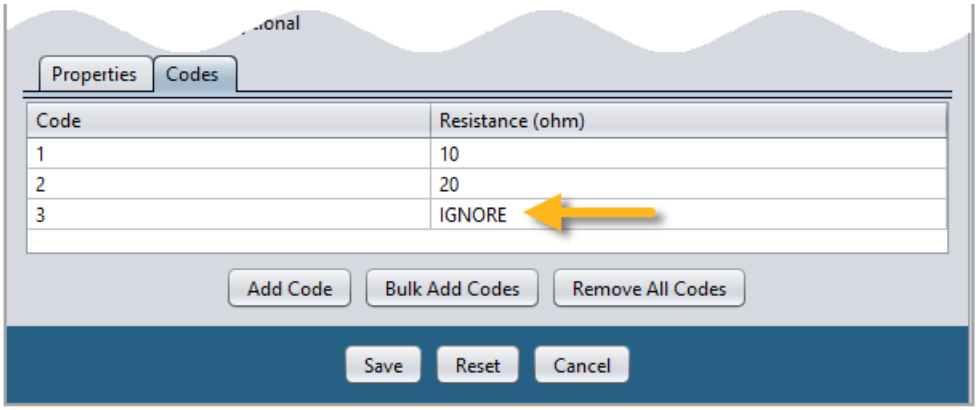
To assign or edit any of the part properties assigned to a code, select the **Codes** tab. There you may edit existing values in the table directly, or you can double-click or right-click a row and select **Edit Code** which will open the **Edit Part Wizard Code** dialog shown below.



Multi-cell cut and paste also works from the table shown in the **Codes** tab, allowing codes to be edited in an external program and copied from the clipboard. To copy the current code table to the clipboard, select all rows and columns desired, then press Ctrl-C or right-click the table and select **Copy** from the menu. To paste values back to the table, select the first row and column to receive the data, and press Ctrl-V or right-click the table and select **Paste** from the menu. If the data you paste has more columns than are what defined, they will be ignored. Additional rows, however, will be added.

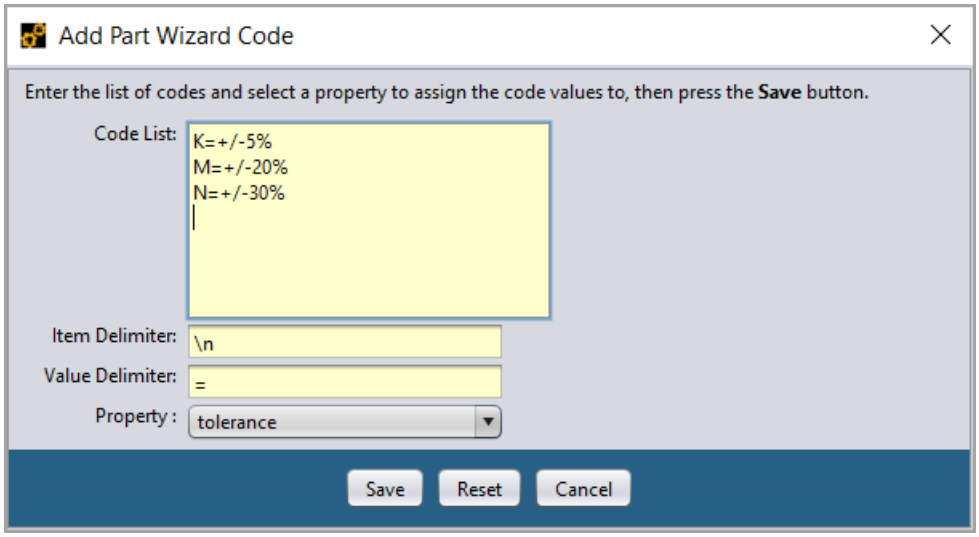
When editing a code, you can assign the value IGNORE for a part property as shown in the **Edit Part Wizard Field** dialog below. This means the code is not applicable for that property and should not be not included in the pattern result. In the example below, if **Code** = 1, the **Resistance** is set to 10. If the **Code** = 3, **Resistance** is ignored and will not be returned in the patterns match test.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you are finished editing code fields, press the **Save** button to make the changes permanent.

The **Bulk Add Codes** button on the **Part Wizard** Field dialog allows a user to more quickly add a large set of codes instead of using the **Add Code** dialog for each code. In the dialog, enter each code and code values in the text area separated by the value delimiter and each code separated by the item delimiter. When part properties are defined for a code, select the part property to receive the defined code value. If multiple part properties are defined and they need to be assigned values, each code needs to be manually edited as indicated above and the part property assigned.



4.7.7. Managing Part Wizard Patterns

4.7.7.1. Adding / Copying a Part Wizard Pattern

A new pattern can be added by pressing the **Add Pattern ...** button displayed in the **Matching Criteria** section of the **Part Wizard** dialog. At that point, the edit pattern dialog will be displayed with empty fields. At a minimum you must enter a new, unique Pattern ID, a Manufacturer name and at least one pattern matching field to save the new pattern.

You can also create a copy of a pattern by right-clicking a pattern list entry in the **Patterns** list and selecting **Copy Selected Pattern** from the context menu. At that point, the edit pattern dialog will be displayed with the same properties defined for the pattern being copied. At a minimum you must enter a new, unique Pattern ID to save the new pattern.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.7.2. Deleting Part Wizard Patterns

You may delete one or more patterns by selecting them in the **Patterns** list, right-clicking any hilighted item and selecting **Delete Selected Patterns** from the context menu. A confirmation dialog will appear asking you to confirm deletion of the selected patterns.

4.7.7.3. Exporting Part Wizard Patterns

One or more patterns can be exported by selecting them in the **Patterns** list, right-clicking any hilighted item and selecting **Export Selected Patterns** from the context menu. At that point, an export dialog will appear allowing you to specify the file to be exported.

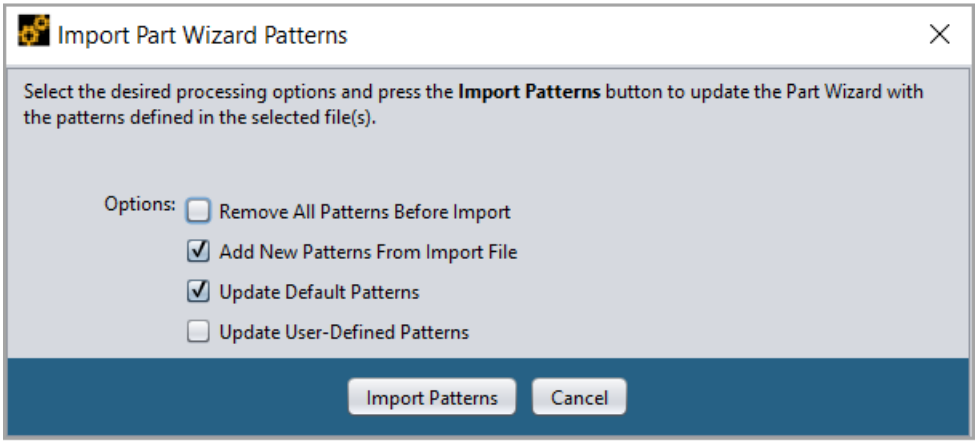
You may export the pattern definitions to a single XML file or to a ZIP file containing individual XML files. In either case, the exported patterns can be emailed to another Sherlock user, archived in the file system for safe-keeping or maintained in a configuration management system.

4.7.7.4. Importing Part Wizard Patterns

One or more patterns can be imported from a ZIP file or a directory by pressing the **Import Patterns** button displayed in the **Matching Criteria** section of the **Part Wizard** dialog. At that point, an import dialog will appear allowing you to select the file to be imported.

You may import pattern definitions from a single XML file or from a ZIP file containing individual XML files.

After the import file has been selected, you will be prompted to select the desired processing options.

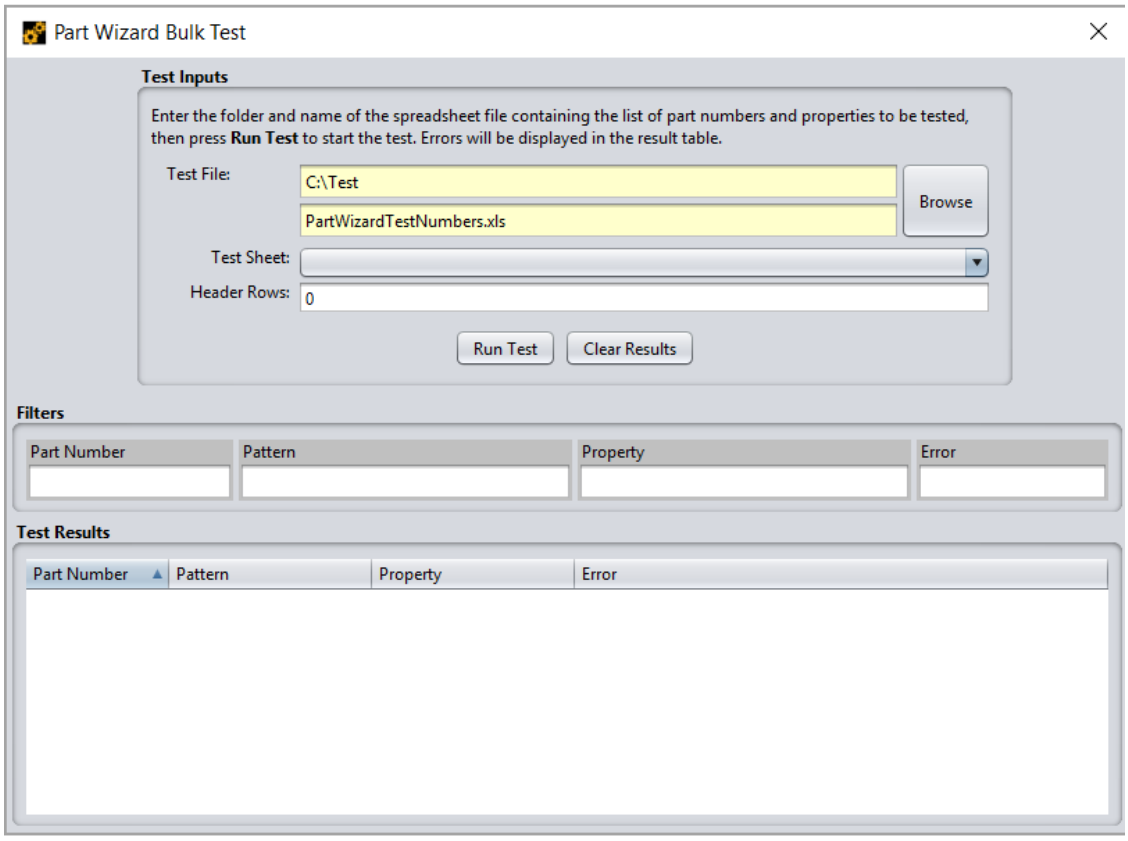


In normal operations, the options shown here should suffice to add new patterns and update any default patterns defined by Sherlock. If you have previously edited patterns (that is, **User-Defined Patterns**) and you want to update those patterns as well using the import file, then you should enable the last processing option. If you want to completely replace all patterns with those in the import file, then you should select the first processing option.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.8. Bulk Pattern Testing

The **Bulk Testing** feature allows you to maintain a collection of part numbers and associated part properties in one or more spreadsheet files and then test all **Part Wizard** Patterns against those part numbers. To use the bulk test feature, simply press the **Bulk Test** button in the **Part Wizard** Dialog to display the following dialog:



The **Test File** field is used to designate the location and name of the spreadsheet file containing the part numbers to be tested, with the following spreadsheet file formats supported:

- Excel 97/2000/XP (*.xls)
- Excel XML (*.xlsx)
- Comma Separated Value (*.csv)

Excel spreadsheet files may include multiple spreadsheets. In that case, the **Test Sheet** selection list is used to designate the sheet to be processed. Whenever the **Test File** field is changed, Sherlock will open the file and update the **Test Sheet** list with the list of spreadsheets found. For CSV files, only a single spreadsheet exists in the file and the **Test Sheet** list will be empty.

The **Header Rows** field is used to designate the number of rows to be skipped at the top of the spreadsheet before finding the row that specifies the column names. Sherlock attempts to guess the number of header rows whenever the **Test Sheet** selection is made, but you may need to enter the value manually for unusually formatted spreadsheets.

The spreadsheet used to store the part numbers and associated part properties must be defined as follows to be used properly by the Bulk Test feature:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The first row of data (after the header rows have been skipped) must contain column names consisting of valid Part Property **identifiers**, which should correspond to the part properties generated by the patterns being tested. For a complete list of part property identifiers, select **Help > Part Properties** from the Sherlock main menu.
- The special column name **PartNumber** (case-sensitive) must be defined in the spreadsheet to indicate the column containing the part numbers to be tested.
- The remaining columns should contain the part property values expected to be generated by a **Part Wizard** pattern. If a part property is not expected for a given part number, the cell value should be empty.

For example, here's an excerpt from the top of a spreadsheet containing AVX part numbers:

	A	B	C	D	E	F	G	H
1	PartNumber	partManufacturer	partDescription	capacitance	packageMount	packageName	partType	ratedVoltage
2	08055A101J	AVX	CERAMIC, 0805, 50V, 100pF, +/-5%	1.00e-4	SMT	0805	CAPACITOR	50.0
3	08055A101JAT2A	AVX	CERAMIC, 0805, 50V, 100pF, +/-5%	1.00e-4	SMT	0805	CAPACITOR	50.0
4	08055C103KAT1A	AVX	CERAMIC, 0805, 50V, 10.0nF, +/-10%	0.010	SMT	0805	CAPACITOR	50.0
5								

MISC AVX TDK

The AVX pattern should generate seven part properties for each of the part numbers listed. Since spreadsheet cells may contain either text or numeric data, Sherlock automatically attempts to compare values numerically if possible. This eliminates nuisance differences, like numbers formatted as decimal values in the spreadsheet, but as exponential numbers by the pattern.

Note:

It is best to store all cell values as TEXT values in the spreadsheet to avoid confusion when cutting and pasting values from the **Part Wizard** editor.

4.7.9. Bulk Test Results

When the **Run Test** button is pressed, Sherlock will process all part numbers found in the designated spreadsheet. For each part number, all matching patterns will be executed, and the part properties generated by each pattern will be compared to those defined in the spreadsheet. If part property values differ or no pattern is found that matches the part number, an error will be displayed in the **Test Results** table.

Each row in the table shows the pattern that matched the part number (if any), the property generated by the pattern and type of problem detected. You can clear individual rows in the **Test Results** table by selecting one or more rows, right-clicking one of the selected rows and selecting **Clear Result(s)** from the context menu. This allows the **Test Results** table to be used as a to-do list when fixing patterns and/or test files. Press the **Clear Results** button at the top of the dialog to clear the entire **Test Results** table.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7.10. Bulk Pattern Matching

The **Bulk Test** feature can also be used to perform simple pattern matching tests to determine if part numbers are being matched properly by specific patterns. Pattern matching tests are automatically performed if the following three columns are defined in each test spreadsheet:

- PartNumber
- Pattern
- Match

The **PartNumber** and **Pattern** columns should contain a list of part numbers and patterns to be tested. If the corresponding value in the **Match** column is **yes**, then the specified pattern should match the specified part number. Otherwise, the pattern should not match the specified part number. For each part number, Sherlock will determine if the pattern matches the part number and will compare that result to the expected result specified by the **Match** column. Any inconsistencies will be noted in the **Test Results** table.

Note:

All other columns defined in the spreadsheet will be ignored when the **Pattern** and **Match** columns are defined.

4.8. Sherlock Part Properties

Sherlock makes use of an assortment of part properties to identify parts, locate them on the PCB, determine package characteristics, and reliability aspects, as well as other processing needs. This document lists all of the part properties used by Sherlock. The identifier shown in parentheses for each property is the unique value used internally by Sherlock to designate the property. When importing part properties, Sherlock will automatically recognize spreadsheet columns using either the part property identifier or the part property label. Properties tagged with [DB] may be stored in the Sherlock Part Database to be shared across CCAs and projects. Properties tagged with [PKG] may be stored in the Package Database and copied to each part assigned a given package name.

Tip:

This document is available in PDF form within Sherlock. Click **Part Properties** in Sherlock's **Help** menu.

Section Links:

4.8.1. A - Part Properties
4.8.2. B - Properties
4.8.3. C - D Properties
4.8.4. E - J Properties
4.8.5. L - Properties
4.8.6. M - Properties

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.8.7. N - O Properties
4.8.8. P - Properties
4.8.9. Q - R Properties
4.8.10. S - Properties
4.8.11. T - Y Properties

4.8.1. A - Part Properties

Return to top of list (p. 294)

Activation Energy (Ea) (activationEnergy)

Arrhenius based acceleration factor (in eV). This is only applicable for Prediction (Type 2) and Test Data input types.

Applied Current (A) (opCurrent) [DB]

Operating current (A) for the component.

Applied Duty Cycle (%) (opDutyCycle)

The applied duty cycle.

Applied Freq (MHz) (opFrequency) [DB]

Operating frequency in MHz for crystal components and for IC components used in the calculation of acceleration factors in Semiconductor Wearout analysis.

Applied Power (W) (powerDisp) [DB]

Average amount of power in watts dissipated by the component.

Applied Ripple (mA) (rippleCurrentApplied)

Operating ripple current (mA).

Applied Voltage (V) (opVoltage) [DB]

Operating voltage (V) for IC components used in the calculation of acceleration factors in Semiconductor Wearout analysis.

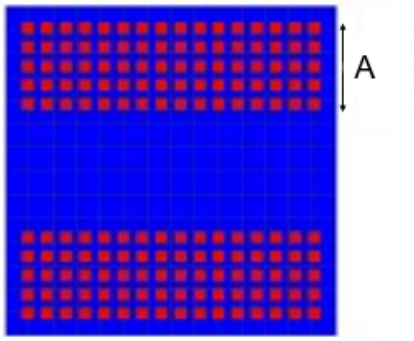
4.8.2. B - Properties

Return to top of list (p. 294)

Ball Chan Width (ballChanWidth) [DB] [PKG]

Width of the outer ball region for BGA components with PERIMETER, ISLAND or CHANNEL layouts (A, in the image below). The "Ball Units" property specifies the units for this property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

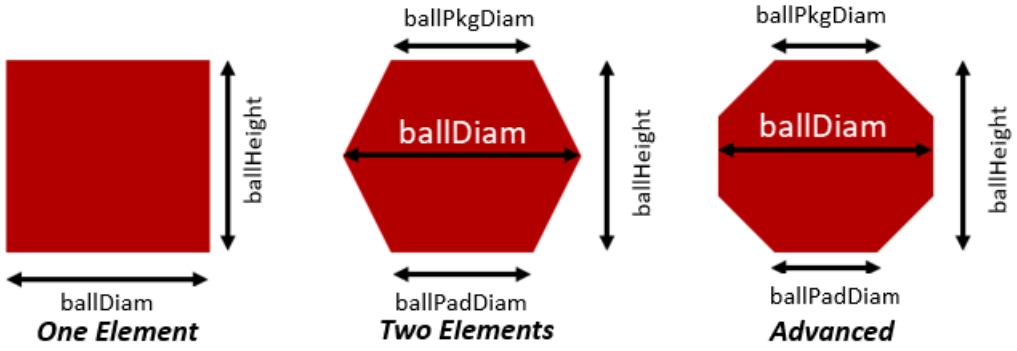


Ball Count (ballCount) [DB] [PKG]

Number of solder balls for a BGA component.

Ball Diameter (ballDiam) [DB] [PKG]

Maximum diameter of BGA solder ball measured in the horizontal plane midway between package and pad. (See the image below.) The "Ball Units" property specifies the units for this property.

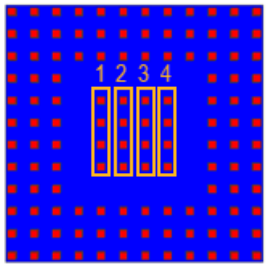


Ball Height (ballHeight) [DB] [PKG]

Maximum height of a solder ball for a BGA component. (See the image above.) The "Ball Units" property specifies the units for this property.

Ball Island Cols (ballIslandCols) [DB] [PKG]

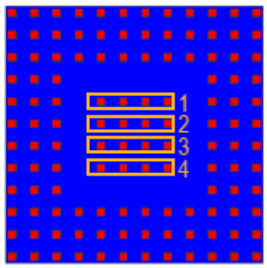
Number of solder ball columns in the BGA island. In the example below, four columns are highlighted.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ball Island Rows (ballIslandRows) [DB] [PKG]

Number of solder ball rows in the BGA island. In the example below, four rows are highlighted.

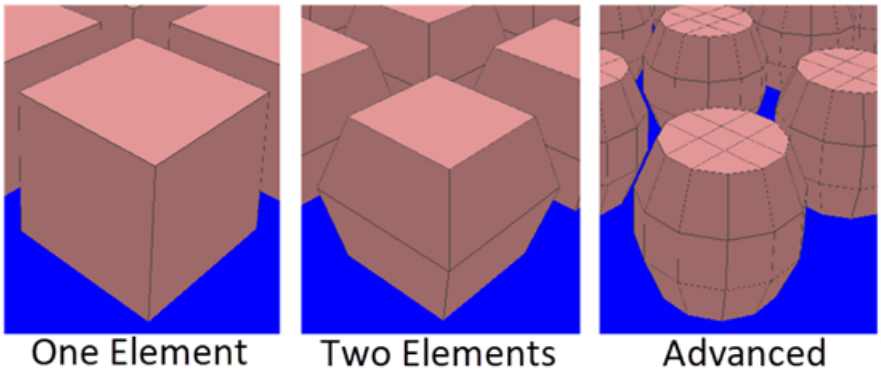


Ball Material (ballMaterial) [DB] [PKG]

Type of solder ball used to connect the BGA to the PCB. Supported values are defined in the solder data file provided by Sherlock and/or a user-defined data file.

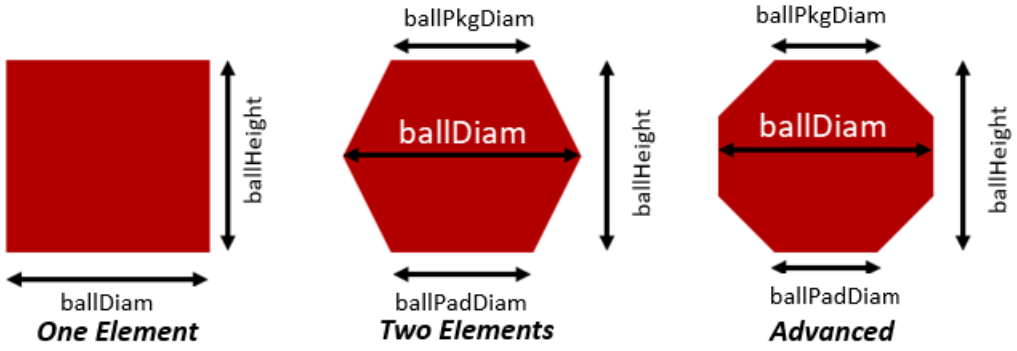
Ball Modeling (ballModel)

Indicates if the solder balls associated with this part should be modeled when building FEA models. Supported values are NONE, ONE ELEMENT, TWO ELEMENTS, and ADVANCED.



Ball Package Diameter (ballPkgDiam) [DB] [PKG]

Diameter of BGA solder ball where the ball attaches to the package. (See the image below.) The "Ball Units" property specifies the units for this property.



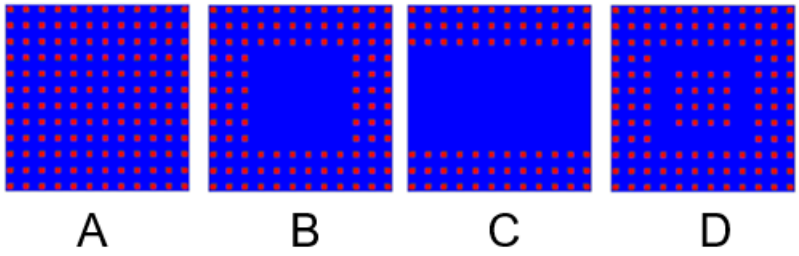
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ball Pad Diameter (ballPadDiam) [DB] [PKG]

Diameter of BGA solder ball where the ball attaches to the pad. The "Ball Units" property specifies the units for this property. See the image above.

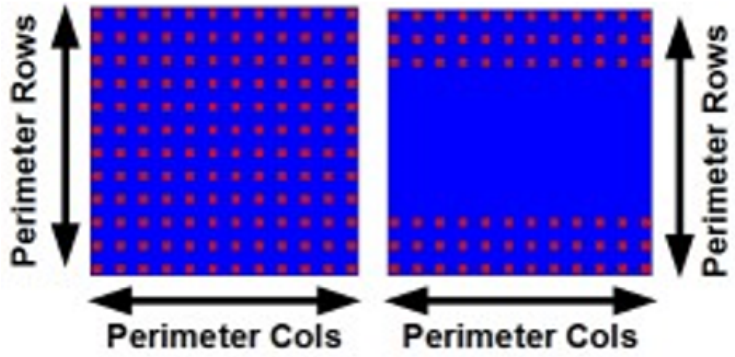
Ball Pattern (ballPattern) [DB] [PKG]

Arrangement pattern of all solder balls for a BGA component. Supported values are FULL (A, below), PERIMETER (B), CHANNEL (C), ISLAND (D), NETLIST, and UNKNOWN.



Ball Perimeter Cols (ballPerimeterCols) [DB] [PKG]

Number of solder ball columns along the BGA perimeter. See the image below.

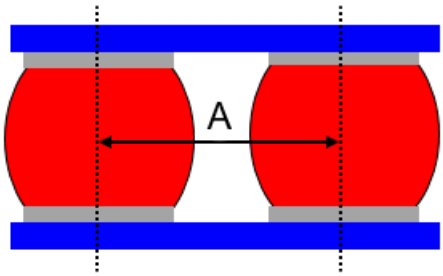


Ball Perimeter Rows (ballPerimeterRows) [DB] [PKG]. See the image above.

Number of solder ball rows along the BGA perimeter.

Ball Pitch (ballPitch) [DB] [PKG]

Distance between adjacent solder balls for a BGA component (A in the image below). The "Ball Units" property specifies the units for this property.



Ball Units (ballUnits) [DB] [PKG]

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit of length for all BGA properties. Supported values are IN, MM, M, MIL, MICRON and NM.

Board Side (locSide)

Side of board on which the component exists. Supported values are TOP and BOTTOM.

4.8.3. C - D Properties

Return to top of list (p. 294)

Capacitance (uF) (capacitance) [DB]

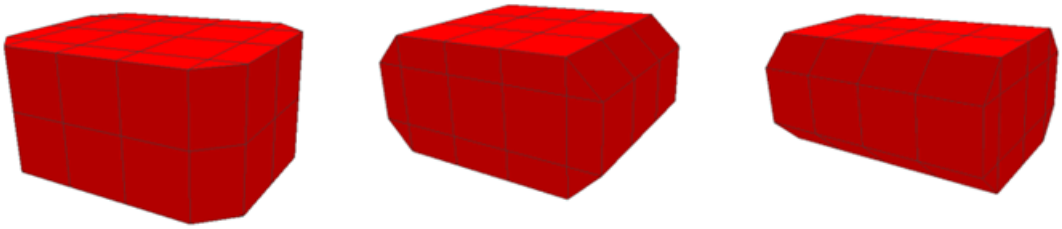
Capacitance in micro-Farads.

Confidence Bounds (%) (testConfidence)

Desired confidence bounds extracted from the test for semi-conductor wearout.

Corner Face (packageCornerFace) [DB] [PKG]

Orientation of the package corners. Supported values are TOP_BOTTOM, FRONT_BACK and LEFT_RIGHT.



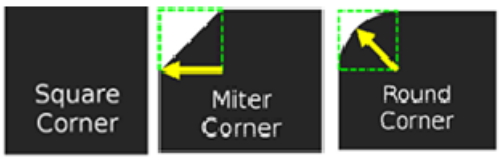
Corner Face = TOP_BOTTOM Corner Face = FRONT_BACK Corner Face = LEFT_RIGHT

Corner Radius (packageCornerRadius) [DB] [PKG]

Radius of the package corner when "Package Corner" is MITER or ROUND. The "Package Units" property specifies the units for this property.

Corner Shape (packageCorner) [DB] [PKG]

Shape of the package corners. Supported values are SQUARE, MITER and ROUND.



Created By (userCreatedBy) [DB]

The username (and timestamp) of the person who created the library item definition.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Die Length (dieLength) [DB] [PKG]

Length of the die (typically the maximum dimension). The "Die Units" property specifies the units for this property.

Die Material (dieMaterial) [DB] [PKG]

Primary material of the DIE. Material names are defined in the material data file provided by Sherlock and/or a user-defined data file.

Die Thickness (dieThickness) [DB] [PKG]

Thickness (height) of the die. The "Die Units" property specifies the units for this property.

Die Units (dieUnits) [DB] [PKG]

Unit of length for all die properties except "Feature Size". Supported values are IN, MM, M, MIL, MICRON and NM.

Die Width (dieWidth) [DB] [PKG]

Width of the die (typically the minimum dimension). The "Die Units" property specifies the units for this property.

Documents (wearoutDocuments)

These are documents referenced for the input semiconductor wearout data. (Optional)

4.8.4. E - J Properties

Return to top of list (p. 294)

Failure Class (frType) [DB]

The electronic component characterization pertaining to device type/application, package construction, general complexity, and various other factors used to determine a base handbook failure rate.

Failure Mechanism (componentFailureMode)

This property selects specific Component Failure Mechanism Analysis options which are vendor specific. When a vendor is selected which uses these options, this field will be enabled.

Flag Length (flagLength) [DB] [PKG]

Flag length (usually maximum dimension) for the IC component. The "Flag Units" property specifies the units for this property.

Flag Thickness (flagThickness) [DB] [PKG]

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flag thickness (height) for the IC component. The "Flag Units" property specifies the units for this property.

Flag Units (flagUnits) [DB] [PKG]

Unit of length for all flag properties. Supported values are IN, MM, M, MIL, MICRON and NM.

Flag Width (flagWidth) [DB] [PKG]

Flag width (usually minimum dimension) for the IC component. The "Flag Units" property specifies the units for this property.

Frequency Exponent (frequencyExponent)

Power law based accelaration factor (m). This is only applicable for Prediction (Type 2) and Test Data input types.

Input Data Type (wearoutType)

The Input Data Type is the type of reliability data being provided.

Prediction (Type 1) is for predictions specific for the Thermal Events defined in the Sherlock Life Cycle.

Prediction (Type 2) is for predictions at a different environment (different temperature, voltage, and/or frequency) that require an acceleration factor (AF) to extrapolate to the Thermal Events in the Sherlock Life Cycle.

Field Data is collected from the device in the field.

Test Data is collected during test conditions.

Supported values are PREDICTION (TYPE 1), PREDICTION (TYPE 2), FIELD DATA and TEST DATA.

JEP95 (packageJEP95) [PKG]

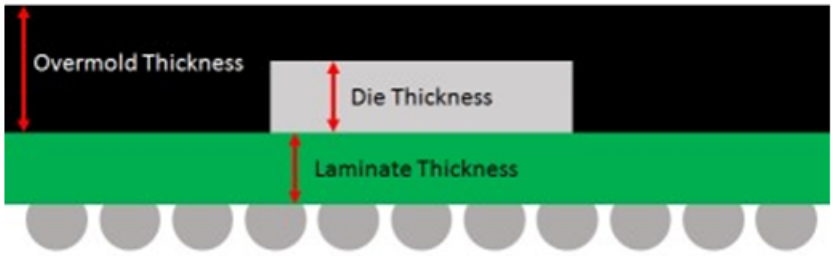
Unique JEP95 designator.

4.8.5. L - Properties

Return to top of list (p. 294)

Laminate Thickness (packageLaminateThickness) [DB] [PKG]

Thickness of the package laminate. The "Package Units" property specifies the units for this property.



Lead Bend (degrees) (leadBend) [DB] [PKG]

For ThruHole lead geometries, lead bend indicates how far the lead is bent from vertical in degrees. When zero, the lead will be entirely vertical. When greater than zero, the bottom vertical lead segment will have a length specified by the "Lead Height" property, and the upper lead segment will have a length specified by the "Lead Shoulder" property. For SIP layouts, the angle may be between 0 and 90 (inclusive). For DIP layouts, the angle may be 0° or 90°, where an angle of 0° indicates that the leads are inside the part outline, and an angle of 90° indicates the leads are outside the part outline. For Axial layouts, the angle may be 90° or 180°, where an angle of 180° indicates that the radial part is standing on edge, with the top lead bent 180°. For Gullwing lead geometries, lead bend indicates how far the lead leg is bent from vertical in degrees. The angle may be 45° to 90° inclusive. When 90°, the lead leg will be entirely vertical.



Lead Bend Package Radius (leadBendPackageRadius)

Radius of the inner bend of the lead nearest the package. The "Lead Units" property specifies the units for this property. Sherlock only uses this property when the **Advanced Lead Mesh Settings** (p. 85) feature is enabled.



Lead Bend Radius (leadBendRadius)

Radius of the inner bend of the lead. For footed leads, this is the lead bend at the lead foot. For Axial-180 leads, this is the lead bend farthest from the package. The "Lead Units" property specifies the units for this property.



Lead Count (leadCount) [DB] [PKG]

Number of component leads (aka pin count).

Lead Frame Percentage (leadFramePercentage) [DB] [PKG]

Percentage of lead material in the layer of the package beneath the die. This is only used for thermal conductivity calculations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

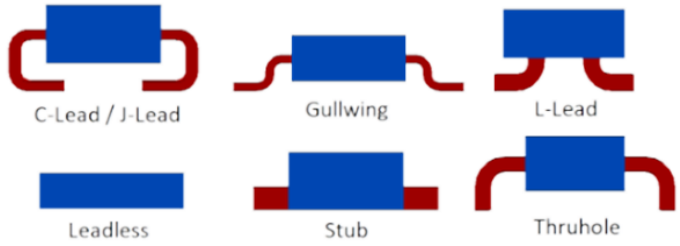
Lead Foot Length (leadFootLength) [DB] [PKG]

Length of the lead foot measured perpendicular to the component body. The "Lead Units" property specifies the units for this property.



Lead Geometry (leadGeometry) [DB] [PKG]

Defines the lead shape. Supported values are C_LEAD, GULLWING, J_LEAD, LEADLESS, L_LEAD, STUB and THRUHOLE.



Lead Height (leadHeight) [DB] [PKG]

Height of the lead. The "Lead Units" property specifies the units for this property.



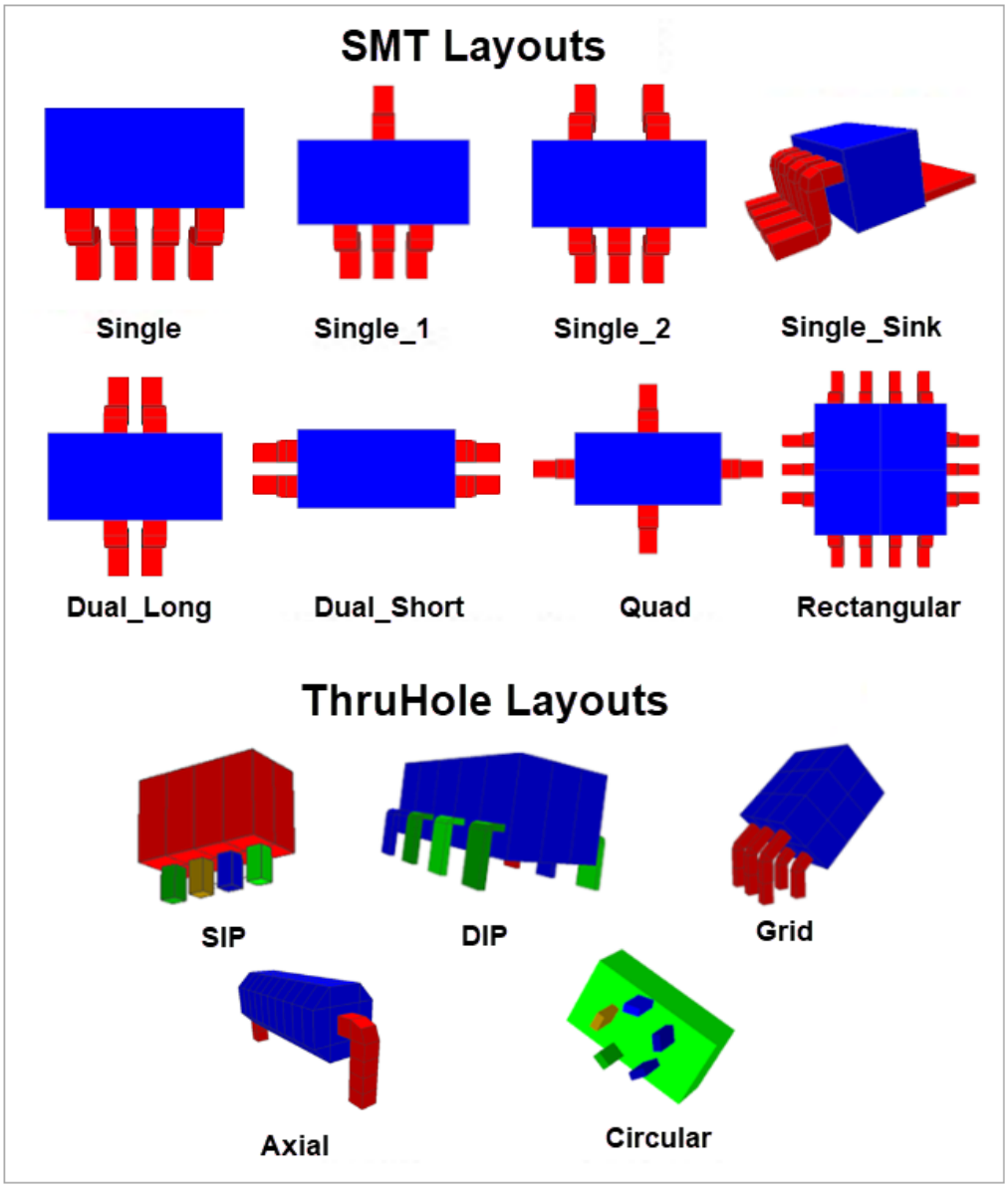
Lead Hole Diameter (leadHoleDiam) [DB] [PKG]

Diameter of the hole used to solder the lead. The "Lead Units" property specifies the units for this property.

Lead Layout (leadLayout) [DB] [PKG]

Defines how leads are arranged for a thru-hole part. Supported values are SIP, DIP, AXIAL, CIRCULAR, GRID, SIP_SINK, SINGLE, SINGLE_1, SINGLE_2, SINGLE_SINK, DUAL_LONG, DUAL_SHORT, QUAD and RECTANGULAR.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Lead Long Count (leadLongCount) [DB] [PKG]

Number of leads on long side of component.

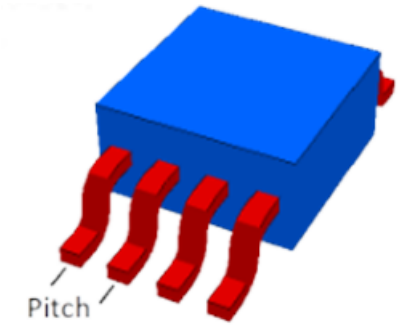
Lead Material (leadMaterial) [DB] [PKG]

Primary material of which the lead is made. Material names are defined in the material data file provided by Sherlock and/or a user-defined data file.

Lead Pitch (leadPitch) [DB] [PKG]

The center-to-center distance between adjacent leads in a SIP or DIP pattern. The "Lead Units" property specifies the units for this property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Lead Short Count (leadShortCount) [DB] [PKG]

Number of leads along short side of component.

Lead Shoulder (leadShoulder) [DB] [PKG]

The length of the non-vertical lead segment when a lead is bent. The "Lead Units" property specifies the units for this property.



Lead Standoff (leadStandoff) [DB] [PKG]

The distance between the PCB and the bottom of the body of a part with bent leads. The standoff distance must be greater than 0.01 mm and less than the lead height minus one-half the lead thickness. The "Lead Units" property specifies the units for this property.



Lead Stiffness (leadStiffness) [DB] [PKG]

The lead stiffness used by the Leaded Solder Joint Fatigue analysis model.

The default lead stiffness calculation can be found in the Sherlock Theory Reference.

Lead Thickness (leadThickness) [DB] [PKG]

Thickness of the lead measured perpendicular to the component body. The "Lead Units" property specifies the units for this property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

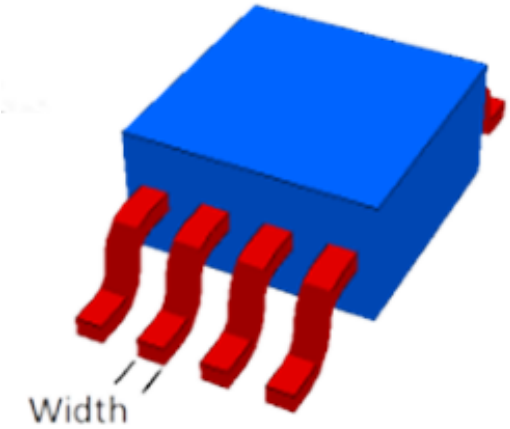


Lead Units (leadUnits) [DB] [PKG]

Unit of length for all lead properties. Supported values are IN, MM, M, MIL, MICRON and NM.

Lead Width (leadWidth) [DB] [PKG]

Width of the lead. The "Lead Units" property specifies the units for this property.



Lifetime (lifetime) [DB]

Endurance test time for a component measured in hours.

Location Units (locUnits)

Unit of length for the component location. Supported values are IN, MM, M, MIL, MICRON and NM.

4.8.6. M - Properties

Return to top of list (p. 294)

Manufacturer (partManufacturer) [DB]

Manufacturer of the part.

Material (partMaterial) [DB] [PKG]

Primary material of which the part is made. Material names are defined in the material data file provided by Sherlock and/or a user-defined data file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Rated Current (A) (ratedCurrent) [DB]

Maximum amount of current in amps that can be handled by the component.

Max Rated Power (W) (ratedPower) [DB]

Maximum amount of power in watts that can be dissipated by the component.

Max Rated Temp (C) (ratedTemp) [DB]

Maximum temperature in degrees C at which the component can be operated.

Max Rated Voltage (V) (ratedVoltage) [DB]

Maximum voltage in volts that can be handled by the component.

Max Storage Temp (C) (storageTempMax) [DB]

Maximum temperature in degrees C at which the component can be stored. If not specified then the max rated operational temperature will be used.

Min Rated Temp (C) (ratedTempMin) [DB]

Minimum temperature in degrees C at which the component can be operated.

Min Storage Temp (C) (storageTempMin) [DB]

Minimum temperature in degrees C at which the component can be stored. If not specified then the min rated operational temperature will be used.

Mirrored (locMirrored)

Indicates if the component is mirrored across the Y-axis. Supported values are YES, NO, TRUE, FALSE, ON, OFF, ENABLED and DISABLED.

Model Leads (modelLeads)

Indicates if the leads associated with this part should be modeled when building FEA models. Supported values are YES, NO, TRUE, FALSE, ON, OFF, ENABLED and DISABLED.

Model Part (modelPart)

Indicates if this part should be modeled when building FEA models. Supported values are ENABLED and DISABLED.

Modified By (userModifiedBy) [DB]

The username (and timestamp) of the person who last modified the library item definition.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.8.7. N - O Properties

Return to top of list (p. 294)

Notes (userNotes) [DB]

Additional information associated with the library item.

Number of Devices (testDeviceCount)

Number of parts or samples tested during the test period.

Number of Failures (testFailures)

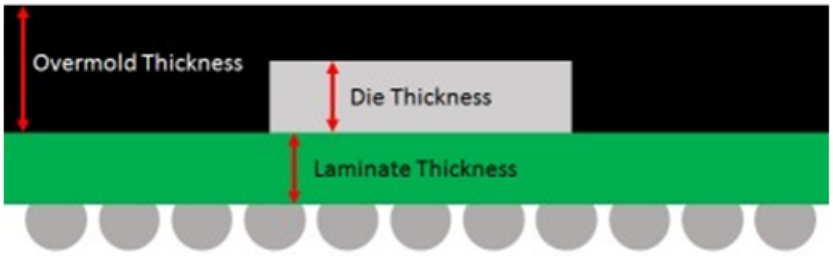
Number of part failures that occurred during the test period.

Overmold Material (overmoldMaterial) [DB] [PKG]

Primary material of the package overmold. Material names are defined in the material data file provided by Sherlock and/or a user-defined data file.

Overmold Thickness (overmoldThickness) [DB] [PKG]

Thickness of the package overmold. The "Package Units" property specifies the units for this property.



4.8.8. P - Properties

Return to top of list (p. 294)

Package Length (packageLength) [DB] [PKG]

Length of the package (usually maximum dimension). The "Package Units" property specifies the units for this property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Package Mount (packageMount) [DB] [PKG]

Package mounting type. Supported values are SMT and TH.

Package Name (packageName) [DB] [PKG]

Unique package name. Supported values are defined in the package data files provided by Sherlock or user-defined package data files.

Package Thickness (packageThickness) [DB] [PKG]

Thickness (height) of the package. The "Package Units" property specifies the units for this property.



Package Type (packageType) [DB] [PKG]

Package type assigned to this package. The package type is used to determine the solder fatigue analysis model to use and controls required fields for a given part. Supported values are AVX, AXIAL, BGA, BOX, BTC, CBEND, CBGA, CC, CDIP, CGA, CQFP, DFN, DIE, DIP, DISC, DPAK, HSOP, KEMET, LCCC, LSOP, MELF, MSOP, OTHER, PDIP, PDSO, PLCC, PQFP, QFJ, QFN, QFP, RADIAL, SIP, SM, SOD, SOIC, SOJ, SON, SOT, SPAK, SSOP, TO, TSOP, TSSOP, USON and VCHIP.

Package Units (packageUnits) [DB] [PKG]

Unit of length for all package properties. Supported values are IN, MM, M, MIL, MICRON and NM.

Package Width (packageWidth) [DB] [PKG]

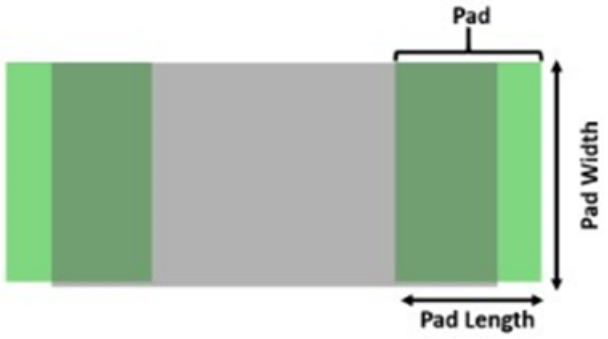
Width of the package (usually minimum dimension). The "Package Units" property specifies the units for this property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Pad Length (padLength) [DB] [PKG]

Length of the pad measured parallel to the attachment side of the component. The "Pad Units" property specifies the units for this property.



Pad Pitch (padPitch) [DB] [PKG]

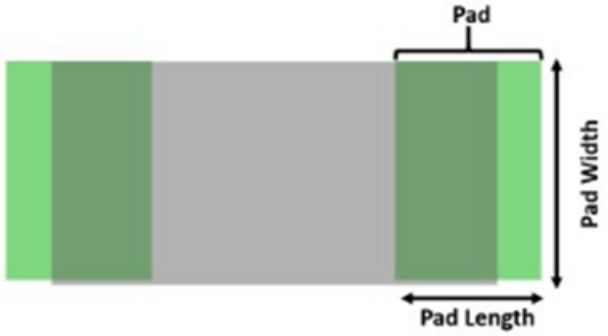
Center-to-center distance between pads. The "Pad Units" property specifies the units for this property.

Pad Units (padUnits) [DB] [PKG]

Unit of length for all board-side pad properties. Supported values are IN, MM, M, MIL, MICRON and NM.

Pad Width (padWidth) [DB] [PKG]

Width of the board-side pad measured perpendicular to the attachment side of the component. The "Pad Units" property specifies the units for this property.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part Description (partDescription) [DB]

Description of the part.

Part Number (partNumber) [DB]

Manufacturer or internal part number for the component.

Part Technology (componentTechnology)

Component Lifetime Part Technology. Supported values are OTHER and ELECTROLYTIC.

Part Temp Rise (thermalTemp)

Default temperature rise (above ambient) for the part used when no thermal maps exist for the PCB. If **DEFAULT** is specified, then the default part temperature rise specified in the Solder Fatigue analysis module will be used during the analysis. The "Thermal Units" property specifies the units for this property.

Part Type (partType) [DB]

Functional part type (for example, capacitor). Supported values are defined in the part type data file provided by Sherlock and/or a user-defined data file.

Process Node (dieProcessNode) [DB] [PKG]

Die process node size and type.

4.8.9. Q - R Properties

Return to top of list (p. 294)

Rated Ripple (mA) (rippleCurrentRated)

Maximum amount of ripple current (mA) that can be handled by the component.

Ref Des (refDes)

Reference designator for a specific component.

Reliability Data Source (wearoutDataSource)

Source type of the semi-conductor wearout data Supported values are MANUFACTURER, USER and JEDEC (GENERIC).

Reliability Metric (reliabilityMetric)

The reliability metric provided by the manufacturer, user, or from a generic source (like JEDEC). If reliability or probability of failure is selected, use the Test Duration (hrs) field to indicate the time period over which the reliability metric is valid. The "Reliability Metric Units" property specifies the units for this property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reliability Metric Units (reliabilityMetricUnits)

Units of measure for data provided by manufacturer or field data. Supported values are RELIABILITY (%), PROB OF FAILURE (%), MTBF (YEARS), MTBF (HOURS), FITS (1E6 HRS) and FITS (1E9 HRS).

Resistance (ohm) (resistance) [DB]

Resistance in ohms.

Ripple Temp Rise (rippleCurrentTempRise)

Temperature rise due to the heat generation by the ripple current. The "Thermal Units" property specifies the units for this property.

Rjb (C/W) (rjb)

Junction resistance to board.

Rjc (C/W) (rjc)

Junction resistance to case.

Rotation (locRotation)

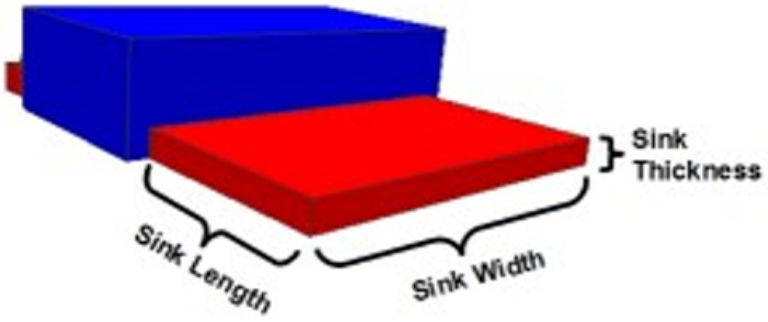
The orientation of the component on the board in degrees. Valid values are between -360 and 360.

4.8.10. S - Properties

Return to top of list (p. 294)

Sink Length (sinkLength) [DB] [PKG]

Length of the sink stub measured perpendicular to the component body. The "Lead Units" property specifies the units for this property.



Sink Thickness (sinkThickness) [DB] [PKG]

Thickness of the sink stub lead. The "Lead Units" property specifies the units for this property. See the image above.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sink Width (sinkWidth) [DB] [PKG]

Width of the sink stub lead measured parallel to the component body. The "Lead Units" property specifies the units for this property. See the image above.

Solder Material (partSolderType)

Type of solder used to connect the part to the PCB. Supported values are defined in the solder data file provided by Sherlock and/or a user-defined data file.

Solder Model (solderModel)

Solder fatigue model used by Sherlock. The default is determined from the Package Type. Supported values are BGA MODEL, CBGA MODEL, CC MODEL, CGA MODEL, DIE MODEL, IMS MODEL, LCCC MODEL, LEADED MODEL, QFN MODEL and THRUHOLE MODEL.

Solder Thickness (solderThickness)

Thickness (height) of the solder joint that connects the part to the PCB used for analysis. If not provided, the default value is the Stencil thickness divided in half. Changing this value will have an impact on analysis results. The "Stencil Units" property specifies the units for this property.

Source of AF (accelFactorsSource)

Determines the source of acceleration factors (AF).

If the source is MANUFACTURER or USER, acceleration factors for temperature, voltage, and frequency must be provided and are independent of the process node under the Die tab.

If you do not have acceleration factors, choose SHERLOCK.

If source is SHERLOCK, acceleration factors are calculated based on the process node. The referenced process node must be defined in the Acceleration Factors Manager.

Supported values are MANUFACTURER, USER and SHERLOCK.

Stencil Thickness (stencilThickness)

Thickness (height) of the solder joint that connects the part to the PCB. For QFN packages, this is the solder paste thickness with the assumption that it is 50% solids. Stencil thickness is NOT used directly by any analysis tasks. This value is simply used to provide a default Solder thickness value. Refer to the Solder Thickness property for more information. The "Stencil Units" property specifies the units for this property.

Stencil Units (solderUnits)

Unit of length for all solder properties. Supported values are IN, MM, M, MIL, MICRON and NM.

Subcircuit Name(s) (subcircuitList)

Comma separated list of subcircuits (aka functional blocks) associated with the component.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.8.11. T - Y Properties

Return to top of list (p. 294)

Test Duration (hrs) (testDuration)

Total time in hours spent testing a part.

Test Duty Cycle (%) (testDutyCycle)

The duty cycle of the test.

Test Frequency (MHz) (testFrequency)

The frequency in MHZ of the test.

Test Method (testMethod)

Test method used to test a batch of parts. Supported values are BLACK BOX, LAB and FIELD.

Test Temperature (C) (testTemperature)

The temperature of the test.

Test Voltage (V) (testVoltage)

The voltage in V of the test.

Thermal Units (thermalUnits)

Unit of temperature for all thermal properties. Supported values are C, F and K.

Voltage Exponent (voltageExponent)

Power law based accelaration factor (n). This is only applicable for Prediction (Type 2) and Test Data input types.

Weight (gram) (weight) [DB] [PKG]

Calculated weight of the component, in grams. This is used to calculate density in any FEA analysis.

X Coordinate (locX)

Location of component along the X axis. The "Location Units" property specifies the units for this property.

X Offset (locOffsetX)

Offset to apply to the X coordinate so that the component will be centered over that location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y Coordinate (locY)

Location of component along the Y axis. The "Location Units" property specifies the units for this property.

Y Offset (locOffsetY)

Offset to apply to the Y coordinate so that the component will be centered over that location.

4.9. Package Management

The Sherlock **Package Manager** allows for a user to create and modify Sherlock package definitions. The Package Manager is installed as a standalone application and is accessible from the Sherlock **Libraries > Packages** menu.

4.9.1. Package Manager Listing
4.9.2. Selecting a Package to Edit
4.9.3. Using the Package Editor
4.9.4. Adding and Copying Packages
4.9.5. Deleting Packages
4.9.6. Package Export and Import
4.9.7. Column Mappings
4.9.8. Package Previewer

Changes to the Package Database in Sherlock 24 R1

- **CBEND**: Eight **CBEND** packages which previously had part material set to EPOXYENCAPSULANT have been changed to OVERMOLD-LEADED.
- **DFS (DFN)**: This package has been removed.
- **DIP:**
  - The lead material for all DIP packages is now COPPER. Previously, 32 packages had their lead material value set to Alloy42.
  - The packageMaterial value for all DIP packages has been set to OVERMOLD-LEADED. Previously the CDIP packages were set to Alumina for this property and all others were set to EpoxyEncapsulant.
- **LCCC:**
  - For all LCCC packages, the part material has been changed from EPOXYENCAPSULANT to ALUMINA.
  - The LCCC package previously named SMD 7X5 has had its name changed to LCCC-2.
  - The LCCC package previously named SMD-0.5 has had its name changed to LCCC-3
  - The LCCC package named SMD-2 has been removed.
- **MELF:**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - The default part material for all MELF packages is now GLASS.
  - The package previously named SOD-80 has is now named MELF (SOD-80).

- **PLCC:** Previously, the part material for all PLCC packages was EPOXYENCAPSULANT. Now, the part material for all PLCC packages is set to OVERMOLD-LEADED.
- **Radial:** The part material for all Radial packages is now ALUMINUM.
- **SIP:**
  - Previously, the lead material for three of the SIP packages was KOVAR. Now, the lead material for all SIP packages is set to COPPER.
  - Previously, the part material for three of the SIP packages was KOVAR. Now, the part material for all packages is set to OVERMOLD-LEADED.
- **SON:** Every SON package with a lead geometry value of LEADLESS has been removed. In total, 16 packages were removed.
- **SOT:** The SOT package named SOT-883 has been removed.
- **TO:** The part material for all TO packages has changed from EPOXYENCAPSULANT to OVERMOLD-LEADED.
- **TSOP:** Lead material is now set to COPPER for all TSOP packages.

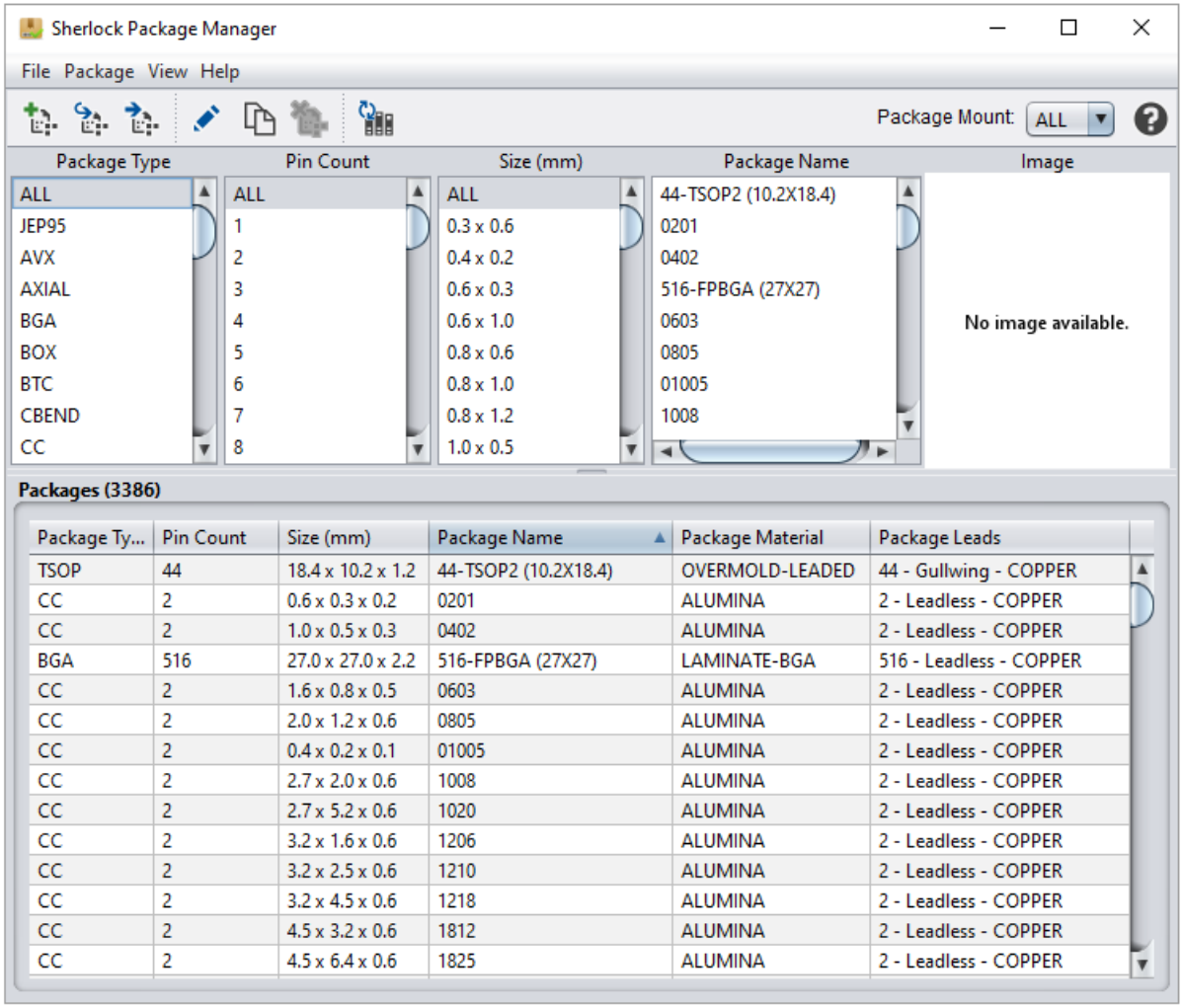
In this section, the following topics are covered:

4.9.1. Package Manager Listing
4.9.2. Selecting a Package to Edit
4.9.3. Using the Package Editor
4.9.4. Adding and Copying Packages
4.9.5. Deleting Packages
4.9.6. Package Export and Import
4.9.7. Column Mappings
4.9.8. Package Previewer

4.9.1. Package Manager Listing

The main **Package Manager** window is organized into two sections. The top section provides a set of columns to allow filtering of the packages in the manager. The bottom table is a list of all packages or those packages matching any selected filters from above.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For example, to filter the package definitions so only the CBEND package types are displayed, select **CBEND** in the **Package Type** column. This will immediately filter the package table to display packages of the type CBEND only. You may select multiple filters per column by pressing the CTRL key while clicking additional package definitions.

When you filter the packages based on the **Package Type**, **Pin Count**, and **Size** columns, the **Package Name** column of the *upper* table is filtered along with the results displayed in the lower table.

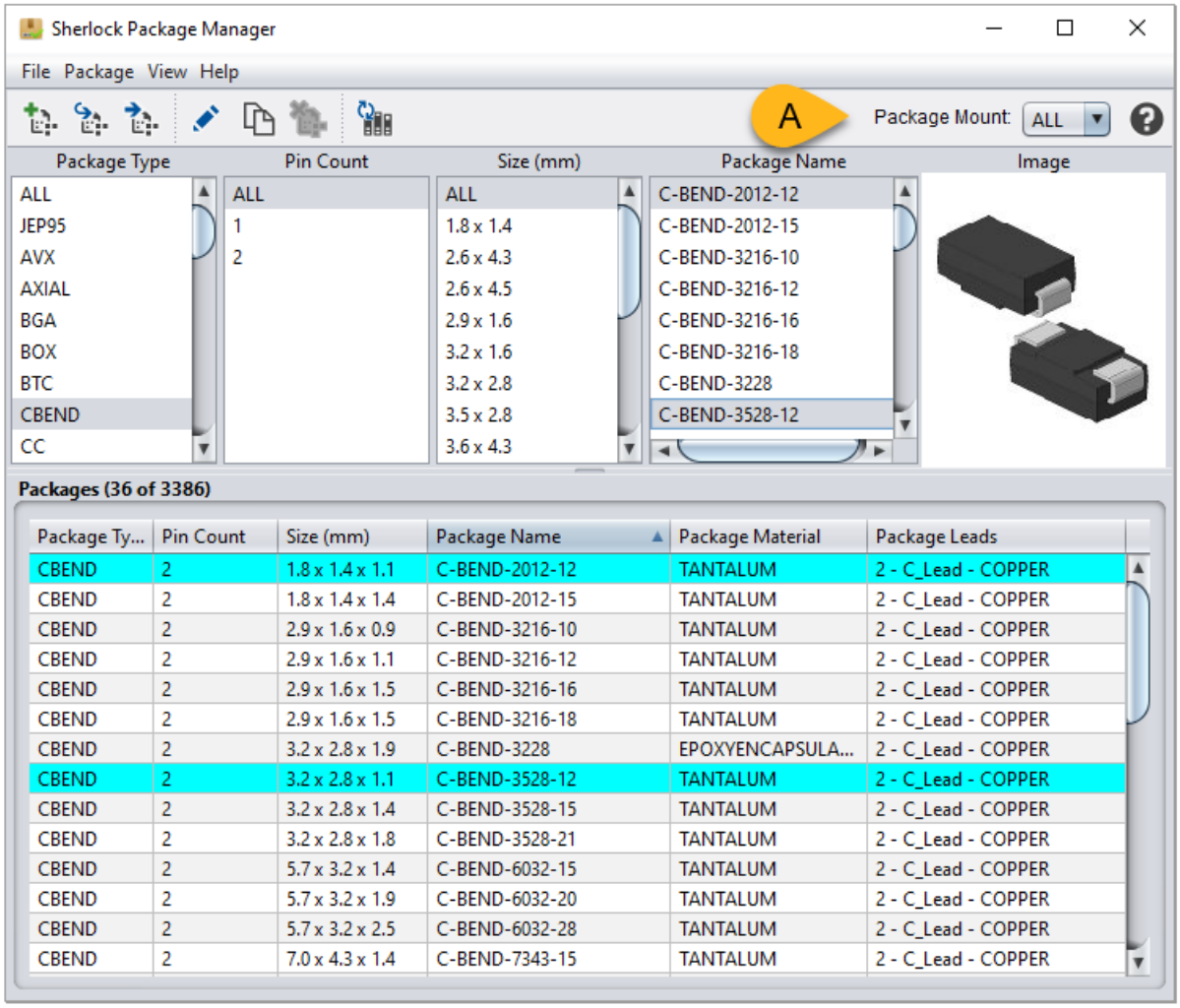
When you select one or more items from the **Package Name** column, Sherlock does not reduce the number of rows in the lower table but, instead, highlights the rows of the selected packages, as shown below. When you select a filter from the **Package Name** column, the **Image** field will display a model of the first selected package in **Package Name** column.

The image below shows an example where the CBEND filter has been selected under **Package Type**. And the **C-BEND-2012-12** and **C-BEND-3528-12** packages have been selected under **Package Name**, resulting in those rows being highlighted in the lower table.

Note:

Use the **Package Mount** filter (A, below) to reduce the list of displayed package types to through-hole (**TH**) or surface-mount (**SMT**) package types.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

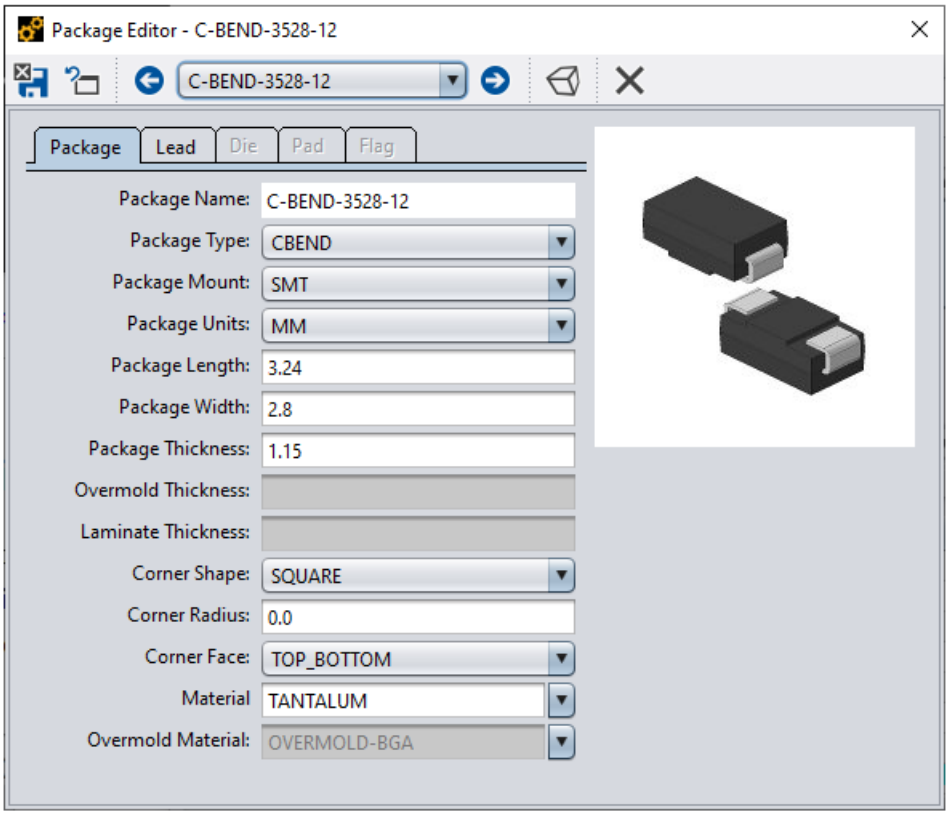


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

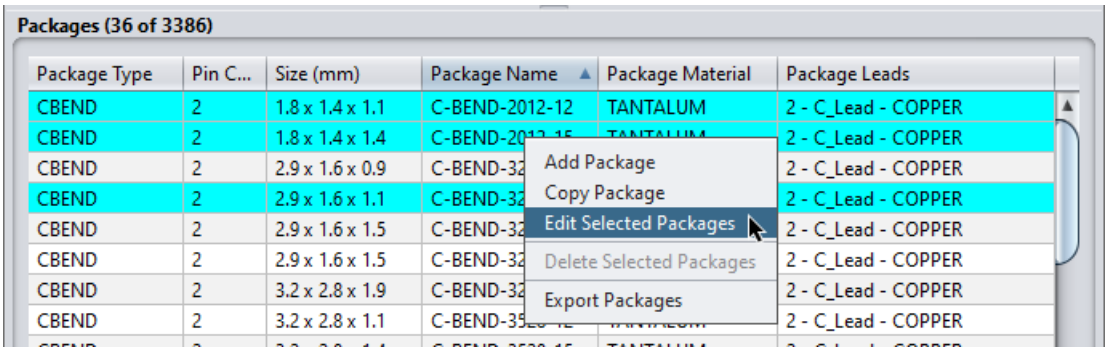
4.9.2. Selecting a Package to Edit

From the **Sherlock Package Manager**, you can open the **Package Editor** (shown below) to view or edit a specific package. You can also edit multiple package definitions at the same time. To open a single package in the **Package Editor**:

- Double-click the table row containing the **Package Name** you wish to view or edit.
- Or right-click the table row and select **Edit Selected Package** from the context menu.



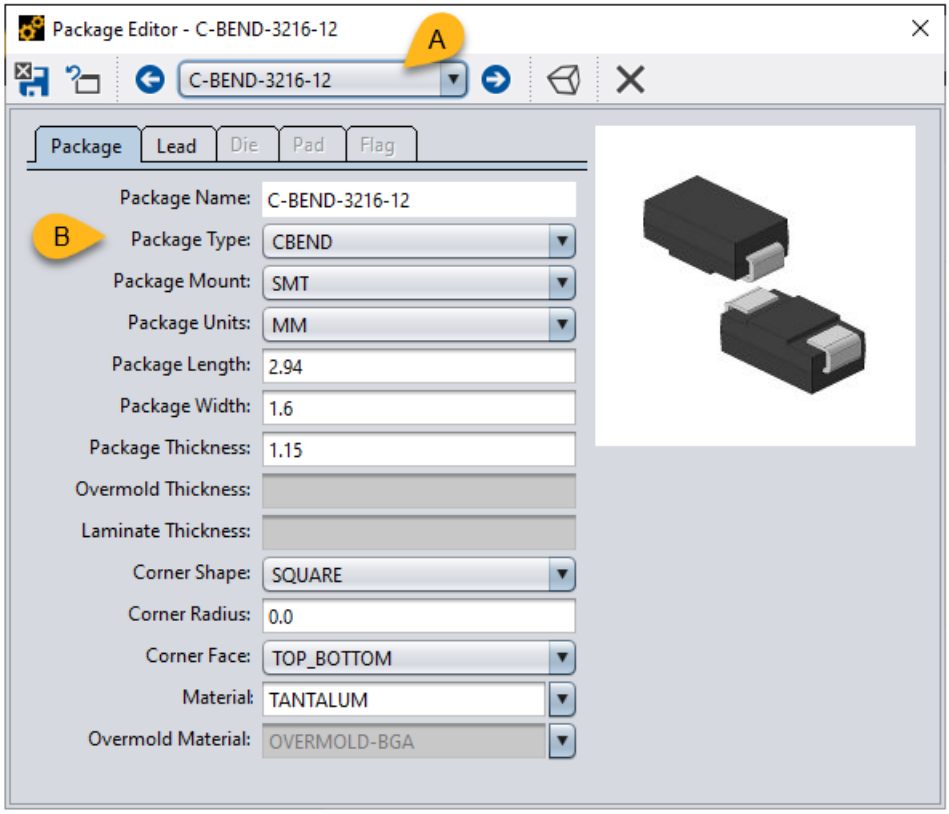
To edit multiple package definitions at the same time, highlight the table rows of the packages you wish to edit. (You can highlight multiple rows by holding the SHIFT or CTRL key while clicking.) Once you have selected the applicable rows, right-click one of the highlighted rows and select **Edit Selected Packages** from the context menu.



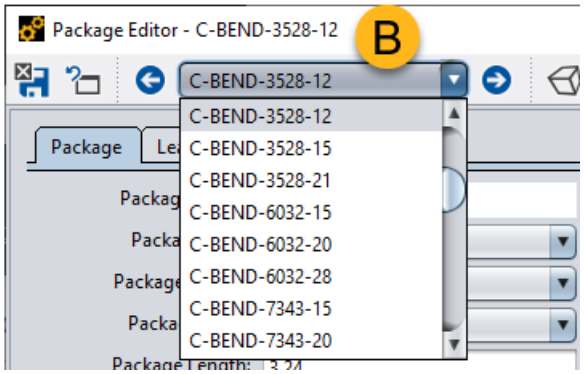
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.9.3. Using the Package Editor

After selecting a package to edit, a new dialog will appear with a listing of all the available package properties. The tabs and properties displayed will vary depending on the **Package Type** selected. From the package editor, you can view and make changes to any of the core package properties (such as package length, width, thickness and so on), lead properties, ball properties, die properties, flag properties, and pad properties.



If you wish to view additional package definitions, you may do so without closing the package editor. Simply select the package name from the drop-down menu (B, below) or use the arrow buttons. The package selector and the arrow buttons are both limited to the *filtered* packages from the **Package Manager** itself. If no filters were applied, then the entire list of packages is available. The example below shows the effects of the filter: Only the CBEND packages are displayed.



Package Editor interface:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

 Save changes and exit the editor.

 Reset the form to the currently saved values.

 Show the previous package.

 Show the next package.

 View the 3D model of the package.

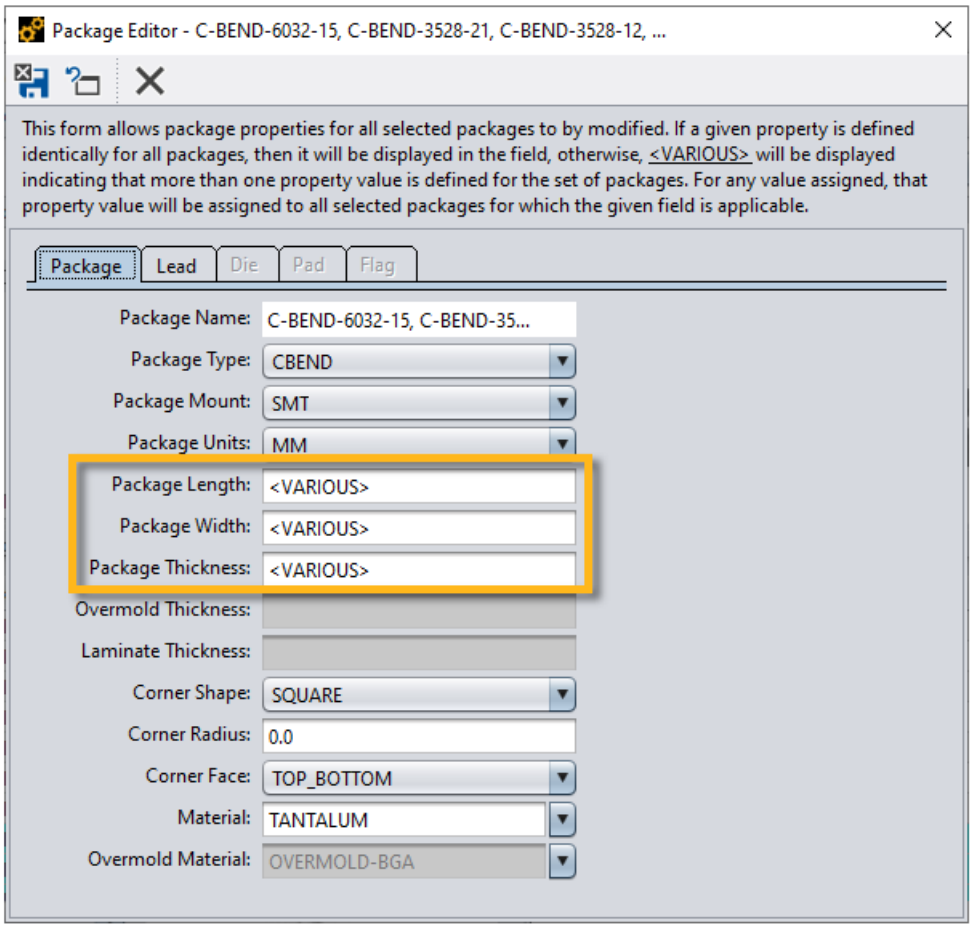
 Close the editor.

After making any changes to the package properties, click the **Save** button to save the changes to the *local* copy of the Sherlock package library. (See the section Sherlock User Guide - User Data Files (p. 166) for more information about the location and format of the local package library.) If you choose not to save your changes or you are simply viewing the package definitions, click the **Close** icon when done. The **Reset** button will revert any changes to the last saved property settings.

When selecting multiple packages to edit at the same time, you will see **<VARIOUS>** displayed for some properties as shown below. This means the selected packages have different (or *various*) values for that property. In the example shown, the selected packages have various length, width, and thickness dimensions.

If you enter a value in a field to replace **<VARIOUS>**, the new value will be applied to *all* the selected packages when you click the **Save** icon.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



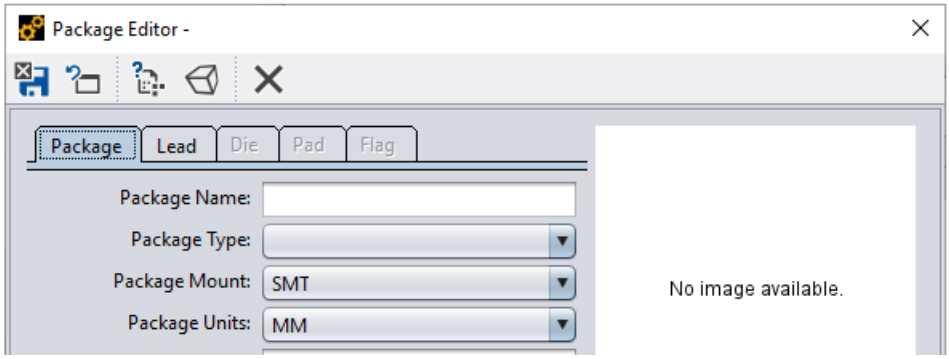
4.9.4. Adding and Copying Packages

You can add new packages to the Package Library by doing the following. From the **Sherlock Package Manager**,

- Select **Add Package** from the **Package** menu
- Click the **Add Package** icon 
- Or right-click anywhere in the lower table and select **Add Package** from the context menu

The **Package Editor** will open, but this time the **Package Name** field is now editable. Depending on the package name provided, Sherlock will enable additional package property items in the editor dialog. Sherlock will also guess the **Package Type** when you first enter the **Package Name**, but you can manually change the **Package Name** if needed. When you are finished defining a package, click the **Save** button to add the new package.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To copy a package, do the following: In the **Sherlock Package Manager**, right-click the table row of the package to be copied and select **Copy Package** from the context menu. Sherlock will create a copy of the package and display it in the **Package Editor**. From there, you can enter a **Package Name** and make any changes desired to the package properties. When finished, click the **Save** button to add the new package.

4.9.5. Deleting Packages

Any *user-defined* package (*not* a Sherlock system package) may be deleted. In the **Sherlock Package Manager**, right-click the table row of the package to be deleted and select **Delete Package** from the context menu. If the **Delete Package** item is grayed out, this means the package is *not* a user-defined package and cannot be deleted.

If a system package has been *modified*, the **Delete Package** menu item will be available. However, choosing **Delete Package** will not delete the package completely. Rather, all user-defined properties will simply revert to system-defined settings.

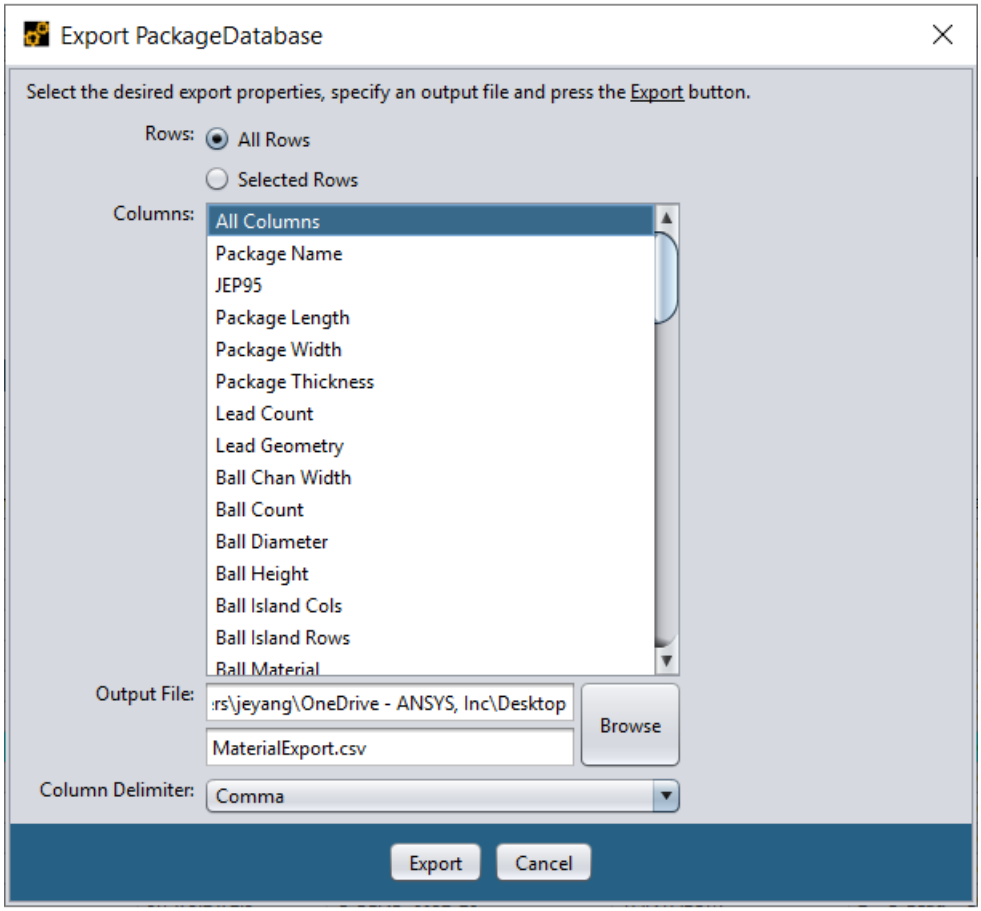
4.9.6. Package Export and Import

Package definitions may be exported to CSV or XLS spreadsheet files. These files can also be imported back into the Package Manager. To export all package definitions, select from the **File** menu **Export Packages** or right-click in the package table listing and select **Export Package** from the context menu. If you wish to export only specific packages, select the packages you wish to export from the package table listing, then right-click the table and select **Export Packages.**

After choosing to export package definitions, the **Export Package Database** dialog will appear (see below). Choose whether all rows are to be exported or only the selected rows in the package table listing. Then choose if all available columns or only selected columns should be exported

Choose the export file location, the click the Export button. Supported file extensions are **.csv**, **.xls**, and **.xlsx**. If successful, a message will appear indicating the number of rows selected. That file then may be opened with a spreadsheet program.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

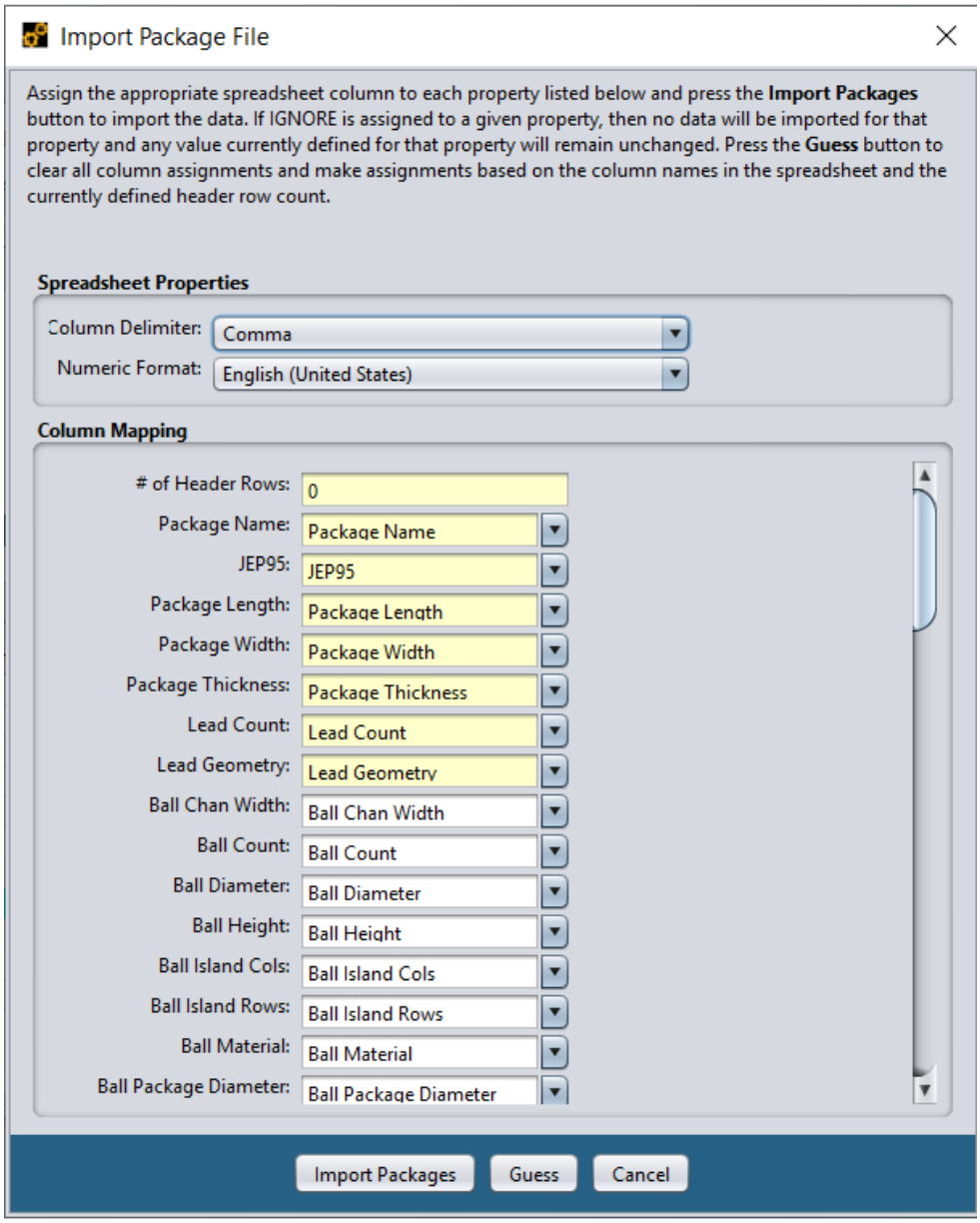


You may also import modified package definitions from a spreadsheet or CSV file. To do so, select from the **File** menu the **Import Packages** option. A file dialog will appear to allow you to locate and the select the file to be imported. For convenience, the last imported/exported file will be used as the default, but you are free to select any file on your system.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.9.7. Column Mappings

After selecting the file, the **Import Package File** dialog will appear (shown below), allowing you to verify the column mapping for each file/sheet that is imported. Sherlock scans each file in an attempt to determine the number of header rows and the column names. Such guesses are usually right, but you should check them anyway by viewing the CSV file in the content panel or Excel.



In addition to guessing the column names, Sherlock also attempts to guess which columns should be mapped to Sherlock package properties.

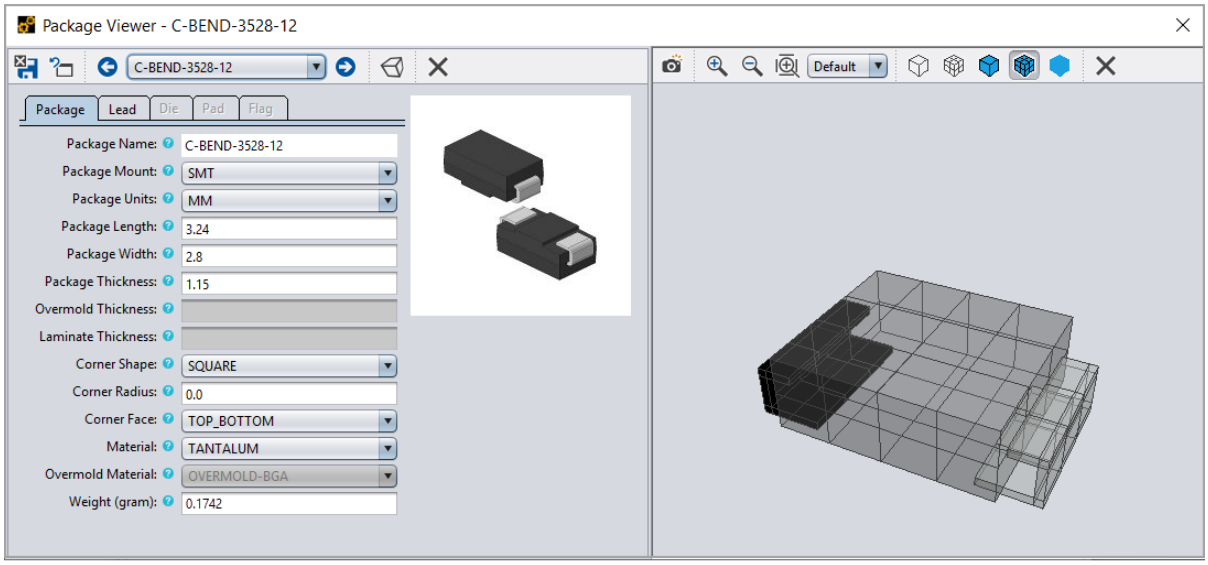
When you have the column mappings correct, press the **Import Packages** button to import the data.

When the import process is complete, you will see a confirmation dialog. If any packages fail to import, then none of the rows in the file will be imported. When this happens an error dialog will be displayed with the information regarding why a particular package failed to import.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.9.8. Package Previewer

The 3D visual representation of any package definition can be viewed when editing the package definition for an individual part. Double-click any given package in the **Package Manager** list of packages to open the package editor dialog for the selected package. At the top of the dialog is a button labeled **View the 3D model of the package,** denoted by a box icon. Click this button to view the 3D visual representation of this package. The window presented for viewing the 3D visual representation has similar controls to that of the Sherlock 3D Viewer. You may pan, rotate and zoom the display using simple mouse controls. Press the **Help** button for a complete list of the mouse controls supported.



 View the 3D model of the package

When making changes to any package properties, simply reselect **View the 3D Model of the Package** and the currently open package viewer window will update with the new properties reflected.

The drop-down menu at the top of the dialog allows the user to select the various viewer options similar to options available in the Sherlock 3D Viewer (see Viewing and Managing Analysis Results (p. 785) for more details).

4.10. Tables: Required Part Properties per Analysis Type

Sherlock tracks over a hundred properties for each part in the Parts Lists, but each analysis uses only a subset of all those properties. The tables below indicate which properties Sherlock uses for each type of analysis. This information can streamline your workflow by clearly defining which properties are crucial to the analysis you are running.

4.10.1. Required Part Properties for: Component Failure Analysis, DFMEA, and PTH Fatigue

4.10.2. Required Part Properties for: Semi-Conductor Wearout, Solder Fatigue, Thermal Derating, and Part Validation

4.10.3. Required Part Properties for Each FEA Analysis

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.10.1. Required Part Properties for: Component Failure Analysis, DFMEA, and PTH Fatigue

Table Legend: Y = Yes, this property is used in the analysis. **N** = No, this property is not used in the analysis. **n/a** = Not applicable.

Sherlock Part Properties Tab	Part Properties	Component Failure Analysis	DFMEA	PTH Fatigue
ID	Ref Des	Y	Y	n/a
	Part Type	N	N	n/a
	Part Number	N	N	n/a
	Manufacturer	N	N	n/a
	Part Description	N	N	n/a
	Subcircuit Name(s)	N	N	n/a
	Failure Mechanism	Y	N	n/a
	Failure Class	Y	N	n/a
Package	Package Name	Y	Y	n/a
	Package Type	Y	Y	n/a
	Package Mount	Y	Y	n/a
	Package Units	N	N	n/a
	Package Length	Y	Y	n/a
	Package Width	Y	Y	n/a
	Package Thickness	Y	Y	n/a
	Overmold Thickness	N	N	n/a
	Laminate Thickness	N	N	n/a
	Model Part	N	N	n/a
	Corner Shape	Y	Y	n/a
	Corner Radius	Y	Y	n/a
	Corner Face	Y	Y	n/a
	Material	Y	Y	n/a
	Overmold Material	Y	Y	n/a
	Junction Res (C/W)	N	N	n/a
	Weight (gram)	N	N	n/a
Location	Board Side	N	N	n/a
	Location Units	N	N	n/a
	X Coordinate	Y	Y	n/a
	Y Coordinate	Y	Y	n/a
	Rotation	Y	Y	n/a
	Mirrored	N	N	n/a

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lead	**Lead Count**	Y	Y	n/a
	Lead Geometry	Y	Y	n/a
	Lead Material	Y	Y	n/a
	Lead Stiffness	N	N	n/a
	Lead Units	Y	Y	n/a
	Lead Height	Y	Y	n/a
	Lead Width	N	N	n/a
	Lead Thickness	Y	Y	n/a
	Lead Foot Length	Y	Y	n/a
	Lead Hole Diamater	N	N	n/a
	Lead Modelling	N	N	n/a
	Lead Layout	Y	Y	n/a
	Lead Pitch	Y	Y	n/a
	Lead Bend (degrees)	Y	Y	n/a
	Lead Shoulder	Y	Y	n/a
	Lead Standoff	Y	Y	n/a
	Lead Long Count	Y	Y	n/a
	Lead Short Count	Y	Y	n/a
	Sink Length	N	N	n/a
	Sink Width	N	N	n/a
	Sink Thickness	N	N	n/a
Pad	**Pad Units**	N	N	n/a
	Pad Length	N	N	n/a
	Pad Width	N	N	n/a
	Pad Pitch	N	N	n/a
Ball	**Ball Pattern**	Y	Y	n/a
	Ball Count	Y	Y	n/a
	Ball Units	Y	Y	n/a
	Ball Pitch	Y	Y	n/a
	Ball Diameter	Y	Y	n/a
	Ball Package Diameter	Y	Y	n/a
	Ball Pad Diameter	Y	Y	n/a
	Ball Height	Y	Y	n/a
	Ball Channel Width	Y	Y	n/a
	Ball Material	Y	Y	n/a
	Ball Modeling	Y	Y	n/a
	Ball Perimeter Rows	Y	Y	n/a
	Ball Perimeter Columns	Y	Y	n/a
	Ball Island Rows	Y	Y	n/a

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Ball Island Columns	Y	Y	n/a
Solder	**Solder Material**	Y	Y	n/a
	Stencil Units	Y	Y	n/a
	Stencil Thickness	N	N	n/a
	Solder Model	N	N	n/a
Electrical	**Max Rated Power (W)**	Y	N	n/a
	Applied Power (W)	Y	N	n/a
	Max Rated Current (A)	Y	N	n/a
	Applied Current (A)	Y	N	n/a
	Max Rated Voltage (V)	Y	N	n/a
	Applied Voltage (V)	Y	N	n/a
	Applied Freq (MHz)	Y	N	n/a
	Capacitance (uF)	Y	N	n/a
	Resistance (ohm)	Y	N	n/a
	Electrical Derating (0-1)	Y	N	n/a
	Rated Ripple (mA)	Y	N	n/a
	Applied Ripple (mA)	Y	N	n/a
Thermal	**Thermal Units**	N	N	n/a
	Part Temp Rise	N	N	n/a
	Min Rated Temp (°C)	Y	N	n/a
	Max Rated Temp (°C)	Y	N	n/a
	Min Storage Temp (°C)	Y	N	n/a
	Max Storage Temp (°C)	Y	N	n/a
	Lifetime	Y	N	n/a
	Ripple Temp Rise	Y	N	n/a
	Rjc (C/W)	N	N	n/a
	Rjb (C/W)	N	N	n/a
Die	**Die Units**	N	N	n/a
	Die Length	N	N	n/a
	Die Width	N	N	n/a
	Die Thickness	N	N	n/a
	Die Material	N	N	n/a
	Process Node	N	N	n/a
Flag	**Flag Units**	N	N	n/a
	Flag Length	N	N	n/a
	Flag Width	N	N	n/a
	Flag Thickness	N	N	n/a
Test	**Test Duration**	Y	N	n/a
	Number of Devices	Y	N	n/a

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Number of Failures	Y	N	n/a
	Confidence Bounds (%)	n/a	n/a	n/a
	Test Temperature	Y	N	n/a
	Test Voltage	Y	N	n/a
	Test Frequency	Y	N	n/a
	Test Duty Cycle	Y	N	n/a
Semi-conductor Wearout	**Input Data Type**	N	N	n/a
	Reliability Data Source	N	N	n/a
	Documents	N	N	n/a
	Reliability Metric	N	N	n/a
	Reliablity Metric Units	N	N	n/a
	Source of AF	N	N	n/a
	Activation Energy (Ea)	N	N	n/a
	Voltage Exponent	N	N	n/a
	Frequency Exponent	N	N	n/a

4.10.2. Required Part Properties for: Semi-Conductor Wearout, Solder Fatigue, Thermal Derating, and Part Validation

Table Legend: Y = Yes, this property is used in the analysis. **N** = No, this property is not used in the analysis. **n/a** = Not applicable.

Sherlock Part Properties Tab	Part Properties	Semi-conductor Wearout	Solder Fatigue	Thermal Derating	Part Validation
ID	**Ref Des**	Y	Y	Y	Y
	Part Type	N	Y	N	N
	Part Number	N	N	N	N
	Manufacturer	N	N	N	N
	Part Description	N	N	N	N
	Subcircuit Name(s)	N	N	N	N
	Failure Mechanism	N	N	N	N
	Failure Class	N	N	N	N
Package	**Package Name**	Y	Y	Y	Y
	Package Type	Y	Y	Y	Y
	Package Mount	Y	Y	Y	Y
	Package Units	N	Y	N	N
	Package Length	Y	Y	Y	Y
	Package Width	Y	Y	Y	Y
	Package Thickness	Y	Y	Y	Y

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Overmold Thickness	N	Y	N	N
	Laminate Thickness	N	Y	N	N
	Model Part	N	Y	N	N
	Corner Shape	Y	Y	Y	Y
	Corner Radius	Y	Y	Y	Y
	Corner Face	Y	Y	Y	Y
	Material	Y	Y	Y	Y
	Overmold Material	Y	Y	Y	Y
	Junction Res (C/W)	N	N	N	N
	Weight (gram)	N	N	N	N
Location	**Board Side**	N	N	N	N
	Location Units	N	N	N	N
	X Coordinate	Y	Y	Y	Y
	Y Coordinate	Y	Y	Y	Y
	Rotation	Y	Y	Y	Y
	Mirrored	N	N	N	N
Lead	**Lead Count**	Y	Y	Y	Y
	Lead Geometry	Y	Y	Y	Y
	Lead Material	Y	Y	Y	Y
	Lead Stiffness	N	Y	N	N
	Lead Units	Y	Y	Y	Y
	Lead Height	Y	Y	Y	Y
	Lead Width	N	Y	N	N
	Lead Thickness	Y	Y	Y	Y
	Lead Foot Length	Y	Y	Y	Y
	Lead Hole Diamater	N	N	N	N
	Lead Modelling	N	N	N	N
	Lead Layout	Y	Y	Y	Y
	Lead Pitch	Y	Y	Y	Y
	Lead Bend (degrees)	Y	Y	Y	Y
	Lead Shoulder	Y	Y	Y	Y
	Lead Standoff	Y	Y	Y	Y
	Lead Long Count	Y	Y	Y	Y
	Lead Short Count	Y	Y	Y	Y
	Sink Length	N	N	N	N
	Sink Width	N	N	N	N
	Sink Thickness	N	N	N	N
Pad	**Pad Units**	N	Y	N	N
	Pad Length	N	Y	N	N

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Pad Width	N	Y	N	N
	Pad Pitch	N	Y	N	N
Ball	**Ball Pattern**	Y	Y	Y	Y
	Ball Count	Y	Y	Y	Y
	Ball Units	Y	Y	Y	Y
	Ball Pitch	Y	Y	Y	Y
	Ball Diameter	Y	Y	Y	Y
	Ball Package Diameter	Y	Y	Y	Y
	Ball Pad Diameter	Y	Y	Y	Y
	Ball Height	Y	Y	Y	Y
	Ball Channel Width	Y	Y	Y	Y
	Ball Material	Y	Y	Y	Y
	Ball Modeling	Y	Y	Y	Y
	Ball Perimeter Rows	Y	Y	Y	Y
	Ball Perimeter Columns	Y	Y	Y	Y
	Ball Island Rows	Y	Y	Y	Y
	Ball Island Columns	Y	Y	Y	Y
Solder	**Solder Material**	Y	Y	Y	Y
	Stencil Units	Y	Y	Y	Y
	Stencil Thickness	N	Y	N	N
	Solder Model	N	Y	N	N
Electrical	**Max Rated Power (W)**	N	N	N	N
	Applied Power (W)	N	N	N	N
	Max Rated Current (A)	N	N	N	N
	Applied Current (A)	N	N	N	N
	Max Rated Voltage (V)	N	N	N	N
	Applied Voltage (V)	N	N	N	N
	Applied Freq (MHz)	N	N	N	N
	Capacitance (uF)	N	N	N	N
	Resistance (ohm)	N	N	N	N
	Electrical Derating (0-1)	N	N	N	N
	Rated Ripple (mA)	N	N	N	N
	Applied Ripple (mA)	N	N	N	N
Thermal	**Thermal Units**	N	N	N	N
	Part Temp Rise	N	Y	N	N
	Min Rated Temp (°C)	N	N	Y	N
	Max Rated Temp (°C)	N	N	Y	N
	Min Storage Temp (°C)	N	N	Y	N
	Max Storage Temp (°C)	N	N	Y	N

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Lifetime	N	N	N	N
	Ripple Temp Rise	N	N	N	N
	Rjc (C/W)	N	N	N	N
	Rjb (C/W)	N	N	N	N
Die	**Die Units**	N	Y	N	N
	Die Length	N	Y	N	N
	Die Width	N	Y	N	N
	Die Thickness	N	Y	N	N
	Die Material	N	Y	N	N
	Process Node	Y	N	N	N
Flag	**Flag Units**	N	Y	N	N
	Flag Length	N	Y	N	N
	Flag Width	N	Y	N	N
	Flag Thickness	N	Y	N	N
Test	**Test Duration**	N	N	N	N
	Number of Devices	N	N	N	N
	Number of Failures	N	N	N	N
	Confidence Bounds (%)	n/a	n/a	n/a	n/a
	Test Temperature	N	N	N	N
	Test Voltage	N	N	N	N
	Test Frequency	N	N	N	N
	Test Duty Cycle	N	N	N	N
Semi-conductor Wearout	**Input Data Type**	Y	N	N	N
	Reliability Data Source	Y	N	N	N
	Documents	Y	N	N	N
	Reliability Metric	Y	N	N	N
	Reliablity Metric Units	Y	N	N	N
	Source of AF	Y	N	N	N
	Activation Energy (Ea)	Y	N	N	N
	Voltage Exponent	Y	N	N	N
	Frequency Exponent	Y	N	N	N

4.10.3. Required Part Properties for Each FEA Analysis

Table Legend: Y = Yes, this property is used in the analysis. **N** = No, this property is not used in the analysis. **n/a** = Not applicable.

Sherlock Part Properties Tab	Part Properties	Harmonic Vibe (FEA)	ICT Analysis (FEA)	Natural Freq. (FEA)	Random Vibe (FEA)	Thermal Mech (FEA)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ID	**Ref Des**	Y	Y	Y	Y	Y
	Part Type	N	N	N	N	N
	Part Number	N	N	N	N	N
	Manu-facturer	N	N	N	N	N
	Part Description	N	N	N	N	N
	Subcircuit Name(s)	N	N	N	N	N
	Failure Mechanism	N	N	N	N	N
	Failure Class	N	N	N	N	N
Package	**Package Name**	Y	Y	Y	Y	Y
	Package Type	Y	Y	Y	Y	Y
	Package Mount	Y	Y	Y	Y	Y
	Package Units	Y	Y	Y	Y	Y
	Package Length	Y	Y	Y	Y	Y
	Package Width	Y	Y	Y	Y	Y
	Package Thickness	Y	Y	Y	Y	Y
	Overmold Thickness	Y	Y	Y	Y	Y
	Laminate Thickness	Y	Y	Y	Y	Y
	Model Part	Y	Y	Y	Y	Y
	Corner Shape	Y	Y	Y	Y	Y
	Corner Radius	Y	Y	Y	Y	Y
	Corner Face	Y	Y	Y	Y	Y
	Material	Y	Y	Y	Y	Y
	Overmold Material	Y	Y	Y	Y	Y
	Junction Res (C/W)	N	N	N	N	N
	Weight (gram)	N	N	N	N	N
Location	**Board Side**	Y	Y	Y	Y	Y
	Location Units	Y	Y	Y	Y	Y
	X Coordinate	Y	Y	Y	Y	Y
	Y	Y	Y	Y	Y	Y

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Coordinate					
	Rotation	Y	Y	Y	Y	Y
	Mirrored	Y	Y	Y	Y	Y
Lead	**Lead Count**	Y	Y	Y	Y	Y
	Lead Geometry	Y	Y	Y	Y	Y
	Lead Material	Y	Y	Y	Y	Y
	Lead Stiffness	Y	Y	Y	Y	Y
	Lead Units	Y	Y	Y	Y	Y
	Lead Height	Y	Y	Y	Y	Y
	Lead Width	Y	Y	Y	Y	Y
	Lead Thickness	Y	Y	Y	Y	Y
	Lead Foot Length	Y	Y	Y	Y	Y
	Lead Hole Diamater	N	N	N	N	N
	Lead Modelling	Y	Y	Y	Y	Y
	Lead Layout	Y	Y	Y	Y	Y
	Lead Pitch	Y	Y	Y	Y	Y
	Lead Bend (degrees)	Y	Y	Y	Y	Y
	Lead Shoulder	Y	Y	Y	Y	Y
	Lead Standoff	Y	Y	Y	Y	Y
	Lead Long Count	Y	Y	Y	Y	Y
	Lead Short Count	Y	Y	Y	Y	Y
	Sink Length	Y	Y	Y	Y	Y
	Sink Width	Y	Y	Y	Y	Y
	Sink Thickness	Y	Y	Y	Y	Y
Pad	**Pad Units**	Y	Y	Y	Y	Y
	Pad Length	Y	Y	Y	Y	Y
	Pad Width	Y	Y	Y	Y	Y
	Pad Pitch	Y	Y	Y	Y	Y
Ball	**Ball Pattern**	Y	Y	Y	Y	Y
	Ball Count	Y	Y	Y	Y	Y
	Ball Units	Y	Y	Y	Y	Y
	Ball Pitch	Y	Y	Y	Y	Y

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Ball Diameter	Y	Y	Y	Y	Y
	Ball Package Diameter	Y	Y	Y	Y	Y
	Ball Pad Diameter	Y	Y	Y	Y	Y
	Ball Height	Y	Y	Y	Y	Y
	Ball Channel Width	Y	Y	Y	Y	Y
	Ball Material	Y	Y	Y	Y	Y
	Ball Modeling	Y	Y	Y	Y	Y
	Ball Perimeter Rows	Y	Y	Y	Y	Y
	Ball Perimeter Columns	Y	Y	Y	Y	Y
	Ball Island Rows	Y	Y	Y	Y	Y
	Ball Island Columns	Y	Y	Y	Y	Y
Solder	**Solder Material**	Y	Y	Y	Y	Y
	Stencil Units	Y	Y	Y	Y	Y
	Stencil Thickness	N	N	N	N	N
	Solder Model	N	N	N	N	N
Electrical	**Max Rated Power (W)**	N	N	N	N	N
	Applied Power (W)	N	N	N	N	N
	Max Rated Current (A)	N	N	N	N	N
	Applied Current (A)	N	N	N	N	N
	Max Rated Voltage (V)	N	N	N	N	N
	Applied Voltage (V)	N	N	N	N	N
	Applied Freq (MHz)	N	N	N	N	N
	Capacitance (uF)	N	N	N	N	N
	Resistance (ohm)	N	N	N	N	N

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Electrical Derating (0-1)	N	N	N	N	N
	Rated Ripple (mA)	N	N	N	N	N
	Applied Ripple (mA)	N	N	N	N	N
Thermal	**Thermal Units**	N	N	N	N	Y
	Part Temp Rise	N	N	N	N	Y
	Min Rated Temp (°C)	N	N	N	N	N
	Max Rated Temp (°C)	N	N	N	N	N
	Min Storage Temp (°C)	N	N	N	N	N
	Max Storage Temp (°C)	N	N	N	N	N
	Lifetime	N	N	N	N	N
	Ripple Temp Rise	N	N	N	N	N
	Rjc (C/W)	N	N	N	N	N
	Rjb (C/W)	N	N	N	N	N
Die	**Die Units**	Y	Y	Y	Y	Y
	Die Length	Y	Y	Y	Y	Y
	Die Width	Y	Y	Y	Y	Y
	Die Thickness	Y	Y	Y	Y	Y
	Die Material	Y	Y	Y	Y	Y
	Process Node	N	N	N	N	N
Flag	**Flag Units**	Y	Y	Y	Y	Y
	Flag Length	Y	Y	Y	Y	Y
	Flag Width	Y	Y	Y	Y	Y
	Flag Thickness	Y	Y	Y	Y	Y
Test	**Test Duration**	N	N	N	N	N
	Number of Devices	N	N	N	N	N
	Number of Failures	N	N	N	N	N
	Confidence Bounds (%)	n/a	n/a	n/a	n/a	n/a

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Test Temperature	N	N	N	N	N
	Test Voltage	N	N	N	N	N
	Test Frequency	N	N	N	N	N
	Test Duty Cycle	N	N	N	N	N
Semi-conductor Wearout	**Input Data Type**	N	N	N	N	N
	Reliability Data Source	N	N	N	N	N
	Documents	N	N	N	N	N
	Reliability Metric	N	N	N	N	N
	Reliablity Metric Units	N	N	N	N	N
	Source of AF	N	N	N	N	N
	Activation Energy (Ea)	N	N	N	N	N
	Voltage Exponent	N	N	N	N	N
	Frequency Exponent	N	N	N	N	N

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Life Cycle Management

For the most part, Ansys Sherlock analysis modules attempt to predict the reliability of an electronic circuit card and its components based on the circuit card design and the expected environmental conditions to be experienced by the circuit card over its expected service life. As such, it is very important for Sherlock users to enter an accurate model of the expected service life. To help facilitate such activities, Sherlock allows users to create and maintain various **Life Cycles** that can be shared across projects and used as the basis for more customized definitions. Life Cycles can be refined over time as needed to increase their accuracy or account for previously unexpected environmental effects. Life Cycles can also be defined to mimic the conditions experienced during laboratory testing, allowing users to validate the reliability results predicted by Sherlock with those predicted by lab testing.

Life Cycles consist of one or more Phases which, in turn, consist of one or more Events. For the most part, Phases are used to group Events into logical collections for management purposes, allowing users to organize things however they want. In this document, we'll discuss a couple different ways of organizing Life Cycle definitions to demonstrate this flexibility.

In this section, the following topics are covered:

5.1. Tutorial Life Cycle

5.2. Phase Definitions

5.3. Project Life Cycles

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1. Tutorial Life Cycle

The examples provided in this document make use of the Tutorial Project. If you haven't already imported the Tutorial Project, proceed as follows:

- Select the **Import Project** option from the **Project** menu located in the **Main Menu.**
- Enter **Tutorial Project** (or anything you want) as the project name and press the **Browse** button to locate the project archive file to be imported. In this case, we're looking for the `Tutorial Project.zip` file located in the **tutorial** sub-directory of the Sherlock installation directory:

  Windows:

  ```
  C:/Program Files/Ansys Inc/v251/sherlock/tutorial
  ```

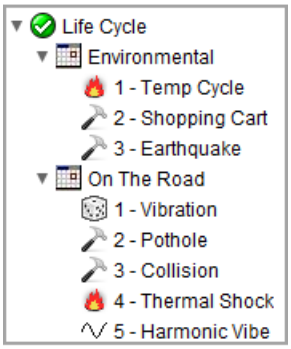
  Linux:

  ```
  /ansys_inc/v251/sherlock/tutorial
  ```

- Press the **Import Project** button to import all files associated with the tutorial project.

If you previously imported the Tutorial Project and then closed it, do the following: Right-click the **Tutorial Project** folder (or whatever you named it) in the **Project Tree** and select the **Open Project** menu option.

In either case, the Tutorial project should now be shown in the **Project Tree** (see below) and, if necessary, you should expand the **Life Cycle** folder to show all Life Cycle elements. We see that the Life Cycle consists of two Phases (Environmental and On The Road) and a total of eight Events across those Phases. Let's take a look at each of the Phases and Events to see how they relate to each other and what environmental affects they represent.



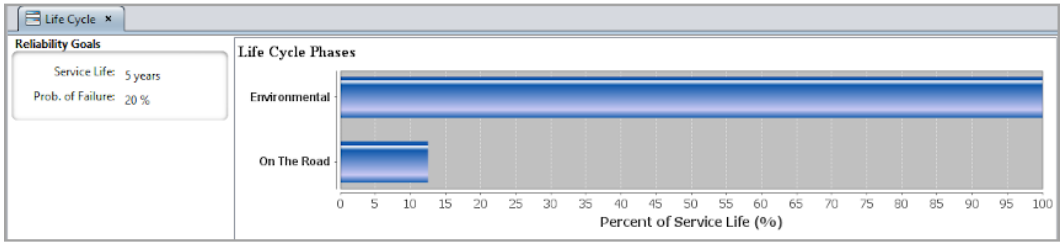
Note:

Life Cycle definitions used in the tutorial project don't represent a very realistic world. The definitions were contrived to allow use to show various aspects of the Sherlock analysis process and results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

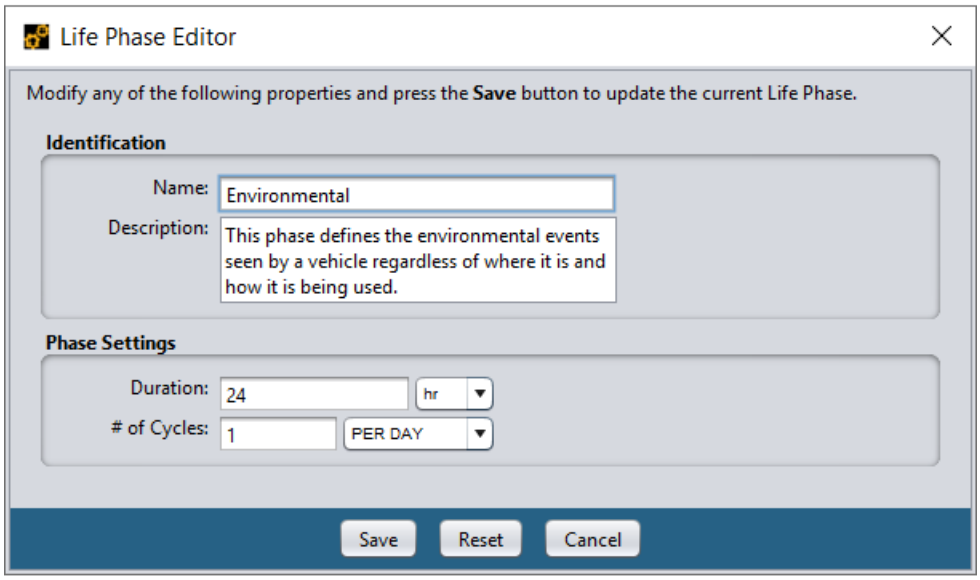
5.2. Phase Definitions

Right-click the **Life Cycle** folder in the **Project Tree** and select **View Life Cycle** from the context menu to display the **Life Cycle** tab in the **Content Panel**. If necessary, select the **Phases** sub-tab at the bottom of the **Life Cycle** tab to graphically display the relative duration of each Phase, as shown here:



We can clearly see that the **Environmental** Phase spans the entire five years' service life, whereas the **On The Road** Phase spans less than 15% of the service life. For the most part, the single most important property associated with a Phase is its duration measured over the entire service life. Although Sherlock provides many ways to specify a Phase duration, they can always be reduced to the percentage of time that the Phase is active.

Let's dig a little detail to see how these two Phases are defined. Right-click the **Environmental** Phase entry in the **Project Tree** and select the **Edit Properties** menu option to display the properties assigned to that Phase. (You may also simply double-click a Phase entry to display the property dialog.)



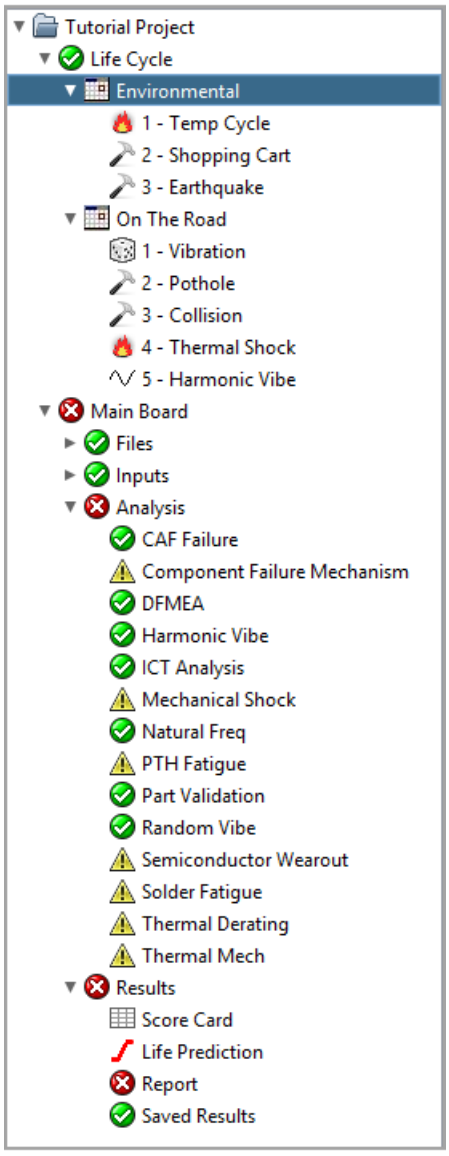
The **Name** and **Description** properties are used for organizational purposes and can be anything you want. The description is displayed in the Sherlock Report when Life Cycle details are selected.

The remaining two properties, **Duration** and **# of Cycles**, are used by Sherlock to determine how often the Phase is in effect with respect to the Service Life (specified for the Life Cycle itself) either in absolute or relative terms. In this example, the Environmental Phase is active for a period of 24 hours and the period repeats itself once per day. In other words, the Environmental Phase is always active, spanning every day of the Service Life. This might seem like a silly way to specify 100%, but we'll see shortly why two properties are needed and why they make Phase and Event definitions far more flexible.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While we've got the **Life Phase Editor** open, let's go ahead and change something. Set the number of cycles to 180 per year and press the Save button. At that point, the Life Phase Editor dialog will be dismissed, and the **Life Cycle** tab will be updated to show that the Environmental Phase now covers roughly 50% of the service life. If you want exactly 50% of the time, you can enter 0.5 per day as the number of cycles. In theory, we could also reduce the Phase duration to 12 hours to achieve the same coverage, but that won't work in this example because of the way the Events are defined, as we'll see shortly.

In addition to the **Life Cycle** tab being updated, you might also have noticed that quite a few entries in the **Project Tree** were updated, with lots of red X and yellow caution icons appearing alongside analysis and result entries. (See below.) This is the result of one of Sherlock's key features: **If any of the input values used by an analysis module are modified, then the results previously generated by that module are cleared.**



In this case, since we change the effective duration of one of the Life Cycle Phases, then all analysis modules that rely on that Phase will clear their results. If you hover the mouse over any of the caution

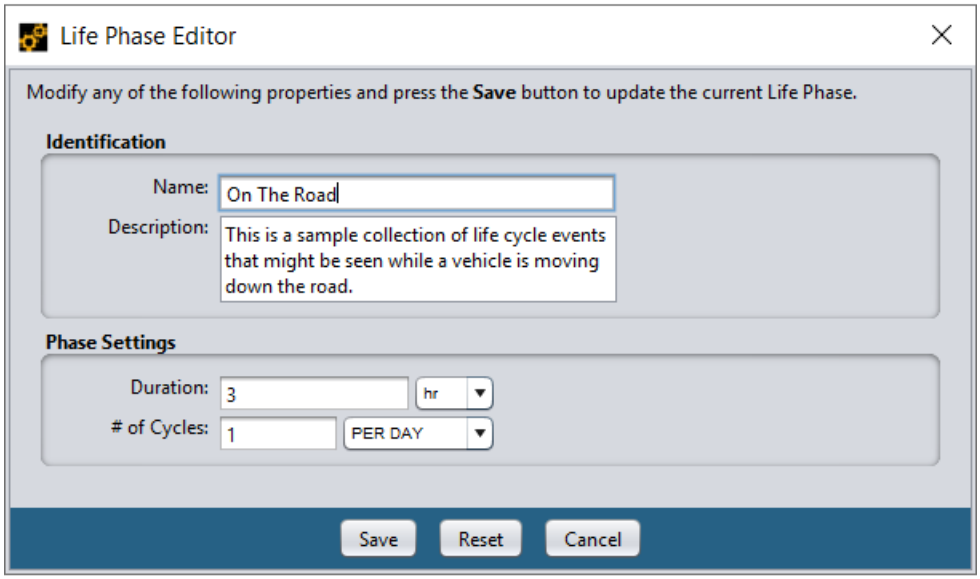
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

icons, you'll see a message indicating that the analysis needs to be executed (that is, it currently has no results).

Note:

When changing a fundamental property like **Time**, it is in your best interest to make all such changes at the same time and double-check your changes before re-executing all the analysis tasks.

Now, let's take a look at the **On The Road** Phase definition to see how it differs by double-clicking the **On The Road** folder in the **Project Tree** to display it's properties. Focusing on the **Phase Settings**, we see that the Phase is only active for 3 hours, once per day. This corresponds to the 12.5% of service life shown in Phase bar chart on the **Life Cycle** tab.



As the description states, this Phase is defined to model the environmental affects seen by a circuit card when it is moving down the road on a vehicle of some kind. The duration and number of cycle settings indicate that such activities only happen 12.5% of the service life.

Note:

When defining Phases and Events, we don't know (or care) *when* the Events for a given Phase will occur, we are simply stating the probability that they will occur during the service life. The Sherlock analysis modules only need to know the total time period during which the Events could cause damage.

5.3. Project Life Cycles

Topics in this Section:

5.3.1. Event Definitions

5.3.2. Thermal Event

5.3.3. Shock Event

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.4. Random Event
5.3.5. Harmonic Event
5.3.6. Load Direction and PCB Orientation
5.3.7. Life Cycle Editing
5.3.8. Life Cycle Import / Export
5.3.9. Life Cycle Manager

5.3.1. Event Definitions

Phases are defined to group together Events, while Events are defined to model physical affects from the real world on the circuit card. Phase and Event definitions are similar in that they both span certain time periods, but Events also have properties to define some real-world condition. Sherlock supports the following different types of Events to model environmental conditions:

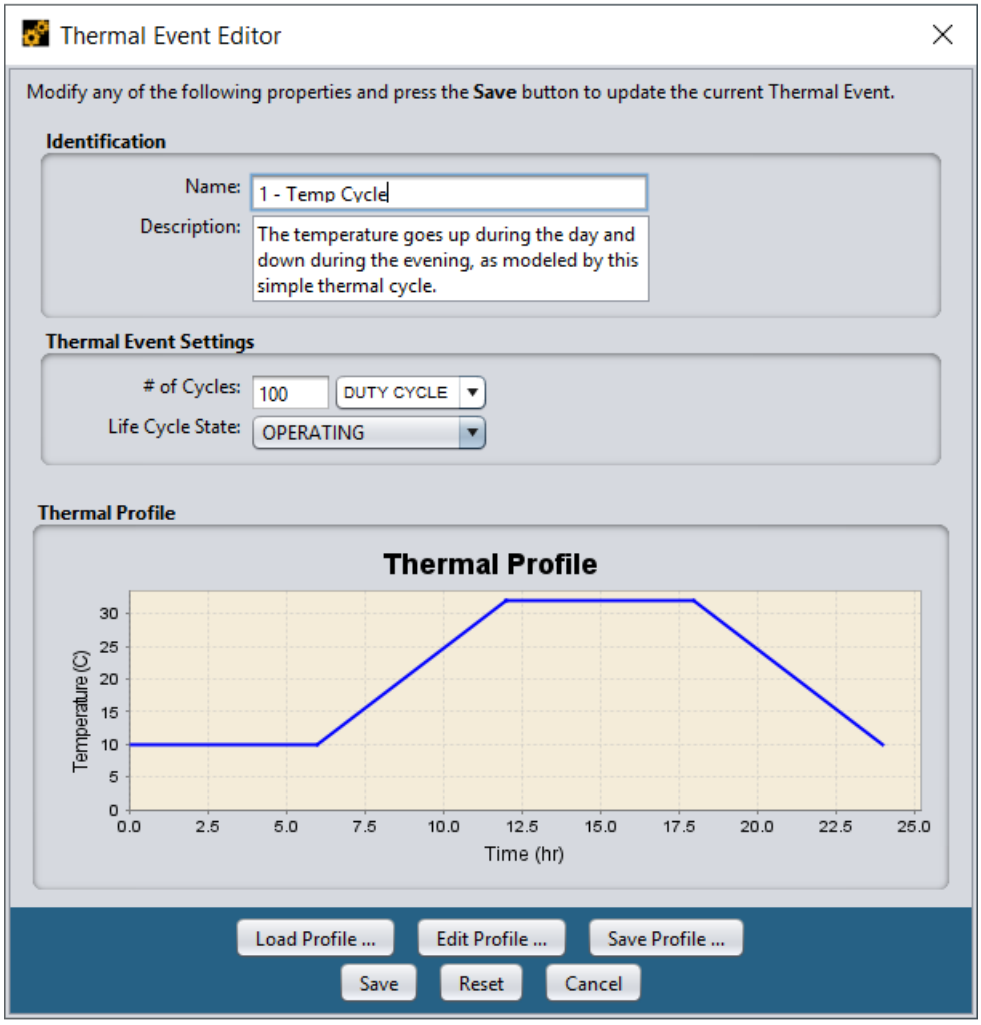
- **Thermal Event** (p. 344)– arbitrary temperature cycle
- **Shock Event** (p. 347)– mechanical shock pulse
- **Random Event** (p. 350) – random vibration
- **Harmonic Event** (p. 354) – harmonic vibration

Let's look at these types of Events by examining the examples provided by the Tutorial Life Cycle.

5.3.2. Thermal Event

Double-click the **Temp Cycle** Event defined in the **Environmental** Phase to see how a common temperature cycle is defined.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this case, the **Thermal Event** is being used to model a typical (albeit overly simplified) daily temperature cycle that oscillates between 10C and 32C over a 24-hour period. The Thermal Profile shows four 6-hour steps (hold, ramp up, hold, ramp down).

The **# of Cycles** setting is specified as 100% Duty Cycle which means that the Thermal Profile is repeated as many times as needed to cover 100% of the Phase duration specified. Since the Event duration is 24 hours, we can achieve the same overall specification by setting the number of cycles to one of the following:

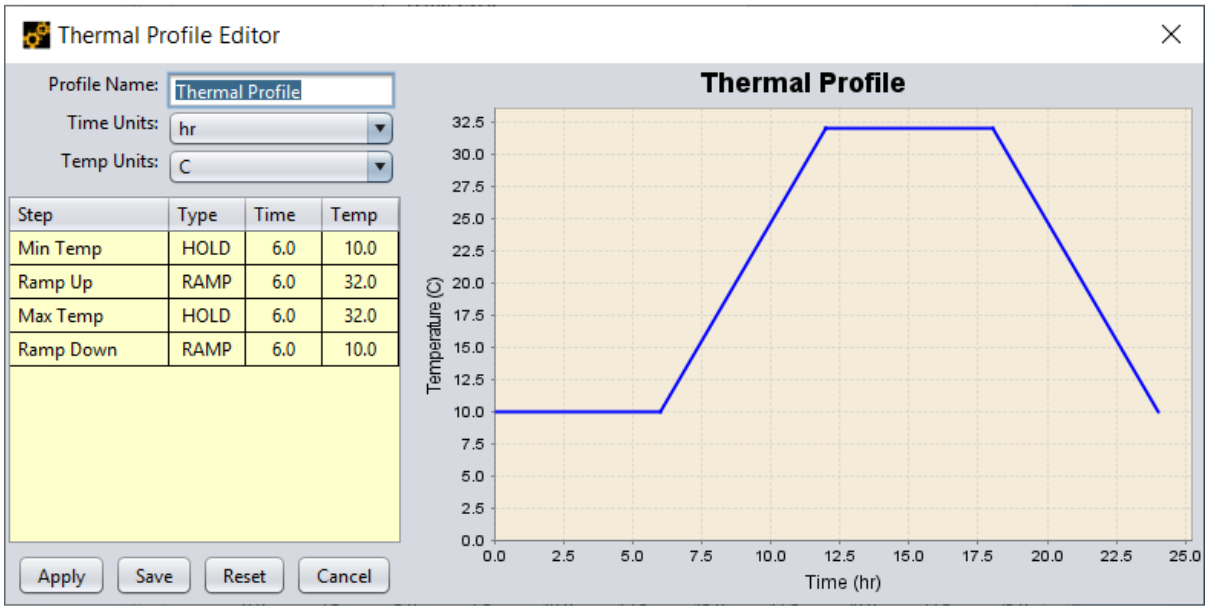
- 1 per day
- 365 per year
- or 100 duty cycle.

Note:

Duration values can be specified in a variety of ways because different people think about time in different ways. Sherlock allows users to use whatever approach that they think is proper for the type of Phases and Events being modeled.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are many different types of thermal profiles needed to model real-world thermal affects. As such, Sherlock provides some convenient features to help users maintain and modify such profiles. The features are provided by the three buttons displayed below the Thermal Profile in the **Thermal Event Editor**. Press the **Edit Profile ...** button to display the **Thermal Profile Editor**, as shown here:



This editor allows you to change the data points associated with the Thermal Profile by adding, removing or editing rows in the table on the left side of the dialog.

To add a row to any Life Cycle profile table, right-click a row on the table then select the **Add Row** item from the menu that appears.

To delete one or more rows, select the rows to be deleted, then right-click and select the **Delete Row** or **Delete Rows** item from the menu that appears.

To edit a cell, double-click the cell or select the row, then right-click and select **Edit Cell** from the menu that appears.

Rows can also be selected and copied to the clipboard to be used in another profile or taken into an out of another program such as a spreadsheet application. After selecting the rows to copy, right-click and select **Copy Row** or **Copy Rows** from the menu that appears.

To paste new rows from the clipboard into the table, select the row that should follow the pasted data then right-click and select **Paste Row** or **Paste Rows** from the menu that appears. Pasted rows are always inserted before the first selected row. If no rows are selected, then it will be appended to the table. The paste menu item is only enabled if the data in the clipboard contains the correct number of columns for the table to be pasted too. If invalid data is pasted, the table will not be able to be saved.

The overall duration of the profile is automatically computed, and the editor ensures that the profile is consistent (first and last temperatures are the same) before allowing you to proceed. After modifying the table, press the **Apply** button to see the corresponding profile in the graphical region of the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dialog. After you've got the profile you want, press the Save button to commit the changes and return to the **Thermal Event Editor**, where the profile will be updated.

Note:

Although Sherlock allows you to define thermal profiles with any number of steps, the analysis modules currently only use the minimum and maximum hold temperatures during their delta strain calculations.

Thermal profiles can be saved to named files and shared between Sherlock projects by using the **Save Profile ...** and **Load Profile ...** buttons displayed below the Thermal Profile. In both cases, either binary data file or CSV file formats can be specified, allowing users to edit thermal profiles outside of Sherlock and then import the profile definitions into appropriate **Thermal Events**. The ability to share thermal profiles across projects, Life Cycle Phases and Events allows users to develop profile libraries that can be refined over time to improve real-world modeling across many projects.

You can find additional information on Thermal Events in the FEA chapter. See Thermal Mech Properties Dialog (p. 713) under *Thermal Mech Analysis*.

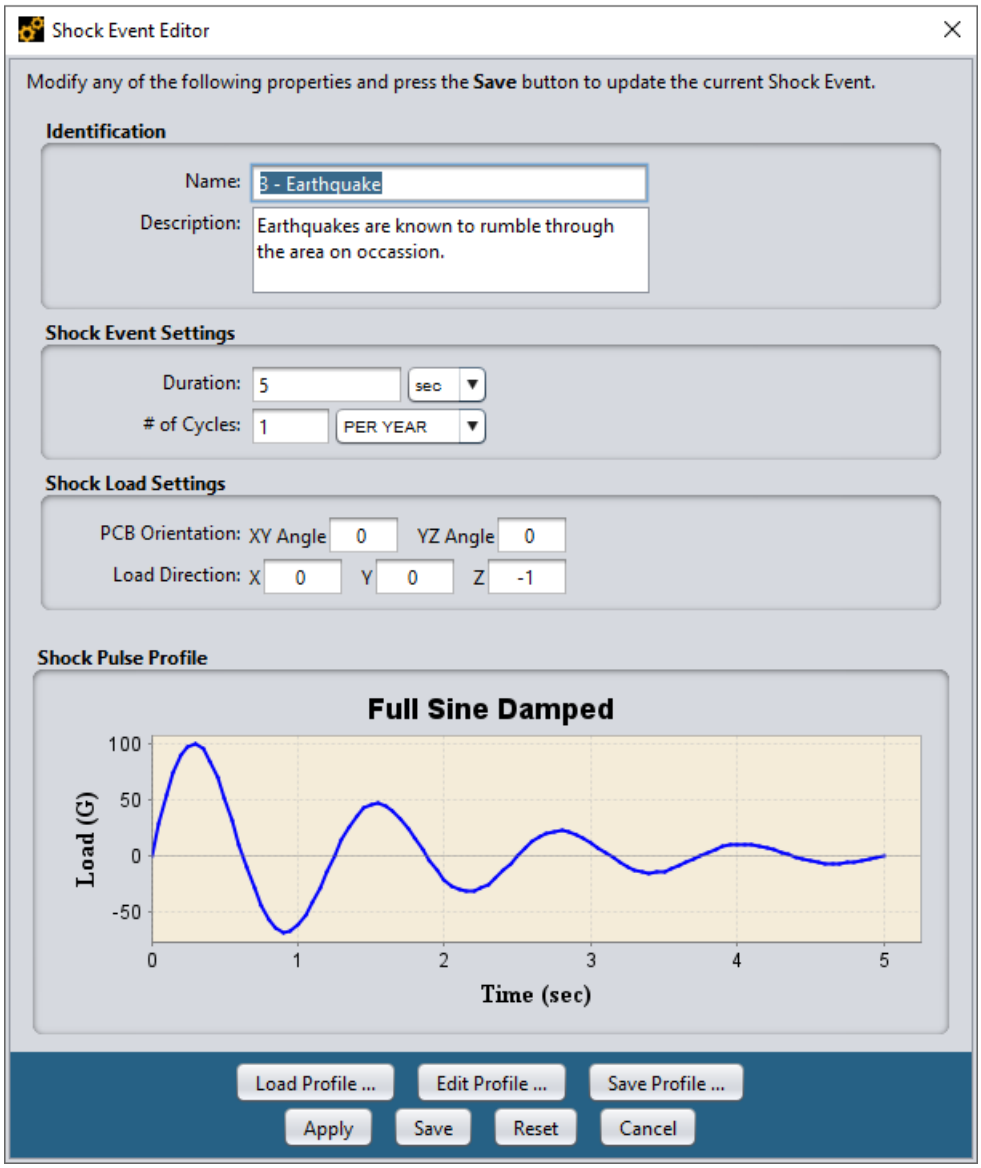
5.3.3. Shock Event

Double-click the **Earthquake** Event in the **Environmental** Phase to see how a mechanical shock Event is defined. As with Phases and other Events, a shock Event is defined by the **Duration** and **# of Cycles** properties, which indicate how long and how often the Event occurs. In this case, an earthquake is expected to occur just once per year, lasting 5 seconds.

As with **Thermal Events**, a shock Event is also defined by a shock profile, which shows the load exerted by the shock over time.

The direction of the shock load relative to the PCB can be specified using the **PCB Orientation** and **Load Direction** properties. By default, the PCB is assumed to be mounted parallel to the XY plane and the load is assumed to be directed in the downward Z direction. You can change one or both properties to change the direction of the load relative to the PCB. For example, if you want the load to be exerted from the lower-left corner of the PCB towards the upper-right corner, parallel to the XY plane, then you should specify (1, 1, 0) as the **Load Direction**. On the other hand, if you want the load to be exerted at a 45-degree down-angle you should specify a **PCB Orientation YZ Angle** of 45, while leaving the **Load Direction** as (0, 0, -1). See the Load Direction and PCB Orientation (p. 355) section below for more details.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

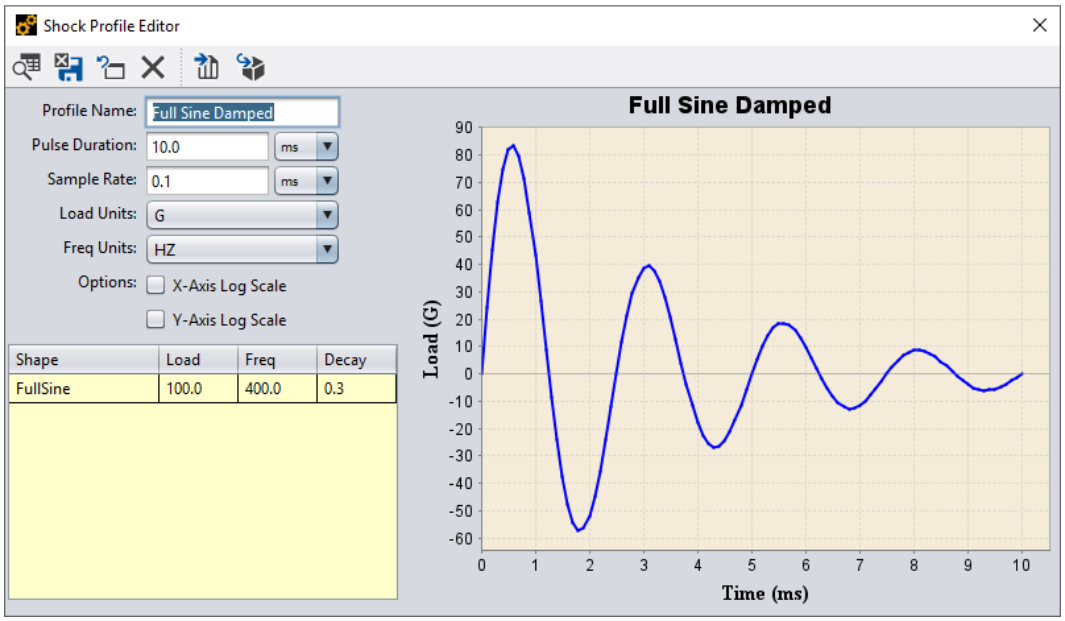


There are two ways to change the shape of the shock profile itself. The easiest way is to press the **Load Profile ...** button in the **Shock Event Editor** to load a different shock profile from a collection of previously defined profiles. Profile definition files can be stored as binary data files or CSV files, allowing users to create and customize profile shapes using whatever tools they want.

The second way to modify the shape of the shock profile is to press the **Edit Profile ...** button to display the **Shock Profile Editor** as shown below. Much like the **Thermal Profile Editor**, you can easily alter the profile by adding, removing or modifying rows in the table and pressing the **Apply** button to see the graphical results. Unlike the **Thermal Profile Editor**, however, the table does not define specific data points along the profile. Rather, it defines one or more primary shapes that Sherlock combines to define the overall shock profile.

If you have loaded a user-defined profile into the **Shock Event Editor** and you click **Edit Profile…**, a confirmation dialog will appear, alerting you that the data points will be deleted when the profile is edited.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameters and interface of the **Shock Profile Editor**:

- **Sample Rate**: This is the rate at which the shock pulse is sampled during the analysis. The number of sampling points is equal to the pulse duration divided by the sample rate. A minimum of ten points is required.
- **Load Units**: Select the units of acceleration.
- **Freq Units**: Select the units of frequency.

 Apply changes

 Save changes and exit editor

 Reset and discard table entries changes

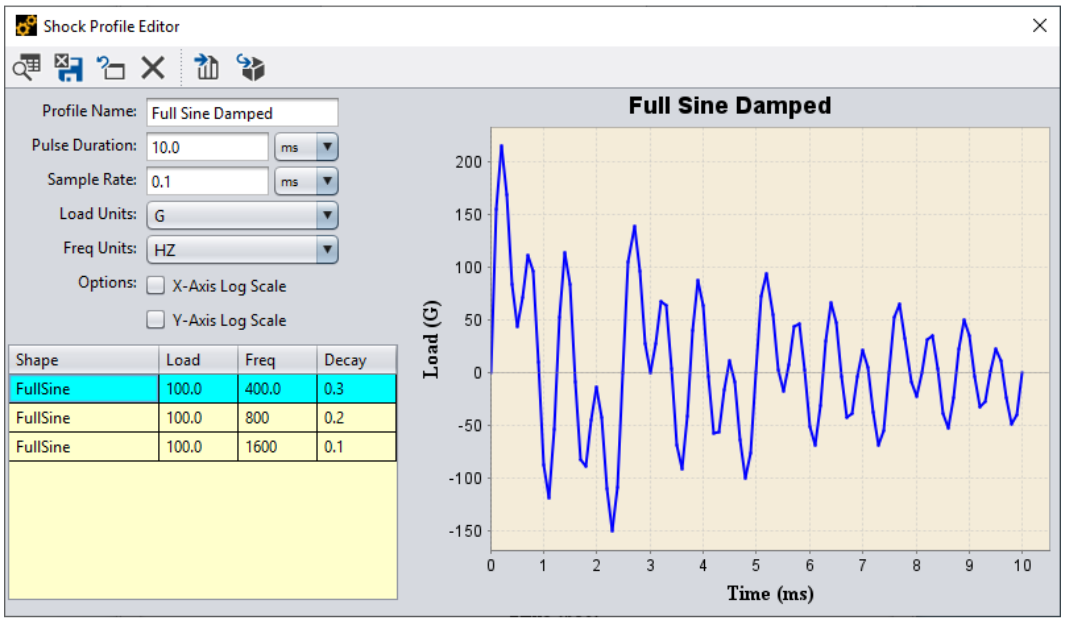
 Cancel and exit profile editor

 Export profile data entries

 Import entries into profile table

To see how this works, let's add a couple more primary shapes to the existing profile. Right-click inside the table and select the **Add Row** menu option. At that point, a duplicate of the first row will be appended to the table. Repeat the process to add a third identical row to the table. Now, change the **Frequency** and **Decay** entries of the second and third rows to define the harmonic components of the profile. Finally, change the **Profile Name** so that we can use this shape in future projects. After you've made your changes, press the **Apply** button and the resulting shape should look like this:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When the **Apply** button is pressed, Sherlock combines the three primary pulse shapes to generate the final pulse shape. You can use any number and type of primary shapes, with different individual loads, frequencies, and decay rates to create a large variety of pulse profiles.

Additional parameters include:

- **Sample Rate**: This is the rate at which the shock pulse is sampled during the analysis. The number of sampling points is equal to the pulse duration divided by the sample rate. A minimum of ten points is required.
- **Load Units**: Select units of acceleration.
- **Freq Units**: Select units of frequency.

Note:

Keep in mind, the Pulse Duration and Load values defined in the **Shock Profile Editor** are *relative* values used solely within the editor to define the profile shape. A combination of all the primary pulse shapes is then used to define the pulse profile.

You may also click **Export profile data entries** to export the data entries within the **Shock Profile Editor** for future use. When you're done modifying the shock profile, press the **Save** button to commit the changes and return to the **Shock Event Editor** where the profile will be updated. At that point, you can click **Save Profile ...** to save a copy of the profile for future use, choosing whatever file name you want.

We're done making changes to the **Shock Event**, so click **Save** to commit the changes and clear all analysis results that depend on shock Events.

5.3.4. Random Event

Topics:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.4.1. Random Vibe Profile

5.3.4.2. Shock Response Spectrum

5.3.4.1. Random Vibe Profile

Double-click the **Vibration** Event in the **On The Road** Phase to see how Random Events are defined. In image A below, we see the Event is defined by the **Duration and # of Cycles** properties, as well as a profile shape. In this case, the profile shape is a Power Spectral Density (PSD) curve, which defines the power of the vibration pulse as a function of frequency. This is a very efficient way to define how the vibration would affect a circuit card across a range of frequencies.

The **PCB Orientation** and **Load Direction** properties are used to define the orientation of the random load relative to the PCB, as discussed previously for Mechanical Shock loads. The **Profile Type** choice currently only supports **Uniaxial** forces. It is a placeholder for other types of random forces, such as **Triaxial**, that will be supported in the future.

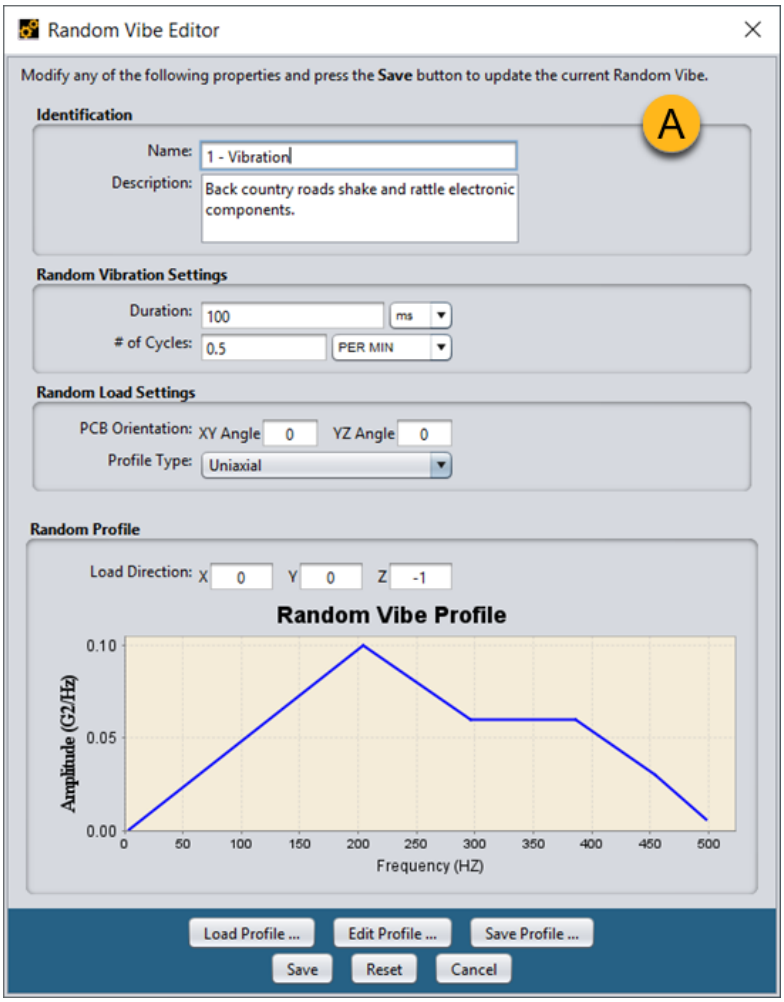
As with other profile types, you can import profiles from the file system by pressing the **Load Profile ...** button and can export profiles to the file system by pressing the **Save Profile ...** button. Random profiles can be stored as either binary data files or CSV files, allowing you to create and modify the profile data outside of Sherlock. In many cases, PSD curves are determined from measurements taken from real-world environments. In those cases, such curves can be easily loaded into Sherlock as CSV data files.

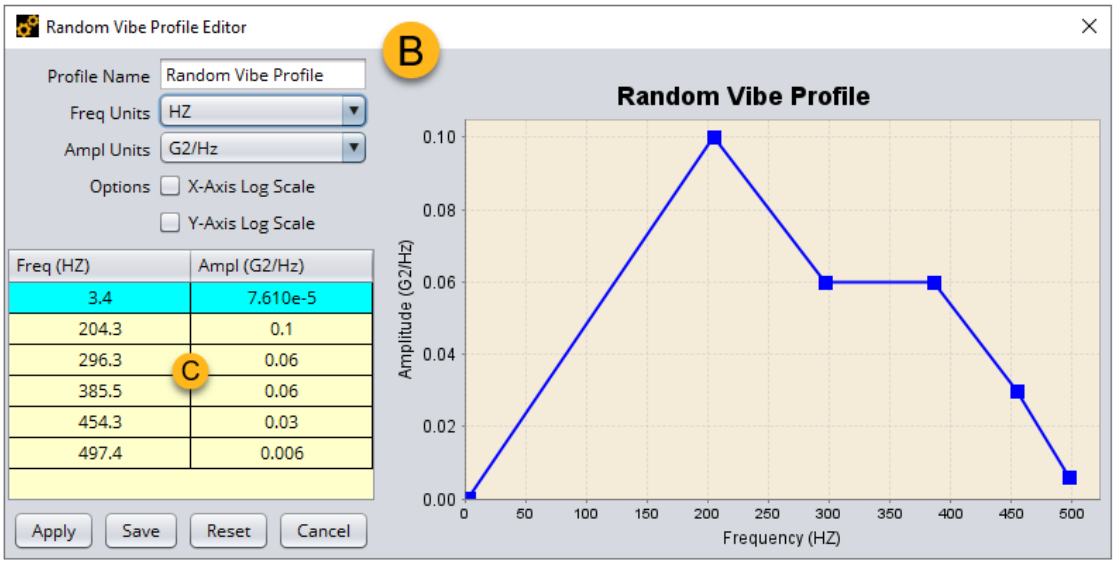
You may edit random profiles in Sherlock by pressing the **Edit Profile** button to display the **Random Vibe Profile Editor** (B, below).

Like the Thermal Profile editor, a table is provided which displays all the data points. This table allows you to add, remove, and modify them as needed to define the curve. As a convenience, the editor also allows you to move the data points in the graphical region by left-clicking any node and dragging it to a new location. The axes will automatically scale as needed, so don't be surprised when the node stops moving but the axes keep changing. With a little practice you'll get used to the process. Go ahead and drag some of the nodes around to see how it works. You can press the **Reset** button at any time to abort pending changes and reload the table and graph with the original profile data.

After you're done practicing, press the **Cancel** button to abort all changes and return to the **Random Vibe Editor**. Then, press the **Cancel** button in that dialog to cancel all changes to the random vibration Event.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Note:

Starting with the 2024 R2 release, Sherlock formats the numerical values in the **Profile Editor** tables as follows: Values less than 0.0001 are formatted in exponential notation

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

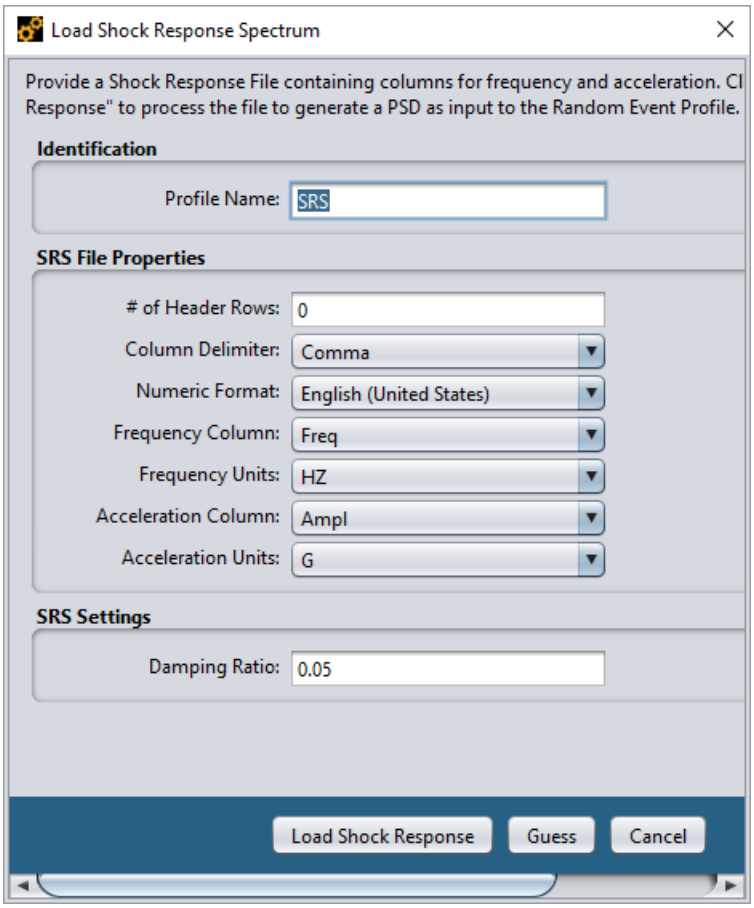
using three decimal places as shown in the top row of table C above. All values greater than or equal to 0.0001 are expressed using up to four decimal places as necessary. This applies to all the Event Profiles.

5.3.4.2. Shock Response Spectrum

Random vibration profiles may also be defined by providing a shock response spectrum specification (SRS). The SRS is converted to a PSD which is then used to populate the random profile. This process is done by loading a profile input SRS file. To see an example, select the **Load Profile ...** button, however, change the file type filter to **Shock Response Spectrum (*.srs)**. An SRS file is a text formatted CSV or white-space delimited file where each row is an SRS input consisting of the columns **Frequency** and **Amplitude**. An example file is here:

```
Freq,Ampl
10,9.4
80,75
2000,75
```

After selecting a file, the **Load Shock Response Spectrum** dialog will appear (shown below) to help define the delimiter, numeric format, column names, and the units of the frequency and acceleration data in the file. After clicking the **Load Shock Response** button, Sherlock will convert the response spectrum to a PSD and use it to populate the random profile. Once complete, the **Random Vibe Editor** profile will reflect the converted SRS data.



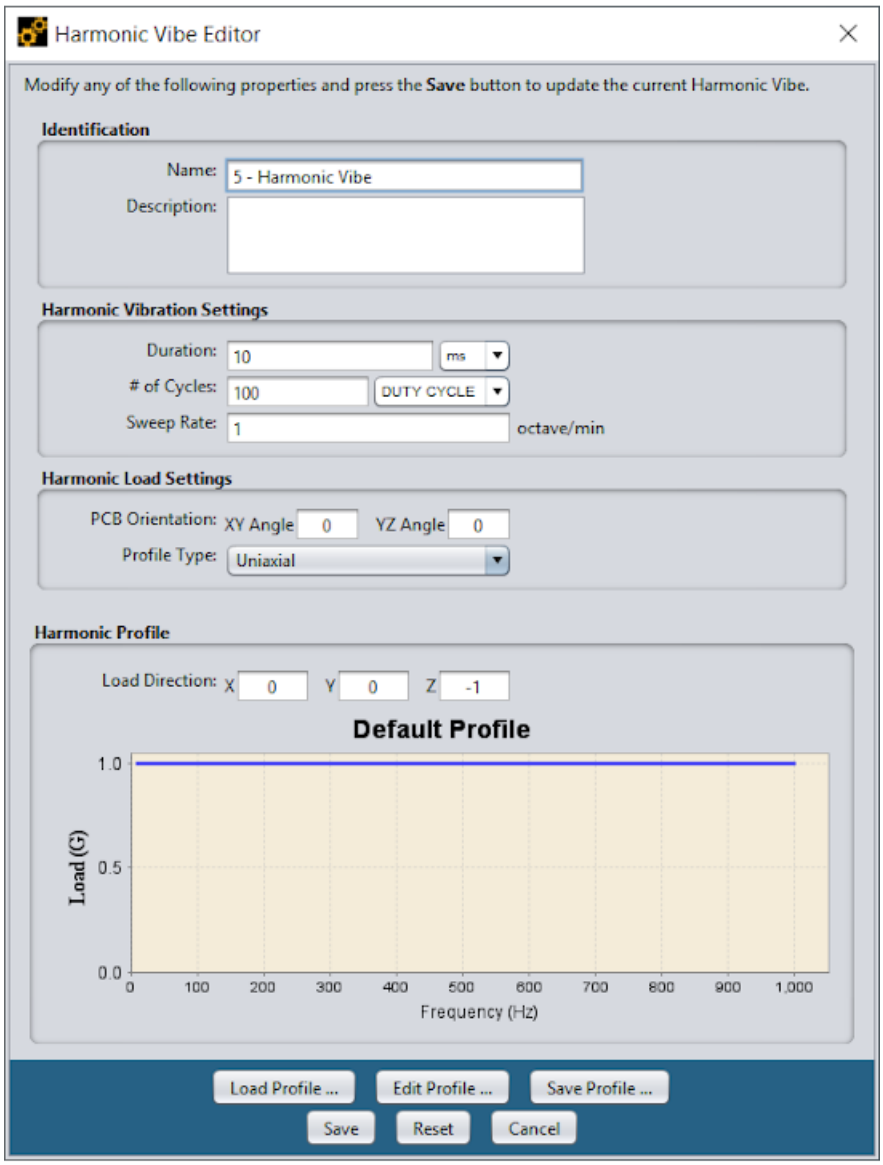
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you're done, press the **Cancel** button to abort all changes and return to the Random Vibe Editor. Then, press the **Cancel** button in that dialog to cancel all changes to the Random Vibration Event.

5.3.5. Harmonic Event

Harmonic Events are similar to Random Events except that the load profile is defined in terms of simple acceleration (G) vs. frequency (Hz) instead of a PSD curve. In addition, harmonic load profiles can be defined as either **Uniaxial** or **Triaxial**. As the names imply, uniaxial loads are defined in a single direction, as defined by the Load Direction property, whereas triaxial loads are defined in each of the X, Y and Z directions.

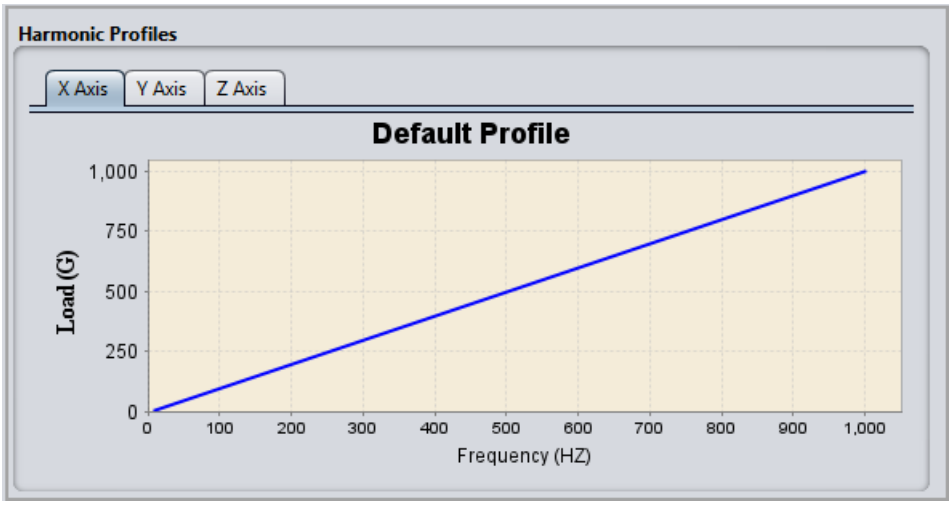
Let's see how triaxial loads are defined by adding a **Harmonic Event** to our **Life Cycle**. To that end, right-click the **On The Road** Phase icon in the **Project Tree** and select **Add Harmonic Vibe** from the context menu. The default dialog shown here will be displayed, allowing you to create a new Event.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter **Name** and **Description** values of your choosing and then select a Profile Type of **Triaxial** in the **Harmonic Load Settings** sub-form. At that point, the **Harmonic Profiles** sub-form will show three separate profiles, one for each direction.

Select the **X Axis profile**, then press the **Edit Profile** ... button to change the load profile for the X direction. In the example shown below, we've defined a linear profile for the X direction that ranges from 1G @ 10Hz to 1,000G @ 1KHz.

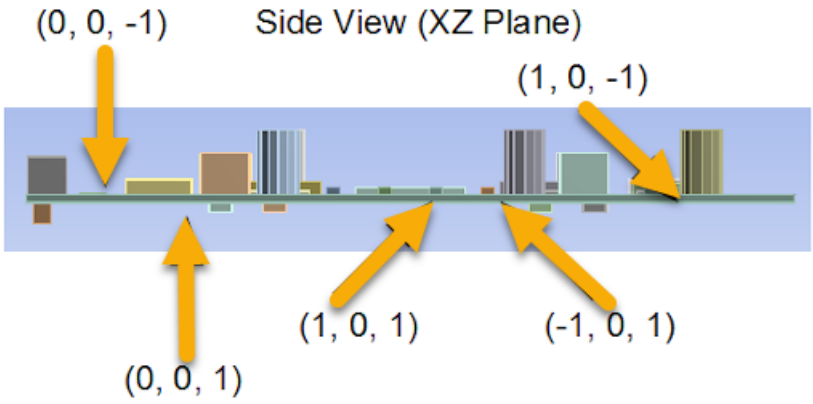


It is important to note that triaxial loads are always defined using the canonical X, Y and Z directions (that is, you cannot specify the load direction). You may change the **PCB Orientation** to change the direction of the triaxial loads relative to the PCB.

5.3.6. Load Direction and PCB Orientation

It's not always easy to visualize the effects of the **Load Direction** and **PCB Orientation** properties in 3D, especially when combined with each other, but the following diagrams should help. It is important to remember that both properties are specified relative to the global coordinate system, not to the PCB coordinate system.

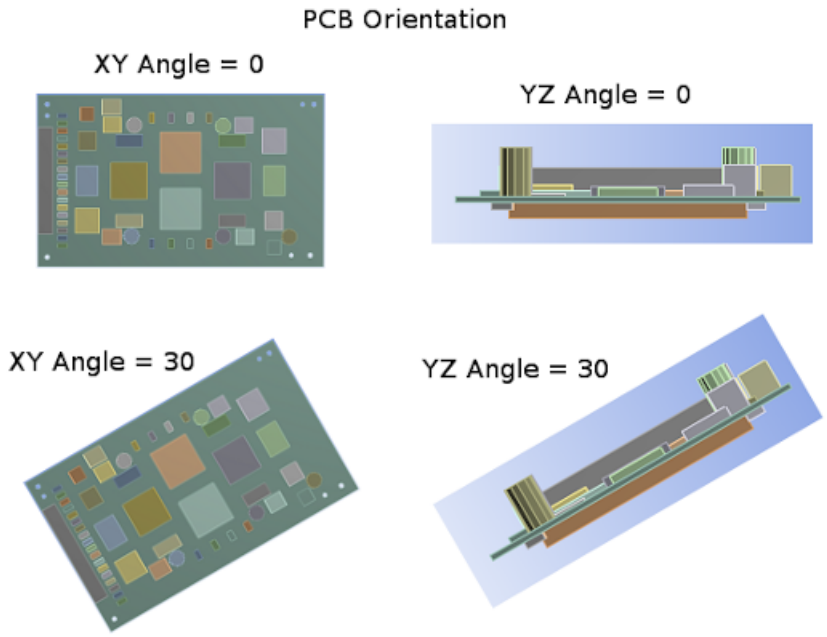
In the following view, we're looking at the bottom edge of the circuit board (XZ Plane).



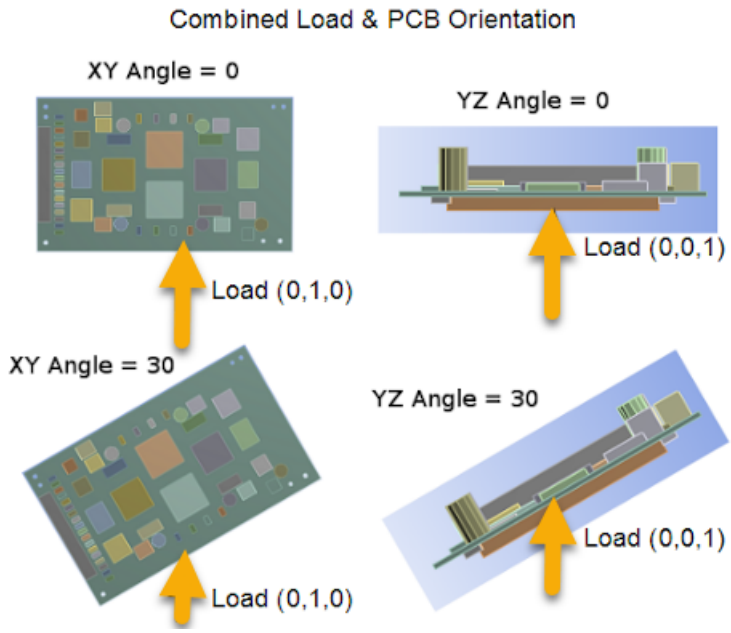
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The load arrows shown furthest to the left are pushing straight down on the top of the PCB (0, 0, -1) and straight up on the bottom of the PCB (0,0,1). The other arrows show loads angled in the X direction as well.

The views shown below illustrate the effect of the **PCB Orientation** angles. The views on the left are looking down on the top of the PCB (XY Plane), while the views on the right are looking at the side of the PCB (YZ Plane). Any 3D orientation can be achieved by setting the two angles appropriately.



Finally, the following views show a combination of **Load Directions** and **PCB Orientations**.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.7. Life Cycle Editing

We've shown you how to modify Life Cycle, Phase and Event properties, but not how to edit the Life Cycle itself by adding or removing Phases and Events. The process is quite simple, performed by a collection of menu options in the **Project Tree**.

Topics:

5.3.7.1. Adding Life Cycle Phases
5.3.7.2. Removing a Life Cycle Phase
5.3.7.3. Duplicating a Life Cycle Phase
5.3.7.4. Copy and Paste a Life Phase
5.3.7.5. Adding a Life Cycle Event
5.3.7.6. Duplicating a Life Cycle Event
5.3.7.7. Copy and Paste a Life Cycle Event
5.3.7.8. Deleting a Life Cycle Event

5.3.7.1. Adding Life Cycle Phases

To add a Phase to the Life Cycle, right-click the **Life Cycle** folder in the **Project Tree** and select the **Add Life Phase** menu option. At that point, the Life Phase Editor will appear, requiring you to enter a name for the new Phase. All Phase names must be unique and are sorted alphabetically when displayed in the Project Tree or Project Report.

5.3.7.2. Removing a Life Cycle Phase

To remove a Phase from the Life Cycle, right-click the Phase folder in the **Project Tree** and select the **Delete Life Phase**.

5.3.7.3. Duplicating a Life Cycle Phase

By right-clicking a given life Phase folder in the **Project Tree**, then selecting **Duplicate Life Phase**, a copy of the selected life Phase will be added to the project. An editor will automatically open so the Phase can be named and updates to other Phase properties can be made.

5.3.7.4. Copy and Paste a Life Phase

Similar to the duplicating of a life Phase, the ability to copy a Phase into the clipboard then to paste a life Phase into a Life Cycle may be done. Whereas duplicating the life Phase requires only a single selection, when you copy the Phase, the Phase must also be pasted back to the Life Cycle to be added. The advantage to this is that the Phase can be pasted into another open project in Sherlock. To copy a Phase, right-click a given life Phase folder in the **Project Tree** and select **Copy Life Phase**. Next right-click the **Life Cycle** folder or any **Life Phase** in the project you wish to copy the life Phase to and select **Paste Life Phase**. The **Paste Life Phase** menu option is only available when there is a life Phase in the clipboard ready to be copied.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.7.5. Adding a Life Cycle Event

To add an Event to any Phase, right-click the Phase folder in the **Project Tree** and select one of the following menu options:

- Add Thermal Cycle
- Add Harmonic Vibe
- Add Random Vibe
- Add Shock Event

As when adding a Phase, you'll be required to enter a unique name for the Event. If you want Events to be displayed in a specific order, simply prefix the name accordingly, as was done with the tutorial Life Cycle Events.

5.3.7.6. Duplicating a Life Cycle Event

You can quickly create similar Events in the same Phase by right-clicking the **Event** name in the **Project Tree** and selecting the **Duplicate Event** menu option. At that point, a copy of the selected Event will be created. The Sherlock application generates a default **Name** for the new Event (name of the original Event plus an incremented numerical identifier), but you may edit the **Name** as you wish. You may also modify any or all properties associated with the new Event as needed.

5.3.7.7. Copy and Paste a Life Cycle Event

Similar to the duplicating of a Life Cycle Event, the ability to copy an Event into the clipboard then to paste an Event into a Life Cycle may be done. Whereas duplicating the Life Cycle Event requires only a single selection, when you copy the Event, the Event must also be pasted back to the Life Cycle to be added. The advantage to this is that the life Event can be pasted into another Life Cycle Phase or open project in Sherlock. To copy a life Event, right-click a given life Event and select **Copy Event**. Next right-click the life Phase folder or any life Event folder in the project and life Phase you wish to copy the Event to and select **Paste Event**. The **Paste Event** menu option is only available when there is a life Event in the clipboard ready to be copied.

5.3.7.8. Deleting a Life Cycle Event

To remove an Event from a Life Cycle Phase, right-click the Event name in the **Project Tree** and select the **Delete Event** menu option.

5.3.8. Life Cycle Import / Export

Clearly, the amount of time spent making accurate Life Cycle definitions can be large depending on the complexity of the Life Cycle and Life Cycles can apply to multiple projects. For these reasons, Sherlock allows users to export Life Cycle definitions to the file system and import them at a later time as needed. That is, users can maintain libraries of Life Cycle definitions, refining them over time to be more accurate and re-using them across multiple projects and analysis tasks.

Topics:

5.3.8.1. Exporting a Life Cycle

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.8.2. Importing a Life Cycle

5.3.8.1. Exporting a Life Cycle

An entire Life Cycle definition can be exported by right-clicking the **Life Cycle** folder in the Project Tree and selecting the **Save Life Cycle** menu option to display the **Save Life Cycle Dialog**. By default, Life Cycle definition files are stored in the lifeCycles sub-directory of the Sherlock User Directory, but you are free to store Life Cycle definitions in any directory of your choosing. As a convenience, Sherlock will remember the selected directory and return to it automatically whenever you load or save Life Cycles.

When the **Save Life Cycle** dialog is displayed, the name assigned to the Life Cycle (using the **Life Cycle Properties** dialog) will be used as the default file name. Feel free to change the file name to whatever suits your purpose.

Note:

Sherlock currently stores all Life Cycle definitions as binary data files, which means that they can only be edited in Sherlock. Future Sherlock versions may support additional file formats, such as XML, to allow users to create and modify Life Cycle definitions outside of Sherlock.

5.3.8.2. Importing a Life Cycle

An entire Life Cycle definition can be imported by right-clicking the **Life Cycle** folder in the **Project Tree** and selecting the **Load Life Cycle** menu option to display the **Load Life Cycle** dialog. By default, Sherlock will return to the directory last used to import or export a Life Cycle, but you are free to navigate to any directory to locate the appropriate definition file. Once the proper directory is found, simply select the desired file and press the **Load Life Cycle** button to replace the current **Life Cycle** definition with that from the definition file.

5.3.9. Life Cycle Manager

All of the examples shown so far in this document made use of the Life Cycle edit capabilities that are built into the Sherlock Client application. Such capabilities are also provided by the **Life Cycle Manager**, which is a standalone application that can be launched from the Windows desktop. The Life Cycle Manager is automatically installed by the Sherlock Installer.

5.3.9.1. Life Cycle Definition Files

5.3.9.2. Launching the Life Cycle Manager

5.3.9.3. Life Cycle Manager Window

5.3.9.4. Life Cycle Manager Menu Options

5.3.9.1. Life Cycle Definition Files

In previous versions of Sherlock, project Life Cycles were saved in files ending with a *.dat file name suffix. In more recent versions of Sherlock, Life Cycle files are saved using the unique ***.dfr-lc** file name suffix to differentiate them from other files in the file system. The **Life Cycle Manager** is capable of reading both file formats but will only generate the ***.dfr-lc** format.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Life Cycle definition files can be freely copied, moved and deleted using common Windows desktop and file system tools, thereby allowing users to make back-up copies and otherwise manage them using whatever file utilities are appropriate.

5.3.9.2. Launching the Life Cycle Manager

If running on Windows, the Sherlock **Life Cycle Manager** can be launched using any of the following methods:

- From Sherlock's main menu, select **Libraries > Life Cycles**
- Double-click any Life Cycle file with a file name suffix of ***.dfr-lc**
- Select **Sherlock Life Cycle Manager** from the Ansys folder accessible from the **Windows Start Menu**
- Double-click the `SherlockLifeCycle.exe` executable file located in the Sherlock installation folder (usually `c:\Program Files \Ansys Inc\v251\sherlock`).

Linux

If Sherlock is running on Linux, you can open the Life Cycle Manager a couple of ways:

- From Sherlock's main menu, select **Libraries > Life Cycles**
- Run the command `runSherlock -lifecycle`

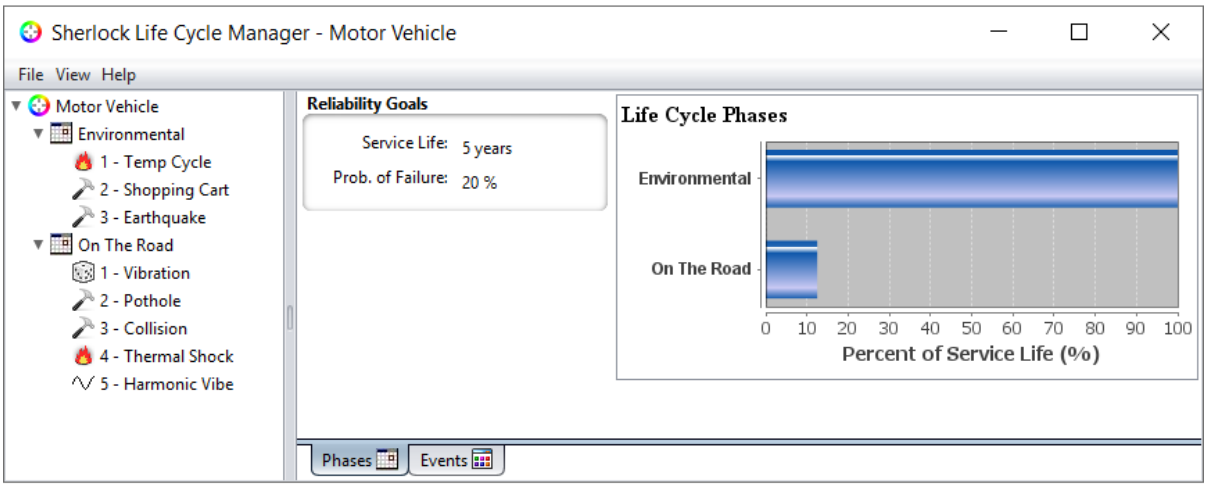
By default, you cannot open Sherlock files by double-clicking them. You must manually associate file types to specific applications within Linux.

5.3.9.3. Life Cycle Manager Window

The **Life Cycle Manager** window is relatively simple, containing the Life Cycle navigation tree, two summary panels and a menu bar, as shown below.

All other functionality is provided by the same Phase and Event edit dialog windows described in previous sections. As in Sherlock Client, to edit any Phase or Event, simply double-click the corresponding node in the Life Cycle tree and the edit dialog will be displayed. After any changes are made, the **Phases** and **Events** summary panels will be automatically updated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5.3.9.4. Life Cycle Manager Menu Options

The following menu options are provided by the Life Cycle Manager:

5.3.9.4.1. Create Life Cycle

5.3.9.4.2. Open Life Cycle

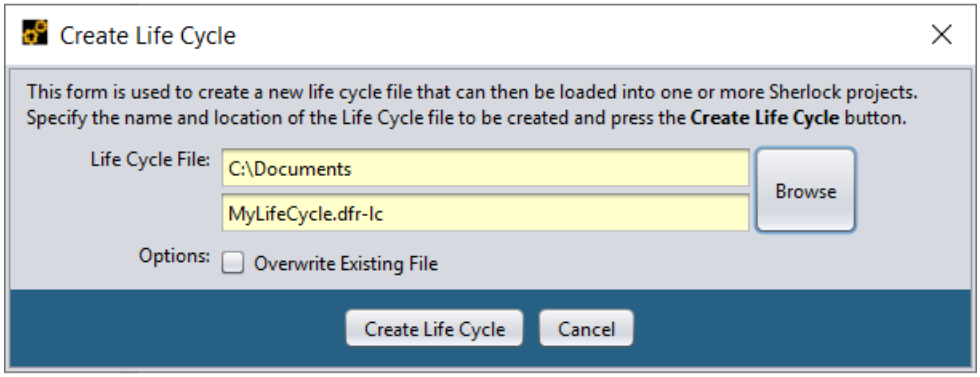
5.3.9.4.3. Close Life Cycle

5.3.9.4.4. Generate Report

In addition to these functions, menu options exist to view the console window and exit the application. We now discuss each of these functions.

5.3.9.4.1. Create Life Cycle

The **File > Create Life Cycle** menu option is enabled when there is no Life Cycle currently opened. When selected, a dialog will appear to enter the location and name of the file to be created.



When the **Create Life Cycle** button is pressed, an empty Life Cycle file will be created in the designated folder and all editing functions will be enabled.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.9.4.2. Open Life Cycle

Existing Life Cycle files can be opened using the **Open Life Cycle** menu option (when no Life Cycle is currently opened). When this menu option is chosen, a file dialog will appear to allow you to select the **Life Cycle** file to be opened.

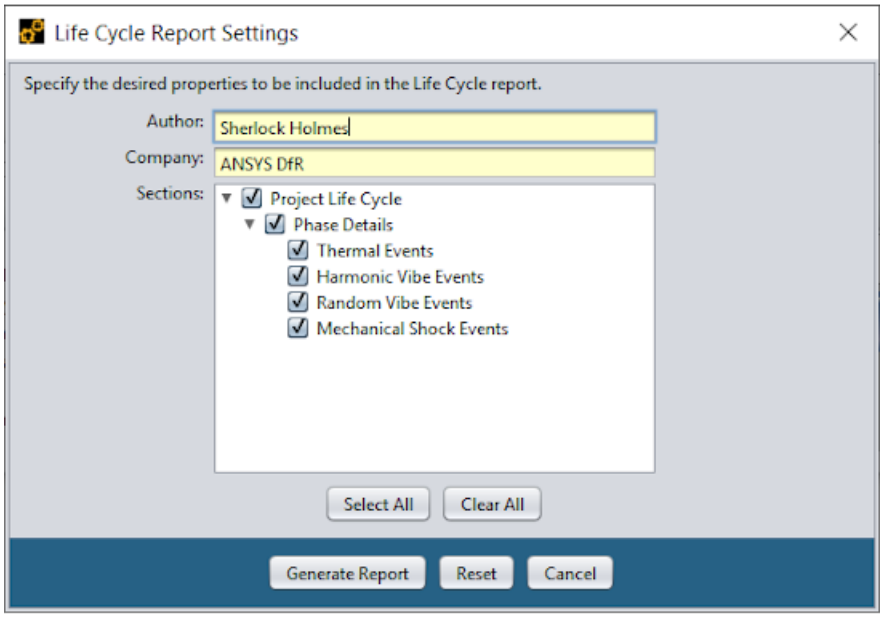
When the **Open Life Cycle** button is pressed, the designated Life Cycle file will be opened, and all editing functions will be enabled.

5.3.9.4.3. Close Life Cycle

Select the **File > Close Life Cycle** menu option to close the currently opened Life Cycle and clear the **Life Cycle Manager** window.

5.3.9.4.4. Generate Report

The **File > Generate Report** menu option allows users to generate a PDF report that summarizes the current Life Cycle definitions. The **Sections** selection list can be used to include specific types of information in the report.



When the **Generate Report** button is pressed, a PDF file will be generated and stored in the same folder as theLife Cycle definition file, using the same file name with a *.pdf suffix.

The PDF will also be displayed using the application currently associated with PDF files in Windows.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Failure Analysis

Topics Covered in this Chapter

6.1. Design Failure Mode and Effects Analysis (DFMEA)

6.2. Component Failure Analysis

6.3. Plated Through Hole (PTH) Fatigue Analysis

6.4. Semiconductor Wearout

6.5. Solder Fatigue Analysis

6.6. Solder Fatigue Analysis Module

6.7. Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis

6.8. Solder Fatigue Tool

6.9. Thermal Derating

6.1. Design Failure Mode and Effects Analysis (DFMEA)

Design Failure Mode and Effects Analysis (DFMEA) is the application of the Failure Mode and Effects Analysis method specifically to product design. It is a paper-and-pencil analysis method used in engineering to document and explore ways that a product design might fail in real-world use. DFMEA documents the key functions of a design, the primary potential failure modes relative to each function and the potential causes of each failure mode. The DFMEA method allows the design team to document what they know and suspect about a product's failure modes prior to completing the design, and then use this information to design out or mitigate the causes of failure.

Ansys Sherlock supports DFMEA by providing convenient mechanisms for maintaining a DFMEA matrix using an effective hierarchical GUI, exporting all or part of the matrix to an XLS spreadsheet using one or more custom templates, and importing changes made to such spreadsheets back into Sherlock for subsequent use. Sherlock assists in the preparation of the DFMEA matrix by leveraging data from other modules, such as providing data extracted from PCB design files, data stored in the **Part Database**, and analysis of pin locations, to name a few examples. Finally, Sherlock provides a collection of analysis results, such as the distribution of **Risk Priority Numbers (RPNs)** calculated for each part in a PCB to help users understand the overall impact of their DFMEA activities.

Note:

The examples shown in this document are based on the ODB++ Tutorial archive provided as part of the Sherlock Tutorial package. Follow the instructions in the ODB++ Project Creation (p. 23) tutorial to load that archive.

In this section, the following topics are covered:

6.1.1. Managing DFMEA Data

6.1.2. Customizing DFMEA Data

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.3. Exporting DFMEA Data
6.1.4. Importing DFMEA Data
6.1.5. DFMEA Results
6.1.6. Net List

6.1.1. Managing DFMEA Data

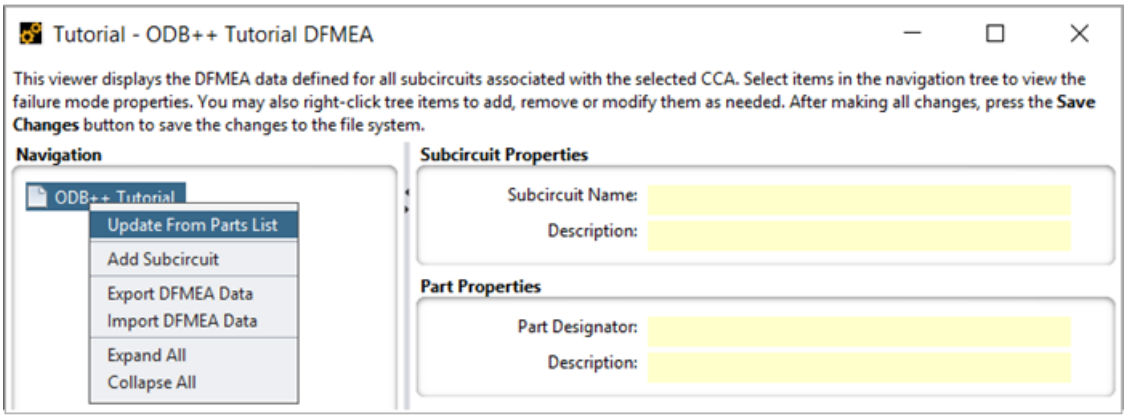
Sherlock maintains DFMEA data as a hierarchical collection of the following types of data:

- One or more Subcircuits per Circuit Card
- One or more Parts per Subcircuit
- One or more Failure Modes per Part

For maximum usability, DFMEA data is displayed and managed as a graphical data tree, allowing users to easily expand, collapse, browse and edit the data.

6.1.1.1. Updating DFMEA Data from Parts List

To view the **DFMEA Data Tree**, simply double-click the **DFMEA** entry located in the **Circuit Card Inputs** folder of the Project Tree. At that point, the **DFMEA Viewer** will appear as a separate window.



For each new circuit card, the **DFMEA Viewer** will initially contain no data, but you can easily add default DFMEA data entries by right-clicking the circuit card name in the **Navigation** panel and selecting the **Update From Parts List** menu option.

At that point, a dialog will appear (see below) allowing you to select various processing options. You may optionally create:

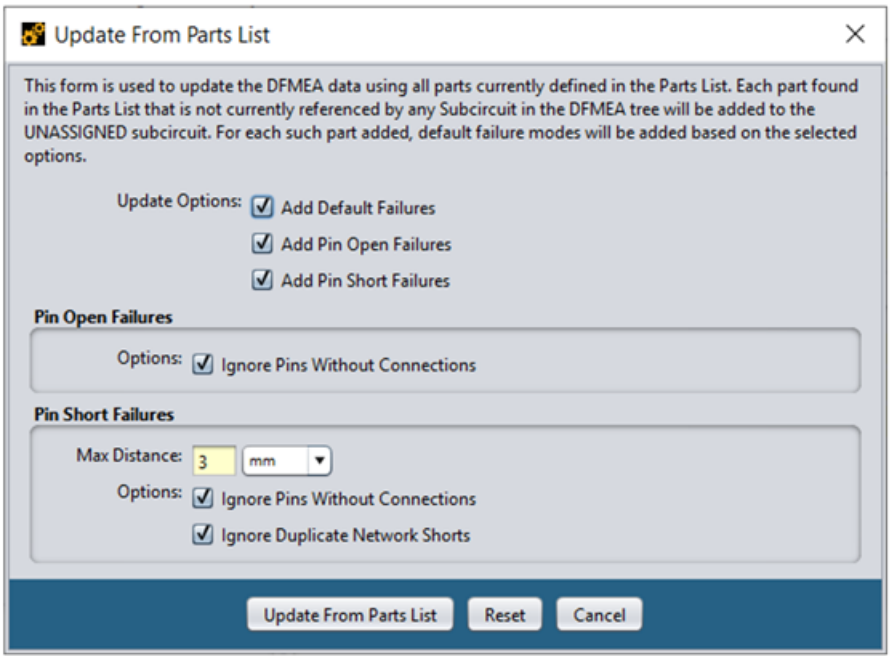
- **Default Failures** based on part types
- Default **Open Failures** for all part pins
- Default **Short Failures** for all pairs of pins

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

We'll discuss these options in more detail later. For now, simply select all the update options and press the **Update From Parts List** button to create default DFMEA data entries for all parts in the Parts List.

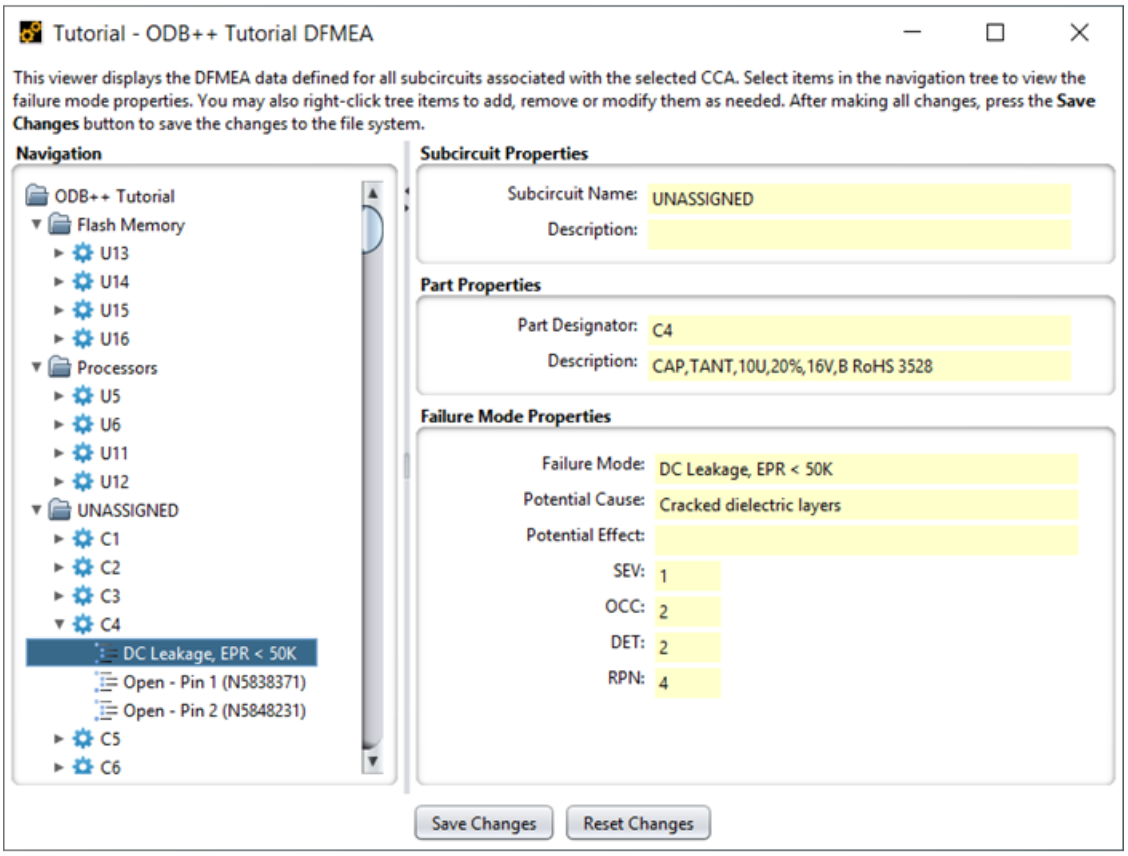
Tip:

The following section explains which part properties Sherlock uses in the DFMEA analysis: Tables: Required Part Properties per Analysis Type (p. 326).



At that point, the **Flash Memory**, **Processors** and **UNASSIGNED** folders will be added to the **Navigation panel.** The Navigation panel is used to quickly find and display the data associated with any Subcircuit, Part or Failure Mode. As each entry is selected in the tree, the data corresponding to that entry, and all higher-level entries, will be displayed in the property panels located on the right-side of the main panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can navigate the tree using either the mouse or the following keyboard keys:

Table 6.1: DFMEA Viewer, Keyboard Navigation

Keyboard Key	Purpose
Up Arrow	Move up one row in the tree
Down Arrow	Move down one row in the tree
Right Arrow	Expand the current entry (if sub-rows exist)
Left Arrow	Collapse the current entry (if sub-rows exist)

You may also double-click any row containing sub-rows to expand/collapse that entry.

By default, Sherlock places all parts found in the **Parts List** into one or more subcircuits designated by the **Subcircuit Name(s)** property value assigned to each part. In this tutorial, subcircuit names were assigned to eight of the parts (U5-U6, U11-U16) when the ODB++ archive was imported because the FUNCTIONAL_BLOCK ODB property was mapped to the **Subcircuit Name(s)** Sherlock property during the import process. Users can also edit the **Parts List** manually to assign parts to one or more subcircuits using a comma-separated list. When the **Update From Parts List** menu option is executed, Sherlock automatically creates the necessary subcircuit folders and assigns all parts to

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the appropriate subcircuit folders based on their current **Subcircuit Name(s)** property values. If a part is not assigned to any subcircuit, then Sherlock places it in the **UNASSIGNED** subcircuit folder.

Note:

If you change the **Subcircuit Name(s)** property for one or more parts, simply re-execute the **Update From Parts List** menu option to move the existing DFMEA entries to the proper subcircuit folders.

You can execute the **Update From Parts List** menu option at any time to copy current part properties from the **Parts List** to the **DFMEA data**. Sherlock automatically copies all part property values in the Parts List to DFMEA data fields that have the same name. In the simplest case, the **Description** property from the Parts List is copied into the **Description** DFMEA field, but any part property available in the Parts List can be copied into a DFMEA field with the same name.

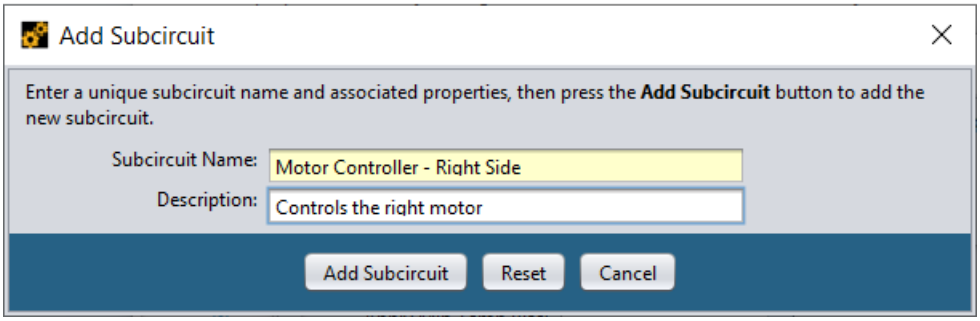
Note:

The Customizing DFMEA Data (p. 373) section below describes how to create custom DFMEA data fields for Subcircuits, Parts and Failure Modes.

6.1.1.2. Manually Organizing DFMEA Data

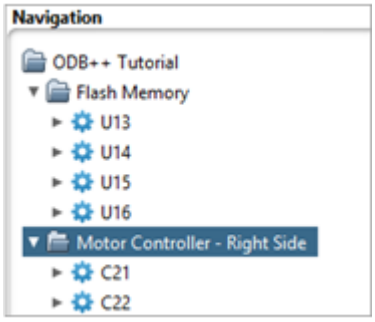
In addition to using the **Subcircuit Name(s)** property to organize parts in the DFMEA tree, you can easily assign one or more parts to a specific subcircuit using the following procedure.

Right-click the top tree entry and select **Add Subcircuit** from the context menu. In the **Add Subcircuit** window, enter the desired Subcircuit name and optional description, then press the **Add Subcircuit** button to create the subcircuit definition and update the **DFMEA Tree**.



To assign one or more parts to the new subcircuit, select the appropriate part designators (C21 & C22 for example) in the **UNASSIGNED** folder, right-click any of the selected parts and select **Cut Part(s)** from the context menu. At that point, the parts will be copied into an internal clipboard and a confirmation dialog will appear asking if you want to remove the parts from the **UNASSIGNED** folder. Press **Yes** to remove the parts. Then, right-click the desired destination subcircuit and select **Paste Part(s)** from the context menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

All standard Windows-style mouse controls are supported for selecting one or more entries in the Navigation tree. Use the SHIFT key to include all entries up to a selected entry and the CONTROL key to toggle individual entries. Once the set of data entries has been selected, right-click any selected entry to remove, cut, or copy the selected entries.

Note:

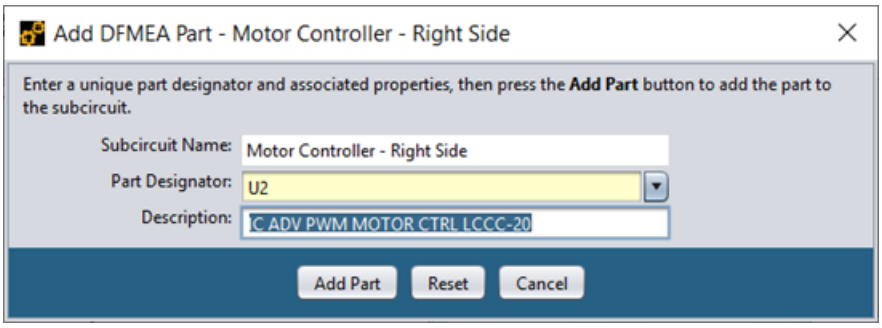
After one or more DFMEA data modifications are made you need to press the **Save Changes** button to save all changes. If you decide not to save the changes, you may press the **Reset Changes** to reset the **DFMEA Viewer** to show the previously saved data.

6.1.1.3. Manually Adding Part Data

In some cases, you may want to add DFMEA data for parts or groups of parts that are not listed in the **Parts List.** Parts can be added simply by right-clicking any existing subcircuit and selecting the **Add Part** menu item in the context menu.

For example, right-click the new **Motor Controller - Right Side** subcircuit in the DFMEA tree and select **Add Part** from the context menu. Enter the desired **Part Designator** (or select it from the pull-down choice list). After you enter or select the Part Designator, Sherlock will automatically fill in the **Description** field using the value found in the current **Parts List**, if any.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



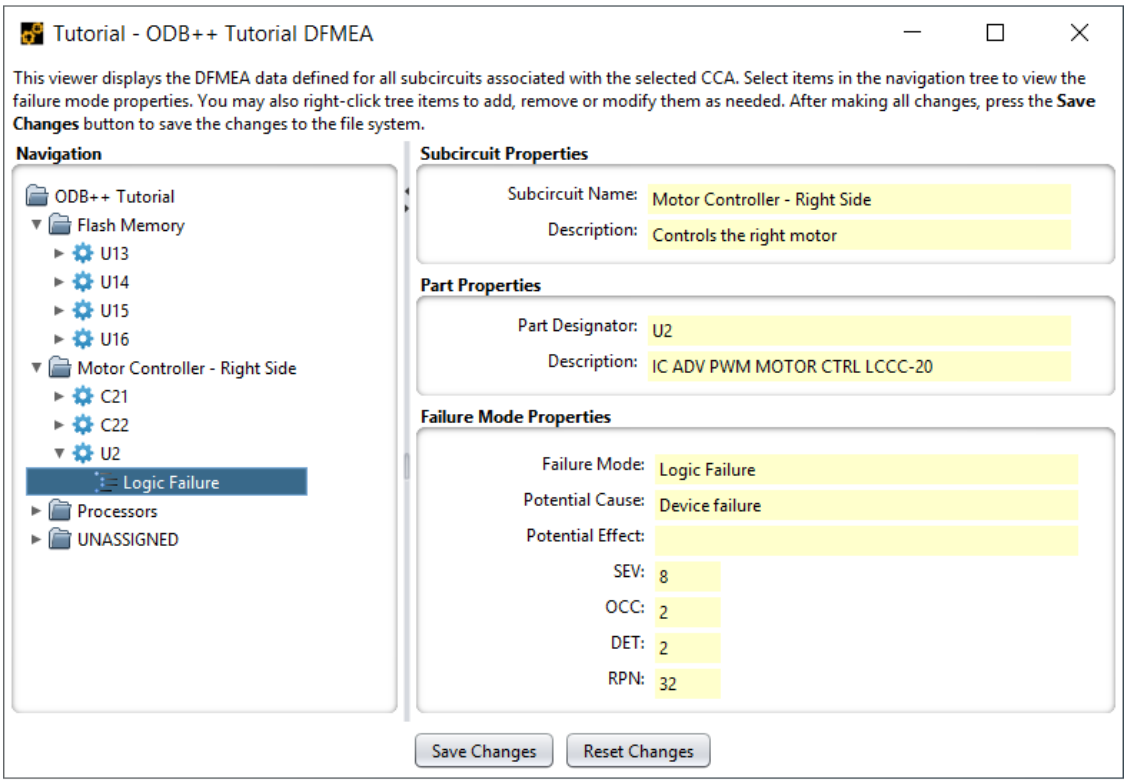
Note:

The same part may be assigned to any number of subcircuit folders to allow you to organize DFMEA data as needed to suit your requirements. In the previous example, we added U2 to the **Motor Controller - Right Side** subcircuit, but there is also a default entry in the **UNASSIGNED** folder.

You are not limited to a single part reference in the **Part Designator** field. Although that is normally the case, you may also enter a list and/or range of part designators, such as R1-R5, R7, R13.

6.1.1.4. Adding Default Part Failures

Now that we've added a part, we can use Sherlock to help create the failure modes associated with each of those parts, similar to how the **Update From Parts List** menu option worked. To that end, right-click the U2 entry just created and select **Add Default Failures** from the context menu. At that point, Sherlock will automatically add one or more **Failure Mode** entries to the U2 part folder, as shown here:



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By examining the new entry, we can see that Sherlock has automatically filled in many of the properties for each Failure Mode added. Such default entries significantly reduce the time needed to manually enter DFMEA data. We'll shortly discuss how to customize the default failure modes to suit your specific project needs.

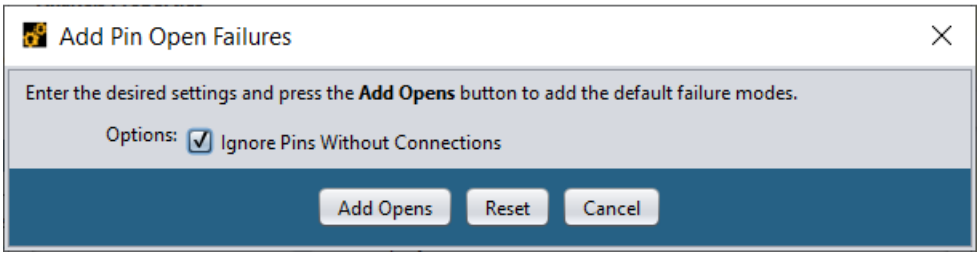
6.1.1.5. Adding Default Pin Open Failures

In addition to default part failures, Sherlock can generate default Pin Open Failures for all pins associated with a given part based on the Net List data imported by Sherlock.

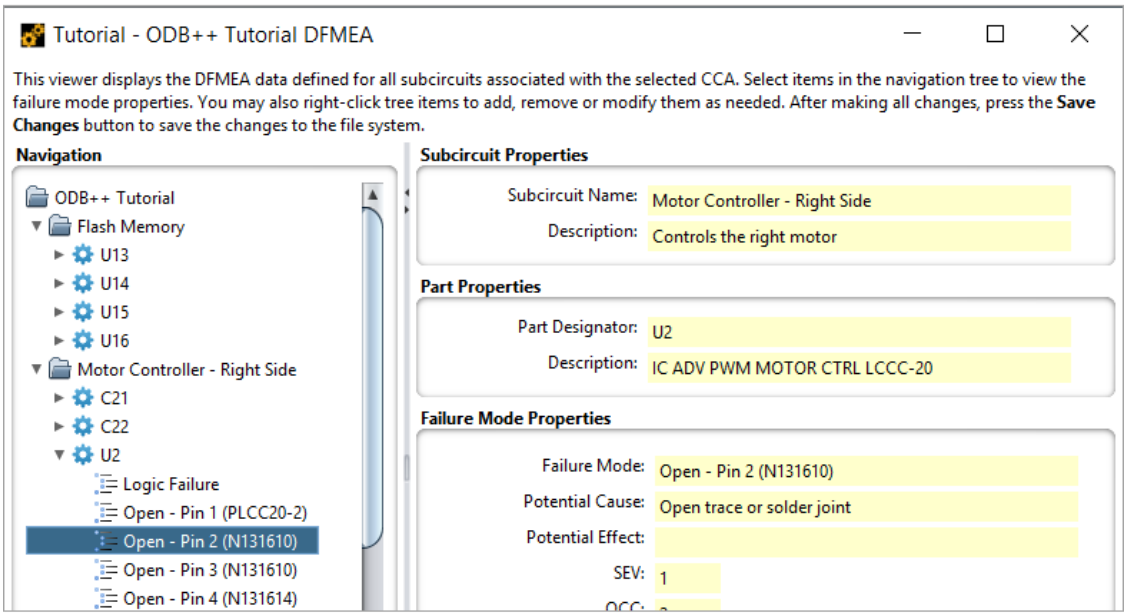
Note:

Net List data is automatically imported by Sherlock when processing ODB++, IPC-2581, and Eagle CAD files with no additional user input required. For other projects, a standard IPC-D-356 file can be used to import the necessary data. See the Net List (p. 395)section for more details.

To generate default Pin Open Failures, right-click the U2 part entry in the **DFMEA Tree** and select the **Add Pin Open Failures** option from the context menu.



Typically, pins that are not connected to any network are ignored, but you may include those pins if necessary. After the **Add Opens** button is pressed, an **Open - Pin XX** failure entry will appear under the selected part entry in the **DFMEA Tree**.

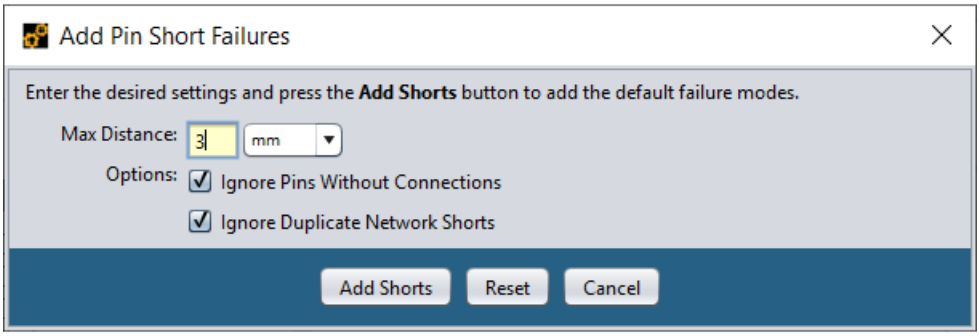


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.1.6. Adding Default Pin Short Failures

Perhaps the most tedious DFMEA data entry chore is that of accounting for shorts between adjacent pins for each part being analyzed. Sherlock can significantly speed-up such a task by automatically determining which pins are adjacent to others, by generating default failure entries for all such pin pairs and by eliminating duplicate short failure modes.

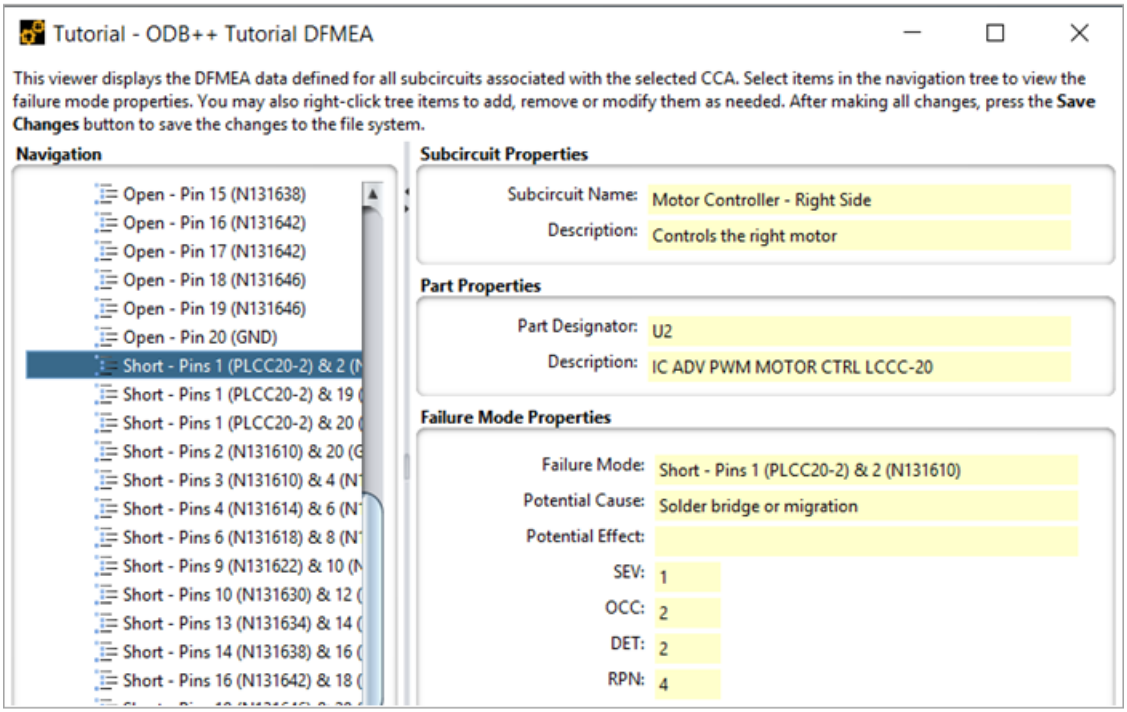
To add default Pin Short Failures, right-click the U2 part entry in the **DFMEA Tree** and select the **Add Pin Short Failures** option from the context menu.



Much like Pin Open Failures, you may choose to ignore pins without connections. You may also choose to ignore duplicate network shorts. Sherlock considers two shorts to be duplicates if a pin shorts to two different pins, but both pins are connected to the same network. The **Max Distance** property can be used to specify the maximum distance used to determine pin adjacency.

After the **Add Shorts** button is pressed, a **Short - Pins XX & YY** entry will appear under the selected part in the **DFMEA Tree** for each pair of adjacent pins found by, as shown below.

By default, displays the pin number and network associated with each pair of adjacent pins.



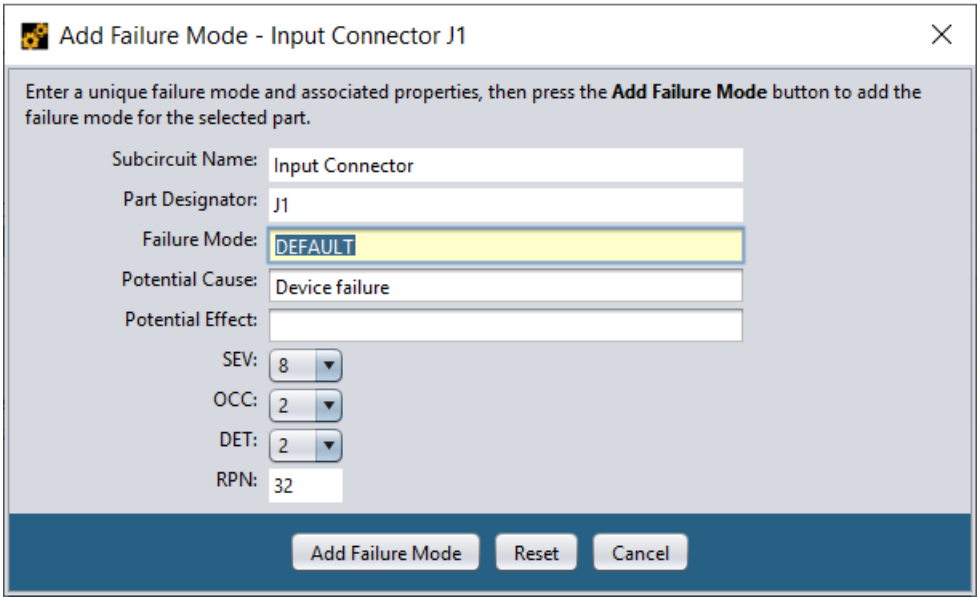
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The Customizing DFMEA Data (p. 373) section below describes how to customize the data values used when generating default **Part Failures**, **Pin Open Failures,** and **Pin Short Failures**.

6.1.1.7. Manually Adding Failure Modes

If the default failure modes describe above don't cover all the possibilities for a given part, you can create a **Failure Mode Item** manually by right-clicking the part and selecting **Add Failure Mode** to display the dialog shown here.



One or more of the fields in the dialog may be initially filled in by Sherlock based on the current default failure mode definition. In either case, modify the properties as necessary and press **Add Failure Mode** to create the failure mode item for the selected part.

For part J1, we have also created a new subcircuit called Input Connector, and used the same process described earlier in the Manually Organizing DFMEA Data (p. 367) section.

6.1.1.8. Editing / Removing DFMEA Data

DFMEA data can be edited at any time simply by right-clicking the desired item in the tree and selecting either **Edit Subcircuit**, **Edit Part** or **Edit Failure Mode** from the context menu. At that point, the dialogs shown above will be displayed, allowing you to modify any or all the properties associated with that entry.

DFMEA data removal is accomplished by right-clicking the desired item in the tree and selecting either **Remove Subcircuit**, **Remove Part** or **Remove Failure Mode** from the context menu. Sub-entries are automatically removed from the tree along with the selected entry.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

After making one or more changes to the **DFMEA Data Tree**, press the **Save Changes** button at the bottom of the DFMEA panel to save the changes to the file system or the **Reset Changes** button to undo all changes made since the last time changes were saved.

6.1.2. Customizing DFMEA Data

DFMEA data is used in different ways by different organizations. Although there are a few standard properties across all organizations, it is often necessary and/or desirable to manage a set of properties that are organization specific. Sherlock supports such flexibility by using two user-customizable definition files to determine the properties to be displayed and edited, as well as the values to be entered automatically for default failure modes.

6.1.2.1. Custom DFMEA Properties

Sherlock uses the **dfmeaProperties.csv** file located in the **DFMEA** sub-folder in the **Sherlock User Directory** (**C:\Users\USERID\AppData\Roaming\Sherlock\DFMEA**) to determine the properties to be displayed and edited for a given DFMEA data entry. That file is created automatically as needed by Sherlock from a standard template containing a standard list of **Subcircuit**, **Part** and **Failure** Mode properties. You may modify that file as needed to define whatever properties are suitable for your organization. The next time that Sherlock is started, it will make use of those properties in all DFMEA displays and dialogs. If you are unsure of the location of your Sherlock User Directory, select **Settings > General Settings** and navigate to the **Advanced** subtab. **The Default Project Directory** shows the location of the Sherlock User Directory.

The **dfmeaProperties.csv** file is not complicated or hard to understand. For example, here is the standard property definition file provided by Sherlock.

Table 6.2: dfmeaProperties.csv

Level	Property
Subcircuit	Description
Part	Description
FM	Potential Cause
FM	Potential Effect

The first row contains the column names used by Sherlock to parse the file and should always contain exactly the column names shown here. All subsequent rows define the property names to be used for each level of data (for example, Subcircuit, Part or Failure). In this example, we've defined **Description** fields for both **Subcircuit** and **Part data** items. We've also defined two other properties (**Potential Cause** and **Potential Effect**) for **Failure Mode** items.

Aside from the properties defined in the property definition file, Sherlock hard-codes the following properties so that they always exist:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 6.3: Hardcoded DFMEA Properties

Level	Property	Description
Subcircuit	name	Unique subcircuit name
Part	refDes	Unique part reference designator
FM	mode	Unique failure mode
FM	SEV	Severity Rating (1-10)
FM	OCC	Occurrence Rating (1-10)
FM	DET	Detection Rating (1-10)

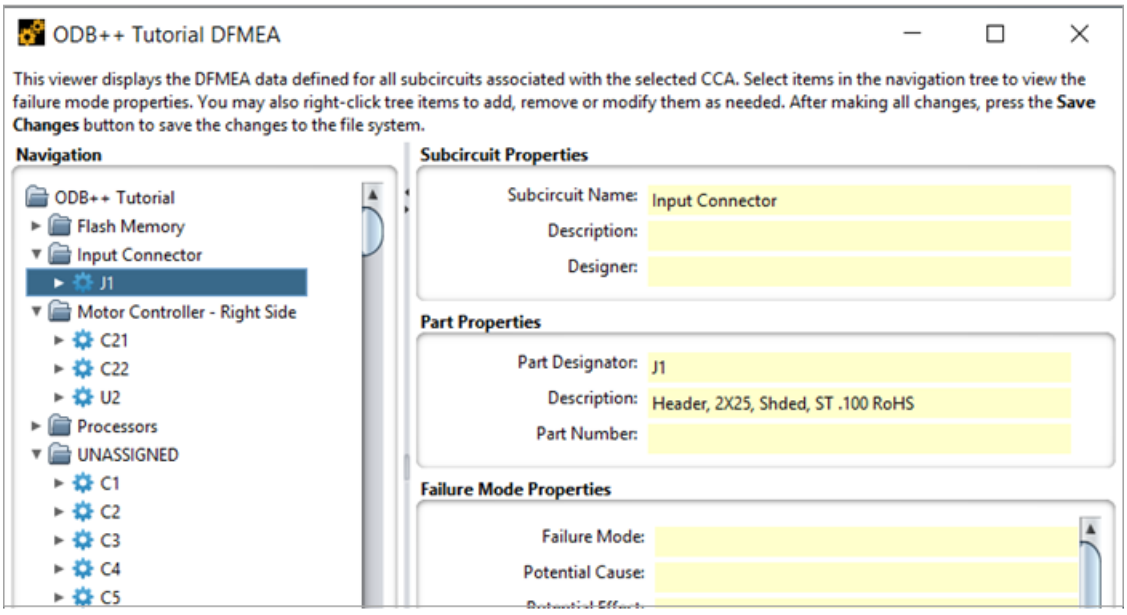
You should NOT create duplicate definitions for these properties in the property definition file.

Let's create some additional properties to see how they can be used. Using your favorite spreadsheet application, edit the **dfmeaProperties.csv** file and add the following rows to the bottom of the spreadsheet:

Table 6.4: Add Rows: dfmeaProperties.csv

Subcircuit	Designer
Part	Part Number

Save the file changes and the restart Sherlock. By examining the **DFMEA Tree**, you'll see that the **Designer** property has been added to the **Subcircuit Properties** panel and that the **Part Number** property has been added to the **Part Properties** panel.

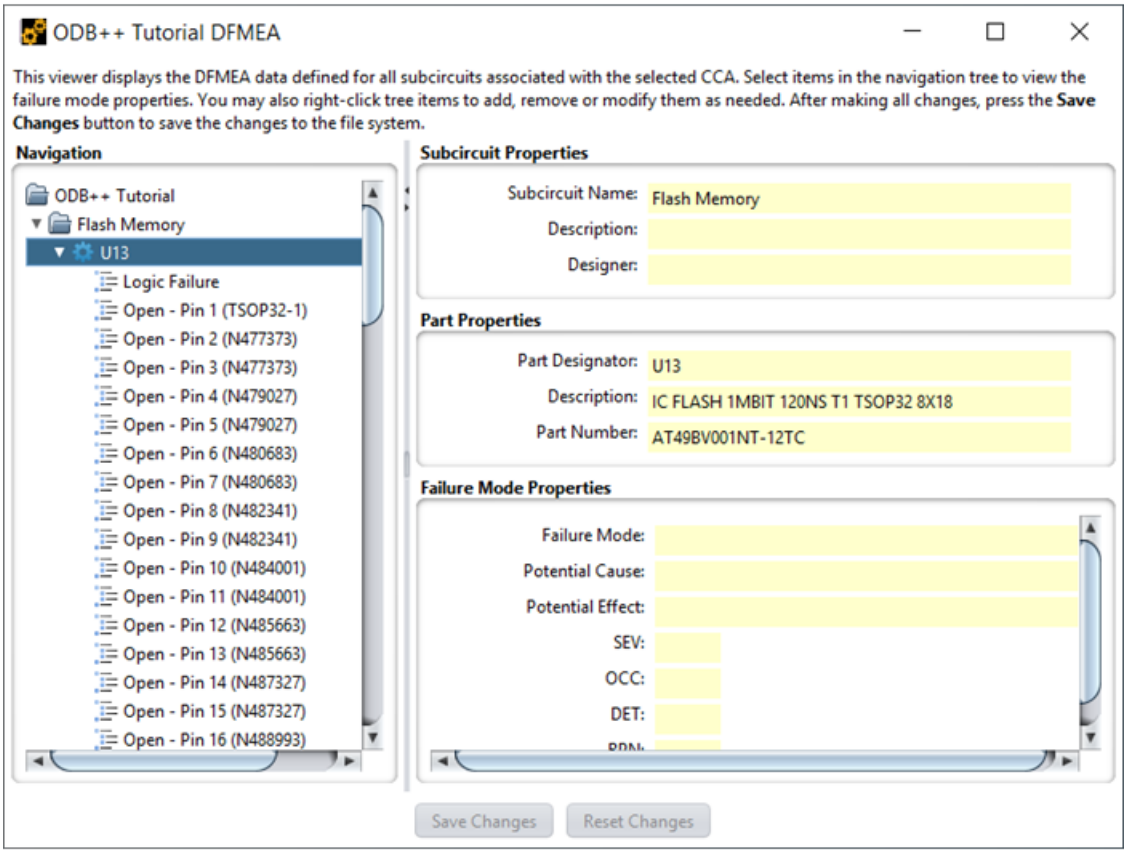


In addition to being added to the read-only panels, those fields have been added to the **Subcircuit** and **Part Edit** dialogs so you can manually assign values to all of the new properties whenever you want.

It is important to note that the **Part Number** DFMEA property we just added is named the same as the **Part Number** property in the **Parts List.** As such, whenever the **Update From Parts List** menu option is executed, Sherlock will automatically copy the **Part Number** value from the Parts

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List into the **Part Number** field of each corresponding DFMEA part entry. To see how this works, simply execute **Update From Parts List** and select U13 in the **DFMEA Tree** (under the Flash Memory subcircuit) to see that the part number has been copied from the **Parts List.**



6.1.2.2. Custom Default Failure Modes

Much like DFMEA properties, Sherlock determines the appropriate default failure modes for a given part by examining the contents of the **dfmeaFailures.csv** file located in the DFMEA sub-folder in the Sherlock User Directory. That file is created automatically by Sherlock from a standard template, but you are free to modify that file at any time to customize the default failure modes to be used for your projects.

The **dfmeaFailure.csv** file is formatted as a standard Comma Separated Value (CSV) file and can be viewed/edited using any spreadsheet application. Below is an excerpt from the default file provided by Sherlock (formatted for readability).

The first row contains the column names used by Sherlock to parse the file. The first two columns are always required to define a default failure mode. If any of the standard numeric columns (SEV, OCC, DET) are not defined, then a default value will be used. Finally, zero or more additional columns may be defined to specify default values for any of the customized **Failure Mode** properties defined in the **Property Definition File** (see previous section (p. 373)). In this case, we've defined only the **Potential Cause** for each Failure Mode, which is one of the two default properties defined by Sherlock.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 6.5: dfmeaFailure.csv

partType	failureMode	SEV	OCC	DET	Potential Cause
DEFAULT	DEFAULT	8	2	2	Device failure
DEFAULT OPEN	Open – Pin <pinName> (<netName>)	1	2	2	Open trace or solder joint
DEFAULT SHORT	Short – Pins <pinName1> (<netName>)	1	2	2	Solder bridge or migration
Capacitor	DC Leakage, EPR < 50K	1	2	2	Cracked dielectric layers
Diode	DC Leakage	1	2	2	Damage to PN Junction
Resistor	High Resistance	1	2	2	Crack propagation through part
Resistor	Shorted	1	2	2	Device failure
Resistor Network	DC Leakage	1	2	2	Device failure
IC	Logic Failure	8	2	2	Device failure

Sherlock uses the **partType** listed in each row to determine which default failure modes to use for a given part. Specifically, if the part type matches the **partType** value (case-insensitive) in the spreadsheet, then that row will be used to create a default failure mode. In the example above, one default failure mode is defined for Capacitor parts and two (2) other default failure modes are defined for Resistor parts.

Part types **DEFAULT**, **DEFAULT OPEN** and **DEFAULT SHORT** are used to generate default Part Failures, default Pin Open Failures and default Pin Short Failures, respectively, as we now discuss.

6.1.2.3. Default Part Failures

The **DEFAULT** row in the **dfmeaFailures.csv** file defines the property values to be used by Sherlock when generating default part failures as a result of the user selecting the **Add Failure Mode** menu option. In such cases, the **mode** property will be specified by the user before the failure mode is added.

6.1.2.4. Default Pin Open Failures

The **DEFAULT OPEN** row in the **dfmeaFailures.csv** file defines the property values to be used by Sherlock when generating default **Pin Open Failures** as a result of the user selecting either the **Update From Parts List** or **Add Pin Open Failures** menu options. Since multiple pin open failures can be generated for a given part, the **failureMode** value specified in the CSV file is used by Sherlock as a template to generate the specific failure mode label. The template value may contain any of the following variables, which will be replaced by the indicated value:

Table 6.6: Template Variables (Default Opens)

Template Variable	Replacement Value
<partName>	ID associated with the selected part

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

<pinName>	ID associated with the current pin
<netName>	ID associated with the connected network

For example, the default template:

Open - Pin <pinName> (<netName>)

will generate the following mode:

Open - Pin 20 (GND)

for pin 20 connected to network GND.

Note:

Net Lists may use arbitrary strings for pin names, so don't be surprised if you don't see pin numbers in some cases. Similarly, networks can be labeled using arbitrary, sometimes meaningless, strings, depending on how the circuit engineer manages them.

6.1.2.5. Default Pin Short Failures

The **DEFAULT SHORT** row in the **dfmeaFailures.csv** file defines the property values to be used by Sherlock when generating default **Pin Short Failures** as a result of the user selecting either the **Update From Parts List** or **Add Pin Short Failures** menu options. Since multiple pin short failures can be generated for a given part, the **failureMode** value specified in the CSV file is used by Sherlock as a template to generate the specific failure mode label. The template value may contain any of the following variables, which will be replaced by the indicated value:

Table 6.7: Template Variables (Default Shorts)

Template Variable	Replacement Value
<partName>	ID associated with the selected part
<pinName1>	ID associated with the first pin
<netName1>	ID associated with the first connection
<pinName2>	ID associated with the second pin
<netName2>	ID associated with the second connection

For example, the default template:

Short - Pins <pinName1> (<netName1>) & <pinName2> (<netName2>)

will generate the following mode

Short - Pins 19 (N131646) & 20 (GND)

for adjacent pins 19 and 20 connected to networks N131646 and GND, respectively.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.3. Exporting DFMEA Data

Sherlock provides a flexible, user-customizable mechanism for exporting all or part of the DFMEA data to one or more spreadsheet files using a collection of spreadsheet templates. **Template files** are formatted as standard XLS spreadsheet files, allowing you use the full range of spreadsheet formatting capabilities to specify the output file layout. In addition, Sherlock defines a set of **template variables** that can be referenced in a template file to include individual DFMEA data values, as well as lists of DFMEA data entries. Using such capabilities, you can easily create an output spreadsheet containing only the data associated with a selected failure more for a selected part, a summary of DFMEA data, or the entire **DFMEA Data Tree**, to name a few examples.

6.1.3.1. Template Files

Sherlock provides the following collection of standard template files as part of the installation package:

- **00-Backup-Template.xls**: Generates a no-frills spreadsheet containing the entire **DFMEA Data Tree** for a selected CCA, including the data properties defined by Sherlock by default.
- **01-CCA-Summary-Template.xls**: Generates a summary spreadsheet containing a list of all sub-circuits defined for a selected CCA that shows the number of components contained in each subcircuit and the range of SEV, OCC, DET and RPN values for each subcircuit.
- **02-CCA-Details-Template.xls**: Generates a detailed spreadsheet containing the entire **DFMEA Data Tree**, with full details for all subcircuits, components and failure modes for a selected CCA. This template also provides summary information (for example, min/max numeric values) for all subcircuits and components.
- **03-Subcircuit-Summary-Template.xls**: Generates a summary spreadsheet containing the min/max numeric values of all subcircuit and component data for a selected CCA.
- **04-Subcircuit-Details-Template.xls**: Generates a detailed spreadsheet containing all component and failure mode data for a selected subcircuit.
- **05-Part-Template.xls**: Generates a detailed spreadsheet containing all failure mode data for a selected subcircuit component.
- **06-Failure-Mode-Template.xls:** Generates a detailed spreadsheet containing the failure mode data for a selected subcircuit component failure mode.

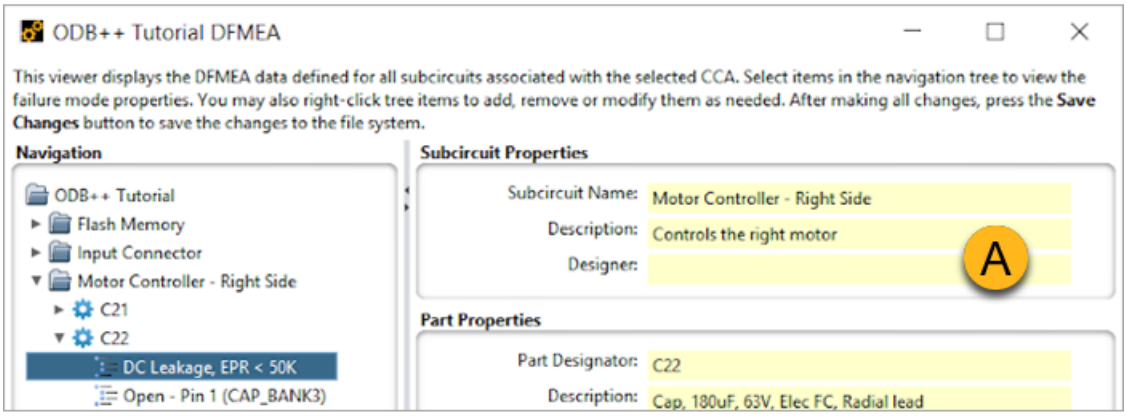
All the template files listed above are created automatically by Sherlock in the **DFMEA** folder located in the Sherlock User Directory. You are free to use the templates as-is or modify them to suit your specific needs, as we now discuss.

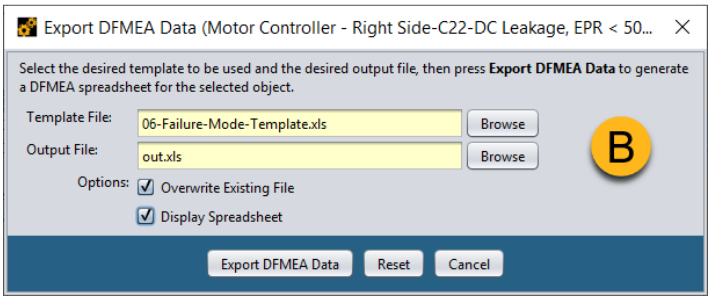
6.1.3.2. Exporting DFMEA Data

Exporting DFMEA data involves simply selecting the desired DFMEA data item to be exported and an appropriate template file. To see how this works, select the **DC Leakage, EPR < 50K** failure mode for the C22 component in the **Input Connector** subcircuit as shown below (A).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Then, right-click that entry and select **Export DFMEA Data** from the context menu. At that point, a dialog will appear (B) allowing you to select an appropriate template file, specify an output file and choose various processing options.





Use the first **Browse** button to select the failure mode template provided by Sherlock from the **DFMEA** folder located in the Sherlock User Directory.

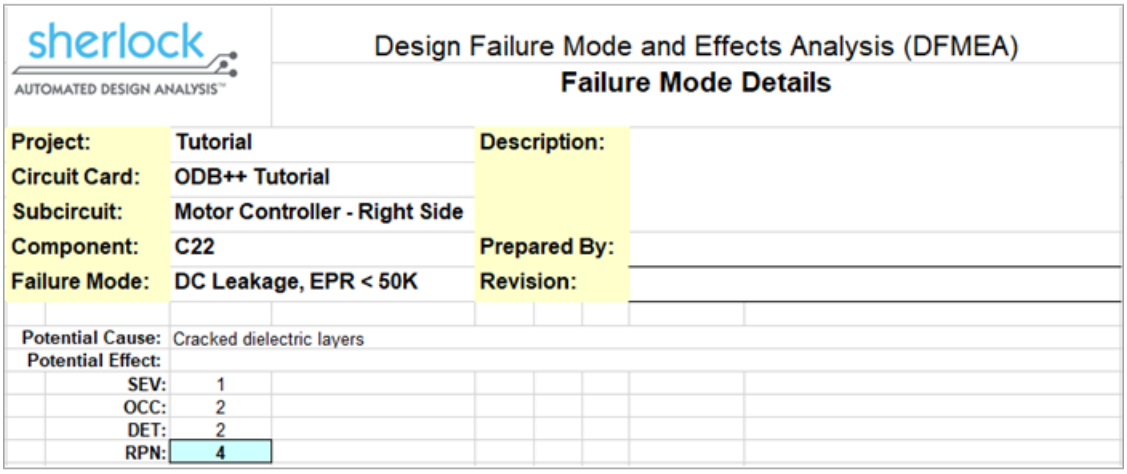
Then use the second **Browse** button to specify the name and location of an output file to be generated.

If the **Overwrite Existing File** option is selected, then any existing output file with the same name will be overwritten without prompting you.

If the **Display Spreadsheet** option is selected, then the application currently associated with XLS files on your system (for example, Excel, Open Office) will be used to display the output file after it has been generated.

Press the **Export DFMEA Data** button to generate the spreadsheet using the selected template file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



sherlock
AUTOMATED DESIGN ANALYSIS™

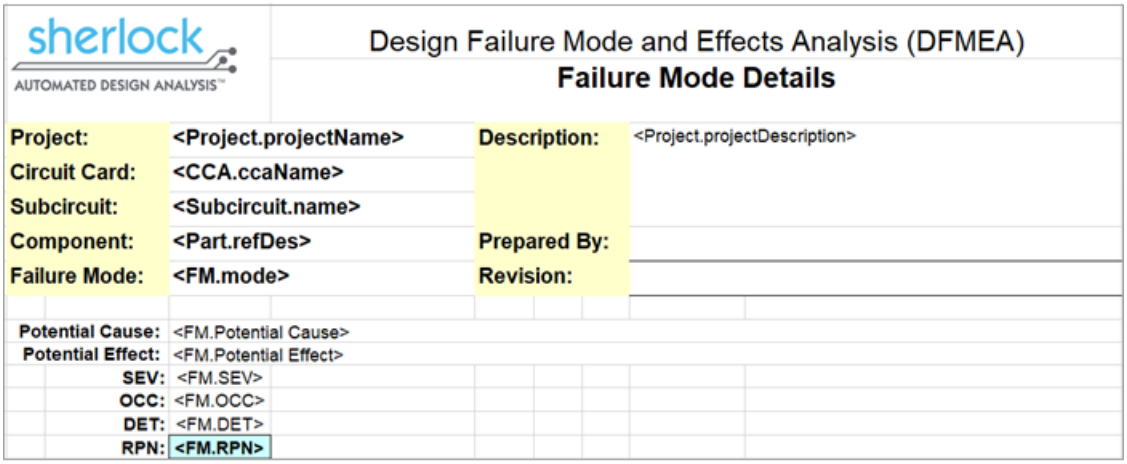
Design Failure Mode and Effects Analysis (DFMEA)
Failure Mode Details

Project:	**Tutorial**	**Description:**	
Circuit Card:	**ODB++ Tutorial**		
Subcircuit:	**Motor Controller - Right Side**		
Component:	**C22**	**Prepared By:**	
Failure Mode:	**DC Leakage, EPR < 50K**	**Revision:**	
Potential Cause:	Cracked dielectric layers		
Potential Effect:			
SEV:	1		
OCC:	2		
DET:	2		
RPN:	4		

This simple spreadsheet includes a header section showing various Project, CCA, Subcircuit and Component properties, as well as the selected Failure Mode. Below the header, the spreadsheet contains a single row for each failure mode property.

6.1.3.3. Template Variables

If we look at the **06-Failure-Mode-Template.xls** that was used in the previous example:



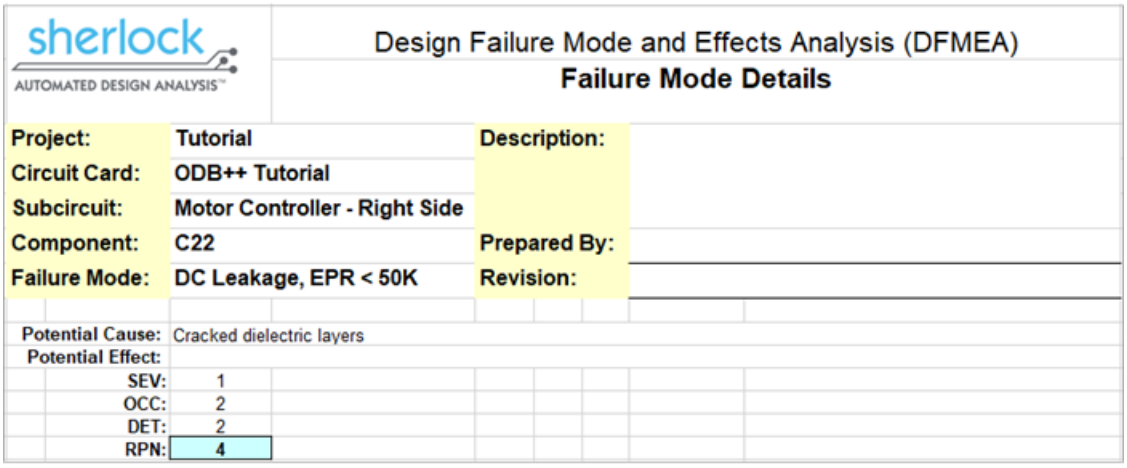
sherlock
AUTOMATED DESIGN ANALYSIS™

Design Failure Mode and Effects Analysis (DFMEA)
Failure Mode Details

Project:	**<Project.projectName>**	**Description:**	<Project.projectDescription>
Circuit Card:	**<CCA.ccaName>**		
Subcircuit:	**<Subcircuit.name>**		
Component:	**<Part.refDes>**	**Prepared By:**	
Failure Mode:	**<FM.mode>**	**Revision:**	
Potential Cause:	<FM.Potential Cause>		
Potential Effect:	<FM.Potential Effect>		
SEV:	<FM.SEV>		
OCC:	<FM.OCC>		
DET:	<FM.DET>		
RPN:	**<FM.RPN>**		

And compare it to the output file:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



We see immediately that the output file mimics the look-and-feel of the template file, including all the same formatting and style settings. The only differences between the template and output files are those cells containing template variables. Specifically, all template variables found in the template file are replaced by their corresponding property values in the output file. For example, the **<Project.projectName>** template variable is replaced by "Tutorial" in the output file. This simple mechanism allows you to customize templates as needed to satisfy a range of needs.

Sherlock supports the following internally defined template variables:

Table 6.8: Internally Defined Template Variables

Variable Name	Description
Project.projectName	Name of selected project
Project.projectDescription	Description defined in the Project Properties
CCA.ccaName	Circuit card name
CCA.ccaDescription	Description defined in the Circuit Card Properties
Subcircuit.name	Subcircuit Name
Subcircuit.componentCount	Number of components contained by the subcircuit
Subcircuit.minSEV	Minimum SEV value for all subcircuit components
Subcircuit.maxSEV	Maximum SEV value for all subcircuit components
Subcircuit.minOCC	Minimum OCC value for all subcircuit components
Subcircuit.maxOCC	Maximum OCC value for all subcircuit components
Subcircuit.minDET	Minimum DET value for all subcircuit components
Subcircuit.maxDET	Maximum DET value for all subcircuit components
Subcircuit.minRPN	Minimum RPN value for all subcircuit components
Subcircuit.maxRPN	Maximum RPN value for all subcircuit components
Part.refDes	Component reference designator
Part.minSEV	Minimum SEV value for all failure modes
Part.maxSEV	Maximum SEV value for all failure modes
Part.minOCC	Minimum OCC value for all failure modes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part.maxOCC	Maximum OCC value for all failure modes
Part.minDET	Minimum DET value for all failure modes
Part.maxDET	Maximum DET value for all failure modes
Part.minRPN	Minimum RPN value for all failure modes
Part.maxRPN	Maximum RPN value for all failure modes
FM.mode	Failure Mode
FM.SEV	SEV value
FM.OCC	OCC value
FM.DET	DET value
FM.RPN	RPN value

In addition to these internally defined variables, Sherlock will also recognize variables based on each of the custom Subcircuit, Part or Failure Modes properties defined in the **dfmeaProperties.csv** definition file (as discussed in previous sections).

For example, since Sherlock defines the DFMEA data properties in the default **dfmeaProperties.csv** file (see columns A and B in table below), the template variables (column C) will also be supported:

Table 6.9: DFMEA Data Properties

A Level	B Property	C Template Variables
Subcircuit	Description	<Subcircuit.Description>
Part	Description	<Part.Description>
FM	Potential Cause	<FM. Potential Cause>
FM	Potential Effect	<FM. Potential Effect>

Note:

Custom property names are always case-sensitive and all blanks must be specified in the template variable referenced exactly as they are specified in the property definition file.

Template variables may be referenced by themselves or as part of an arbitrary string containing one or more template variables. For example, the following string expression in the template file:

Cause (**<FM. Potential Cause>**) & Effect (**<FM. Potential Effect>**)

would produce the following cell value in the output file:

Cause (**Cracked dielectric layers**) & Effect ()

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

using the same inputs as the previous example (substitutions are shown in **bold**).

Note:

If Sherlock cannot determine a value for a given template variable, then the template variable itself will be used as the cell value in the output spreadsheet.

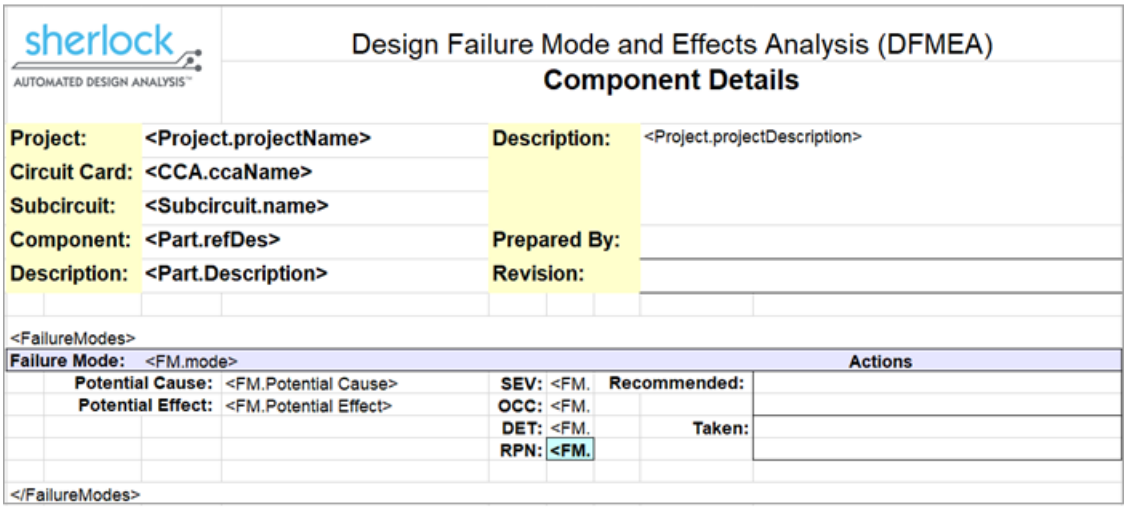
6.1.3.4. Templates Lists

In addition to template variables, templates may reference **template lists** to include a list of data entries in the output file. Specifically, the following template lists are supported by Sherlock:

Table 6.10: Template Lists

Template List	Alias	Description
<Subcircuits>	None	Lists all subcircuits associated with the current circuit card
<Components>	<Parts>	Lists all parts associated with the current subcircuit
<FailureModes>	<Failures>	Lists all failure modes associated with the current part

Such list references can be used individually or in a nested manner to generate a wide variety of output spreadsheets. For example, let's examine how the **05-Part-Template.xls** template file uses both template variables and template lists to show all failure modes associated with a selected part.



As in the previous example, the header section contains references to individual template variables to display various project, circuit card, subcircuit and component property values. Then, the <FailureModes> list is referenced, indicating that the output file should contain data for each failure mode associated with the current part. All of the lines defined between the opening <FailureModes>

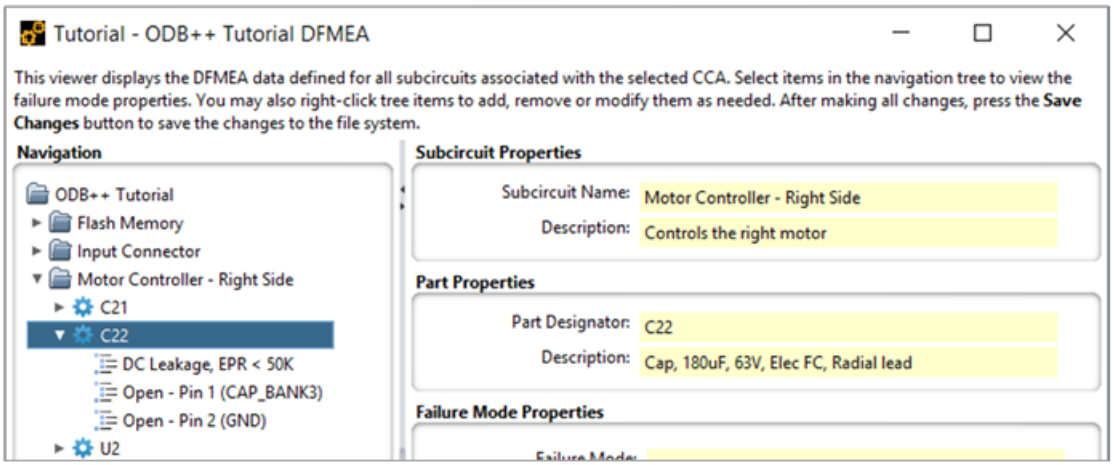
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tag and the closing </FailureModes> tag are part of the template used to display each Failure Mode.

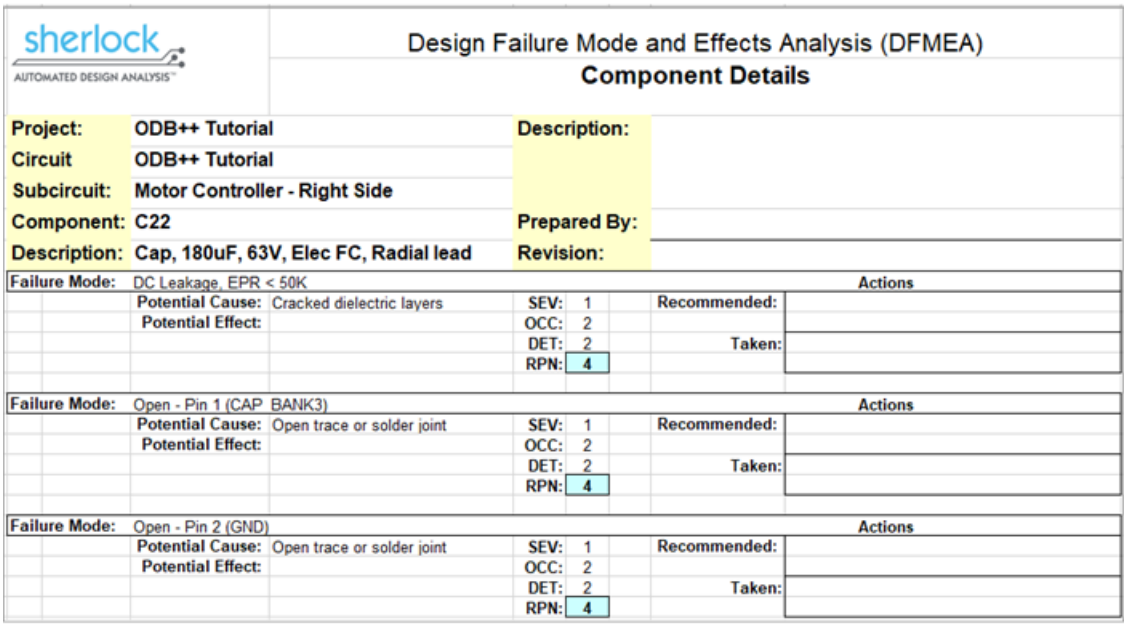
Note:

Template list opening and closing tags must always be paired properly in the template file. If Sherlock cannot find the closing tag for a given opening tag it will generate an error message.

To see how Template Lists are used, select the U2 part (or any failure mode associated with the U2 part) in the **Motor Controller - Right Side** subcircuit as shown here.



Then, right-click that entry and select **Export DFMEA Data** from the context menu. When prompted for the template file, select the **05-Part-Template.xls** file and press **Export DFMEA Data** to generate the following output file:



sherlock
AUTOMATED DESIGN ANALYSIS™

Design Failure Mode and Effects Analysis (DFMEA)
Component Details

Project:	**ODB++ Tutorial**	**Description:**	
Circuit	**ODB++ Tutorial**		
Subcircuit:	**Motor Controller - Right Side**		
Component:	**C22**	**Prepared By:**	
Description:	**Cap, 180uF, 63V, Elec FC, Radial lead**	**Revision:**	

Failure Mode:	DC Leakage, EPR < 50K					**Actions**
	Potential Cause:	Cracked dielectric layers	**SEV:**	1	**Recommended:**	
	Potential Effect:		**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		
Failure Mode:	Open - Pin 1 (CAP_BANK3)					**Actions**
	Potential Cause:	Open trace or solder joint	**SEV:**	1	**Recommended:**	
	Potential Effect:		**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		
Failure Mode:	Open - Pin 2 (GND)					**Actions**
	Potential Cause:	Open trace or solder joint	**SEV:**	1	**Recommended:**	
	Potential Effect:		**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		

As in the previous example, the header section is reproduced exactly as shown in the template file, with property values substituted for each template variable. The remainder of the file was generated

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

by repeatedly using the <FailureModes> template list defined in the template file. For each failure mode associated with the selected part, the template list is used to display the failure mode properties.

6.1.3.5. Nested Templates Lists

Template lists can be nested in a natural order to export any part of the DFMEA data tree. Specifically, the <Subcircuits> template list may include references to the <Components> template list and the <Components> template list may include references to the <FailureModes> template list. For example, the 04-Subcircuit-Details-Template.xls template file shown here...

sherlock AUTOMATED DESIGN ANALYSIS™

Design Failure Mode and Effects Analysis (DFMEA)

Subcircuit Details

Project:	**<Project.projectName>**	**Description:**	<Project.projectDescription>
Circuit Card:	**<CCA.ccaName>**		
Subcircuit:	**<Subcircuit.name>**		
Prepared By:		**Revision:**	

<Components>

Component: <Part.refDes>

Failure Mode	Potential Cause	SEV	Potential Effect	OCC	Controls	DET	RP	Actions Taken
<FailureModes>								
<FM.mode>	<FM.Potential Cause>	<FM.SE	<FM.Potential Effect>	<FM.OCC		<FM.DE	<FM.R	
</FailureModes>								
</Components>								

...uses the <FailureModes> template list nested inside of the <Components> template list to export each failure mode for each component associated with the selected sub-circuit, resulting in the following output:

Component: C21

Failure Mode	Potential Cause	SEV	Potential Effect	OCC	Controls	DET	RP	Actions Taken
DC Leakage, EPR < 50K	Cracked dielectric layers	1		2		2	4	
Open - Pin 1 (CAP_BANK3)	Open trace or solder joint	1		2		2	4	
Open - Pin 2 (GND)	Open trace or solder joint	1		2		2	4	

Component: C22

Failure Mode	Potential Cause	SEV	Potential Effect	OCC	Controls	DET	RP	Actions Taken
DC Leakage, EPR < 50K	Cracked dielectric layers	1		2		2	4	
Open - Pin 1 (CAP_BANK3)	Open trace or solder joint	1		2		2	4	
Open - Pin 2 (GND)	Open trace or solder joint	1		2		2	4	

It's important to note that you may define as many rows inside of each template list as needed to achieve the desired output. In this case, a double-line and some high-lighted cells are output at the start of each Component to improve readability. Then, a basic header and border format is used to list each of the failure modes for a given component.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.3.6. Page Headers

When generating spreadsheets that span multiple printed pages, the <PageHeader> and </PageHeader> tags can be used to designate one or more rows to be repeated at the top of each printed page. As with other template lists, individual template variables may be referenced, but no other template lists may be nested inside of the PageHeader list. Such repeating rows are typically used to display project and circuit card properties, and perhaps repeating column headers. For example, the **00-Backup-Template.xls** template defines a basic page header to display the project and circuit card name on every printed page, as shown here:

<PageHeader>			
Project:	**<Project.projectName>**		
CCA:	**<CCA.ccaName>**		
</PageHeader>			
<Subcircuits>			
Subcircuit:	**<Subcircuit.name>**		
Description:	**<Subcircuit.Description>**		
<Components>			
Component:	Ref Des:	<Part.refDes>	
	Description:	<Part.Description>	
<FailureModes>			
	Failure Mode:	Mode:	<FM.mode>
		Cause:	<FM.Potential Cause>
		Effect:	<FM.Potential Effect>
		SEV:	<FM.SEV>
		OCC:	<FM.OCC>
		DET:	<FM.DET>
</FailureModes>			
</Components>			
</Subcircuits>			

In fact, that template shows how each of the template features can be used to export all DFMEA data to a spreadsheet file that can be easily parsed by another application. Sherlock users need only add template variables whenever they define a new DFMEA property to maintain a comprehensive template.

6.1.4. Importing DFMEA Data

The same template used to export DFMEA data from Sherlock can be used to import DFMEA data from a spreadsheet into Sherlock. This allows Sherlock users to both import DFMEA data initially from existing spreadsheets and to export data from Sherlock, modify it using a spreadsheet application and then use that spreadsheet to update the Sherlock **DFMEA Data Tree.**

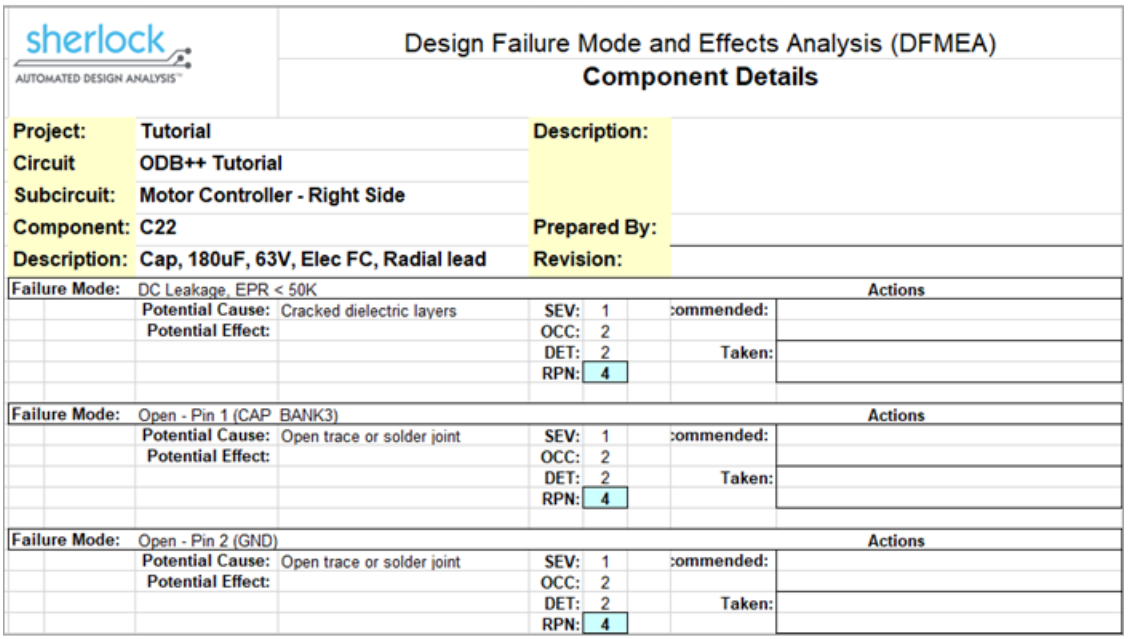
6.1.4.1. Re-Importing a DFMEA Spreadsheet

Any spreadsheet that was generated by Sherlock using a given template file can be easily imported by Sherlock using the same template file. Basically, the same template file defines not only how to generate the spreadsheet, but also how it can be parsed to import data. As long as the spreadsheet format remains consistent with the template, you may add, modify or delete DFMEA data in the spreadsheet and successfully update Sherlock using that spreadsheet. This allows users to seamlessly

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

switch back-and-forth between Sherlock and their spreadsheet application for viewing and maintaining DFMEA data.

To see how the basic export-modify-import cycle works, select the C22 part entry in the DFMEA data tree (assigned to the **Motor Controller - Right Side** sub-circuit) and generate a spreadsheet using the **05-Part-Template.xls** template file:



sherlock
AUTOMATED DESIGN ANALYSIS

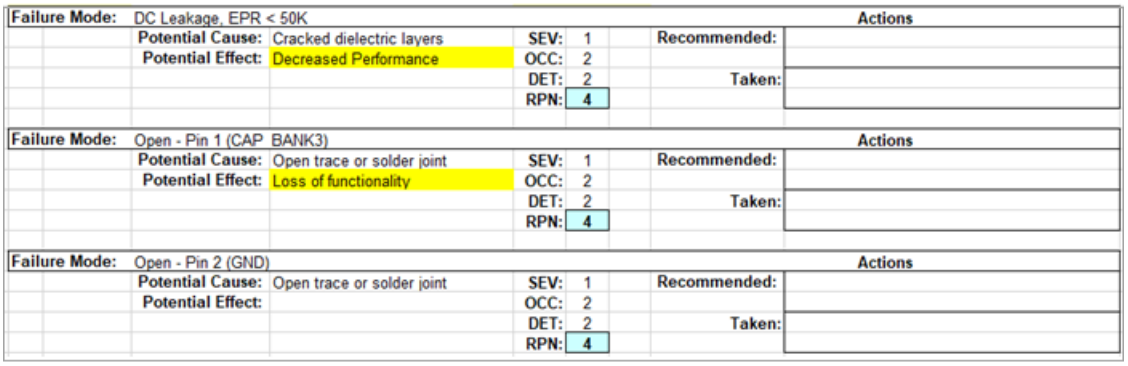
Design Failure Mode and Effects Analysis (DFMEA)
Component Details

Project:	**Tutorial**	**Description:**	
Circuit	**ODB++ Tutorial**		
Subcircuit:	**Motor Controller - Right Side**		
Component:	**C22**	**Prepared By:**	
Description:	**Cap, 180uF, 63V, Elec FC, Radial lead**	**Revision:**	

Failure Mode:	DC Leakage, EPR < 50K					**Actions**
	Potential Cause:	Cracked dielectric layers	**SEV:**	1	ommended:	
	Potential Effect:		**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		
Failure Mode:	Open - Pin 1 (CAP_BANK3)					**Actions**
	Potential Cause:	Open trace or solder joint	**SEV:**	1	ommended:	
	Potential Effect:		**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		
Failure Mode:	Open - Pin 2 (GND)					**Actions**
	Potential Cause:	Open trace or solder joint	**SEV:**	1	ommended:	
	Potential Effect:		**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		

In this example, we see that there are no **Potential Effect** values indicated for any of the failure modes. We could use Sherlock to modify those values individually or we can update them in the spreadsheet file just generated and then update Sherlock using that spreadsheet, as we now demonstrate.

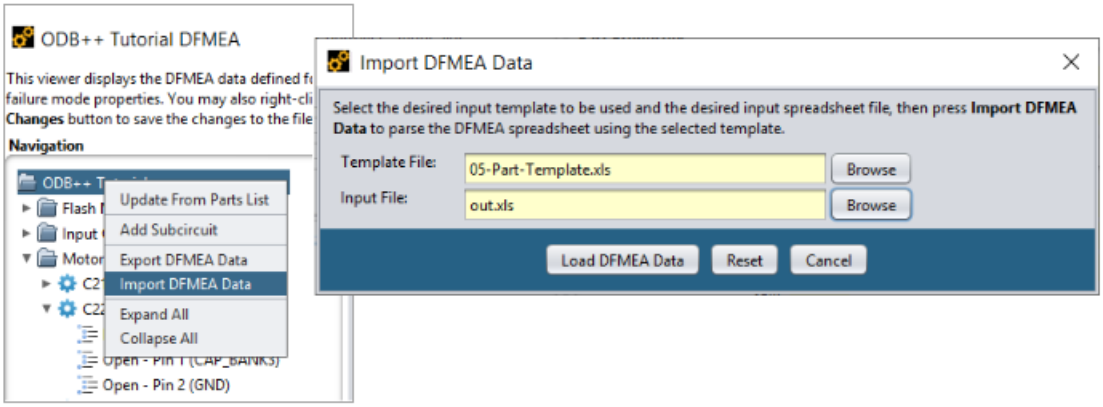
Modify the spreadsheet to define values for the first two failure modes, as highlighted here in yellow:



Failure Mode:	DC Leakage, EPR < 50K					**Actions**
	Potential Cause:	Cracked dielectric layers	**SEV:**	1	**Recommended:**	
	Potential Effect:	Decreased Performance	**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		
Failure Mode:	Open - Pin 1 (CAP_BANK3)					**Actions**
	Potential Cause:	Open trace or solder joint	**SEV:**	1	**Recommended:**	
	Potential Effect:	Loss of functionality	**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		
Failure Mode:	Open - Pin 2 (GND)					**Actions**
	Potential Cause:	Open trace or solder joint	**SEV:**	1	**Recommended:**	
	Potential Effect:		**OCC:**	2		
			DET:	2	**Taken:**	
			RPN:	4		

After saving the spreadsheet changes, right-click the top-level **ODB++ Tutorial** entry in the **DFMEA Data Tree** and select **Import DFMEA Data** from the context menu to display the **Import DFMEA Data** dialog.

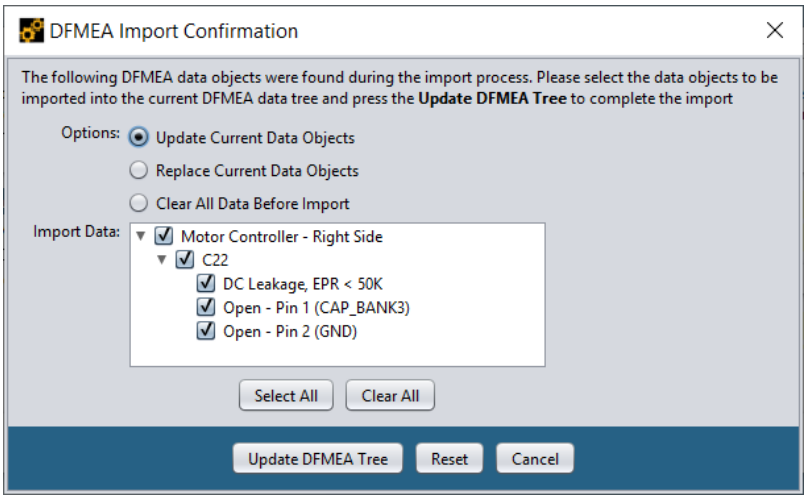
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



As a convenience, Sherlock automatically displays the last template file used, as well as the name of the last output file to be used as the input file. If different files are desired, simply press the **Browse** buttons to locate the proper files.

In this case, we are re-importing the file that we just generated, so the default values are correct. Press the **Load DFMEA Data** button to parse the input file and begin the import process.

If the input file is parsed successfully, the **DFMEA Import Confirmation** dialog will be displayed. The confirmation dialog displays a tree showing all the DFMEA data entries found in the input file, as well as a list of processing options.



In this example, we want to update the current DFMEA data maintained by Sherlock using all the data found in the input file. As such, select **Update Current Data Objects** as the processing option and select all entries in the **Import Data Tree.** Finally, press the **Update DFMEA Tree** button to complete the import process. After the import process has completed, a quick review of the C22 failure modes will show that we now have **Potential Effect** values specified for the first two failure modes, just like they were specified in the spreadsheet.

The **Replace Current Data Objects** processing option is similar to the update option, except that it removes all existing properties and entries associated with a given entry before importing the data found in the spreadsheet. Similarly, the **Clear All Data Before Import** removes all DFMEA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

data from the Sherlock tree before importing data from the input file. These processing options exist to quickly replace all or part of the DFMEA tree with the data contained in the input file.

Note:

Always use caution when selecting the **Replace** or **Clear** processing options. You will not be able to recover any data lost from Sherlock once you've pressed the **Update DFMEA Tree** button.

6.1.4.2. Importing Custom DFMEA Properties

You can combine custom property definitions and the import process to quickly add data values to any DFMEA data tree. For example, in our previous examples, we added the **Subcircuit / Designer** property to the **dfmeaProperties.csv** file, but we haven't specified any values for that property, and we haven't referenced that template variable in any template. Let's remedy that situation with a simple template modification.

First, right-click the **Main Board** entry in the DFMEA data tree and select **Export DFMEA Data** from the context menu. Then, select the **01-CCA-Summary-Template.xls** template file and generate a spreadsheet containing a summary list of all sub-circuits.

sherlock AUTOMATED DESIGN ANALYSIS™

Design Failure Mode and Effects Analysis (DFMEA)
Circuit Card Summary

Project: ODB++ Tutorial **Description:**
Circuit Card: ODB++ Tutorial
Prepared By: **Revision:**

Subcircuit Name	Component Count	Min SEV	Max SEV	Min OCC	Max OCC	Min DET	Max DET	Min RPN	Max RPN	NOTES
Flash Memory	4	1	8	2	2	2	2	4	32	
Input Connector	1	8	8	2	2	2	2	32	32	
Motor Controller - Right Side	3	1	8	2	2	2	2	4	32	
Processors	4	1	8	2	2	2	2	4	32	
UNASSIGNED	210	0	8	0	2	0	2	0	32	

Use the **Notes** cell to enter the designer information for each sub-circuit, such as highlighted here:

sherlock AUTOMATED DESIGN ANALYSIS™

Design Failure Mode and Effects Analysis (DFMEA)
Circuit Card Summary

Project: ODB++ Tutorial **Description:**
Circuit Card: ODB++ Tutorial
Prepared By: **Revision:**

Subcircuit Name	Component Count	Min SEV	Max SEV	Min OCC	Max OCC	Min DET	Max DET	Min RPN	Max RPN	NOTES
Flash Memory	4	1	8	2	2	2	2	4	32	John Doe
Input Connector	1	8	8	2	2	2	2	32	32	Sally Smith
Motor Controller - Right Side	3	1	8	2	2	2	2	4	32	Harry Simpson
Processors	4	1	8	2	2	2	2	4	32	Mike Johnson
UNASSIGNED	210	0	8	0	2	0	2	0	32	Ellen Brown

After saving the data changes in the spreadsheet, we can now modify the template file to import those values using the customized template variable:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

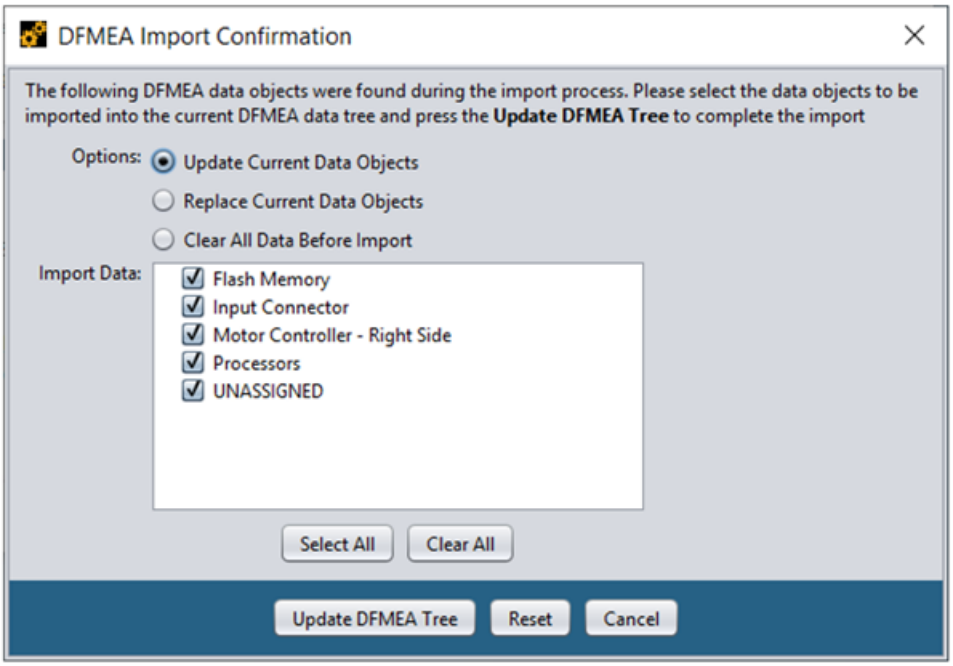
sherlock
AUTOMATED DESIGN ANALYSIS™

Design Failure Mode and Effects Analysis (DFMEA)
Circuit Card Summary

Project: <Project.projectName> **Description:** <Project.projectDescription>
Circuit Card: <CCA.ccaName>
Prepared By: **Revision:**

Subcircuit Name	Component Count	Min SEV	Max SEV	Min OCC	Max OCC	Min DET	Max DET	Min RPN	Max RPN	NOTES
<Subcircuits>										
<Subcircuit.name>	<Subcircuit.comp	<Subci	<Subci	<Subc	<Subci	<Subci	<Subci	<Sub	<Sub	<Subcircuit.Designer>

After saving the template change, we're ready to import the missing data by right-clicking the **Main Board** DFMEA tree entry, selecting **Import DFMEA Data** and using the modified template and spreadsheet files.



As before, we want to update the existing DFMEA data using all the data values imported from the spreadsheet. After the data has been imported, a quick review of the **DFMEA Data Tree** will show that all the designer data has been successfully added.

Note:

When importing data, Sherlock automatically ignores data values associated with the **min**, **max** and **count** internal properties supported for sub-circuits and parts because those values are automatically generated by Sherlock. Similarly, all RPN values imported from a spreadsheet are also ignored.

6.1.4.3. Importing Pre-Existing Spreadsheets

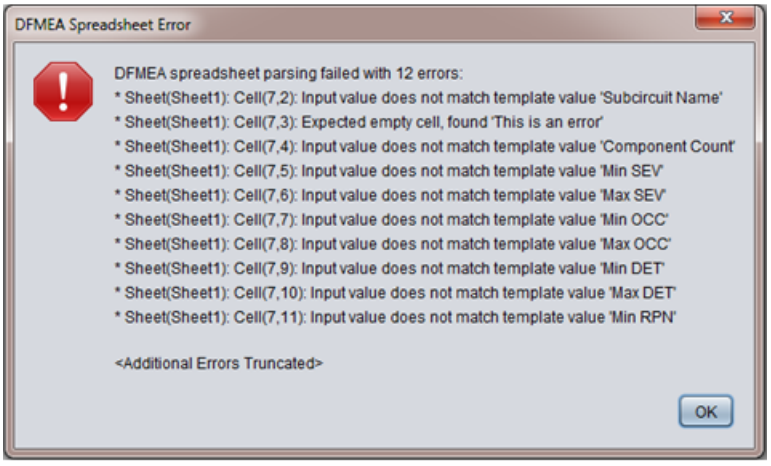
Organizations that have existing DFMEA spreadsheets can import the data from those spreadsheets into Sherlock using the same import approach described above by creating a template file that

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

represents the structure of the existing spreadsheet. For example, if the spreadsheet contains a fixed number of rows containing basic project and management information, those rows can simply be copied to the template file. If the spreadsheet then contains one or more rows for a given failure mode, a single copy of those rows can be copied to the template and placed inside of nested <Subcircuits>, <Components>, and <FailureModes> template lists. Finally, replace the values in that sample row by the appropriate subcircuit, part, or failure mode template variable. If additional properties are needed, simply add them to the **dfmeaProperties.csv** definition file to cause them to appear in the DFMEA data displays in Sherlock and to be imported from the input spreadsheet.

6.1.4.4. Import Errors

When modifying spreadsheet files, it is easy to add unnecessary rows or improperly modify cell values. In such cases, the resulting spreadsheet will probably not be consistent with the template used by Sherlock to process that file. In such cases, Sherlock will display an error dialog like the following:



In this case, Sherlock was not able to match the value found in cell (7,2) with the value expected by the template file (**Subcircuit Name**). This problem occurred because someone added an unexpected row to the spreadsheet file, as highlighted here in orange:

sherlock AUTOMATED DESIGN ANALYSIS™

Design Failure Mode and Effects Analysis (DFMEA)

Circuit Card Summary

Project: ODB++ Tutorial **Description:**

Circuit Card: ODB++ Tutorial

Prepared By: **Revision:**

This is an error

Subcircuit Name	Component Count	Min SEV	Max SEV	Min OCC	Max OCC	Min DET	Max DET	Min RPN	Max RPN	NOTES
Flash Memory	4	1	8	2	2	2	2	4	32	John Doe
Input Connector	1	8	8	2	2	2	2	32	32	Sally Smith
Motor Controller - Right Side	3	1	8	2	2	2	2	4	32	Harry Simpson
Processors	4	1	8	2	2	2	2	4	32	Mike Johnson
UNASSIGNED	210	0	8	0	2	0	2	0	32	Ellen Brown

The problem can be corrected by either (a) removing the offending row from the spreadsheet file or (b) adding a similar row to the template file (if the orange row is really supposed to be there).

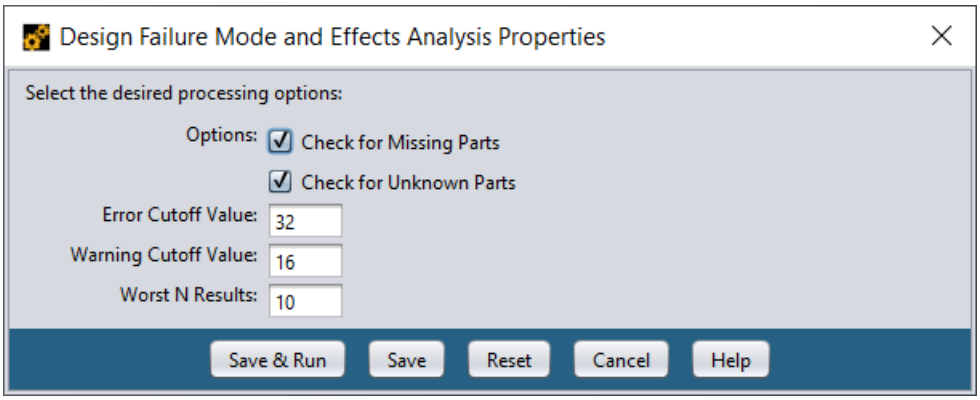
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.5. DFMEA Results

In addition to the DFMEA data management capabilities described above, Sherlock provides an analysis module that generates summary results based on the RPN values assigned in the DFMEA data. Although such results are not used to generate the same types of scores provided by other Sherlock analysis modules, they can be used to either get an overall view of the DFMEA data associate with a given circuit card to determine where best to spend additional effort.

6.1.5.1. Generating DFMEA Results

The DFMEA results are generated by right-clicking the **Analysis > DFMEA** entry in the Sherlock Project Tree for a given circuit card and selecting the **Edit Properties** item from the context menu to display the properties dialog:



The properties dialog allows you to select various processing options, with the following options supported:

Table 6.11: Processing Options

Processing Option	Description
Check for Missing Parts	Generates an error for any part contained in the Parts List that is not referenced by at least one subcircuit in the DFMEA data
Check for Unknown Parts	Generates an error for any part referenced in the DFMEA data that does not exist in the Parts List

The dialog also allows you to specify the **cutoff** values to be used when color coding results. RPN values greater than or equal to the **Error Cutoff Value** are colored as errors (red), while values greater than or equal to the **Warning Cutoff Value** are colored as warnings (yellow). All other results will be colored green. After selecting the desired processing options and specifying the desired cutoff values, press the **Save & Run** button to save the properties and run the analysis task (or you can simply **Save** the properties to be used for a subsequent analysis run).

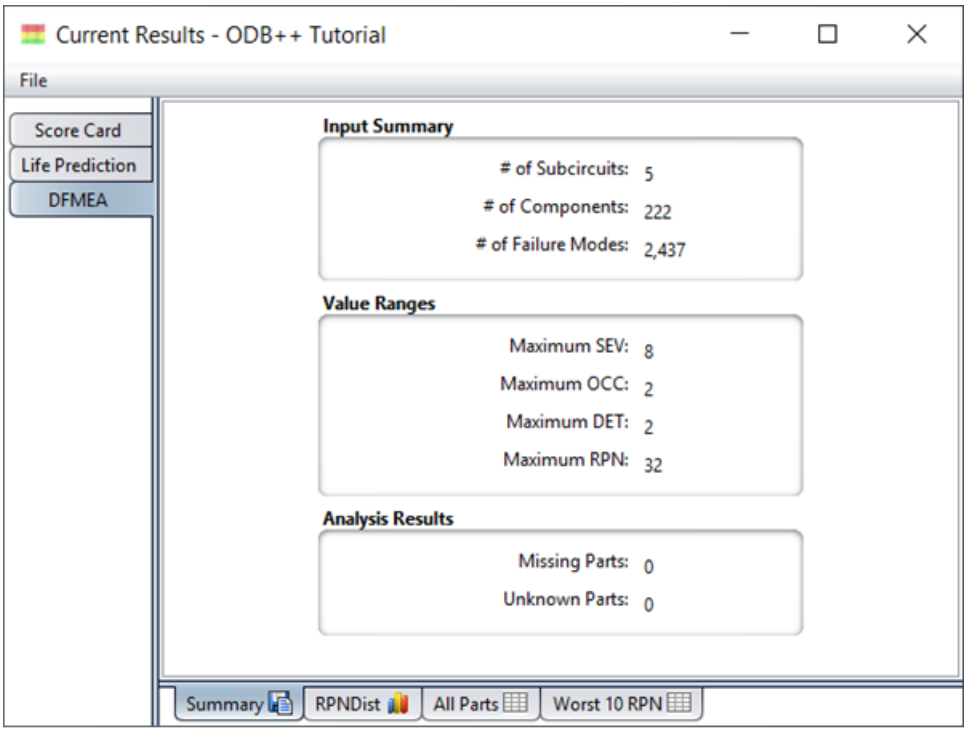
6.1.5.2. DFMEA Results

After the analysis task has completed successfully, double-click the **Analysis > DFMEA** entry in the Sherlock Project Tree to view the DFMEA results, including:

- Summary Results

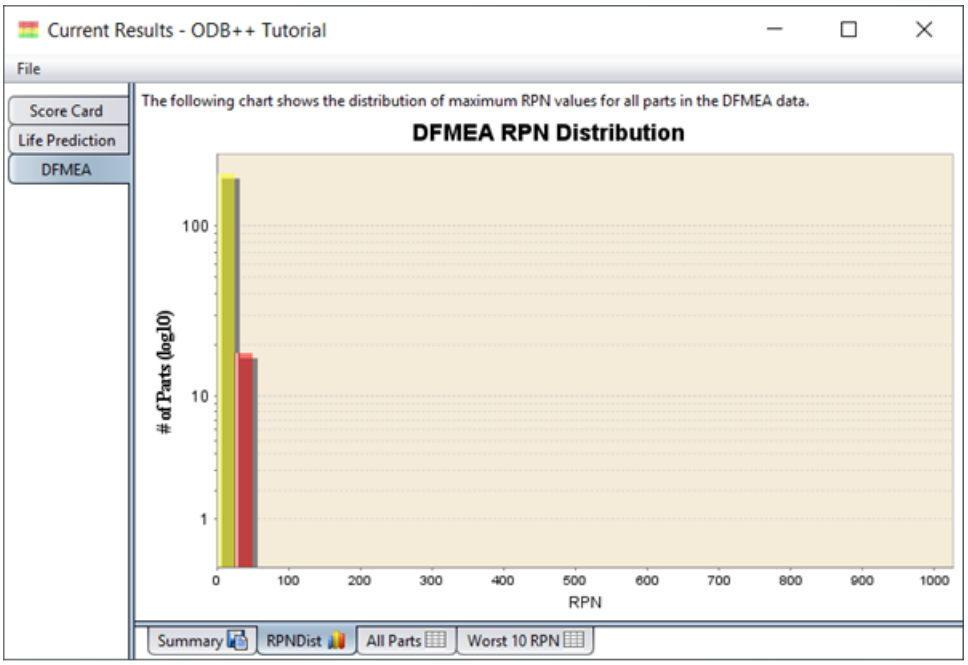
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- RPN Distribution Chart
- DFMEA Ratings Table
- Worst RPN table
- Processing Issues



The **Summary** tab shows various counts and statistics gleaned from the DFMEA data. The maximum values shown are across all subcircuits, parts and failure modes associated with the current circuit card. The **RPN Dist** tab shows a graphical distribution of the maximum RPN values assigned to each part in the DFMEA data.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

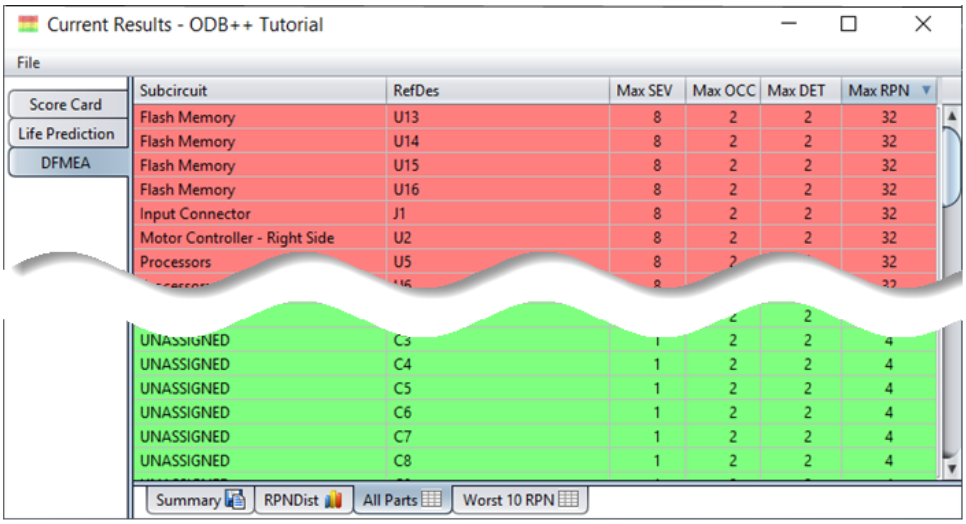


The **All Parts** tab shows a list of all parts defined in the **DFMEA Tree** and the maximum ratings assigned to the failure modes associated with those parts. (See table below.)

The rows in the table are color-coded based on the cutoff values defined in the analysis properties dialog. The table is initially sorted by decreasing RPN values, but you may single-click any column header to sort by that column. (Repeated clicks toggle between ascending and descending order.)

The **Worst N RPN** tab displays a list of the worst number of specified RPN values. This number is specified in the **Worst N Results** which can be changed in the DFMEA Properties. In this example, we have specified ten in the **Worst N Results,** and should therefore expect a table of ten worst RPN values.

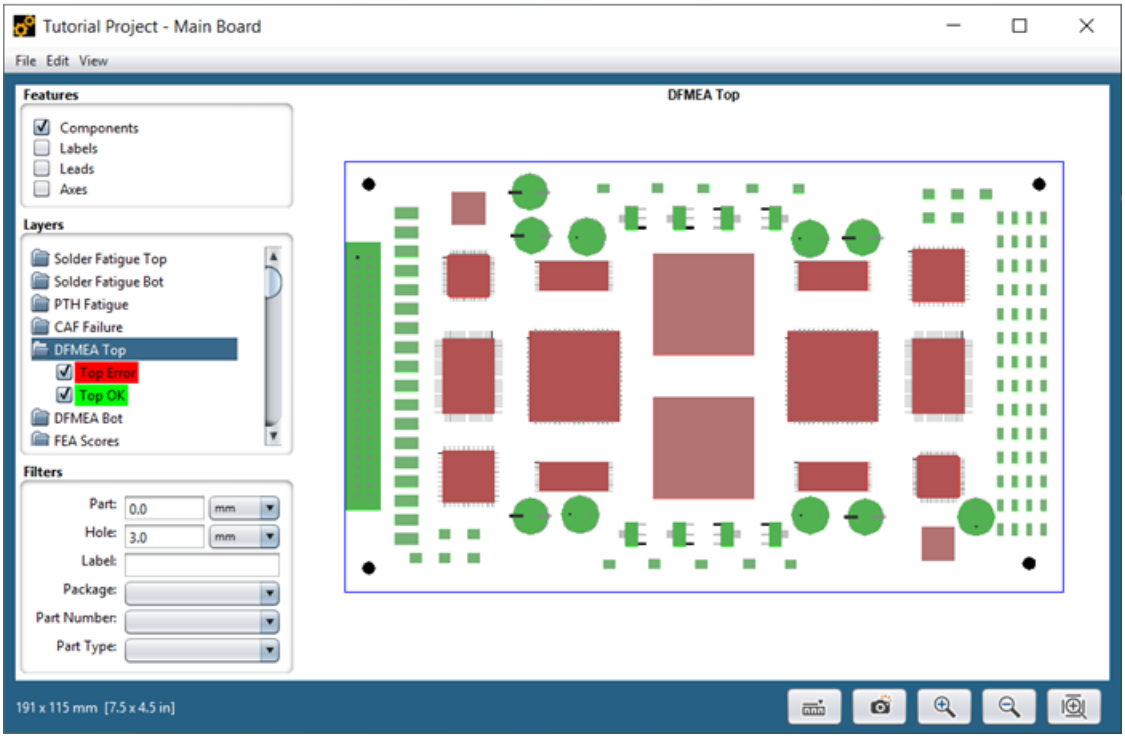
Finally, an **Issues** tab lists may also appear. This tab lists any issues generated during the analysis process.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.5.3. DFMEA Layers

In addition to the tabular and graphical results described above, the **DFMEA module** also generates a collection of graphical layers that display color-coded parts based on the maximum RPN value assigned in the DFMEA data.



In this example, we can see each of the DFMEA parts color-coded the same as their corresponding rows in the tabular result. This location-based view of the results clearly shows the critical regions of the board that need to be examined further to improve their DFMEA ratings.

6.1.6. Net List

A circuit card Net List defines all electrical connections between component pins as a list of pin and network combinations. Pins that are electrically connected will be matched with the same network name. Sherlock uses the Net List when generating default Pin Open Failures and Pin Short Failures.

Importing Net List Data

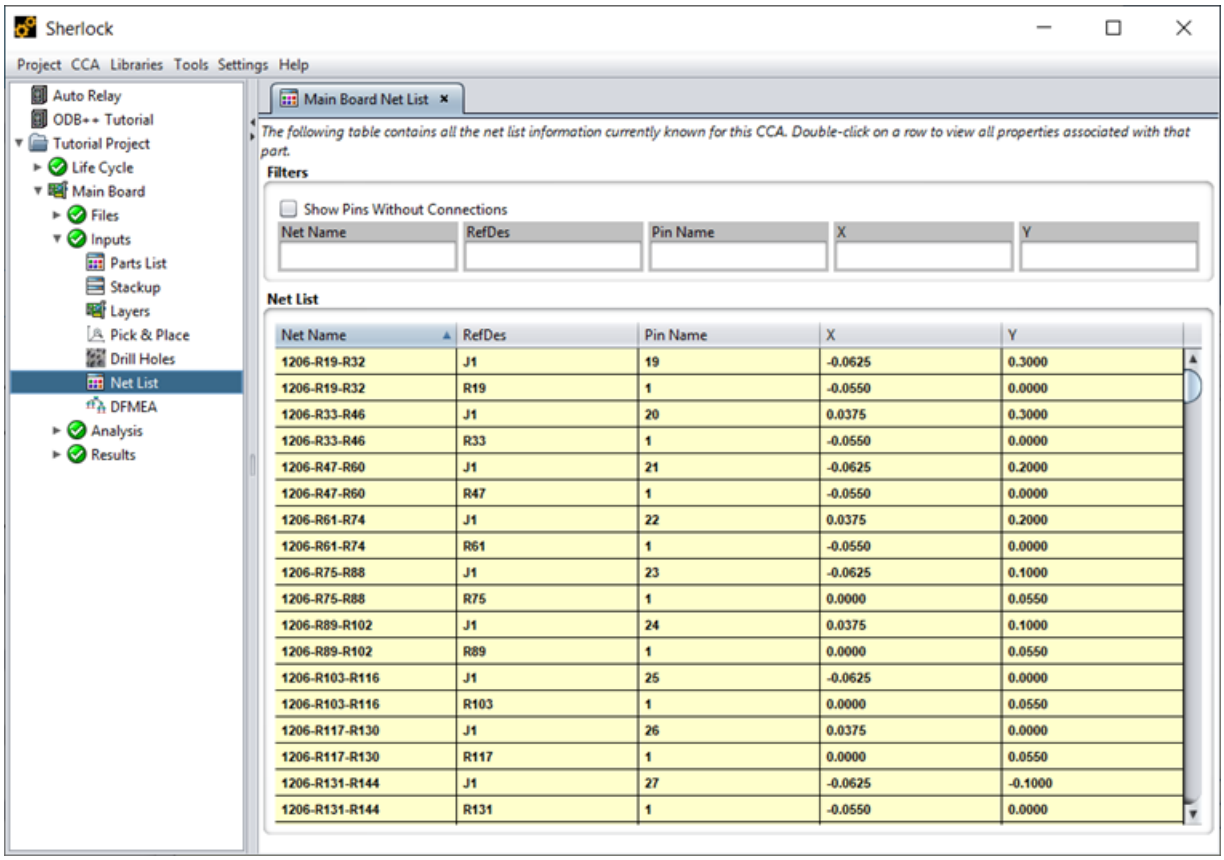
Sherlock automatically imports Net List data when processing ODB++, IPC-2581, or Eagle CAD files, requiring no additional user input. For other projects, however, users must supply a CSV/XLS or a standard IPC-D-356 formatted file containing the Net List data. Once the file has been added to the **Files** list for a given circuit card, select the Net List (CSV/XLS) or Net List (IPC) depending on the imported file type, and Sherlock will load all the net list data found in the file.

Viewing Net List Data

For review purposes, the Net List data associated with a given circuit card can be viewed by double-clicking the **Inputs >> Net List** entry in the Project Tree. At that point, the **Net List** tab will be displayed

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the **Content Panel**, showing the **Net Name**, **Reference Designator** and **Pin Name** for each network connection.



You may sort by any column and/or filter by rows containing specific substrings. You may also double-click any row to see the corresponding part properties. Finally, Net List data can be exported from the table using the standard Sherlock export capabilities.

6.2. Component Failure Analysis

Component Failure Mechanism analysis is used to predict failures due to wearout not related to solder joint fatigue. This is important where a PCB may be in a relatively benign environment in which wearout becomes the dominating influence on the lifetime of a component. Ansys Sherlock currently performs this analysis for ceramic and electrolytic capacitors. The methods used for electrolytic capacitors are vendor-dependent based on equations provided by supported vendors.

In this section, the following topics are covered:

6.2.1. Input Data

6.2.2. Analysis Properties and Analysis Results

6.2.1. Input Data

The **Component Failure Mechanism Analysis Module** makes use of the following input data for the analysis calculations:

- **Life Cycle Reliability Goals**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Parts List**

> **Tip:**
>
> The following section explains which part properties Sherlock uses in the Component Failure Analysis: Tables: Required Part Properties per Analysis Type (p. 326).

- **Thermal Events** and associated **Thermal Maps**

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

With regards to properties defined in the **Parts List**, the **Component Failure Mechanism Analysis Module** uses properties from various tabs defined in the **Part Editor:**

- **Part Type**
- **Package Properties**
- **Electrical Properties**
- **Thermal Properties**

For ceramic capacitors, **Test Properties** are also used.

Currently, Sherlock performs Component Failure Mechanism Analysis for ceramic, electrolytic, and aluminum polymer capacitors from the parts list. For any component defined with the part type of **CAPACITOR CERAMIC**, **CAPACITOR ELECTROLYTIC**, or **CAPACITOR ALUMINUM POLYMER**, the analysis will be performed as well. These part types have the **Component Failure Mechanism** property enabled. The ability to perform this analysis depends on specific properties being defined for the part, and those properties may be vendor dependent. For electrolytic or aluminum polymer capacitors, the **Lifetime** property and **Max Rated Temperature** must be defined. Some vendors may require **Ripple Temperature Rise**, **Applied and Rated Ripple Current**, and **Applied and Rated Voltage**. For any component missing required inputs, the analysis task will provide details on the missing properties in the analysis errors tab for **Component Failure Analysis**.

> **Note:**
>
> Additional component limitations are listed under the relevant vendor in the next section Required Vendor Inputs (p. 397).

6.2.1.1. Required Vendor Inputs

The following shows vendor-specific required parameters as supported by Ansys Sherlock. If a vendor is not listed for which the part type is **CAPACITOR ELECTROLYTIC**, the **Default** properties will be used.

- Default – Lifetime, Max Rated Temperature
- Cornell Dubilier – Lifetime, Max Rated Temperature, Operating Voltage, Applied Voltage

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Illinois Capacitor – Lifetime, Max Rated Temperature, Component Failure Mechanism, Ripple Current Temperature Rise, Applied Ripple Current, Rated Ripple Current, Operating Voltage, Applied Voltage
- Nichicon – Lifetime, Max Rated Temperature, Component Failure Mechanism, Ripple Current Temperature Rise, Applied Ripple Current, Rated Ripple Current
- Nippon ChemiCon - Lifetime, Max Rated Temperature, Component Failure Mechanism, Ripple Current Temperature Rise, Applied Ripple Current, Rated Ripple Current
- Panasonic - Lifetime, Max Rated Temperature
- Rubycon - Lifetime, Max Rated Temperature, Component Failure Mechanism, Ripple Current Temperature Rise, Applied Ripple Current, Rated Ripple Current

---

Note:

For Rubycon, screw-type terminals are not supported.

- Vishay - Lifetime, Max Rated Temperature, Ripple Current Temperature Rise

---

Note:

For Vishay, only solid electrolytic capacitors are supported. The analysis is not meant for liquid-based aluminum electrolytic capacitors.

If a vendor is not listed for which the part type is **CAPACITOR ALUMINUM POLYMER**, the **Default** properties will be used.

- Default – Lifetime, Max Rated Temperature
- Illinois Capacitor – Lifetime, Max Rated Temperature, Operating Voltage, Rated Voltage
- KEMET – Lifetime, Max Rated Temperature
- Nichicon – Lifetime, Max Rated Temperature, Component Failure Mechanism, Ripple Current Temperature Rise, Applied Ripple Current, Rated Ripple Current
- Nippon ChemiCon - Lifetime, Max Rated Temperature, Component Failure Mechanism, Ripple Current Temperature Rise, Applied Ripple Current, Rated Ripple Current
- Panasonic - Lifetime, Max Rated Temperature
- Samxon – Lifetime, Max Rated Temperature
- Wurth Electronics – Lifetime, Max Rated Temperature”

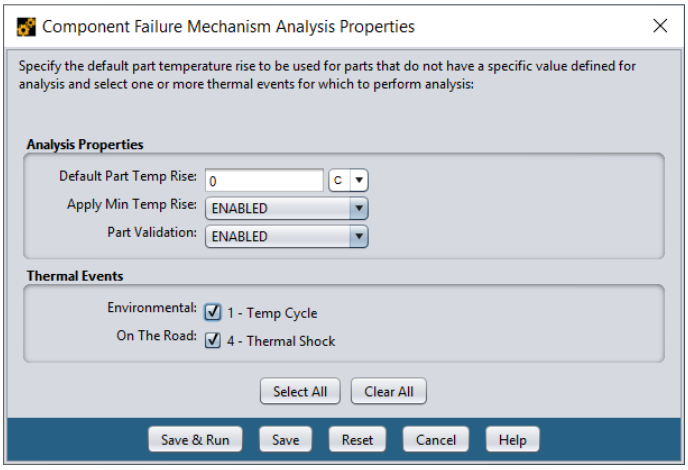
For the **CAPACITOR CERAMIC** part type the properties used are package length, package width, package thickness, max rated temperature, operating voltage, applied voltage, test voltage, test duration, test devices/samples.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.2. Analysis Properties and Analysis Results

6.2.2.1. Analysis Properties

The **Component Failure Analysis Module** allows you to set the properties shown here:



The **Part Temp Rise** property defines the default temperature rise to be used when:

- There are no **Thermal Maps** assigned to a given **Thermal Event** or the **Thermal Map** cannot be used to derive a temperature for a given component, and
- No specific **Part Temp Rise** property is defined for a given component in the Parts List.

The **Apply Min Temp Rise** parameter allows users to prevent the part temperature rise value from being applied to the minimum temperature defined in a thermal cycle. By default, it is applied to the minimum temperature.

The **Part Validation** parameter, when enabled, causes the **Part Validation Analysis** to be performed prior to **Solder Joint Fatigue** analysis if it has not yet been successfully run.

6.2.2.2. Analysis Results

The **Failure Rate Analysis Module** generates the following results:

- **Summary Panel** showing reliability goals and analysis statistics
- **Life Prediction Curve** for the circuit card based on the time to failure predicted for all parts analyzed
- **Result Table** showing key properties, damage, time to failure 5%, and time to failure 63%, for each part analyzed

6.3. Plated Through Hole (PTH) Fatigue Analysis

Plated Through Holes (PTHs), also known as plated through vias (PTVs), are holes drilled through multilayer printed circuit boards (PCBs) that are electrochemically plated with a conductive metal (typically copper). These plated holes provide electrical connections between layers. Because these plated holes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are metallurgically bonded to annular rings on the top and bottom of the printed circuit board, they act like rivets and constrain the PCB. This constraint subjects the PTH to stresses when the PCB experiences changes in temperature. Over time the PTH will experience fatigue and eventually fail due to crack propagation. PTH Fatigue is influenced by number of drivers, including temperature range, PTH diameter, PTH copper plating thickness, copper plating material properties, printed board thickness, printed board out-of-plane material properties, and quality of the copper plating. Ansys Sherlock calculates a time to failure using the industry-accepted model published in IPC-TR-579, *Round Robin Reliability Evaluation of Small Diameter Plated-Through Holes in Printed Wiring Boards*. Life calculation for PTHs subjected to thermal cycling is a three step process, involving a stress calculation, strain range calculation, and an iterative lifetime determination.

In this section, the following topics are covered:

6.3.1. Input Data

6.3.2. Analysis Properties

6.3.3. Analysis Results

6.3.1. Input Data

The **PTH Fatigue Analysis Module** makes use of the following input data for the analysis calculations:

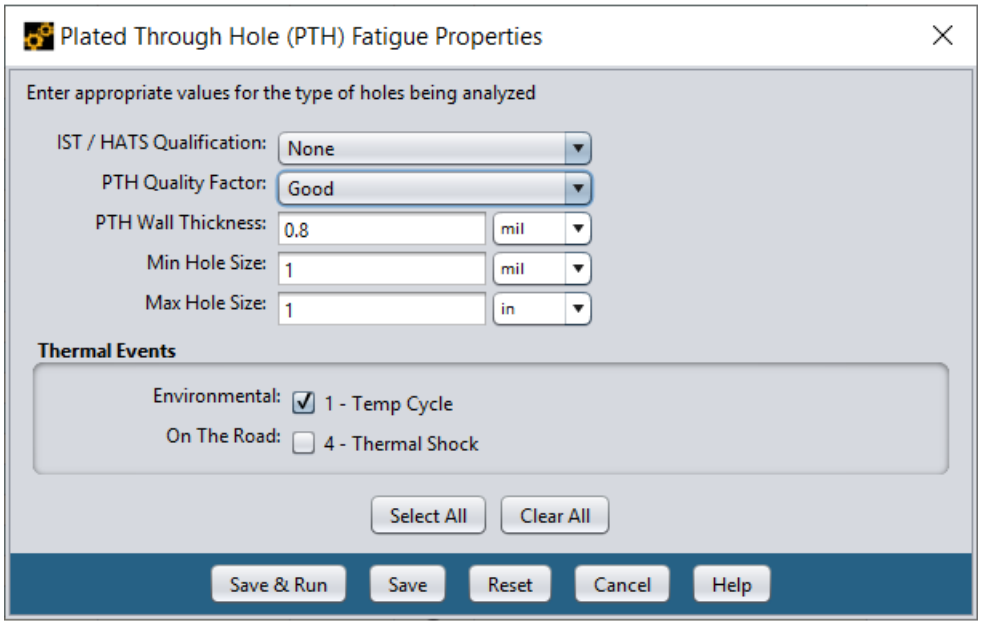
- **Life Cycle** Reliability Goals
- Size and location of all plated through-holes and vias
- Circuit Card mechanical properties (stackup data)
- **Thermal Events** and associated **Thermal Maps**

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.2. Analysis Properties

The **PTH Fatigue Analysis Module** allows the user to set the properties shown here:



The **IST / HATS Qualification** property indicates the type of qualification process performed for the circuit card during production.

The **PTH Quality Factor** property indicates the overall quality of the plated through hole.

The **PTH Wall Thickness** property specifies the average thickness of the conductive material used to coat the PTH.

The **Min Hole Size** and **Max Hole Size** specify the range of hole sizes to be analyzed.

The **Thermal Events** panel allows the specific thermal Events to be selected for the analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.3. Analysis Results

The **PTH Fatigue Analysis Module** generates the following results:

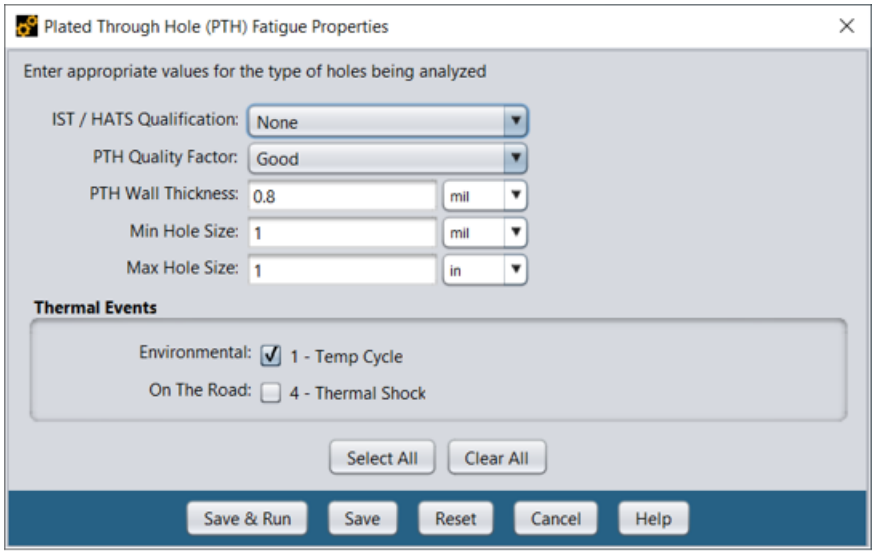
- **Summary Panel** showing overall scores, reliability goals, board properties, hole properties, and analysis statistics
- **Life Prediction Curve** for the circuit card based on the Cycles to Failure predicted for all holes analyzed
- **Score Distribution Chart** showing a histogram of scores for all holes analyzed
- **Result Table** showing key properties, Cycles to Failure and score for each hole analyzed
- **Graphical Layers** showing the score for each hole analyzed

To see how the various input data and properties affect the **PTH Fatigue Results**, let's run a couple of tests with various inputs. We'll start by examining the results provided by the Tutorial Project. If you haven't already done so, import that project now.

Note:

If you've modified data during the tutorials, it's likely that the **PTH Fatigue Results** have been cleared in your project or are different from those shown in this document. If so, the easiest way to clean things up is to delete the existing Tutorial Project and import it again from the ZIP file provided.

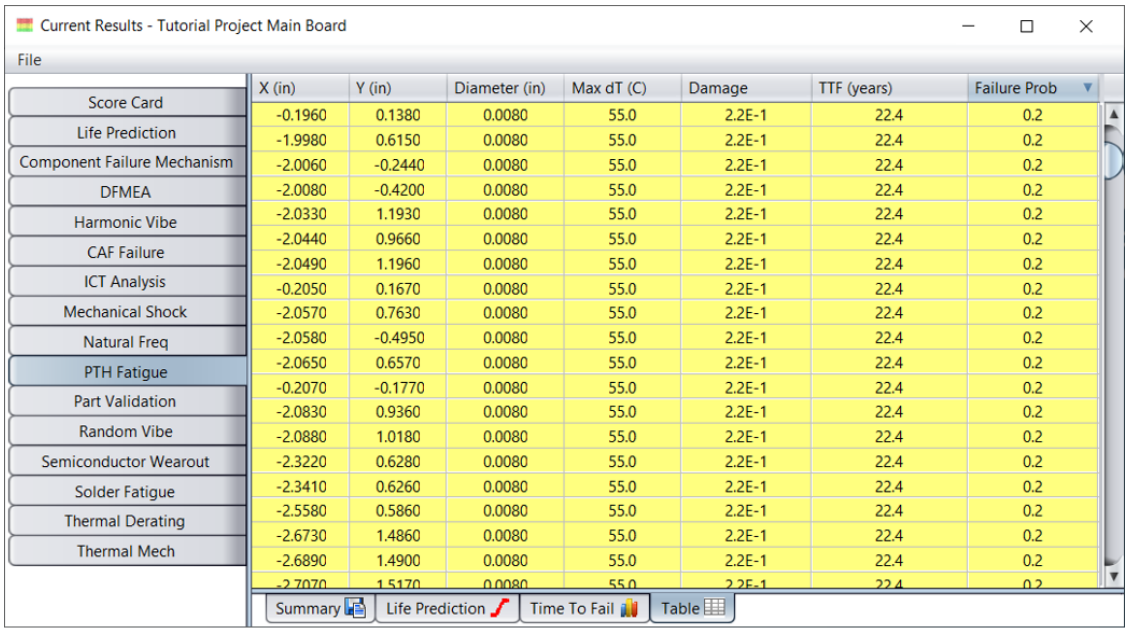
We'll start by setting the PTH Fatigue properties as shown below. Afterwards, select **Save & Run** to commit the changes and run the analysis.



Afterwards, select **Save & Run** to commit the changes and run the analysis. Double-click the **PTH Fatigue** entry of the **Analysis** folder in the Project Tree to display the PTH Fatigue Results. Select the **Table** sub-tab to show the analysis results for all plated through holes analyzed.

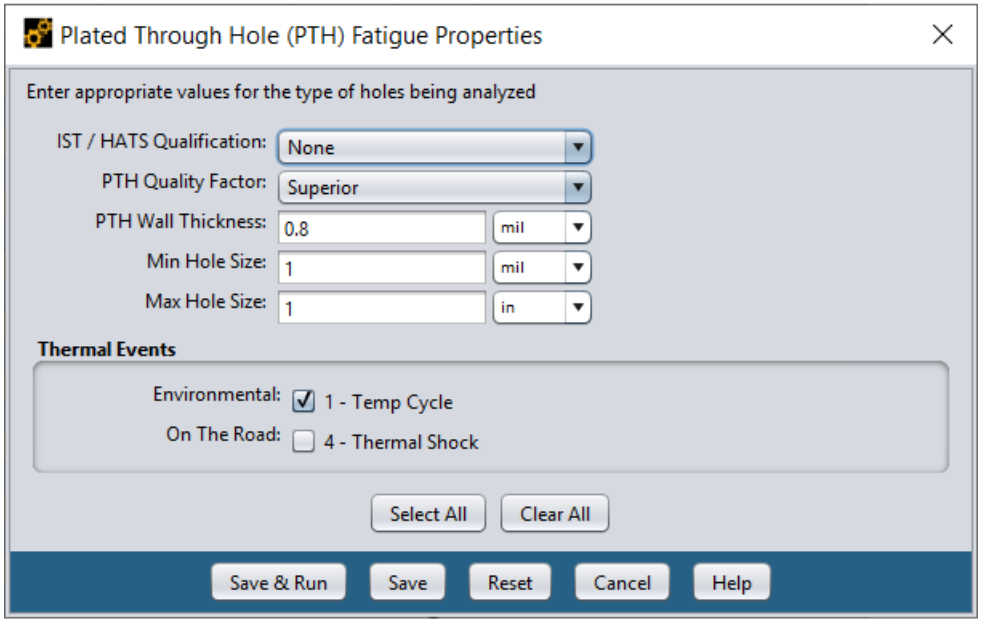
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Now, double-click the **PTH Fatigue** entry of the **Analysis** folder in the **Project Tree** to display the **PTH Fatigue Results**. Select the **Table** sub-tab to show the analysis results for all plated through holes analyzed.



As is clear by the color-coding and data columns, all holes are predicted to fail within 22.4 to 47 years, which represents a safety factor between 3 or 5 more times the desired service life.

To see how the calculations are affected by the input properties, right-click the **PTH Fatigue** entry in the **Project Tree** and select the **Edit Properties** menu option to edit the input properties.

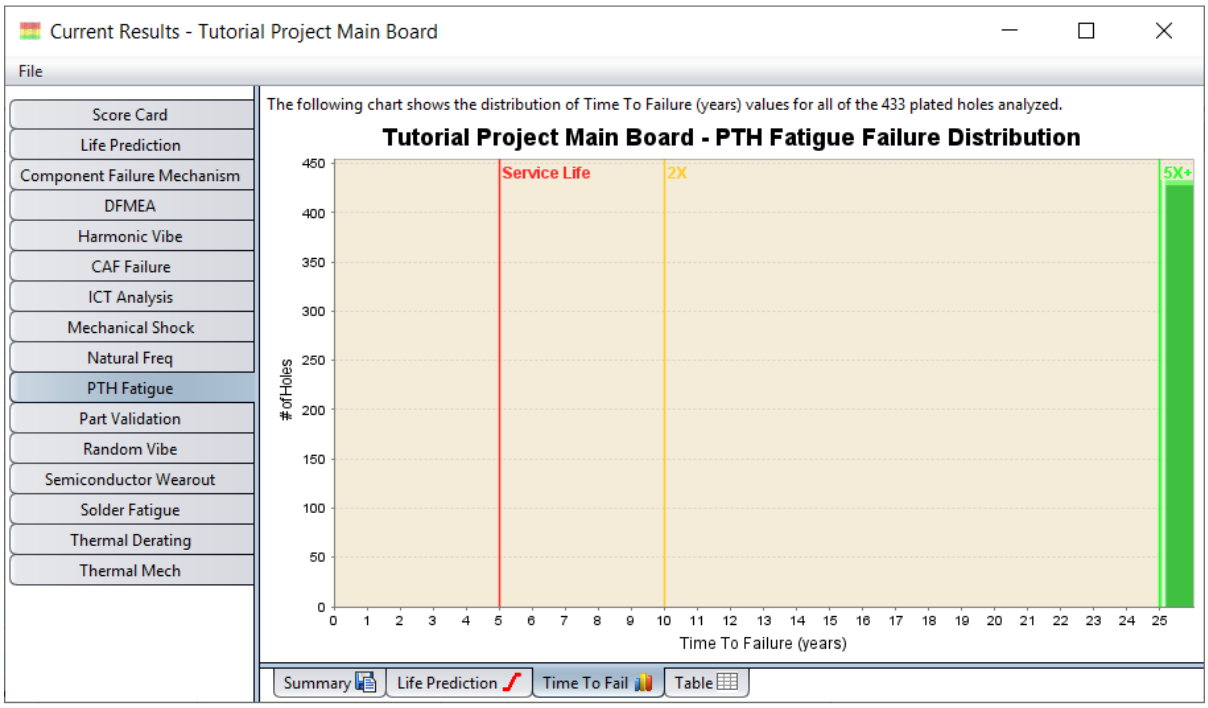


Let's change the **PTH Quality Factor** to **Superior**. After changing the property, press the **Save & Run** button to commit the changes and re-run the analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While the analysis is being performed, the icon next to the **PTH Fatigue** entry will be changed to a clock indicating that a background process is being performed. When the analysis process is complete, the icon will change back to a GREEN check mark, at which point the **PTH Fatigue Results** panel will be refreshed.

Instead of looking at the tabular results, let's look at the **Failure Distribution** chart to see how the changes affected the predicted results.



The **Failure Distribution Chart** is a histogram showing the various **Time to Failure** values predicted for all the holes analyzed. The X-axis shows the Time to Failure values, while the Y-axis shows the number of holes that are predicted to have a given TTF value. The chart allows you to easily see that our input property change has significantly changed the predicted results. Previously, the minimum TTF was 22.4 years, now the TTF is >50 years.

Now, let's see how the **Thermal Events** defined in the **Life Cycle** affect the **PTH Fatigue Results**. You may recall from our previous lesson that two **Thermal Events** are defined in the **Life Cycle** used by the Tutorial Project. The **Temp Cycle** Event models the daily ambient temperate cycle, oscillating between 10C and 32C over a 24 hour period, every day of the year. On the other hand, the **Thermal Shock** Event models a drastic temperature change from -40C to 105C over a short period of time, occurring 3 times per day. Essentially, the **Temp Cycle** Event is a relatively mild event that occurs over a long period of time, while the Thermal Shock Event is a relatively severe event that occurs over a short period of time.

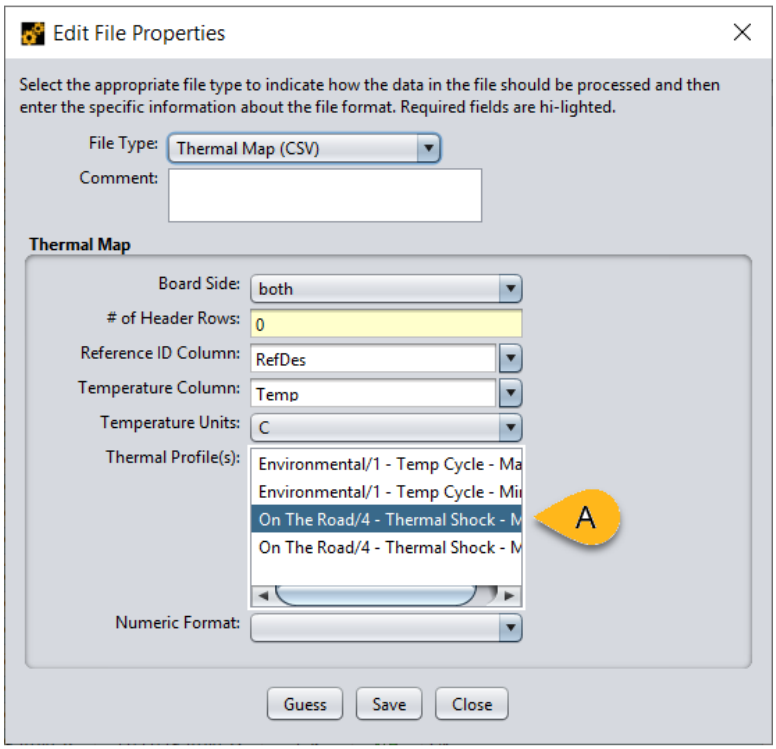
In both cases, the **Thermal Events** themselves define uniform temperate changes across the entire circuit card. Furthermore, each of the input properties is also applied to each hole in the same way (for example, they all have the same PTH Wall Thickness). This uniformity of thermal and input properties means that that the only differentiation in the analysis results comes from the hole sizes.

We can change the thermal properties seen by each hole by assigning one or more **Thermal Maps** to the **Thermal Events** defined in the **Life Cycle**. **Thermal Maps** assign location-specific thermal values to the circuit card, causing variations in temperate across all the plated through holes. The

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tutorial Project includes a **Thermal Map**, but until now it has not been assigned to any **Thermal Events**.

To assign the **Thermal Map**, right-click the **Thermal Map.csv** entry located in the **Files** folder of the **Project Tree** and select the **Edit Properties** menu option. Since this file has already been successfully imported in the past, we know that all the properties, shown below, are correct.

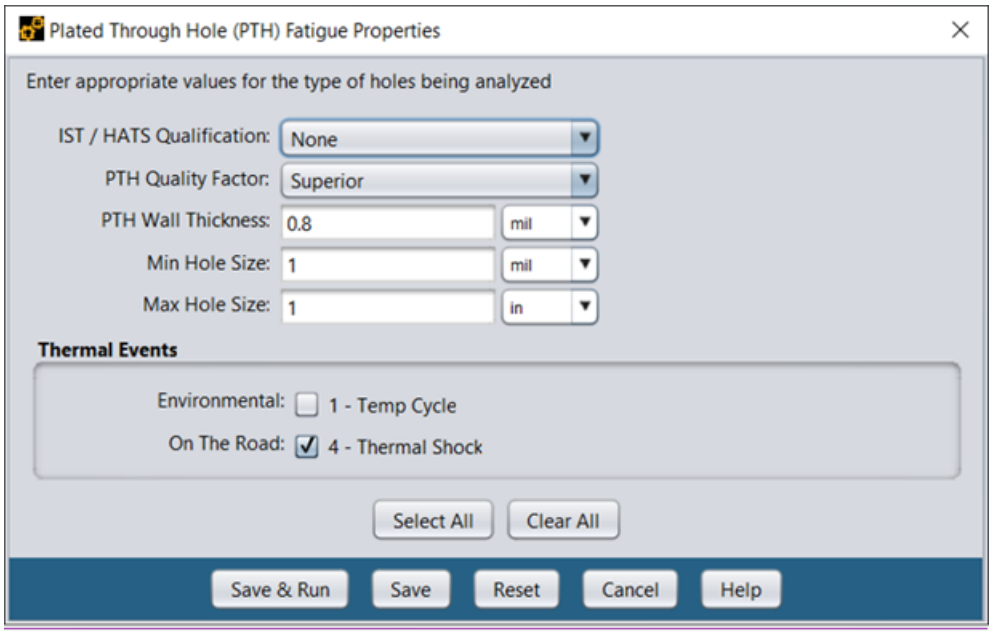


Our main focus, therefore, is on the **Thermal Profile(s)** property, which lists all of the **Thermal Events** defined in the project **Life Cycle**. In this case, we see **Min** and **Max** entries for both **Thermal Events** discussed above. The property dialog allows us to assign the selected **Thermal Map** to one or more of the **Thermal Events**, representing the minimum or maximum temperatures (or both). Once assigned, the temperatures defined in the **Thermal Map** itself will be used for those events, instead of the ambient values defined in the thermal cycle.

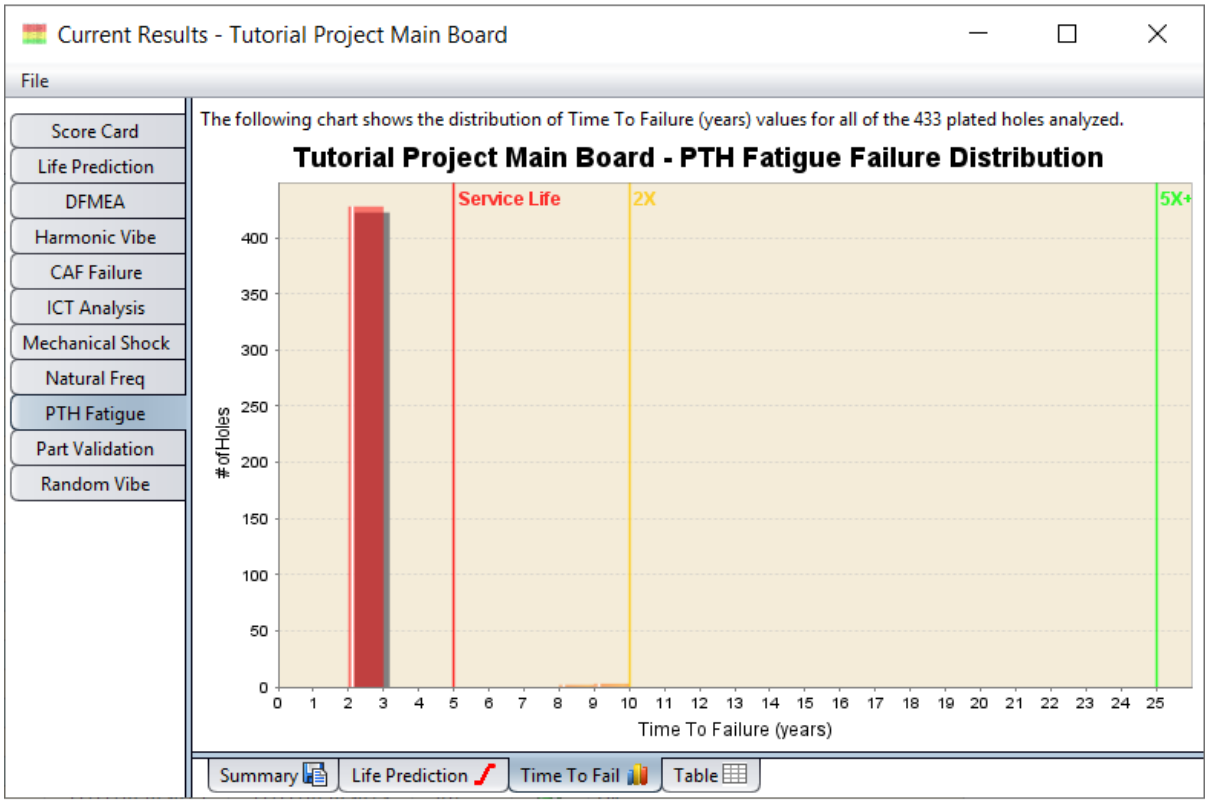
To see how this works, select the profile entry as shown (A, above) and press **Save** to assign the **Thermal Map** to that Thermal Event.

Since we've effectively changed one of the **Thermal Events**, all analysis results that depend on thermal properties will be cleared, including the **PTH Fatigue Results**. Therefore, you'll need to right-click the **PTH Fatigue** entry in the **Project Tree** and select **Edit Properties**. Since we have changed the Thermal Profile to “On The Road/4-Thermal Shock-Max,” we must also toggle the **Thermal Event** to On "The Road: 4-Thermal Shock" as shown below:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Afterwards, click **Save &Run** to run the analysis and generate the results. Displayed below is the Time to Failure Graph.



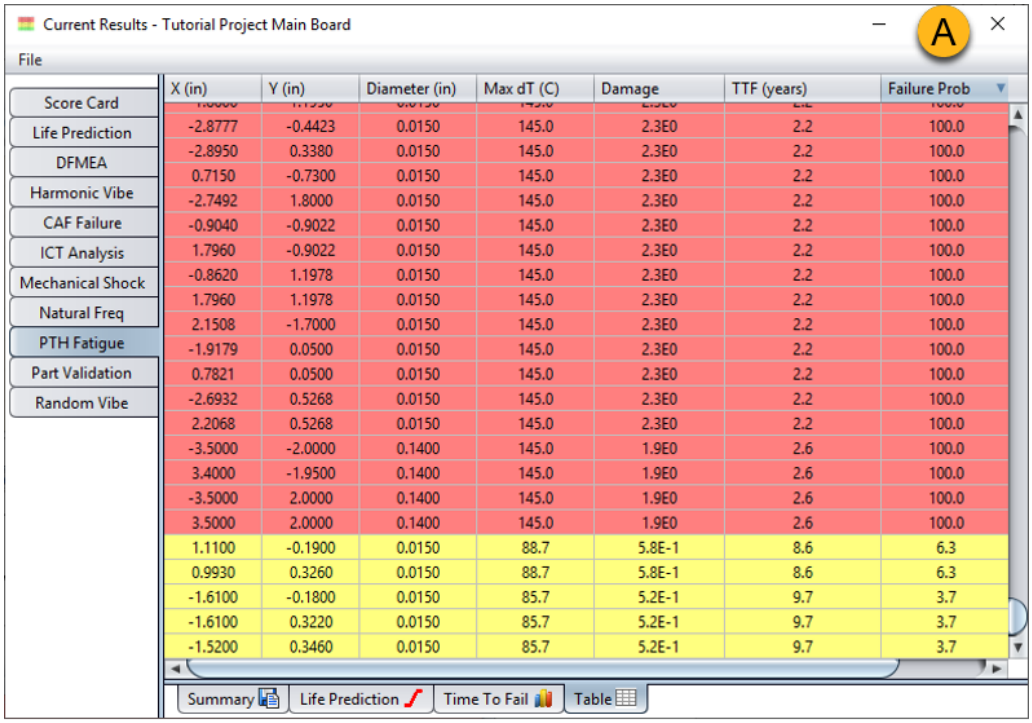
Now if we look at the **Failure Distribution**, we see most holes are expected to fail during the service life period.

To get a more detailed view of what happened, select the **Table** sub-tab to see the analysis details for each hole analyzed. See image A below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

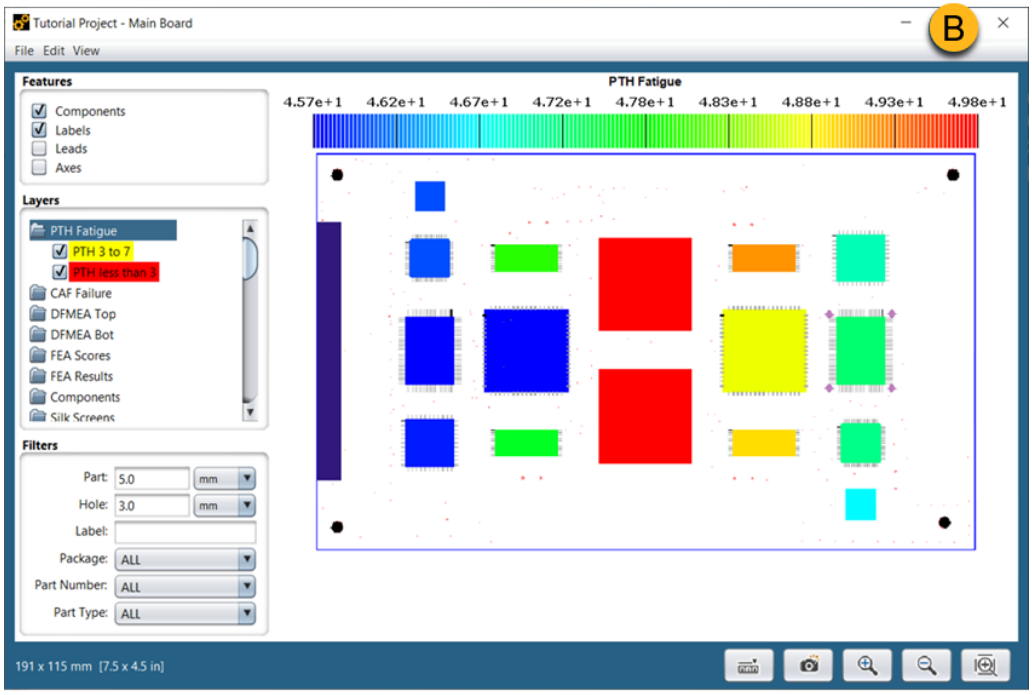
You may also overlay the **PTH Fatigue Results** with the **Thermal Map** in the **Layer Viewer**. To that end, double-click the **Thermal Map.csv** entry in the **Project Tree** to display the **Thermal Map** itself in the **Layer Viewer**. Then, double-click the **PTH Fatigue** folder in the **Layer Viewer** to overlay the PTH Fatigue results. Finally, double-click the **Drill Holes** entry to display holes that weren't analyzed. You may need to enter **0** in the **Hole Filter** to view all holes. The resulting display should look something like image B.

The **Thermal Map** defines uniform temperatures for a small number of circuit card components. The color coding shows the relative temperature of each component, from cool blue to hot red.



X (in)	Y (in)	Diameter (in)	Max dT (C)	Damage	TTF (years)	Failure Prob
-2.8777	-0.4423	0.0150	145.0	2.3E0	2.2	100.0
-2.8950	0.3380	0.0150	145.0	2.3E0	2.2	100.0
0.7150	-0.7300	0.0150	145.0	2.3E0	2.2	100.0
-2.7492	1.8000	0.0150	145.0	2.3E0	2.2	100.0
-0.9040	-0.9022	0.0150	145.0	2.3E0	2.2	100.0
1.7960	-0.9022	0.0150	145.0	2.3E0	2.2	100.0
-0.8620	1.1978	0.0150	145.0	2.3E0	2.2	100.0
1.7960	1.1978	0.0150	145.0	2.3E0	2.2	100.0
2.1508	-1.7000	0.0150	145.0	2.3E0	2.2	100.0
-1.9179	0.0500	0.0150	145.0	2.3E0	2.2	100.0
0.7821	0.0500	0.0150	145.0	2.3E0	2.2	100.0
-2.6932	0.5268	0.0150	145.0	2.3E0	2.2	100.0
2.2068	0.5268	0.0150	145.0	2.3E0	2.2	100.0
-3.5000	-2.0000	0.1400	145.0	1.9E0	2.6	100.0
3.4000	-1.9500	0.1400	145.0	1.9E0	2.6	100.0
-3.5000	2.0000	0.1400	145.0	1.9E0	2.6	100.0
3.5000	2.0000	0.1400	145.0	1.9E0	2.6	100.0
1.1100	-0.1900	0.0150	88.7	5.8E-1	8.6	6.3
0.9930	0.3260	0.0150	88.7	5.8E-1	8.6	6.3
-1.6100	-0.1800	0.0150	85.7	5.2E-1	9.7	3.7
-1.6100	0.3220	0.0150	85.7	5.2E-1	9.7	3.7
-1.5200	0.3460	0.0150	85.7	5.2E-1	9.7	3.7

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

If you hover the mouse over any of the colored regions, the associated temperature will be displayed in the upper left corner of the layer panel. That provides a convenient way to verify data and analyze results involving thermal properties.

6.4. Semiconductor Wearout

Ansys Sherlock provides the ability to predict semiconductor wearout using multiple methods depending on the available component-specific data. The prediction is performed with the Semiconductor Wearout analysis module which will produce wearout results and life prediction for applicable parts. Sherlock uses a model that follow SAE ARP 6338. More information about SAE ARP 6338 can be found at http://standards.sae.org/wip/arp6338/.

Note:

The Sherlock Semiconductor Wearout Analysis is only valid for components with silicon dies.

In this section, the following topics are covered:

6.4.1. Input Data
6.4.2. Acceleration Factors
6.4.3. Documents
6.4.4. Analysis

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1. Input Data

The analysis performed depends on the available data available per component. Sherlock supports five approaches to predict semiconductor wearout.

1. The manufacturer provides a prediction based on the life cycle or the user has actual field data at the same or similar life cycle.
2. The manufacturer provides temperature-specific prediction and the manufacturer, or the user provides the acceleration factor.
3. The manufacturer provides the test results or the user provides or assumes the test results, and the manufacturer or the user provides the acceleration factor.
4. The manufacturer provides temperature-specific prediction and the user uses industry generic models (JEDEC JEP-122) and user-defined parameters or Sherlock-parameters to calculate an acceleration factor.
5. The manufacturer provides the test results or the user provides or assumes the test results and the user uses industry generic models (JEDEC JEP-122) and user-defined parameter or Sherlock parameters to calculate an acceleration factor.

Tip:

The following section explains which part properties Sherlock uses in the Semiconductor Wearout analysis: Tables: Required Part Properties per Analysis Type (p. 326).

Note that options 2 and 4, along with options 3 and 5, are essentially the same basic inputs with the difference between the two groups being that in the first the acceleration factors are provided, whereas in the later, the acceleration factors are computed.

The data for each component is provided in the parts list, most of which will be on the **Semiconductor Wearout** tab. However, some input items are found on other tabs. Service life data comes from the life cycle. The data required by each of the options and the location of that data is provided below. If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

6.4.1.1. Option 1: Prediction and User Field Data

All input for option 1 comes from the reliability goals of the life cycle definition and the data defined on the **Semiconductor Wearout** and **Die** tabs of the parts list.

6.4.1.1.1. Semiconductor Wearout Tab

Input Data Type – informational field must be one of **PREDICTION (TYPE 1) or FIELD DATA** for option 1

Reliability Data Source – optional informational field

Documents – optional informational field containing documents which reference the data

Reliability Metric and Units – required fields from which the life prediction is derived

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1.1.2. Die Tab

Process Node – required field, must be less than the selected analysis property size to be included in analysis

6.4.1.2. Option 2: Temperature Specific and Acceleration Factors

All input for option 2 comes from the reliability goals of the life cycle definition and the data defined on the **Semiconductor Wearout, Die, Elec**, and **Test** tabs of the parts list.

6.4.1.2.1. Semiconductor Wearout Tab

Input Data Type – informational field must be **PREDICTION (Type 2)** for option 2

Reliability Data Source – informational field

Documents – informational field containing documents which reference the data

Reliability Metric and Units – required fields

Source of AF – required field must be one of **MANUFACTURER** or **USER** for option 2

Temperature (Ea) – required field, Arrhenius based acceleration factor (Ea) in electron volts

Voltage Exponent – required field, power law based acceleration factor (n)

Frequency Exponent – required field, power law based acceleration factor (m)

6.4.1.2.2. Die Tab

Process Node – required field, must be less than the selected analysis property size to be included in analysis

6.4.1.2.3. Electrical Tab

Applied Voltage – required field, used to compute the voltage activation energy

Applied Frequency – required field, used to compute the frequency activation energy

6.4.1.2.4. Test Tab

Test Duration – required field, specifies the length of the test

Test Temperature – required field, used to compute the temperature activation energy

Test Voltage – required field, used to compute the voltage activation energy

Test Frequency – required field, used to compute the frequency activation energy

6.4.1.3. Option 3: Test Results and Acceleration Factors

All input for option 3 comes from the reliability goals of the life cycle definition and the data defined on the *Semiconductor Wearout, Die, Elec, and Test* tabs of the parts list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1.3.1. Semiconductor Wearout Tab

Input Data Type – informational field must be **TEST DATA** for option 3

Reliability Data Source – informational field

Documents – informational field containing documents which reference the data

Source of AF – required field must be one of **MANUFACTURER** or **USER** for option 3

Temperature (Ea) – required field, Arrhenius based acceleration factor (Ea) in electron volts

Voltage Exponent – required field, power law based acceleration factor (n)

Frequency Exponent – required field, power law based acceleration factor (m)

6.4.1.3.2. Die Tab

Process Node – required field, must be less than the selected analysis property size to be included in analysis

6.4.1.3.3. Electrical Tab

Applied Voltage – required field, used to compute the voltage activation energy

Applied Frequency – required field, used to compute the frequency activation energy

6.4.1.3.4. Test Tab

Test Duration – required field, specifies the length of the test

Number of Devices – required field, specifies the number of devices tested

Confidence Bounds – required field, specifies the confidence bounds for the test

Test Temperature – required field, used to compute the temperature activation energy

Test Voltage – required field, used to compute the voltage activation energy

Test Frequency – required field, used to compute the frequency activation energy

6.4.1.4. Option 4: Temperature Specific and Calculated Factors

All input for option 4 comes from the reliability goals of the life cycle definition and the data defined on the **Semiconductor Wearout, Die, Elec**, and **Test** tabs of the parts list. Additionally, the **Process Node** specified on the **Die** tab is used to reference parameters used for computing the acceleration factors for analysis.

6.4.1.4.1. Semiconductor Wearout Tab

Input Data Type – informational field must be **PREDICTION (Type 2)** for option 4

Reliability Data Source – informational field

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Documents – informational field containing documents which reference the data

Reliability Metric and Units – required fields

Source of AF – required field must be **SHERLOCK** for option 4

6.4.1.4.2. Die Tab

Process Node – required field, must be less than the selected analysis property size to be included in analysis; specifies set of acceleration factors components to use (see **Acceleration Factors** for more information).

6.4.1.4.3. Electrical Tab

Applied Voltage – required field, used to compute the voltage activation energy

Applied Frequency – required field, used to compute the frequency activation energy

Applied Duty Cycle – required field, used to compute acceleration factors

6.4.1.4.4. Test Tab

Test Duration – required field, specifies the length of the test

Test Temperature – required field, used to compute the temperature activation energy

Test Voltage – required field, used to compute the voltage activation energy

Test Frequency – required field, used to compute the frequency activation energy

Test Duty Cycle – required field, used to compute acceleration factors

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1.5. Option 5: Test Results and Calculated Factors

All input for option 5 comes from the reliability goals of the life cycle definition and the data defined on the **Semiconductor Wearout**, **Die**, **Elec**, and **Test** tabs of the parts list. Additionally, the **Process Node** specified on the **Die** tab is used to reference parameters used for computing the acceleration factors for analysis

6.4.1.5.1. Semiconductor Wearout Tab

Input Data Type – informational field must be **TEST DATA** for option 5

Reliability Data Source – informational field

Documents – informational field containing documents which reference the data

Source of AF – required field must be **SHERLOCK** for option 5

6.4.1.5.2. Die Tab

Process Node – required field, must be less than the selected analysis property size to be included in analysis; specifies set of acceleration factors components to use (see Acceleration Factors (p. 414)for more information).

6.4.1.5.3. Electrical Tab

Applied Voltage – required field, used to compute the voltage activation energy

Operating Frequency – required field, used to compute the frequency activation energy

Applied Duty Cycle – required field, used to compute acceleration factors

6.4.1.5.4. Test Tab

Test Duration – required field, specifies the length of the test

Number of Devices – required field, specifies the number of devices tested

Confidence Bounds – required field, specifies the confidence bounds for the test

Test Temperature – required field, used to compute the temperature activation energy

Test Voltage – required field, used to compute the voltage activation energy

Test Frequency – required field, used to compute the frequency activation energy

Test Duty Cycle – required field, used to compute acceleration factors

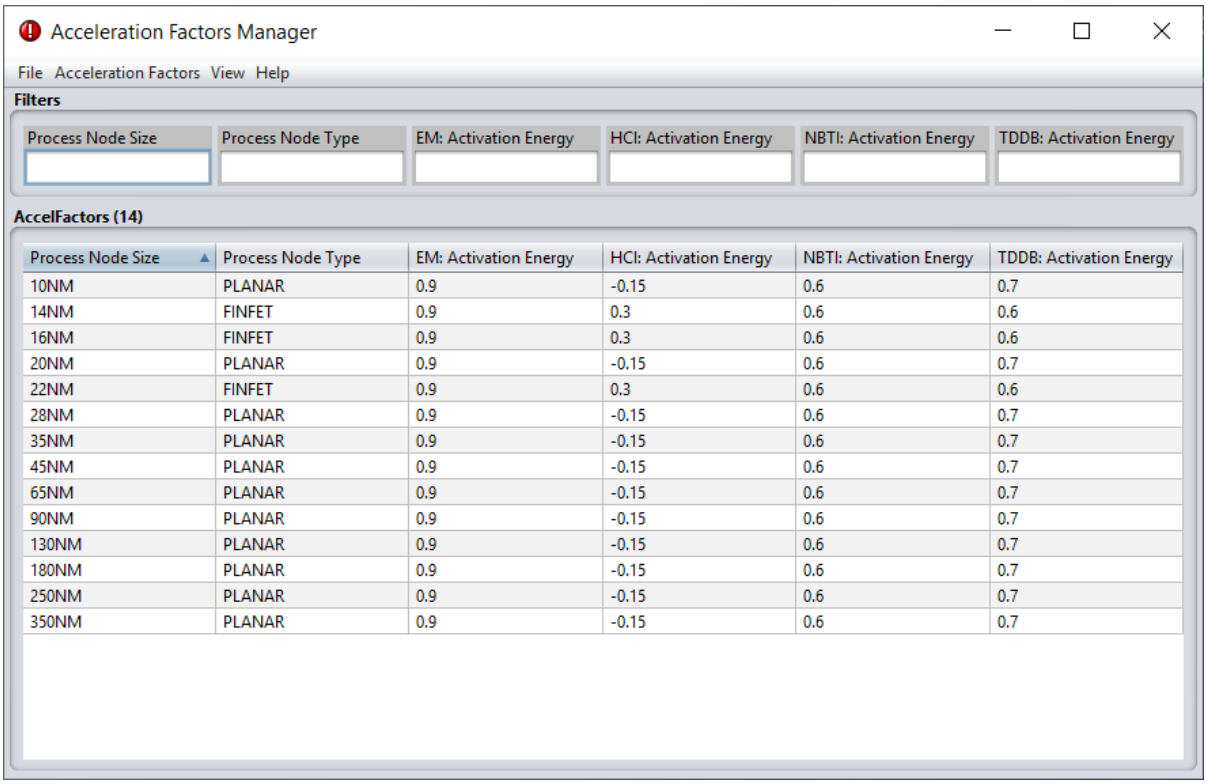
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.2. Acceleration Factors

Acceleration factor inputs are chosen based on the Die process node assigned to a given part. When the inputs are required during analysis, Sherlock will use the Die process node to look up those factors in the acceleration factors manager and provide them as input to the Semiconductor Wearout analysis process. The ability to view, create, or modify acceleration factor inputs, is provided with the **Acceleration Factors Manager** found in the **Libraries** menu of Sherlock.

6.4.2.1. Acceleration Factors Manager Listing

The main acceleration factors manager window is organized into two section. The top section provides a set of fields to allow filtering of the process nodes in the manager. The bottom table is a list of all process nodes and some key acceleration factors input data.



Process Node Size	Process Node Type	EM: Activation Energy	HCI: Activation Energy	NBTI: Activation Energy	TDDB: Activation Energy
10NM	PLANAR	0.9	-0.15	0.6	0.7
14NM	FINFET	0.9	0.3	0.6	0.6
16NM	FINFET	0.9	0.3	0.6	0.6
20NM	PLANAR	0.9	-0.15	0.6	0.7
22NM	FINFET	0.9	0.3	0.6	0.6
28NM	PLANAR	0.9	-0.15	0.6	0.7
35NM	PLANAR	0.9	-0.15	0.6	0.7
45NM	PLANAR	0.9	-0.15	0.6	0.7
65NM	PLANAR	0.9	-0.15	0.6	0.7
90NM	PLANAR	0.9	-0.15	0.6	0.7
130NM	PLANAR	0.9	-0.15	0.6	0.7
180NM	PLANAR	0.9	-0.15	0.6	0.7
250NM	PLANAR	0.9	-0.15	0.6	0.7
350NM	PLANAR	0.9	-0.15	0.6	0.7

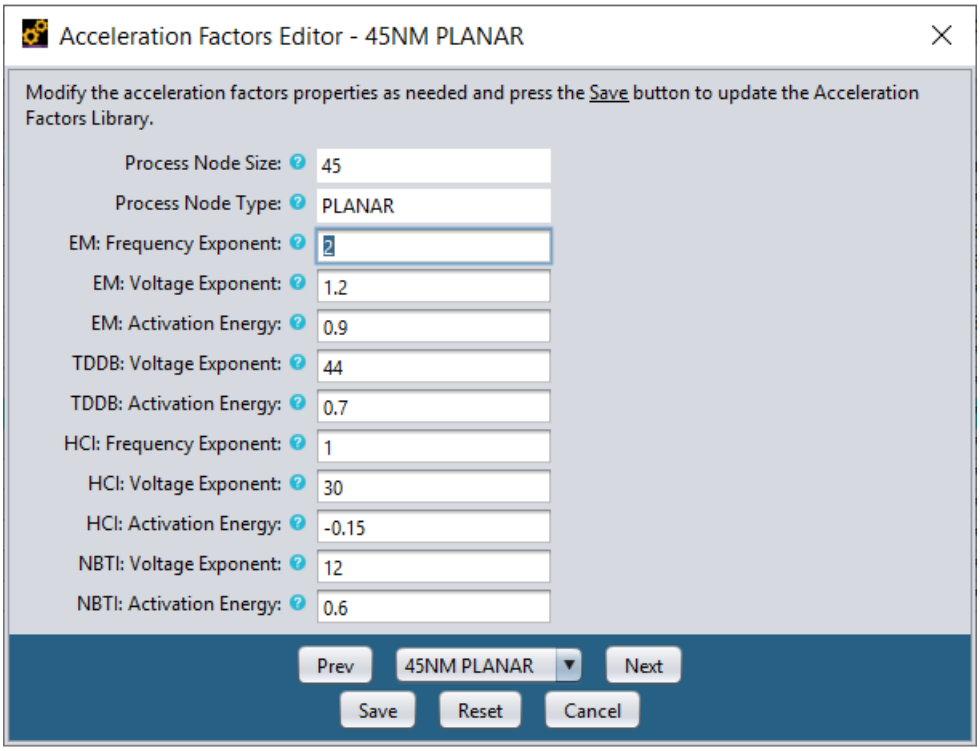
6.4.2.2. Viewing and Editing Acceleration Factors

To view the acceleration factor inputs assigned to a given process node, you may double-click the desired process node from the table. You may also right-click the process node and select **Edit Selected Acceleration Factors** from the context menu.

After selecting a process node to edit, a new dialog window, shown below, will appear with a listing of all available acceleration factors input data available for the selected process node.

After making any desired process node changes, use the **Save** button to save the changes to the local copy of the Sherlock acceleration factors library. See the section on User Data Files (p. 166) for more information about the location and format of the local acceleration factors library. If you don't wish to save any changes or are simply viewing the acceleration factors definition, select the Cancel button when done. The **Reset** button will revert any changes you have made to the definition since opening the dialog to the original settings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you wish to view additional process node definitions, you may do so without closing the editor dialog window by simply selecting the process node from the selector at the bottom of the window or using the **Prev** and **Next** buttons. The process node selector and the **Prev** and **Next** buttons are limited to the filtered process nodes from the acceleration factors manager itself. If no filters were applied, then the entire list of process nodes is available.

6.4.2.3. Modifying Acceleration Factors

To add a new process node acceleration factor, select **Add Acceleration Factors** from the **Acceleration Factors** menu, or right-click in the table listing and select **Add Acceleration Factors** from the menu. The editor dialog will appear with all properties empty, ready for a new process node acceleration factors to be defined. When you are complete defining a process node acceleration factor, click the **Save** button to add the new definition.

To copy a process node acceleration factor, right-click the acceleration factor to be copied from the table listing and select **Copy Acceleration Factors** from the menu. A copy of the process node acceleration factors will be made. Provide a process node size and type for the copied definition and make any changes desired to the properties, then click the **Save** button to add the copied process node acceleration factor.

Any user-defined process node acceleration factor that is not a Sherlock system definition may be deleted. Right-click the process node acceleration factor to be delete from the table listing and select Delete Selected Acceleration Factors from the menu. If the **Delete Selected Acceleration Factors** item is not available, this indicates that it is not a user-defined process node acceleration factor.

If a system process node acceleration factors has been modified, the **Delete Selected Acceleration Factors** menu item will be available. However, it will not actually delete the definition. Deleting a

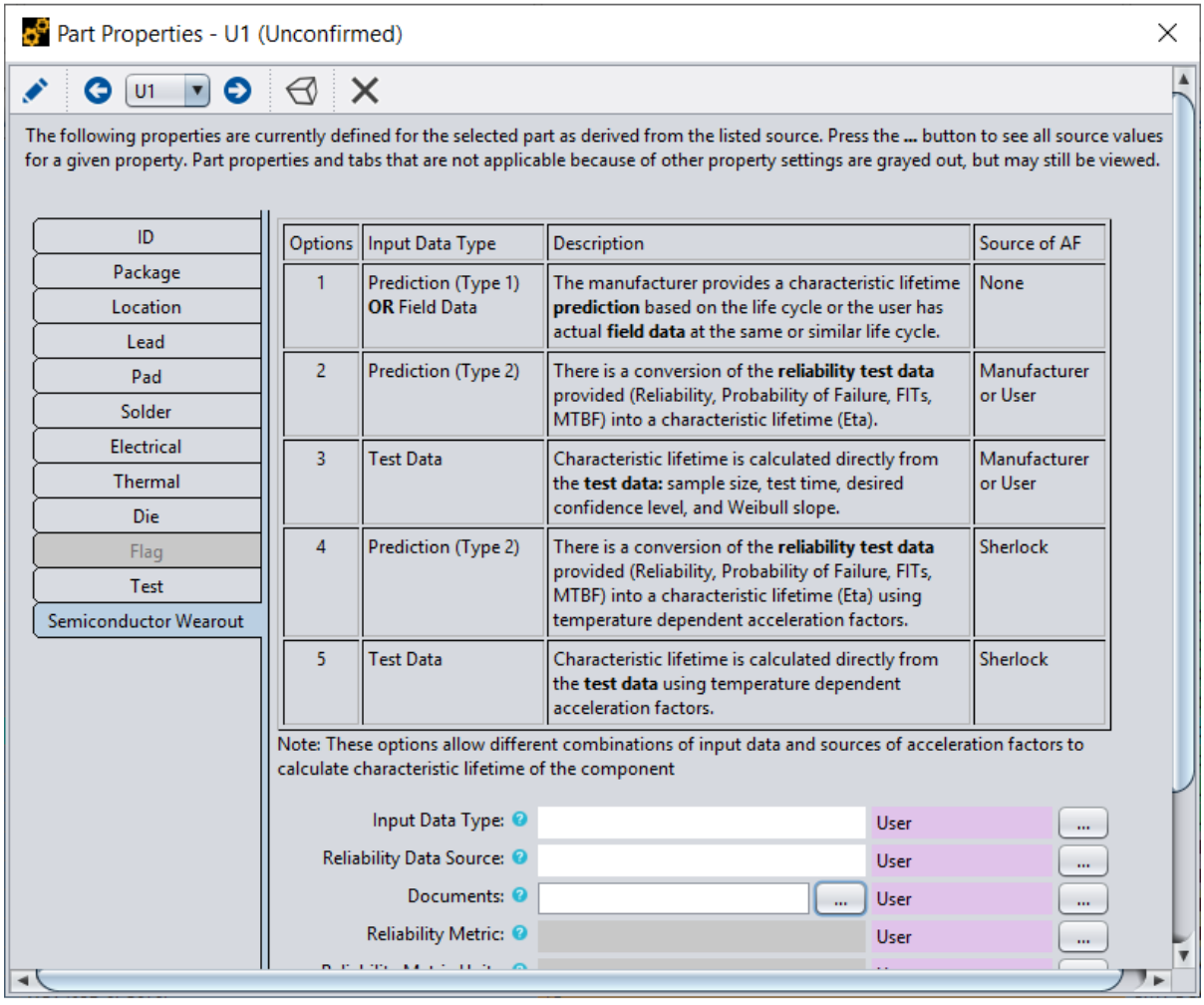
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

system process node acceleration factor that has user-defined properties will simply remove all the user-defined properties and allow the definition to revert to being completely system defined.

6.4.3. Documents

Ansys Sherlock provides a repository of documents received from a supplier for a part, related to semiconductor wearout. Each repository is specific to a circuit card and are not shared amongst projects. However duplicate parts within a project may reference the same set of documents.

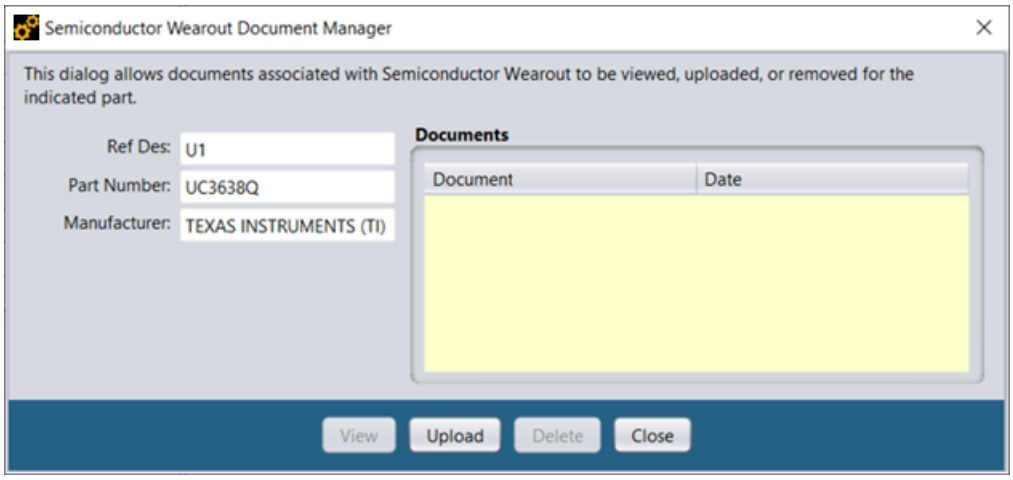
Assigned documents may be viewed from the **Semiconductor Wearout** tab of the **Parts List**. The **Documents** field will list all uploaded documents. To view any of the documents, double-click the desired document from the field.



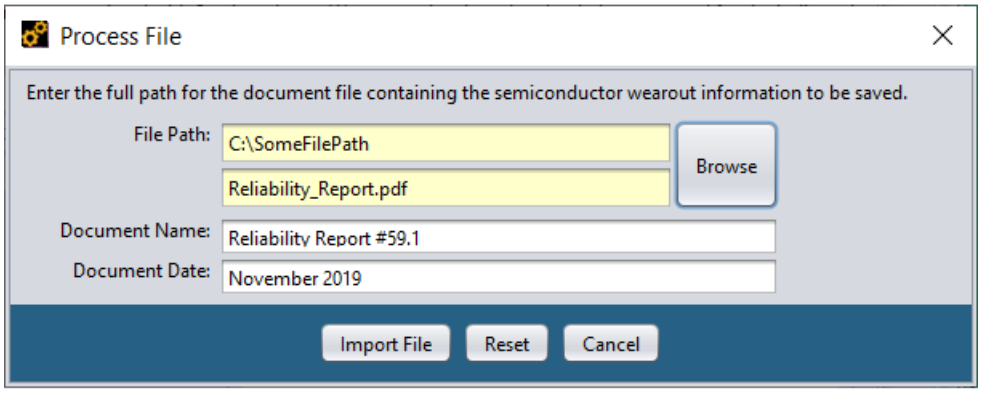
Alternatively, you may click the **"..."** button located next to the available documents. This will open the **Document Manager** shown below. When the underlying part is open for editing, the document manager will allow for documents to be uploaded, deleted or viewed. When the underlying part is being viewed only, the document manager will only allow a given document to be viewed.

To view a given document right-click click or double-click the desired document and Sherlock will open the selected document with the default application assigned by the operating system for viewing such documents. To upload a new document, make sure that the part editor is enabled, or else you will not be able to upload a document.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select the **Upload** button and a dialog will appear to allow for selection of the document to be uploaded along with assigning a name and a document date. If no name is specified, the document will be named the same as the file name of the document.

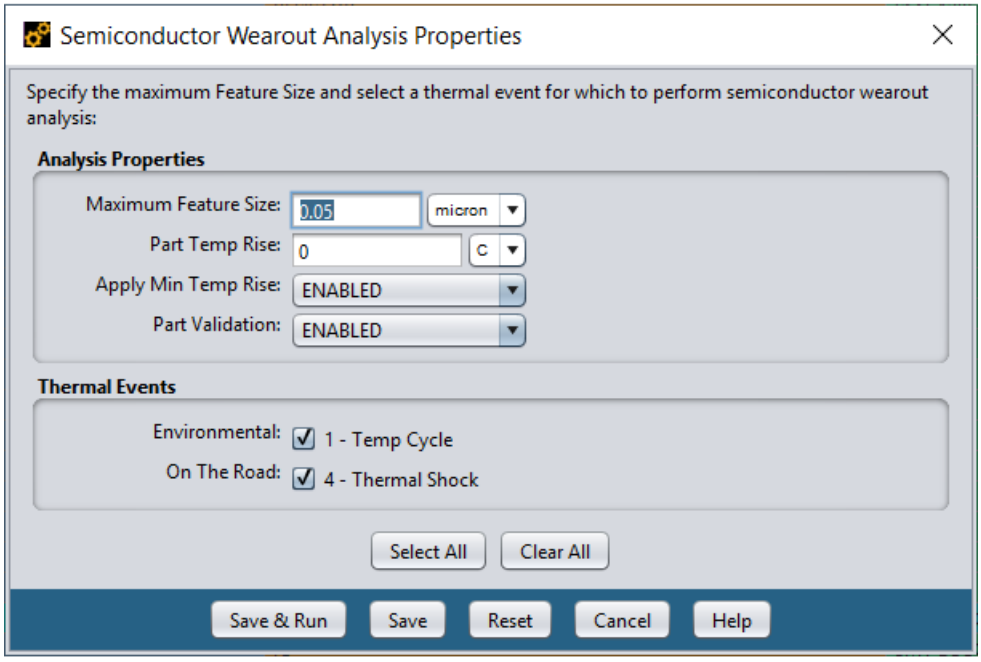


When a document is uploaded and is assigned to another part, a warning dialog will appear and allow the document to be uploaded and assigned to both parts.

6.4.4. Analysis

To perform semiconductor wearout analysis, locate the **Semiconductor Wearout** item from the **Analysis** folder of the circuit card for which to perform analysis. Right-click and select **Run Analysis Task** from the popup menu. The analysis properties dialog will appear. Analysis is performed on IC components with a feature size less than 0.05 microns by default. With the analysis properties dialog, you may change the maximum feature size and to select the required **Thermal Event** to use while performing analysis. Each temperature defined by the selected **Thermal Events** is used as input for **Temperature Specific** and **Test Results** semiconductor wearout data types. The acceleration factors are computed using this temperature and weighted based on the amount of the time the temperature is active.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6.4.4.1. Analysis Results

The Semiconductor Wearout analysis module generates the following results:

- Summary panel showing overall scores, reliability goals, and analysis statistics
- Life prediction curve for the circuit bard based on the part with the worst calculated life prediction
- Result table show key properties, characteristic life, and score for each analyzed part
- Graphical layers showing the score for each part analyzed

6.4.4.1.1. Analysis Computations

For **option 1**, the failure rate is computed from the provided reliability metric, the characteristic lifetime is computed, and the life prediction is computed.

For **option 2**, the failure rate is computed from the provided reliability metric, then acceleration factors are applied to the failure rate to determine the characteristic lifetime.

For **option 3**, the provided test results are converted to reliability and failure rate data, then the acceleration factors are applied to the failure rate to determine the characteristic lifetime.

Options 4 and 5 are done exactly as options 2 and 3 respectively, however, instead of using the provided acceleration factors, the acceleration factors are computed by using acceleration factor input components referenced by the selected **Die Process Node** and uses industry generic models (JEDEC JEP-122) and user-defined parameter or Sherlock parameters to calculate an acceleration factor.

Failure mechanisms that determine long term reliability of ICs are intrinsic in nature and are driven by material degradation over time. Degradation mechanisms at the wear out stage for a semiconductor component are; Time Dependent Dielectric Breakdown (TDDB), Negative Bias

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Instability (NBTI), Hot Carrier Injection (HCI) and Electromigration (EM). Acceleration factors for options 4 and 5 are calculated based on these four failure mechanisms. One or more of these mechanisms are likely to dominate and impact the lifetime of an IC and the overall system.

6.5. Solder Fatigue Analysis

Solder joints provide electrical, thermal, and mechanical connections between electronic components and a printed board. During changes in temperature, the component and printed board will expand or contract by dissimilar amounts due to differences in the coefficient of thermal expansion (CTE). This difference in expansion or contraction will place the second-level solder joint under a shear load. Repeated exposure to temperature changes, such as power on/off or diurnal cycles, can introduce damage into the bulk solder. With each additional temperature cycle this damage accumulates, leading to crack propagation and eventual failure of the solder joint.

Thermo-mechanical solder joint fatigue is influenced by maximum temperature, minimum temperature, dwell time at maximum temperature, component design, component material properties, solder joint geometry, solder joint material, printed board thickness, and printed board in-plane material properties. Sherlock calculates a time to failure using strain energy, which requires determining the applied force, the strain range, and then extrapolating cycles-to-failure from the derived strain energy.

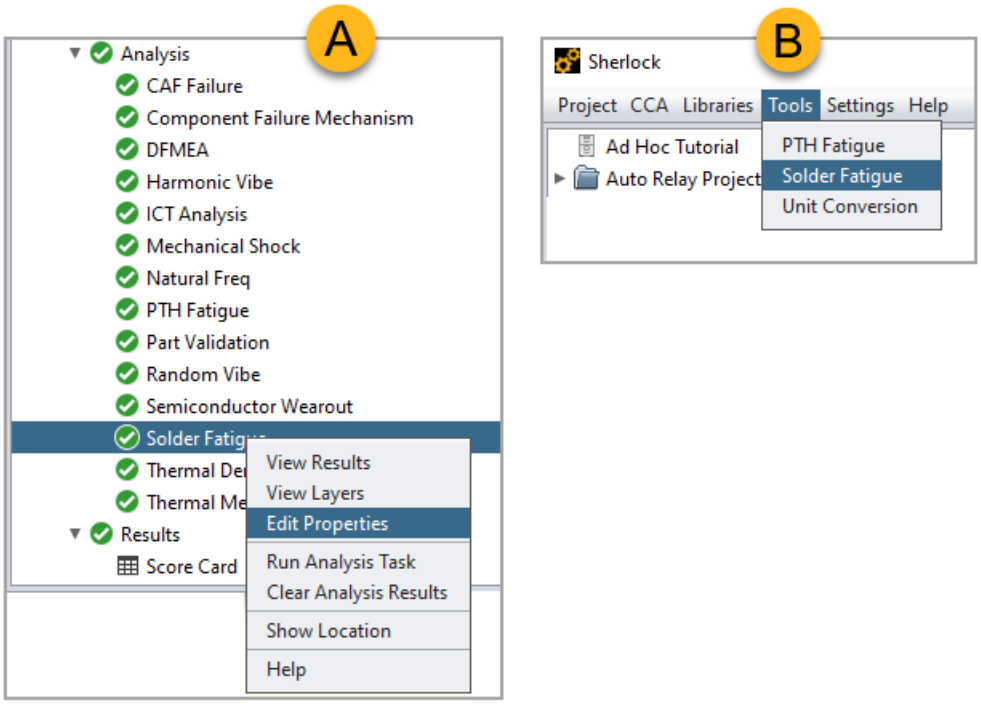
Note:

- Currently, Sherlock does not calculate reliability for connectors.
- The ceramic chip solder fatigue model may not be valid for reverse termination components.
- When calculating solder joint fatigue predictions for stub leads, Sherlock uses a leaded model formula based on the Euler-Bernoulli beam bending theory. Due to the geometric simplifications in the software, Sherlock models the stub leads as short beams which do not meet the assumptions in the Euler-Bernoulli beam theory. When running a Solder Fatigue, you may enter the lead stiffness manually for stub leads. This will override the computed stiffness.

Sherlock offers three tools for running a Solder Fatigue Analysis which are descrbed in subsequent sections of this chapter:

1. The Solder Fatigue Analysis Module provides a comprehensive board-level analysis. To set up an analysis, right-click **Solder Fatigue** in the **Project Tree** (A, below) and select **Edit Properties**. See the section Solder Fatigue Analysis Module (p. 420).
2. Sherlock's Solder Fatigue Analysis Module allows you to input thermal analysis results generated in Ansys Electronics Desktop (AEDT) - Icepak (`.tmap` files). The workflow is presented in Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis (p. 426).
3. The **Solder Fatigue Tool** allows you to run a quick solder fatigue analysis on a specific **Package Type**, calculating the number of cycles-to-failure based on the properties of the solder, the board, the component package, and its leads. To open the tool, select **Tools > Solder Fatigue** from the **Main Menu** (B). See the sectionSolder Fatigue Tool (p. 433).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6.6. Solder Fatigue Analysis Module

Topics covered in this section:

6.6.1. Input Data
6.6.2. Solder Fatigue Properties Dialog
6.6.3. Pad Properties
6.6.4. Editing Solder Properties
6.6.5. Supported Solder Models

6.6.1. Input Data

The **Solder Fatigue Analysis Module** uses the following input data for the analysis calculations:

- Life Cycle Reliability Goals
- Parts List

> **Tip:**
>
> The following section explains which part properties Sherlock uses in the Solder Fatigue Analysis Module: Tables: Required Part Properties per Analysis Type (p. 326).

- Circuit card mechanical properties (stackup data)
- Component sizes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

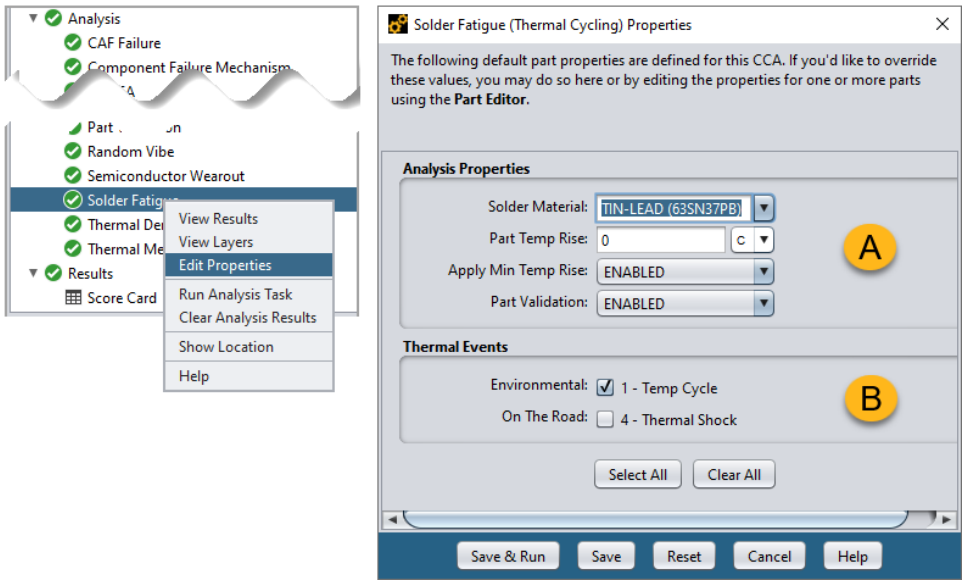
- Solder properties
- Thermal Events and associated Thermal Maps

> **Tip:**
>
> Regarding Thermal Event inputs, you can import analysis results from Ansys Icepak. See the section below, Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis (p. 426).

If any of the input data listed above is changed, Sherlock will automatically clear all results for the analysis module.

6.6.2. Solder Fatigue Properties Dialog

The Solder Fatigue Analysis Module allows you to configure various properties for your analysis using the **Solder Fatigue (Thermal Cycling) Properties** dialog. To open the module, right-click **Solder Fatigue** in the **Project Tree** and select **Edit Properties**.



To set up your analysis, configure the **Analysis Properties** (A):

- **Solder Material**: Sherlock applies the **Solder Material** selected in the dropdown menu to any component in the **Parts List** that does *not* already have a **Solder Material** assigned in its Part Properties. (See Editing Solder Properties (p. 424) below.) This allows you to change the solder type for the entire circuit card, while also assigning specific solder types to individual parts, depending on how the circuit card is assembled.

> **Note:**
>
> Sherlock makes use of an internal library that defines mechanical and chemical properties for a collection of solder types. You can override or augment solder data with the **Solder Manager.** See Solder Management (p. 145) for more details.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Part Temp Rise**: This value defines component temperature rise (that is, the number of degrees above ambient temperature). However, Sherlock uses this value for a component only when other data sources are not available. Sherlock prioritizes temperature data sources in the following order:

  1. If a Thermal Map has been assigned to the Thermal Event, Sherlock will use the average temperature of the map region where the part is located.

  2. The **Ambient Temperature** defined in the **Thermal Event** plus the **Part Temp Rise** assigned to the part in the **Parts List**.

  3. The **Ambient Temperature** defined in the **Thermal Event** plus the default **Part Temp Rise** assigned in **Solder Fatigue Analysis Properties**.

- **Apply Min Temp Rise**: (**Enabled** or **Disabled**) When **Enabled** (default setting), Sherlock applies the **Part Temp Rise** value to the **Minimum Temperature** defined in the Thermal Cycle.

- **Part Validation**: (**Enabled** or **Disabled**) When **Enabled** (default setting), Sherlock runs the Part Validation Analysis (p. 263) prior to the analysis if a successful validation has not yet been run.

Thermal Events (B)

In the **Thermal Events** panel, select which Thermal Event(s) will be applied during the analysis. For a thorough guide to Thermal Events, see Project Life Cycles (p. 343) and the subsection Thermal Event (p. 344).

6.6.3. Pad Properties

This section describes how Sherlock models component pads and how you can edit their properties using the **Update Pad Properties** feature.

6.6.3.1. How Sherlock Models Component Pads

6.6.3.2. Updating Pad Properties

6.6.3.1. How Sherlock Models Component Pads

Sherlock uses a variety of models to predict solder fatigue for individual parts based on the **Package Type**. By default, Sherlock estimates the **Pad Length** and **Pad Width** properties for each part based on the **Package Type** and its dimensions. For the following **Package Types**, however, the solder fatigue models depend specifically on the size of the pads to which the component is soldered:

- Ball Grid Array (BGA / CBGA)
- Chip Component (CC / SMD)
- Lead-less Chip Carrier (LCCC / CLCC / LCC)
- Bottom Terminated Components (BTC)

Sherlock automatically extracts pad data whenever the top or bottom copper layer files are parsed. The pad data is stored as binary data files in the meta-data folder within the Sherlock **CCA Folder**. No user inputs are required for the extraction process. After pad data has been extracted, you can

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

use the **Update Pad Properties** feature (see Updating Pad Properties (p. 423) below) whenever needed.

Note:

Sherlock's pad-size estimations provide only a rough estimate for solder fatigue analysis purposes. You are encouraged to review the pad dimensions for all parts using data from their specific PCB layout.

6.6.3.2. Updating Pad Properties

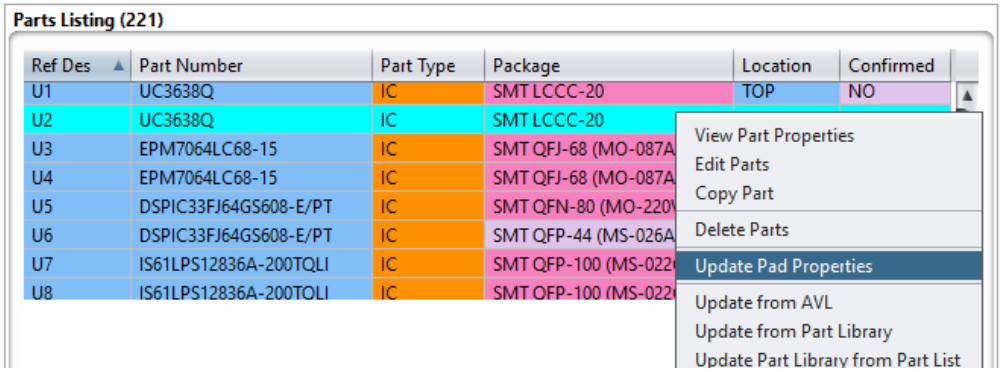
To help automate the process of entering accurate pad dimensions, you can use the **Update Pad Properties** feature to extract pad dimensions from the top and bottom copper layers defined for the PCB. The pad dimensions are stored in the **Parts List** for each updated part. Sherlock assigns these updated values in the **Parts List** a higher priority than data from the **Package Library** or **Part Libraries**.

You can use the **Update Pad Properties** feature on a single part or on multiple parts simultaneously. The feature can be run from the **Parts List** or the **Parts Editor**.

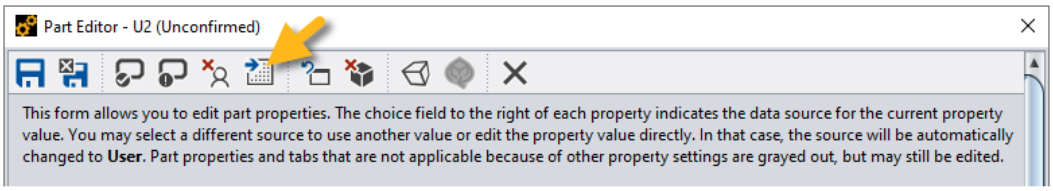
- From the **Parts List**, select one or more parts and click the **Update Pad Properties** icon.



- From the **Parts List**, right-click any selected part(s) and choose **Update Pad Properties** from the context menu.



- From the **Parts Editor**, click the **Update Pad Properties** icon.



Whatever method used to invoke the **Update Pad Properties** feature, Sherlock automatically updates the selected part(s) as follows:

- The current location and package size are used to find all covered PCB pads.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Extraneous pads, such as via pads, are ignored.
- The dimensions of the smallest PCB pad found under a part are assigned to that part.
- The PCB is designated as the source for pad property data.

When you invoke **Update Pad Properties** from the **Parts List**, Sherlock automatically saves all part updates and displays a message indicating the number of parts updated.

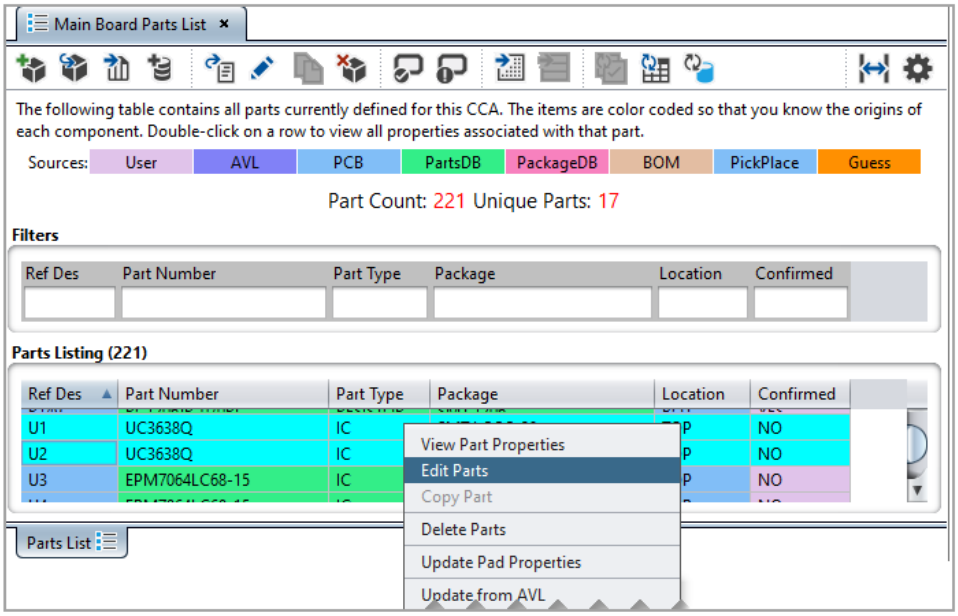
When invoked from the **Part Editor**, Sherlock will update the **Pad** (or **Ball**) tab to reflect the newly assigned properties and will display a status message. Like any other **Part Editor** changes, the updates are temporary. You must click **Apply** or **Save** to update the **Parts List** itself.

Note:

QFN and DFN components (Quad Flat No-Leads and Dual Flat No-Leads) are not affected by the **Update Pad Properties** feature. Their pad properties are based on component-side pads, not board-side pads.

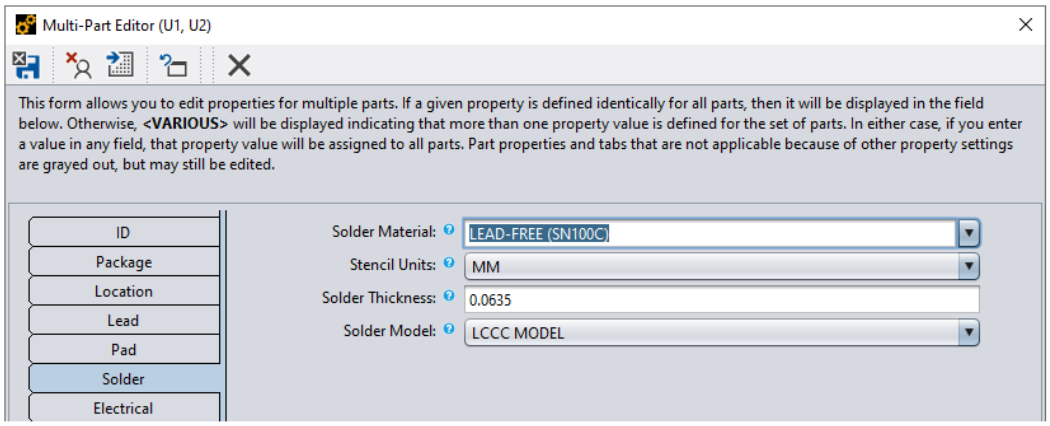
6.6.4. Editing Solder Properties

To edit solder properties, right-click a part or parts from the **Parts List** and select **Edit Parts**.



In this example, parts U1 and U2 have been selected and appear in the **Multi-Part Editor**. To edit the solder for these parts, select the **Solder** tab as shown below. Then edit the **Solder Material** by selecting, for example, LEAD-FREE (SN100C) from the dropdown menu. Then click the  icon to save the change and exit the editor.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



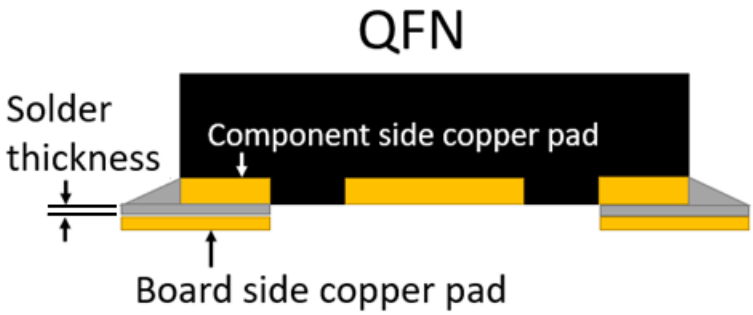
Solder Properties:

- **Solder Material**: The solder-type used to connect the part to the PCB.

  Supported solder materials are defined in Sherlock's solder data file, but you have the option of creating a user-defined data file as well. (Refer to Solder Management (p. 145) and User Data Files and Documents (p. 166).)

- **Stencil Units**: Unit of length for all solder properties. Supported values are **IN**, **MM**, **M**, **MIL**, **MICRON** and **NM**.

- **Solder Thickness**: Thickness (height) of the solder joint that connects the part to the PCB.



- **Solder Model**: The solder model Sherlock will use in the analysis. Supported values are **BGA MODEL**, **CBGA MODEL**, **CC MODEL**, **CGA MODEL**, **DIE MODEL**, **IMS MODEL**, **LCCC MODEL**, **LEADED MODEL**, **QFN MODEL**, and **THRUHOLE MODEL**.

  By default, this is based on **Package Type**, but you can override Sherlock's selection. (The table below explains how Sherlock determines the default model. (p. 426)) When the **Solder Model** for a given component differs from what Sherlock expects, Sherlock will generate a warning during the analysis for each affected component.

6.6.5. Supported Solder Models

When performing a Solder Fatigue Analysis, Sherlock automatically selects the **Solder Model** for each component based on its **Package Type**. (The table below explains how Sherlock determines the default model. (p. 426)) However, you can override Sherlock's model selection if you wish.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To assign a different **Solder Model** to a component, use the **Part Editor** as shown above (p. 424). Typically, you would assign a **Package Type** (from the **Package** tab) and let Sherlock determine what **Solder Model** to use. However, you may change the **Solder Model** directly if you wish.

Note:

When the **Solder Model** for a given component differs from what Sherlock expects (based on the **Package Type**), Sherlock will generate a warning during the analysis for each affected component.

The table below lists the **Solder Models** Sherlock supports. The right-hand column explains the logic Sherlock follows to determine when a particular model is used as the default **Solder Model**.

Default Solder Model	Sherlock Chooses this as the Default Solder Model when...
BGA MODEL	...the **Package Type** is BGA.
CBGA MODEL	...the **Package Type** is CBGA.
CC MODEL	...the **Package Type** is CC or MELF. Or it isn't a leaded SMT and has a **Lead Count** of 2 and **Lead Geometry** of Leadless.
CGA MODEL	...the **Package Type** is CGA.
DIE MODEL	...the **Package Type** is DIE.
IMS MODEL	...the **Package Type** is BTC.
LCCC MODEL	...the **Package Type** is LCCC.
LEADED MODEL	...the **Package Type** is SON or USON (or **Lead Geometry** isn't Leadless or Thruhole).
QFN MODEL	...the **Package Type** is QFN or DFN.
THRUHOLE MODEL	...the **Package Type** isn't BGA, CBGA, CGA and **Lead Geometry** is Thruhole.

6.7. Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis

Sherlock's Solder Fatigue Analysis Module allows you to input thermal analysis results generated in Ansys Electronics Desktop (AEDT) - Icepak (`.tmap` files). Sherlock exports the following (including all associated materials): stackup layers, components, pads, traces, cutouts, vias, drill holes, and net names–although Sherlock does not associate net names with geometry.

The steps below guide you through a typical workflow. The instructions assume you have already created or imported a project in Sherlock and you are familiar with running an analysis in Ansys Icepak. And you should be familiar with the contents of the previous section describing the Solder Fatigue Analysis Module (p. 420).

Note:

When exporting a CCA to AEDT, AEDT may become unresponsive when the CCA has a large number of components (>9000 approximately).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Topics in this section:

6.7.1. Workflow Summary

6.7.2. TMAP File Specifications

6.7.1. Workflow Summary

1. **Prepare the Sherlock CCA for Export:** Before exporting the CCA, you should verify your Sherlock project is properly formatted for a successful export.

   A. **Configure Part Properties:** Sherlock exports the following part properties to AEDT. The values are blank by default in the **Part Properties Editor/Viewer**. You must specify the values manually.

   - **Rjb** (junction resistance to board): Located in the **Thermal** tab in the **Part Properties Editor/Viewer**.
   - **Rjc** (junction resistance to case): Located in the **Thermal** tab.
   - **Applied Power**: Located in the **Electrical** tab.
   - **Max Rated Power**: Located in the **Electrical** tab.

   B. **Generate Traces for all Image Layers:** In order to export traces represented by image layers in Sherlock, you must first generate traces for that layer prior to exporting to AEDT. See Trace Model Generation (p. 497).

   Note:

   For non-image layers, Sherlock uses circles and arcs (instead of discretized small lines) when exporting traces to AEDT. Some traces are exported as lines with BendType, StartCapType, and EndCapType properties that are shown in AEDT.

   Note:

   When generating trace models (p. 497), Sherlock automatically clips traces at the board outline, but this feature is not supported in boards exported to AEDT.

   C. **Configure Material Properties:** When exporting to AEDT, it's helpful to understand how Sherlock handles certain material properties:

   - When exporting a component composed of multiple materials, Sherlock calculates the anisotropic thermal conductivity properties and stores the values with the material in AEDT.
   - You can specify the **Lead Frame Percentage** property for leaded parts and packages. This property pertains to the component's lead frame layer, the layer directly beneath the die.

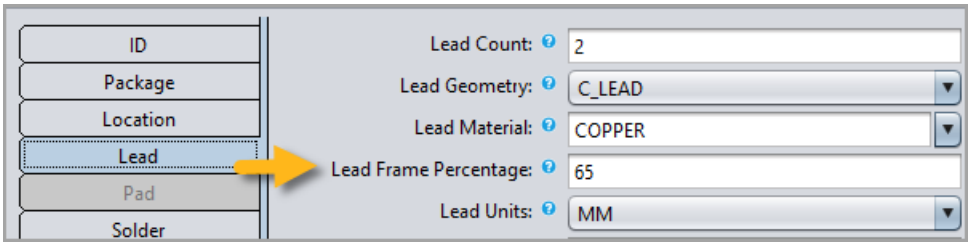
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The value entered represents the percentage of the lead frame layer that consists of the lead material.

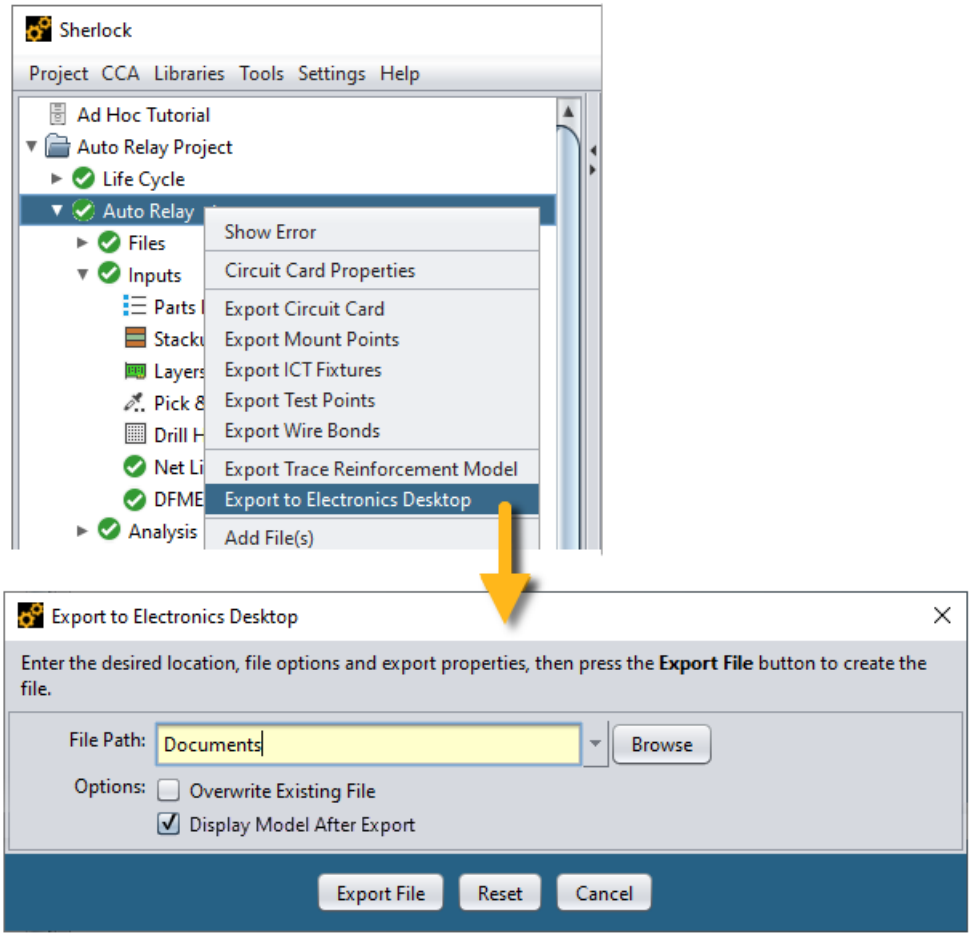
Note:

The **Lead Frame Percentage** property is applicable to surface mounted packages that are leaded. It does not apply to packages with through-hole leads.

Sherlock does not use this property in its own analyses but will export the information to AEDT. As shown below, you can find the **Lead Frame Percentage** property under the **Lead** tab in the **Part Viewer**, the **Part Editor**, the **Package Viewer**, and **Package Editor**. The default value is 65%.



2. **In Sherlock, Export the CCA:** After you have confirmed your CCA is properly formatted for a successful export, right-click the CCA in the **Project Tree** and select **Export to Electronics Desktop** in the context menu. This opens the **Export to Electronics Desktop** dialog.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export Options:

- **File Path**: Use the **Browse** button to navigate to the folder where you want the model to be exported. In the selected folder, Sherlock will create a folder named after the CCA being exported with .aedb appended. The resulting file structure will be

  ```
  <user selected folder>/<CCA name>.aedb/edb.def
  ```

  where `edb.def` is an exported file.

- **Overwrite Existing File**: If checked, Sherlock will automatically overwrite an existing .aedb folder that has the same name as the one being generated. If unchecked, Sherlock will ask for confirmation before overwriting the folder.

- **Display Model After Export**: This option appears only if you have AEDT installed. If selected, AEDT will automatically launch after the export is complete. The installed version of AEDT must match your version of Sherlock. For example: If you are using Sherlock 2025 R1, AEDT must also be 2025 R1.

3. **Run the Thermal Analysis in IcePak:** These instructions assume you are already familiar with running an analysis in Ansys Icepak. Refer to the Ansys Icepak User's Guide.

4. **Export the .tmap file from IcePak.**

   Tip:

   If your `.tmap` file is has multiple boards or subassemblies, review the section below TMAP File Specifications (p. 431) to make sure the file is formatted properly. The section can also be helpful when troubleshooting issues with the export.

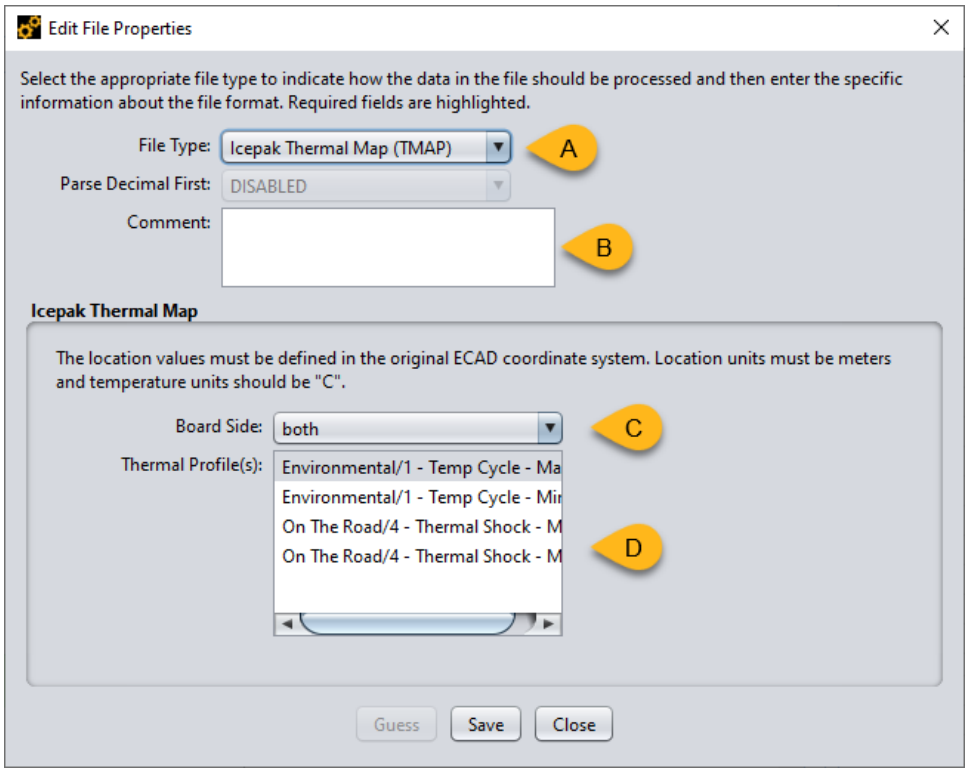
5. **Add the .tmap File to Your Sherlock Project:** In your Sherlock project, add the `.tmap` file to the list of CCA files by using the **Add File(s)** command or by dragging and dropping the file onto your project's list of CCA files (as described in Adding Files to a Circuit Card Assembly (p. 100)).

   Note:

   After Sherlock processes the file, it will display an error icon () in the **Project Tree**. This is normal. Sherlock is letting you know you must assign a Thermal Event as explained below.

6. **Configure the File Properties:** Once Sherlock has processed the file, right-click the file icon in the **Project Tree** and select **Edit Properties** from the context menu. This opens the **Edit File Properties** dialog shown below where you can set the following properties:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



File Properties:

- **File Type** (A): Sherlock automatically sets the file-type to **Icepak Thermal Map** when it sees `.tmap` file extension.
- **Comment** (B): (optional) A text field where you can attach comments to the file.
- **Board Side** (C): (**top**, **bot**, or **both**) Select whether the thermal map represents the top, bottom, or both sides of the circuit board.
- **Thermal Profile(s)** (D): Here, all the Thermal Profiles you have created for your project are listed. The thermal image must be attached to one of these Profiles. To select, click and highlight the desired Profile.

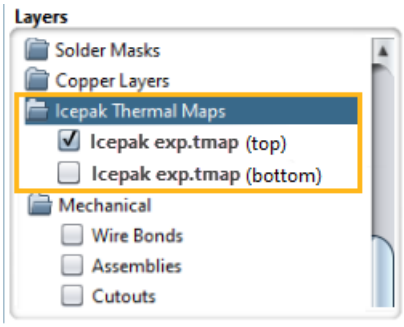
When you have finished setting the file properties, click **Save**.

7. **Run the Solder Fatigue Analysis:** Once the `.tmap` file has been processed and configured, Sherlock will apply the thermal map the next time you run a Solder Fatigue Analysis using the associated Thermal Profile. Also, Sherlock creates an **Icepak Thermal Maps** folder in the **Layer Viewer**, as shown below, allowing you to see your circuit board with a visual overlay of the thermal map.

Note:

When you select **both** for **Board Side** when configuring the `.tmap` file, the **Layer Viewer** will append **(top)** and **(bottom)** to the `.tmap` entries.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6.7.2. TMAP File Specifications

Review this section to make sure your exported `.tmap` file is properly formatted for a successful import into Sherlock and for troubleshooting export issues.

Tip:

If your file contains multiple boards, refer to the section below: *TMAP Files Containing Multiple Boards*. For boards with subassemblies, you will find helpful content under *Other Considerations*.

TMAP File Structure

- There are two sections to the `.tmap` file: **Nodes**: Specifies coordinates for node locations. **Faces**: Specifies the nodes of the face, the coordinate of the center of the face polygon, and the temperature of the thermal map for the face.
- Units: Coordinates are in meters, temperature in Celcius.
- Each node is defined using the node number and the node location (x, y, and z coordinates).
- Each face is defined using the center coordinate (x, y, and z coordinates of the center of the face), the temperature, and the node numbers (3 or 4). These refer to the node numbers defined in the **nodes** section.

  Example:

  [nodes]

  <node number>, x, y, z

  ...

  [faces]

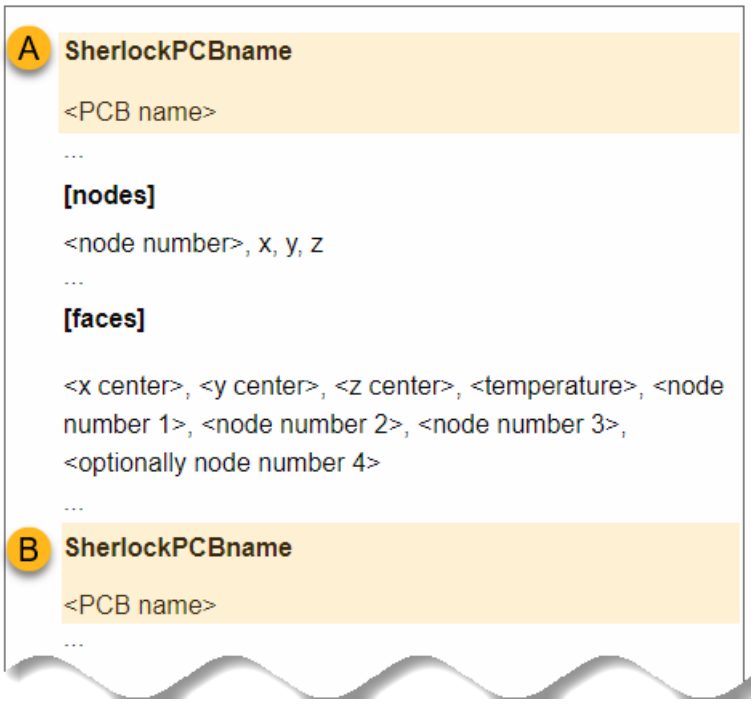
  <x center>, <y center>, <z center>, <temperature>, <node number 1>, <node number 2>, <node number 3>, <optionally node number 4>

  ...

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TMAP Files Containing Multiple Boards

A `.tmap` file containing multiple boards requires special consideration. The image below shows the `.tmap` file structure containing data for multiple boards. The **SherlockPCBname** tag (A and B) identifies the board name, and the board data follows. In order for Sherlock to associate the correct board in the `.tmap` file with your CCA in Sherlock, its name must match the board name in the Icepak file. The check is case sensitive.



If the names do not match, you must do one of the following before importing the data:

- Edit the name of the board in Icepak so it matches the name of the Sherlock CCA.

  Note:

  Sherlock accepts spaces in CCA names, but Icepak/AEDT does *not* accept spaces in board names.

- Edit the name of the CCA in Sherlock so it matches the board name in Icepak. (In the Sherlock **Project Tree**, right-click the CCA and select **Circuit Card Properties** in the context menu. This opens the **Edit Circuit Card Details** window where you can modify the **Circuit Card Name**.)
- Another option is to edit the `.tmap` file by removing data that does not apply to the Sherlock CCA. This option is not ideal because there is a risk of introducing errors into the file.

When the naming conventions are configured properly, Sherlock will add the data pertinent to the CCA and ignore unrelated data.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other Considerations

- If you add a `.tmap` file to a CCA that has subassemblies and the `.tmap` file contains corresponding data, Sherlock imports the data by adding a `.tmap` file to each subassembly of the CCA.
- When a `.tmap` file is added, if any of the <CCA name> items match the CCA to which the file is added, then the new file will contain only the **[nodes]** and **[faces]** sections pertaining to that CCA. The same is true for subassemblies.
- If the contents of the `.tmap` file start with the **SherlockPCBname** tag but there is no PCB name in the file that matches the CCA name in Sherlock, then the `.tmap` file being added will not be modified and Sherlock will indicate an error has occurred. To get more information on the error, right-click the file and select **Show Error**.

6.8. Solder Fatigue Tool

Sherlock provides a **Solder Fatigue Tool** (A, below) for quickly computing **Cycles to Failure** for a selected package. It takes into account the following:

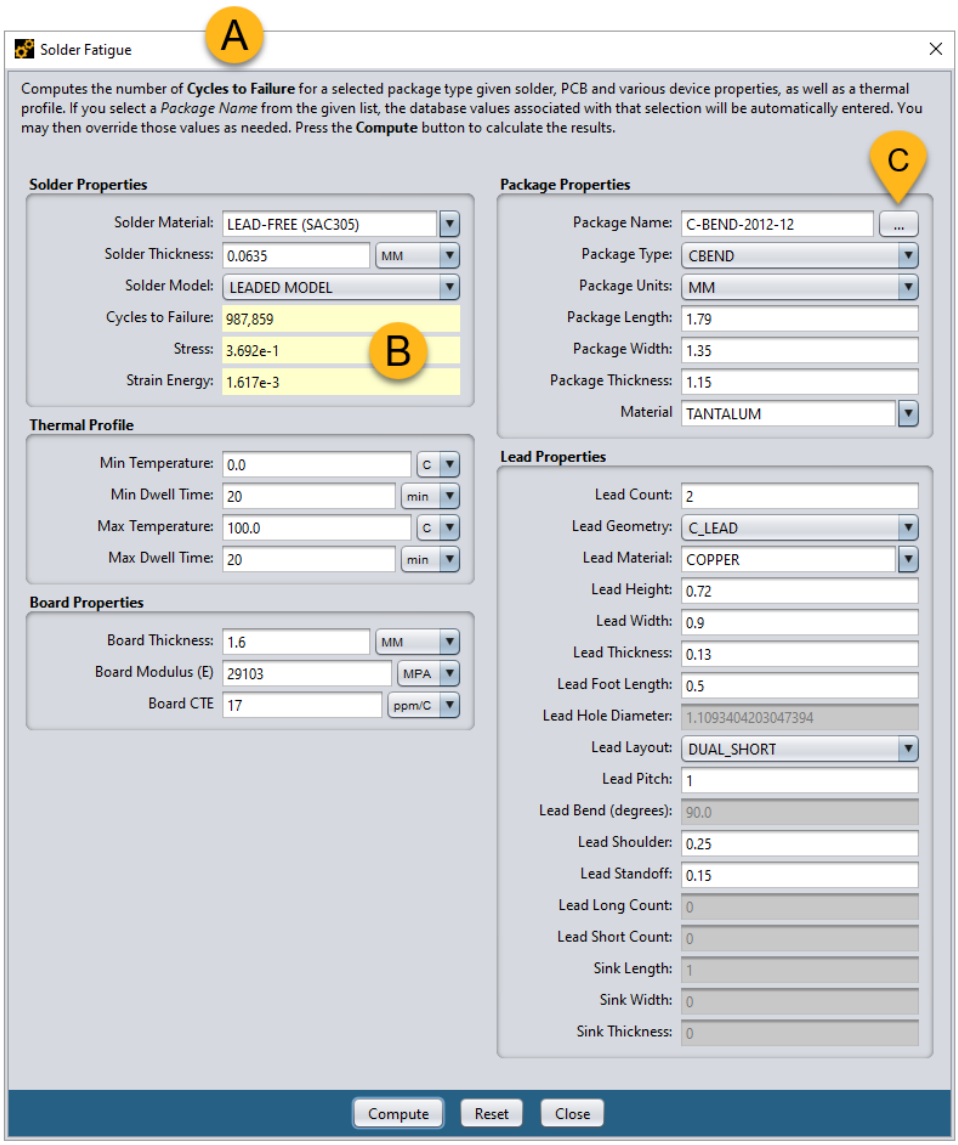
- **Solder Properties**
- The **Thermal Profile** you define
- **Board Properties**
- **Package Properties**
- **Lead Properties**

Note:

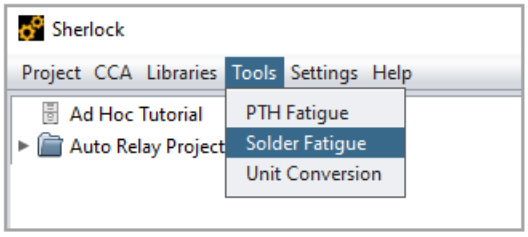
The ceramic chip solder fatigue model may not be valid for reverse termination components.

Sherlock's solutions will appear in the yellow fields (B), displaying **Cycles to Failure**, **Stress**, and **Strain Energy**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To access the tool, select **Tools > Solder Fatigue** from the **Main Menu**.



Use the dropdown menus in the **Solder Fatigue Tool** to specify the properties for the solder, board, package, and leads. Fields are enabled or disabled based on **Package Type** selected. Some considerations when defining the various properties.

- **Package Name**: If you use the **Package Chooser** (by clicking the ... icon, C above) to choose the **Package Name**, Sherlock will automatically update all the fields under **Package Properties**. If you *manually* enter the **Package Name**, however, Sherlock will *not* update the part properties. Regardless of which method you choose, you can still manually edit package properties afterward.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Package Name** and **Package Type**: For a given **Package Name**, you can select a **Package Type** that is different than what is stored in Sherlock's part database. This may or may not be desirable, so double-checking these values is recommended.
- **Package Type** and **Solder Model**: When you change the **Package Type**, Sherlock will guess the **Solder Model**. If desired, you can manually change the **Solder Model** to a different value.

When you are ready for Sherlock to generate a solution, click **Compute**. Clicking **Reset** clears the yellow solution fields and restores the settings to their default values shown above.

6.9. Thermal Derating

The **Thermal Derating Analysis Module** allows Ansys Sherlock users to compare life cycle **Thermal Events** to the minimum and/or maximum rated operating and storage temperature of parts used in a selected circuit card assembly (CCA). To perform this analysis, Sherlock compares the rated temperature properties of each part to the temperature experienced by that part during each of the thermal profiles defined in the product life cycle. Scores are computed for each part, during each Event and combined to yield an overall score for each part and for the entire circuit card over the expected life cycle. Each part may define both max and min operating and storage temperatures. The type of temperature used during the analysis is defined by the Thermal Event life cycle state which may be either operating (default) or storage. When using part storage temperature and no storage temperature is defined for the part, the analysis will fallback and use the operating temperature.

Note:

The examples shown in this document are based on the Tutorial Project provided as part of the Sherlock Tutorial package as discussed in the Project Overview (p. 8) lesson.

In this section, the following topics are covered:

6.9.1. Input Data
6.9.2. Analysis Options
6.9.3. Analysis Results

6.9.1. Input Data

The **Thermal Derating analysis module** makes use of the following input data for the analysis calculations:

- **Parts List**

Tip:

The following section explains which part properties Sherlock uses in the Thermal Derating analysis: Tables: Required Part Properties per Analysis Type (p. 326).

- **Thermal Events**

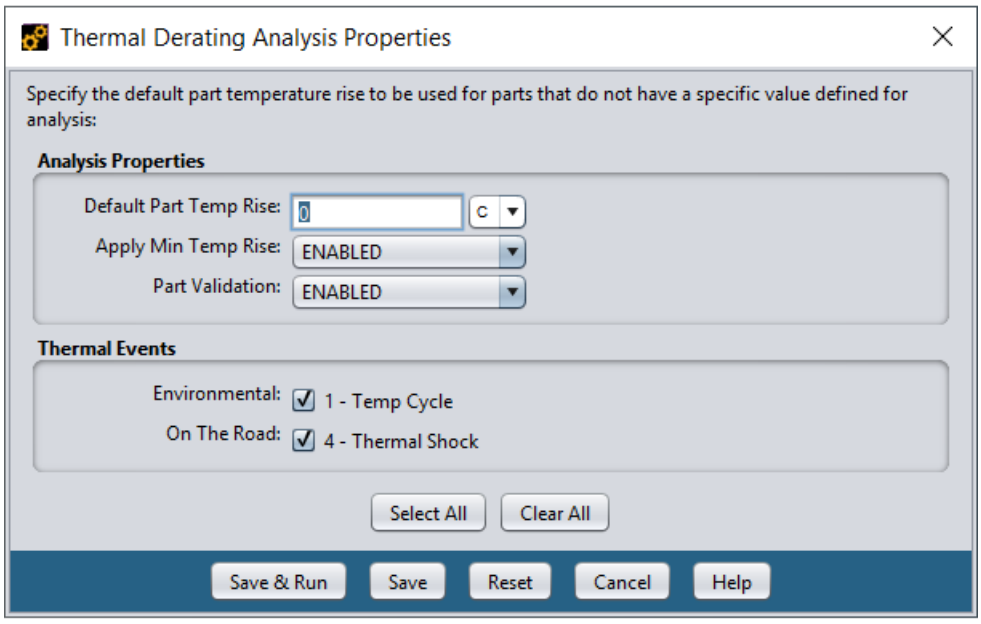
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

6.9.2. Analysis Options

As shown in the dialog box below, the required input properties necessary to perform Thermal Derating analysis are:

The **Default Part Temp Rise** defines the temperature (above/below ambient) to be assigned to each part for which the **Part Temp Rise** property has been assigned the DEFAULT value in the **Parts List**. The **Part Temp Rise** is added to temperatures defined in **Thermal Event** profiles and is ignored when a **Thermal Map** is assigned to a specific **Thermal Event**.



The **Apply Min Temp Rise** parameter allows users to prevent the part temperature rise value from being applied to the minimum temperature defined in a thermal cycle. By default, it is applied to the minimum temperature.

The **Part Validation** parameter, when enabled, causes the Part Validation Analysis (p. 263) to be performed prior to Thermal Derating Analysis if it has not yet been successfully run.

Additionally, the specific **Thermal Event** to perform the analysis for must be selected. At least one **Thermal Event** must be selected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.9.3. Analysis Results

The **Thermal Derating Analysis Module** generates the following results:

- **Summary Panel** showing overall score and analysis statistics
- Table of **Thermal Event** results
- Table of individual part and **Thermal Event** results
- Table of individual part results for all **Thermal Events** analyzed against the part
- Score distribution chart showing a histogram of scores for all parts analyzed
- Score distribution chart showing a histogram of scores for all parts by **Thermal Events** analyzed

To perform **Thermal Derating Analysis**, we'll start by defining some thermal part properties and reviewing all the analysis results. Start by examining the results provided by the Tutorial Project. If you haven't already done so, import that project now.

Note:

If you modified data for the Tutorial project during the tutorial lessons, it is likely that some of the Parts List properties have changed and are different than those shown here in this document. If so, the easiest way to clean things up is to delete the existing Tutorial project and import it again with the ZIP file provided.

Double-click the **Parts List** entry of the **Inputs** folder found in the **Main Board** folder of the **Project Tree**. This will display the Parts List. Right-click on part C1 and select Edit Selected Parts. After the parts editor for part C1 opens, select the **Thermal** tab if it isn't already selected. Note that **Max Rated Temp** is **70** and **Part Temp Rise** is set to **DEFAULT**. The maximum rated temperature is just a guess, so let's set it to the correct value of 155 degrees for this part and click **Save the part editor changes** to apply the change to all parts with the same part number. This should change parts C1-C10.

Using the method employed for modifying part C1, modify part C12-C20 to set the **Max Rated Temp** to 125 degrees and apply the changes. Similarly modify parts C21 – C30 to set the **Max Rated Temp** to 105.

Finally, select all parts in the **Parts List** and choose the **Edit Selected Properties** menu option to set the **Thermal Units** property is set to **C** and the **Min Rated Temp** to **-50** for all parts.

To run the analysis, right-click the **Thermal Derating** entry of the **Analysis** folder in the **Project Tree** and select **Edit Properties**. Use a **Default Part Temperature Rise** of **0** and click **Save & Run**. The analysis of the Parts List against the **Thermal Events** defined in the Environmental and On The Road life cycles will execute. When it has completed, the icon for **Thermal Derating** will change to a GREEN check mark. Right-click again on the **Thermal Derating** entry and select **View Results**.

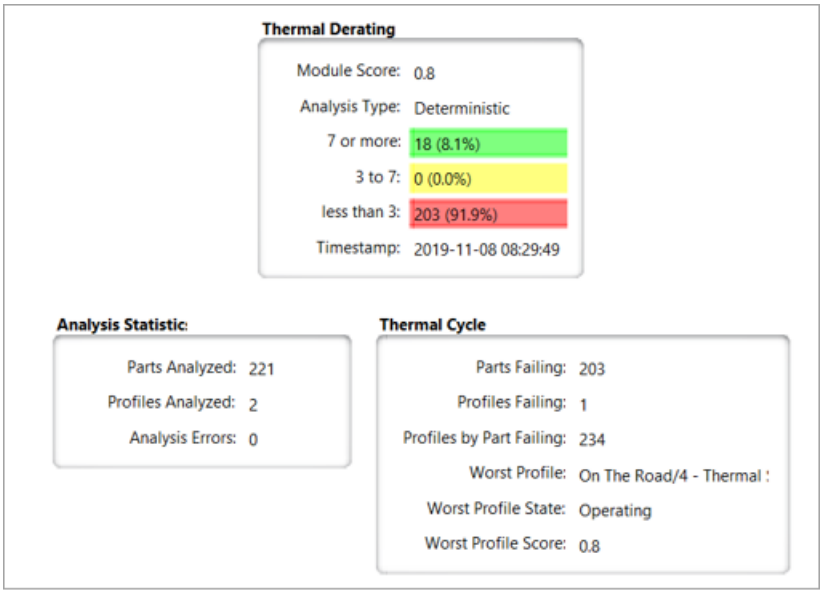
6.9.3.1. Summary Results

The **Summary** tab shows the overall thermal derating score, including the number of parts that fall within the three score levels. The **Analysis Statistics** section shows the number of parts analyzed,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the number of thermal profiles analyzed, the default derating factor, and the number of analysis errors found.

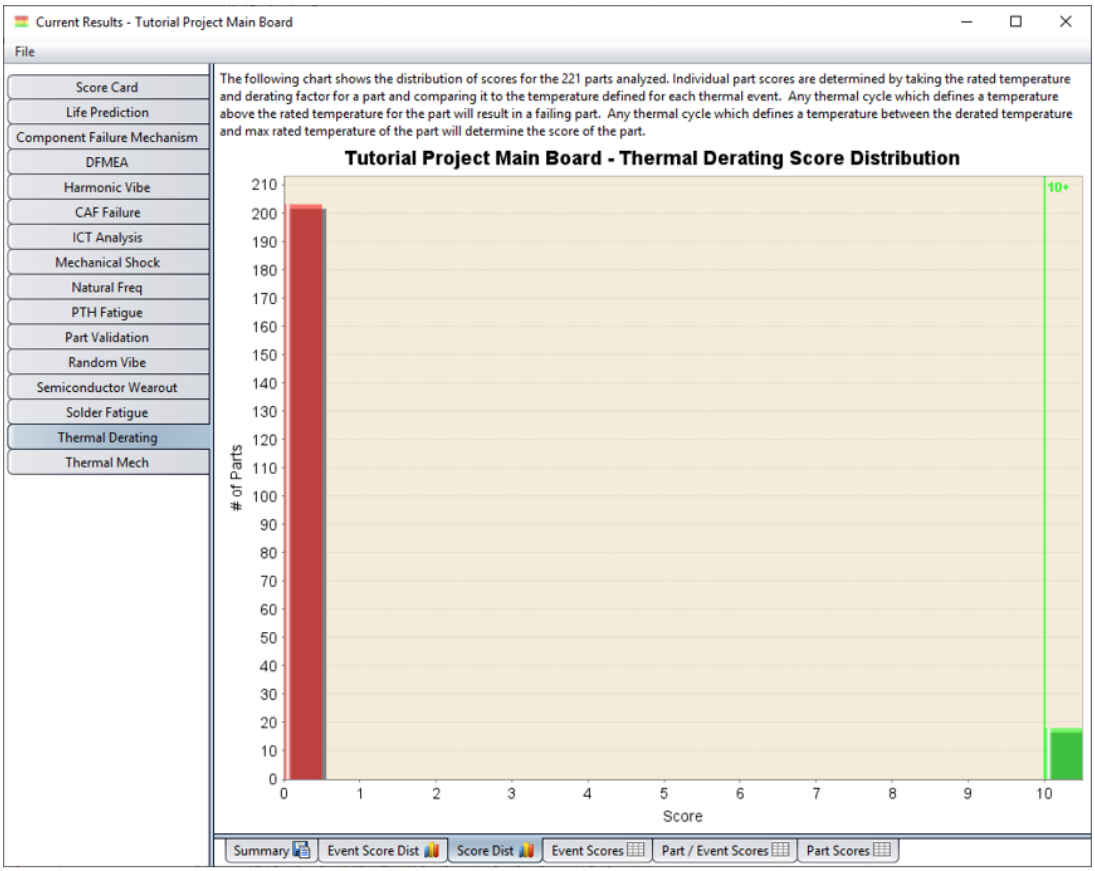
The **Thermal Cycle** section shows the number of failing parts, the number of profiles that produced failing parts, and the number of parts and profiles combined that produced failures. The Thermal Profile that produced the worst score for all parts is listed with the score of that profile against those parts.



6.9.3.2. Score Distribution Chart

The score distribution chart shows a histogram of scores for all parts analyzed. As is evident from this chart, 18 parts passed thermal derating analysis. As will be shown during the review of the individual table results, the parts that passed are the C1 – C10 and C12 - C20 which have a maximum rated temperature above 105 degrees defined by the Thermal Shock Event. Parts C21 -C30 have a maximum rated temperature of 105 degrees Celsius with the remaining parts having a temperature of 70 degrees Celsius which is below the maximum temperature of the Thermal Shock Profile.

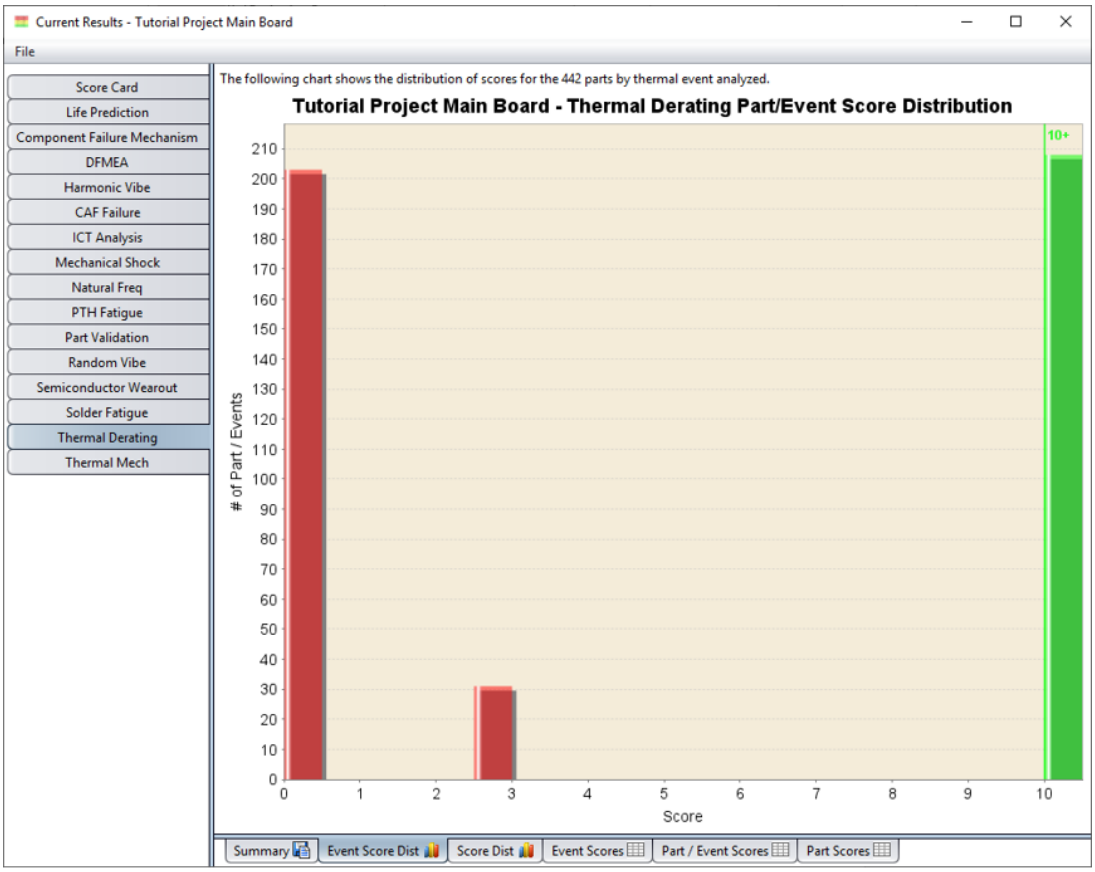
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6.9.3.3. Event Score Distribution Chart

The Event score distribution chart shows a histogram of scores for all parts against each **Thermal Event** analyzed. Compare this chart to the **Score Distribution Chart** and see that many of the parts that failed overall actually pass one of the **Thermal Events**. The details can be found when the Part / Event scores are reviewed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6.9.3.4. Table of Individual Results

Sherlock provides three different tables to provide more analysis detail.

6.9.3.4.1. Event Scores

6.9.3.4.2. Part Scores

6.9.3.4.3. Part / Event Scores

Note:

The **Score** column does not appear in the Results Tables by default. You must add the column manually. To add or remove columns, click the **gear icon** in the upper-right of the table to open the **Column Settings** dialog.

6.9.3.4.1. Event Scores

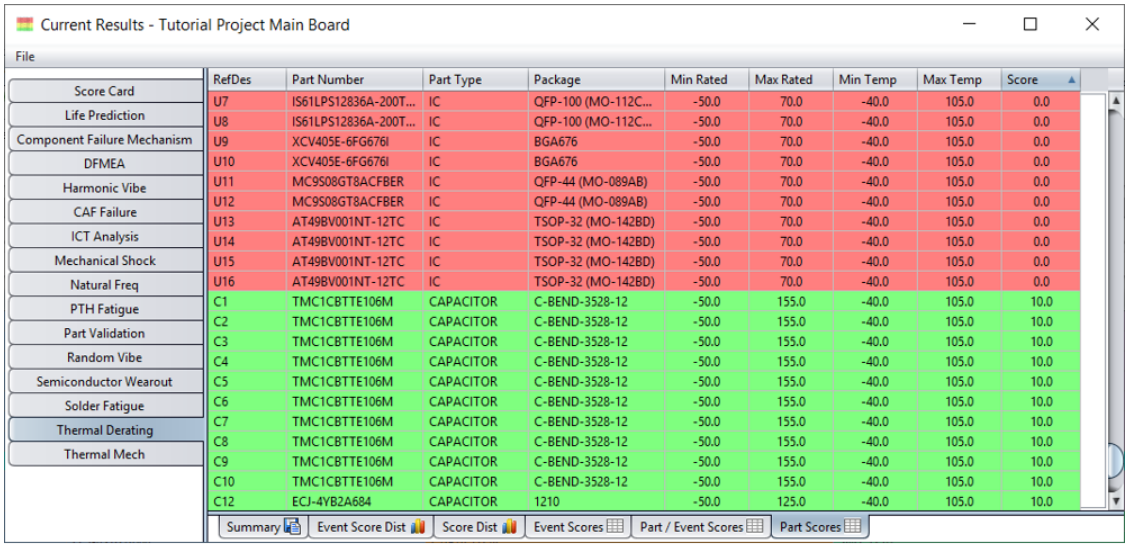
By selecting the **Event Scores** tab, the average score for each **Thermal Event** across all parts is displayed. Reviewing these results shows the **On the Road** Phase produced the lowest score.

Phase	Event	State	Profile	Min Temp	Max Temp	Score
On The Road	4 - Thermal Shock	Operating	Thermal Cycle	-40.0	105.0	0.8
Environmental	1 - Temp Cycle	Operating	Thermal Profile	10.0	65.0	8.9

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.9.3.4.2. Part Scores

To see the overall score for each part for each part analyzed, select the **Part Scores**. This view shows the failing parts in red which is all of the parts except the 19 parts that were modified in the Parts List to update the max rated temperature.



RefDes	Part Number	Part Type	Package	Min Rated	Max Rated	Min Temp	Max Temp	Score
U7	IS61LPS12836A-200T...	IC	QFP-100 (MO-112C...	-50.0	70.0	-40.0	105.0	0.0
U8	IS61LPS12836A-200T...	IC	QFP-100 (MO-112C...	-50.0	70.0	-40.0	105.0	0.0
U9	XCV405E-6FG676I	IC	BGA676	-50.0	70.0	-40.0	105.0	0.0
U10	XCV405E-6FG676I	IC	BGA676	-50.0	70.0	-40.0	105.0	0.0
U11	MC9S08GT8ACFBER	IC	QFP-44 (MO-089AB)	-50.0	70.0	-40.0	105.0	0.0
U12	MC9S08GT8ACFBER	IC	QFP-44 (MO-089AB)	-50.0	70.0	-40.0	105.0	0.0
U13	AT49BV001NT-12TC	IC	TSOP-32 (MO-142BD)	-50.0	70.0	-40.0	105.0	0.0
U14	AT49BV001NT-12TC	IC	TSOP-32 (MO-142BD)	-50.0	70.0	-40.0	105.0	0.0
U15	AT49BV001NT-12TC	IC	TSOP-32 (MO-142BD)	-50.0	70.0	-40.0	105.0	0.0
U16	AT49BV001NT-12TC	IC	TSOP-32 (MO-142BD)	-50.0	70.0	-40.0	105.0	0.0
C1	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C2	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C3	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C4	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C5	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C6	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C7	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C8	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C9	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C10	TMC1CBTTE106M	CAPACITOR	C-BEND-3528-12	-50.0	155.0	-40.0	105.0	10.0
C12	ECJ-4YB2A684	CAPACITOR	1210	-50.0	125.0	-40.0	105.0	10.0

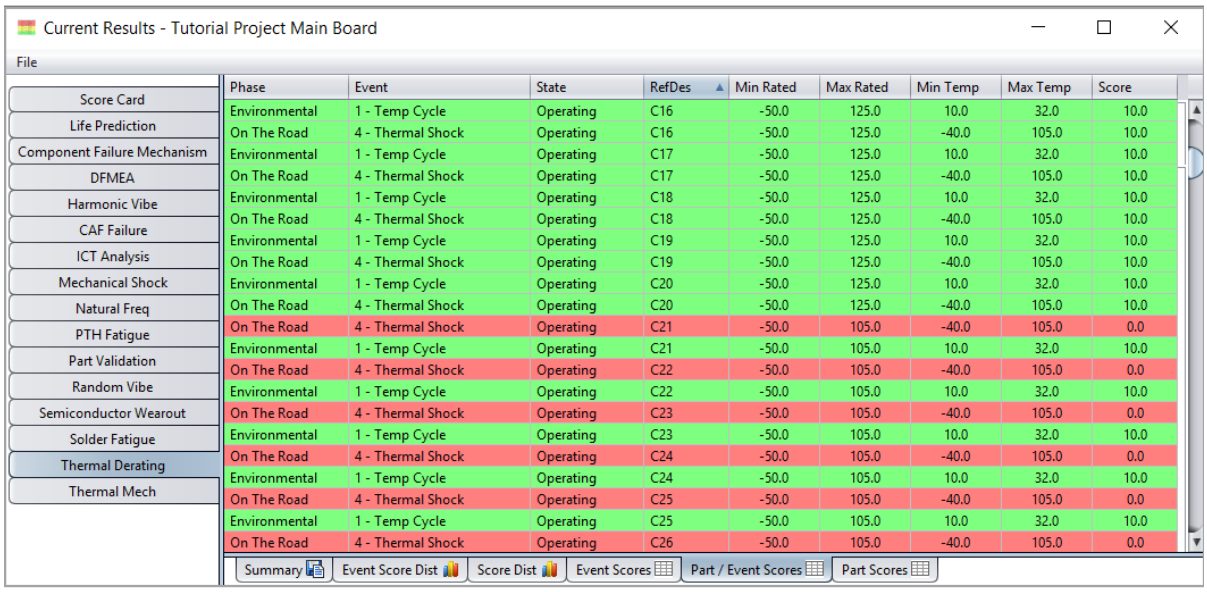
Note:

As with all Sherlock analysis results tables, you can double-click any of the rows to view the part properties for the selected part or right-click any row to export all or part of the results table to a spreadsheet file.

6.9.3.4.3. Part / Event Scores

By selecting the **Part / Event Scores** tab, the **Part Scores** view is broken down such that for each part, the score that part received for each **Thermal Events** profile is provided. Note how part C21 received an overall score of 0.0 on the **Part Scores** tab, but reviewing that same part on the **Part / Event Scores** tab reveals that it received a 10.0 for the Temp Cycle Event profile but received a 0.0 for the **Thermal Shock** profile. Since it failed during one of the Thermal Cycles, it is given an overall score of 0.0 for the entire life cycle.

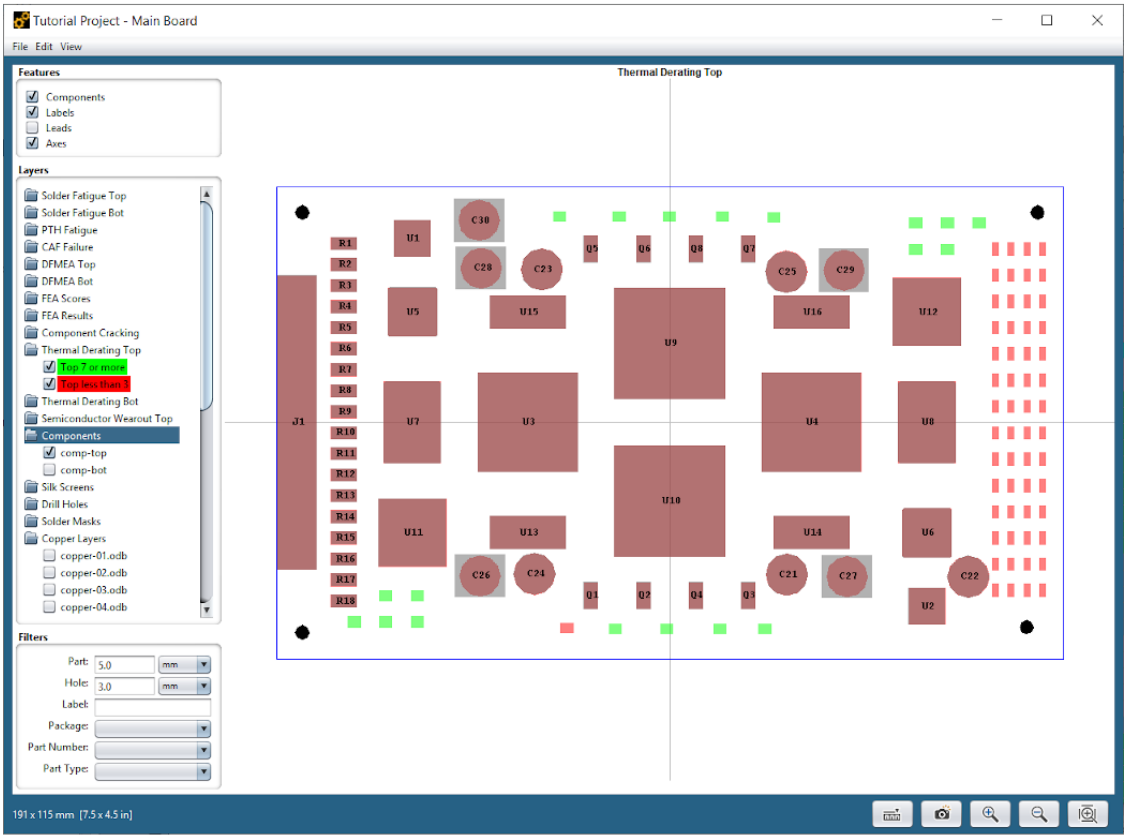
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Phase	Event	State	RefDes	Min Rated	Max Rated	Min Temp	Max Temp	Score
Environmental	1 - Temp Cycle	Operating	C16	-50.0	125.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C16	-50.0	125.0	-40.0	105.0	10.0
Environmental	1 - Temp Cycle	Operating	C17	-50.0	125.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C17	-50.0	125.0	-40.0	105.0	10.0
Environmental	1 - Temp Cycle	Operating	C18	-50.0	125.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C18	-50.0	125.0	-40.0	105.0	10.0
Environmental	1 - Temp Cycle	Operating	C19	-50.0	125.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C19	-50.0	125.0	-40.0	105.0	10.0
Environmental	1 - Temp Cycle	Operating	C20	-50.0	125.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C20	-50.0	125.0	-40.0	105.0	10.0
On The Road	4 - Thermal Shock	Operating	C21	-50.0	105.0	-40.0	105.0	0.0
Environmental	1 - Temp Cycle	Operating	C21	-50.0	105.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C22	-50.0	105.0	-40.0	105.0	0.0
Environmental	1 - Temp Cycle	Operating	C22	-50.0	105.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C23	-50.0	105.0	-40.0	105.0	0.0
Environmental	1 - Temp Cycle	Operating	C23	-50.0	105.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C24	-50.0	105.0	-40.0	105.0	0.0
Environmental	1 - Temp Cycle	Operating	C24	-50.0	105.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C25	-50.0	105.0	-40.0	105.0	0.0
Environmental	1 - Temp Cycle	Operating	C25	-50.0	105.0	10.0	32.0	10.0
On The Road	4 - Thermal Shock	Operating	C26	-50.0	105.0	-40.0	105.0	0.0

6.9.3.5. Layer Viewer

Right-click the **Thermal Derating** entry of the **Analysis** folder in the **Project Tree** and select **View Layers**. The layer viewer will color-code each part analyzed for thermal derating based on the overall score that part received.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Finite Element Analysis (FEA) Overview

Finite Element Analysis (FEA) models 3D shapes by a collection of 3D elements (such as bricks or pyramids for example) that are connected by shared nodes or by surface/node constraints. Each element is assigned mechanical properties based on material content and external loads and/or constraints (for example, gravity, point forces, etc) are applied to the model to simulate the intended operational environment. FEA simulation techniques are then applied to predict the displacements and strains experienced by each node and element in the model.

Sherlock uses FEA simulations to perform the following types of analysis:

- Natural Frequency (p. 655) (that is, modal response)
- In-Circuit Testing (ICT) (p. 661) (that is, static load)
- Mechanical Shock (p. 674) (that is, transient dynamic)
- Harmonic Vibration (p. 688) (that is, modal dynamic)
- Random Vibration (p. 698) (that is, modal dynamic)
- Thermal Mech (p. 708) (that is, thermal cycling)

This document describes the overall FEA processing approach used by Sherlock to predict circuit card and component reliability. All the capabilities described in this document apply to each of the analysis types listed above, which are discussed in detail in subsequent sections.

> **Warning:**
>
> Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate results, see the section Compatibility with Earlier Versions of Sherlock (p. 5).

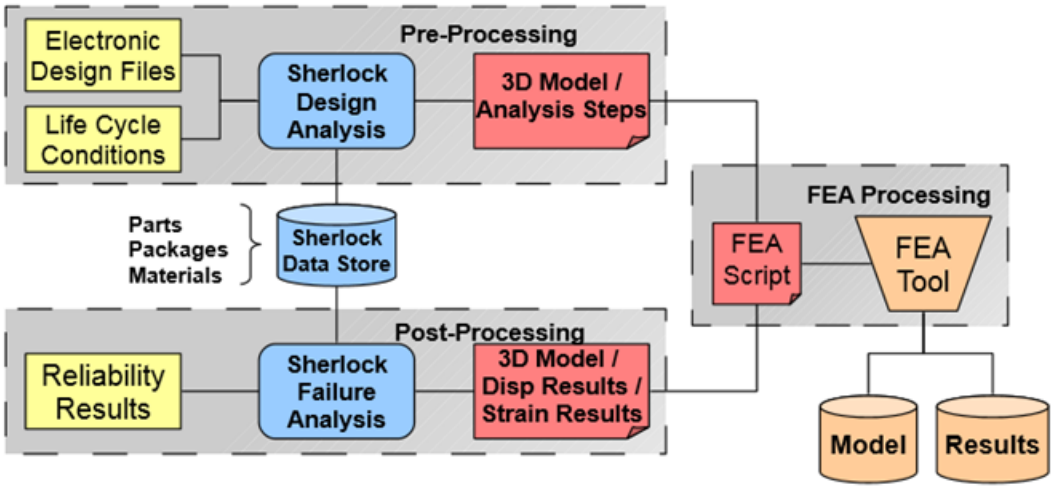
In this section, the following topics are covered:

7.1. FEA Integration
7.2. FEA Engine
7.3. Data Sources
7.4. FEA Models
7.5. FEA Processing
7.6. FEA Results

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1. FEA Integration

Sherlock was designed to work closely with an FEA engine as depicted in the following diagram:



In the pre-processing phase, Sherlock can be used to extract 2D data from electronic design files and combines it with part, package and material data stored in the Sherlock Data Store to create an accurate 3D FEA model of the circuit card and all components selected for modeling. The 3D model is formatted as an input deck for the FEA engine and commands are generated to perform the specific FEA simulations required by each of the appropriate FEA Events defined in the project Life Cycle.

During the FEA processing phase, Sherlock can be used to automatically control all FEA processing, monitor the process for errors and store the analysis results in the file system.

In the post-processing phase, Sherlock can be used to import the 3D model used during FEA processing, as well as all displacement and strain results (which can be voluminous). Sherlock will then analyze all of the FEA results using a collection of reliability models designed specifically for electronic components and will generate a complete set of reliability results, including Predicted Life curves for the circuit card and predicted damage scores for all components, even for those components that were not specifically defined in the 3D model.

Advanced users can choose to manage all or part of the FEA processing phase themselves, while still using Sherlock for the pre-processing and/or post-processing phases. For example, you might use Sherlock to generate a 3D model of a given circuit card, but then define the external loads and constraints yourself using the capabilities of the FEA engine. After manually executing one or more tests, you might then employ Sherlock again to post-process the FEA results to generate reliability results.

7.2. FEA Engine

Topics in this section:

7.2.1. Selecting the FEA Engine
7.2.2. FEA Analysis Settings

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.1. Selecting the FEA Engine

Sherlock supports the following FEA engines:

- Ansys Mechanical Classic / Mechanical APDL
- Abaqus (Windows only)
- NX Nastran (Windows only)

For their FEA analysis, Ansys Mechanical and Abaqus employ the Von-Mises equivalent strain based on elemental mean values and surface average values. NX Nastran's analysis is still based on nodal elastic strain.

Note:

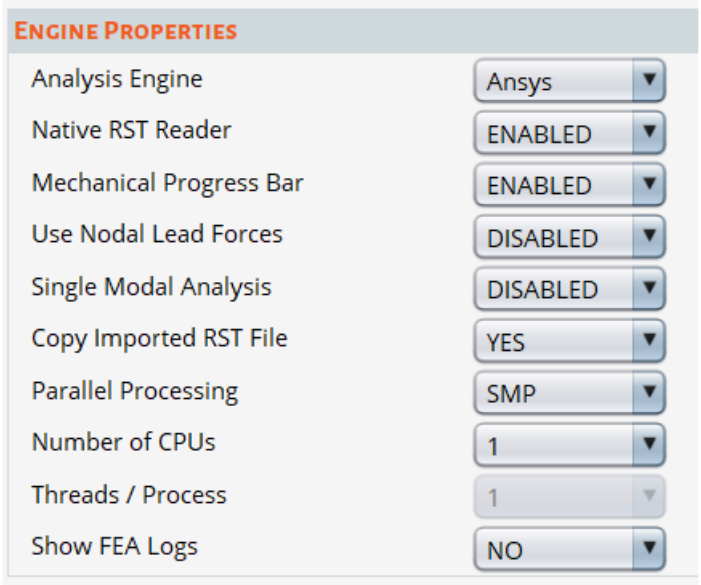
Refer to the Sherlock Theory Reference for a detailed explanation of how Sherlock performs its calculations.

The Ansys Mechanical APDL engine is included in the Sherlock installation directory and is the default engine for FEA simulations. The other engines are commercially available and can be easily integrated with Sherlock if installed on the same workstation

Once installed (following the manufacturer's instructions and licensing procedures), it is relatively easy to configure Sherlock to use a given FEA engine. Simply select **Settings > General Settings** from the Main Menu to display the **Sherlock Settings** dialog. Then click **FEA Analysis** on the sidebar to view the FEA Engine Settings. To change your engine, select the analysis engine you wish to use from the drop-down menu as shown below under Engine Properties (p. 447).

The **Analysis Engine** list shows all FEA engines authorized by your license file. When changing FEA Engines, you must indicate where the Abaqus executable is located on your workstation.

For the Ansys engine, the settings will look something like this:



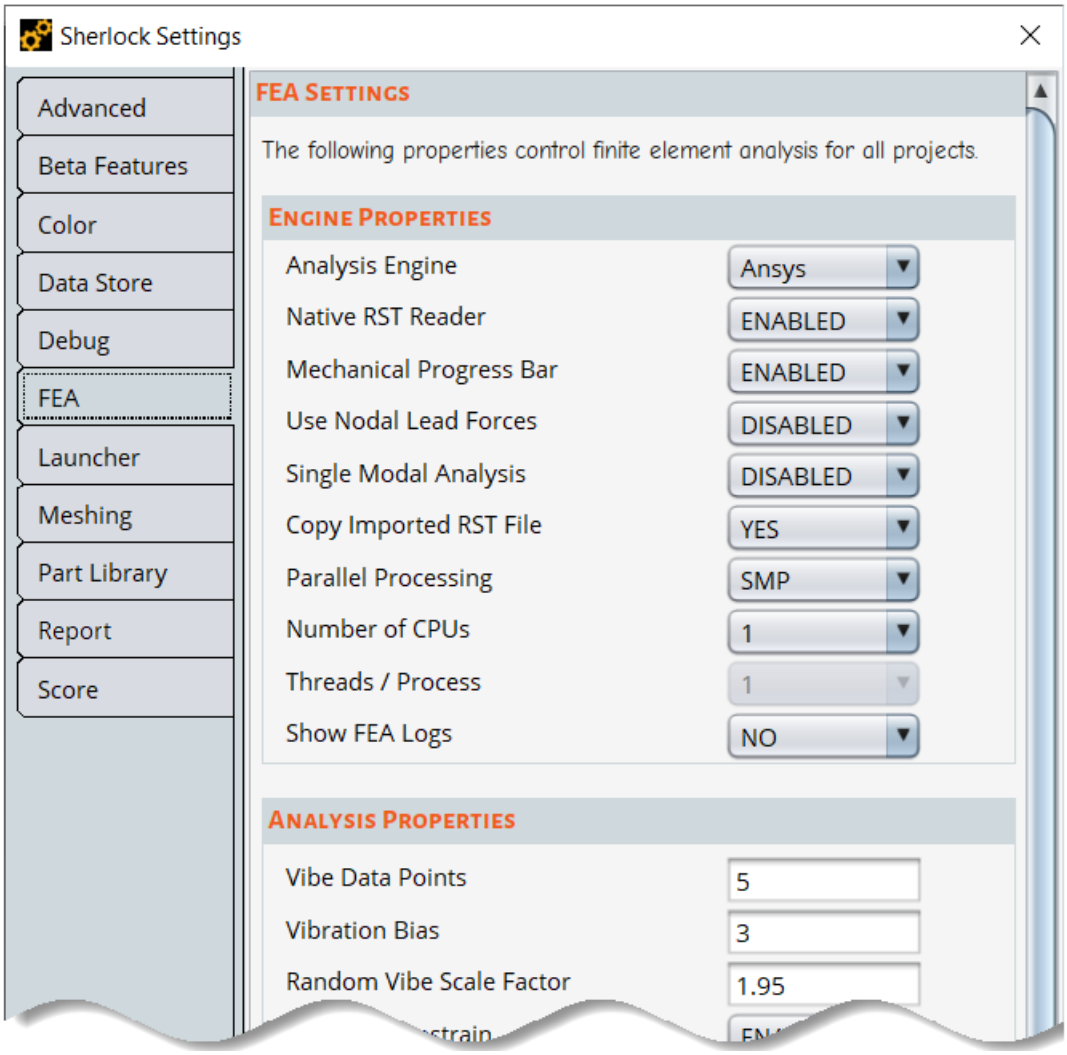
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Ansys executable is the default FEA Engine for Sherlock, so there is no need to indicate where the MAPDL executable is located. The **Number of CPUs** property allows you to specify the maximum number of CPUs allocated to each instance of the FEA engine when executed by Sherlock. See the manufacturer's instructions for the suggested number of CPUs for each engine.

The FEA engine specified in the FEA Settings dialog will be used for all subsequent FEA-related analyses and processing. No other user interaction is required. The engine used during a specific analysis run will be displayed in the Summary results panel for documentation purposes.

7.2.2. FEA Analysis Settings

To access the **FEA Analysis Settings** within Sherlock, select **Settings > FEA Analysis** from the top of the Sherlock window. Here you will find settings for **Engine Properties**, **Analysis properties**, **Displacement Color**, **Strain Color**, and **Stress Color**.



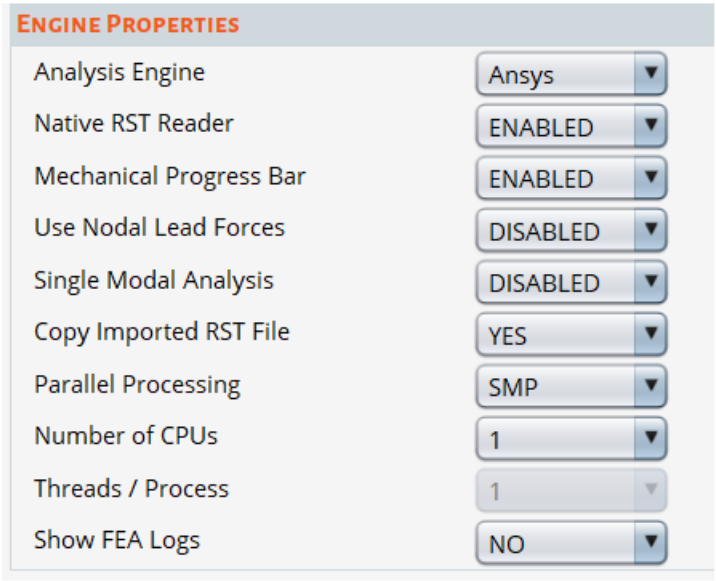
Once you have made changes to the **FEA Analysis Settings**, you must click **Apply** or **Apply and Close** to save your changes. Clicking the **Reset** button will restore all settings to their last saved value. Clicking **Restore Defaults** will revert all settings to their default values. Click **Cancel** to exit the dialog without saving changes.

Each panel in the **FEA Enging Settings** dialog is explained in the sections below:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.2.1. Engine Properties
7.2.2.2. Analysis Properties
7.2.2.3. Analysis Color Settings
7.2.2.4. Strain Color Settings
7.2.2.5. Stress Color Settings
7.2.2.6. Mesh Settings

7.2.2.1. Engine Properties



- **Analysis Engine**: Here you specify the FEA engine that will be used for all subsequent FEA-related analyses and processing. **Ansys** is the default FEA Engine. The Windows version of Sherlock also supports Abaqus and NX Nastran. See also the section above, Selecting the FEA Engine (p. 445).
- **Native RST Reader**: Choose whether the native RST Reader is used for processing results or if Ansys Mechanical APDL commands are used for processing results.
- **Mechanical Progress Bar**: During FEA Export, this option enables/disables the progress bar in Mechanical during material assignment and named geometry creation.
- **Use Nodal Lead Forces**: (Default setting: **DISABLED**) When **ENABLED**, lead stress calculations will be based on forces and moment data.
- **Single Modal Analysis**: (Default setting: **DISABLED**) This setting applies to the Harmonic and Random Vibration analyses with multiple selected Events. When **ENABLED**, Sherlock runs a single modal analysis and applies it to each selected Event. When **DISABLED**, a separate modal analysis is performed for each selected Event which lengthens processing time. To use this feature, the **Analysis Engine** must be set to **Ansys**. For more information, refer to the sections on the Harmonic Vibration (p. 688) and Random Vibration (p. 698) analyses.
- **Copy Imported RST File**: This setting applies to analyses using an Imported Data Source (p. 452).
  - When YES is selected (the default setting), the application creates a *copy* of the imported results file in the working directory instead of directly referencing the original file during processing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When NO is selected, the application accesses the imported RST file directly from its *source* location. In general, this results in faster processing time. However, when the RST file is located on a network drive, it may result in longer processing time.

- **Parallel Processing**: This setting appears only when **Ansys** is selected for the **Analysis Engine**. You must specify the type of parallel processing for FE analysis: distributed-memory parallel (**DMP**), shared-memory parallel (**SMP**, the default setting), or hybrid parallel **(HYBRID**) which is a combination of SMP and DMP. When you select **HYBRID**, the **Threads / Process** setting, explained below, becomes available.

  The **Parallel Processing** selection only affects the performance of the Mechanical APDL solver when running an FEA simulation within Sherlock. It does not impact other processes.

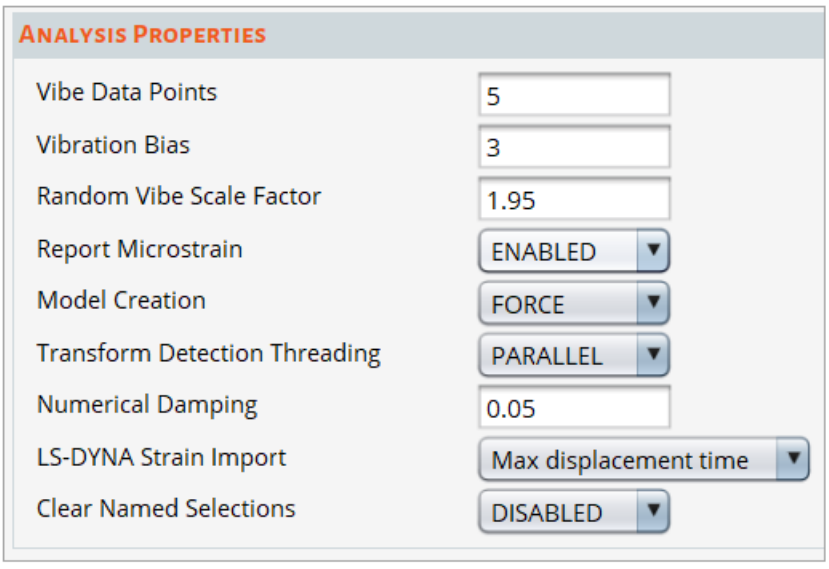
- **Number of CPUs**: This option sets the maximum number of CPUs allocated to each instance of the FEA engine executed by Sherlock, and it also specifies the number of threads used when reading analysis results. For the Ansys engine, the recommended value is 4, as shown above. For other engines, see the manufacturer's instructions for recommendations.

- **Threads / Process**: This setting becomes available when you select **Ansys** as the **Analysis Engine** and **HYBRID** for **Parallel Processing**. From the dropdown menu, specify the number of threads per process that will be used for HYBRID parallel processing during an FE analysis.

- **Show FEA Logs**: Choose whether FEA logs are included in the analysis results viewer.

7.2.2.2. Analysis Properties

Options in the **Analysis Properties** panel are explained below:



- **Vibe Data Points**: (Applies to the Abaqus and Nastran engines only.) This value sets the number of modal analysis database points used for vibration analysis.

- **Vibration Bias**: Specify the number of vibration points around each natural frequency used for the harmonic vibration analysis. Valid values are 1 to 20.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Random Vibe Scale Factor**: This value sets the scaling factor applied to random vibration analysis. A default value of 1.95 is used, which is taken from Steinberg's Vibration Theory.
- **Report Microstrain**: When **ENABLED** (default setting), Sherlock will report analysis results in terms of microstrain. Currently, this setting only applies to Random Vibe, Harmonic Vibe, ICT, Mechanical Shock, and Thermal Mech. Select **DISABLED** to turn off this feature. Any changes to this setting take effect the next time the analysis is run.
- **Model Creation**: This option controls how an FEA model is re-created between each analysis. **FORCE** will force a complete re-creation of the model while **AUTO** will use the previously generated mesh if the mesh settings are identical.
- **Transform Detection Threading**: (**Parallel** or **Single**) Select how many threads Sherlock uses to detect model Transforms when importing results. To use multiple threads, select **Parallel** (default). For only a single thread, select **Single**.

  Tip:

  When handling large models, the **Parallel** setting significantly improves Sherlock's Transform detection.

  Note:

  The Detect Transform function is discussed further in the *Ansys Workbench Integration* section: See *Importing Analysis Results into Sherlock* (p. 763). See also the *Imported Data Source* sections under each analysis type (for example, here in *Mechanical Shock Analysis* (p. 658)).

- **Numerical Damping**: (Known as the **Damping Modifier** in earlier versions of Sherlock.) The default value is based on the **Analysis Engine** selected under **Engine Properties** (p. 447) and should not be modified. This setting specifies the numerical damping to be used in the time-stepping scheme for a transient analysis in Mechanical Shock. The value dictates the convergence criteria based on physics. The method is explained in the Mechanical APDL Theory Reference: See this section under Transient Analysis.

  Note:

  Ansys uses only positive damping. Abaqus and Nastran use negative damping.

Analysis Engine	Numerical Damping Default
Ansys	0.05
Abaqus	–0.05
Nastran	–0.05

- **LS-DYNA Strain Import**: (**Max Over Time** or **Max Displacement Time**) This setting affects how results are imported from LS-DYNA. (See LS-DYNA Imported Data Source (p. 680) under *FEA*

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mechanical Shock.) **Max Over Time**: When reporting strain for an element, Sherlock considers all time steps and reports the maximum stress the element saw. This means the strain result for one element may represent a different time step than that of another element. **Max Displacement Time**: The reported strain results for all elements will come from the same time step—the time when maximum displacement occurred.

- **Clear Named Selection**: When **ENABLED**, the Python script created during the FEA model export will always delete the current Named Selection list in the Mechanical application—including any Named Selections created in the Mechanical application— before reinitializing the Named Selection list with the exported Sherlock data only. This setting is **DISABLED** by default.

7.2.2.3. Analysis Color Settings



- Options in the **Analysis Color Settings** panel are explained in the sections below. To see how these settings affect the graphical display of results, refer to the sections Graphical Result Layers (p. 468) and 3D Results File (p. 471).
- **Max Displacement**: This value sets the maximum displacement as a **PERCENT** or as an **EXACT** amount that will be color-coded in the graphical results display. If you choose **EXACT**, the value you enter represents strain or microstrain, based on which option you chose for **Report Microstrain** under **Analysis Properties** (p. 448).
- **Min Displacement**: This value sets the minimum displacement (as a **PERCENT** or as an **EXACT** amount) that will be color-coded in the graphical results display. If you choose **EXACT**, the value you enter represents strain or microstrain, based on which option you chose for **Report Microstrain** under **Analysis Properties** (p. 448).

7.2.2.4. Strain Color Settings

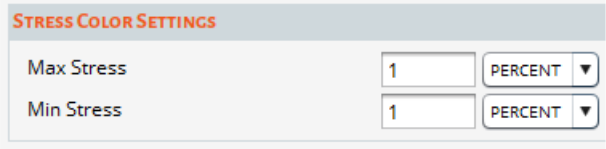


- Options in the **Strain Color Settings** panel are explained below. To see how these settings affect the graphical display of results, see the sections below: Graphical Result Layers (p. 468) and 3D Results File (p. 471).
- **Max Strain**: This value sets the maximum strain (as a **PERCENT** or as an **EXACT** amount) that will be color-coded in the graphical results display. If you choose **EXACT**, the value you enter represents strain or microstrain, based on which option you chose for **Report Microstrain** under **Analysis Properties** (p. 448).
- **Min Strain**: This value sets the minimum strain (as a **PERCENT** or as an **EXACT** amount) that will be color-coded in the graphical results display. If you choose **EXACT**, the value you enter represents strain or microstrain, based on which option you chose for **Report Microstrain** under **Analysis Properties** (p. 448).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.2.5. Stress Color Settings

Options in the **Stress Color Settings** panel are explained below. To see how these settings affect the graphical display of results, see the sections below: Graphical Result Layers (p. 468) and 3D Results File (p. 471).



- **Max Stress**: This value sets the maximum stress as a **PERCENT** or as an **EXACT** amount (in MPa) that will be color-coded in the graphical results display.
- **Min Stress**: This value sets the minimum stress as a **PERCENT** or as an **EXACT** amount (in MPa) that will be color-coded in the graphical results display.

7.2.2.6. Mesh Settings

To access the Mesh Settings within Sherlock, select **Settings** > **Meshing** from the main menu. See the section Meshing Settings (p. 85) for additional information.

7.3. Data Sources

All the analysis modules provided by Sherlock that make use of FEA allow you to specify the Data Source used to acquire the FEA data, supporting Generated and Imported data sources. The Generated data source indicates that Sherlock should handle all aspects of FEA processing, whereas the Imported data source indicates that the user will handle FEA processing tasks and Sherlock will handle only pre-processing and post-processing tasks. Sherlock also supports a third type of Data Source called Strain Map, that becomes available only if strain map files are available under Files in Sherlock.

7.3.1. Generated Data Source
7.3.2. Imported Data Source
7.3.3. Strain Map Import

7.3.1. Generated Data Source

If the **Data Source** property is set to **Generated** then Sherlock will:

- Create a new FEA model based on the currently defined PCB, components, drill holes, etc.
- Submit the model with appropriate commands to the FEA engine for analysis
- Import the results from the FEA engine *and*
- Generate reliability results based on the FEA results

This approach is the most common and easiest method for FEA processing, with Sherlock doing all the work.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.2. Imported Data Source

If the **Data Source** property is set to **Imported** then Sherlock will:

- Import the designated model and results files from the FEA engine *and*
- Generate reliability results based on the FEA results

This allows advanced users to create their own FEA models and/or modify an FEA model created by Sherlock to include elements, constraints and external forces that are not directly supported by Sherlock. After performing the analysis using the FEA engine, users can then import the FEA results into Sherlock to perform the reliability analysis.

Tip:

In **Sherlock Settings > FEA Settings > Engine Properties**, the setting **Copy Imported RST File** (p. 447) determines how the Sherlock application handles imported RST files. Select the option that is most efficient for your specific project.

Note:

- Imported FEA results for Leads, Mount Points, and Wirebonds using NxNastran are not supported.
- Pre-stressed analysis results are not supported.
- When LS-DYNA is selected as the **Data Source** for a Mechanical Shock analysis, Sherlock will not calculate strain results for orthotropic materials. See LS-DYNA Imported Data Source (p. 680).

Note:

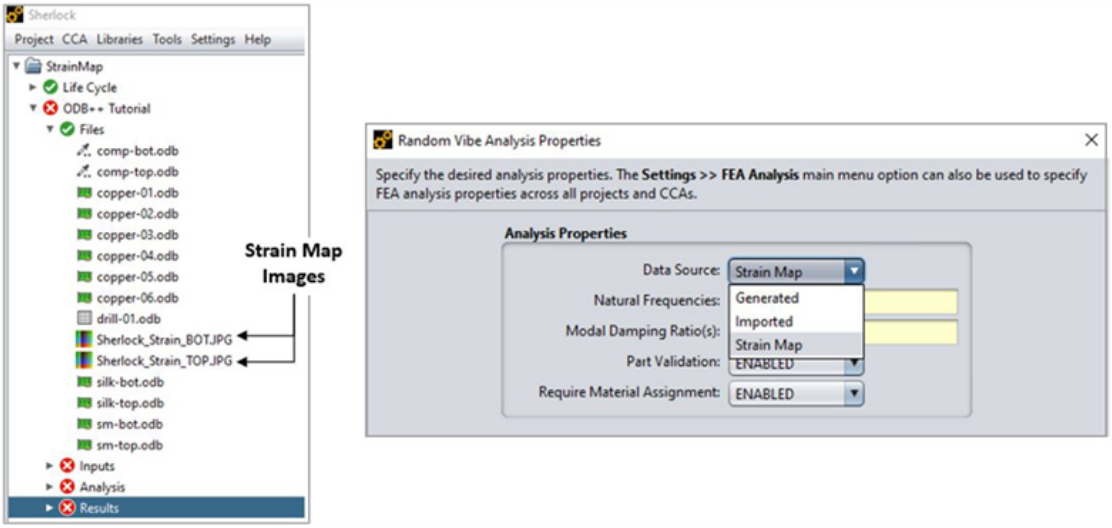
Sherlock will attempt to extract displacement and strain data for each component defined in the circuit card, even if those components were not included in the FEA model itself. In such cases, all result data found near the component will be used to calculate the reliability results.

If you prefer to limit results to modeled parts only, you can select that option in the **Analysis Properties** form. Choose **Generated** for the **Data Source**. Select the **Part Modeling** tab and set **Filtered Part Results** to DISABLED. See FEA Model Properties (p. 562) for additional information.

7.3.3. Strain Map Import

The Strain Mapping feature is available for the Random Vibration and the Mechanical Shock analyses. The feature gives you the flexibility to run board level predictions from images of strain contour or from CSV files. Once these files are added to a Sherlock project, the **Strain Map** option is available under the **Data Source** dropdown menu as shown below. For more information on Strain Mapping, see Strain Mapping in Sherlock (p. 769).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7.4. FEA Models

By default, Sherlock supports a Bonded 3D FEA model which is generated by the Ansys Meshing algorithm. (If you have enabled Sherlock's Legacy Meshing in Mesh Settings (p. 85), you can also select a Merged model.)

7.4.1. Bonded Model

7.4.2. FEA Modeling Properties

7.4.3. Generate 3D Model

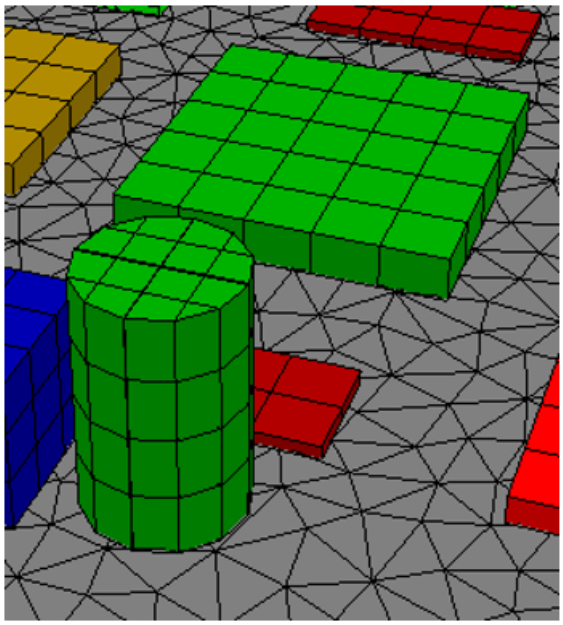
7.4.4. Export FEA Model

7.4.5. Run FEA Analysis

7.4.1. Bonded Model

The **Bonded** model type uses a method in which the PCB and all components are meshed individually. Such individual meshing typically yields a more uniform 3D elements for both the PCB and all components. After all 3D elements are created, Sherlock then automatically "glues" the components to the PCB by creating constraint equations between each component element and the corresponding PCB element(s). Uniform 3D elements can improve the accuracy of results, but the constraints required to define the additional bonds typically increase the analysis time. The bonded method also allows more complicated models to be created.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



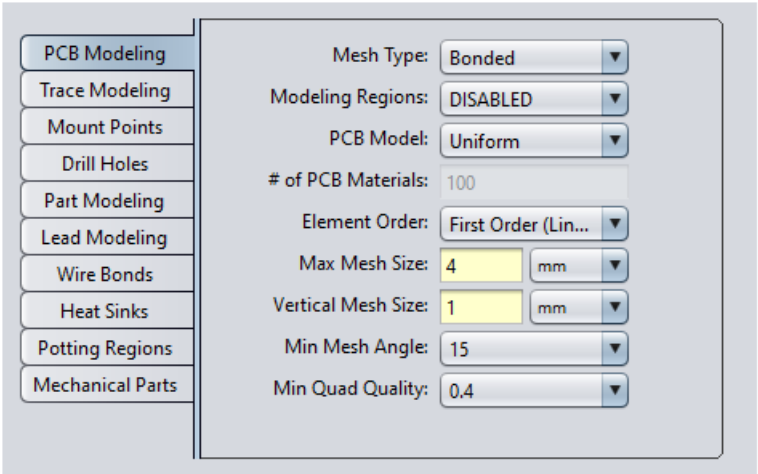
Note:

When using a **Bonded model**, you need to specify a **Maximum Mesh Size** that is small enough to ensure that all components can be bonded to the PCB. In general, the **Maximum Mesh Size** should be no greater than the smallest component being modeled.

7.4.2. FEA Modeling Properties

Sherlock users can indicate how to model the PCB, parts, mount points, leads, drill holes, heat sinks, traces, wire bonds and potting regions each time any of the following actions are performed:

- Generate 3D model
- Export FEA Model
- Run FEA Analysis



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This allows users tremendous flexibility in how they develop, test and use FEA models. For example, you might want to focus exclusively on the model of the PCB itself to ensure that it behaves as expected before loading it with parts and other components. That is easily supported by disabling all other modeling components. As your model is refined, you can selectively add other modeling components to the model.

Note:

When modeling disc shaped components, the leads are not connected to the body when the corner face is set to front back in the 3D model. This is a known limitation in Sherlock. To remedy this, you should define a flat section on the body to ensure the leads are connected.

Descriptions for each of the property tabs included in the FEA modeling dialogs are provided in the following User Guide sections:

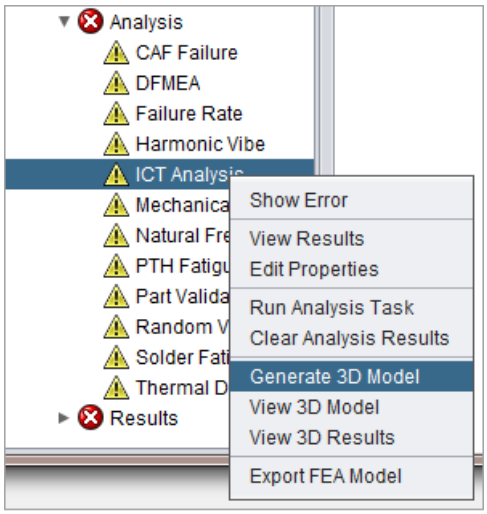
- PCB Modeling (p. 473)
- Trace Modeling (p. 495)
- Part (p. 552) and Lead Modeling (p. 563)
- Mount Points & Fixtures (p. 479)
- Cutouts and Drill Holes (p. 489)
- Heat Sinks (p. 598)
- Potting Regions (p. 616)
- Mechanical Parts (p. 629)

Refer to those User Guide sections for details and examples. The remainder of this section will describe how the FEA dialogs are used to generate and examine FEA models.

7.4.3. Generate 3D Model

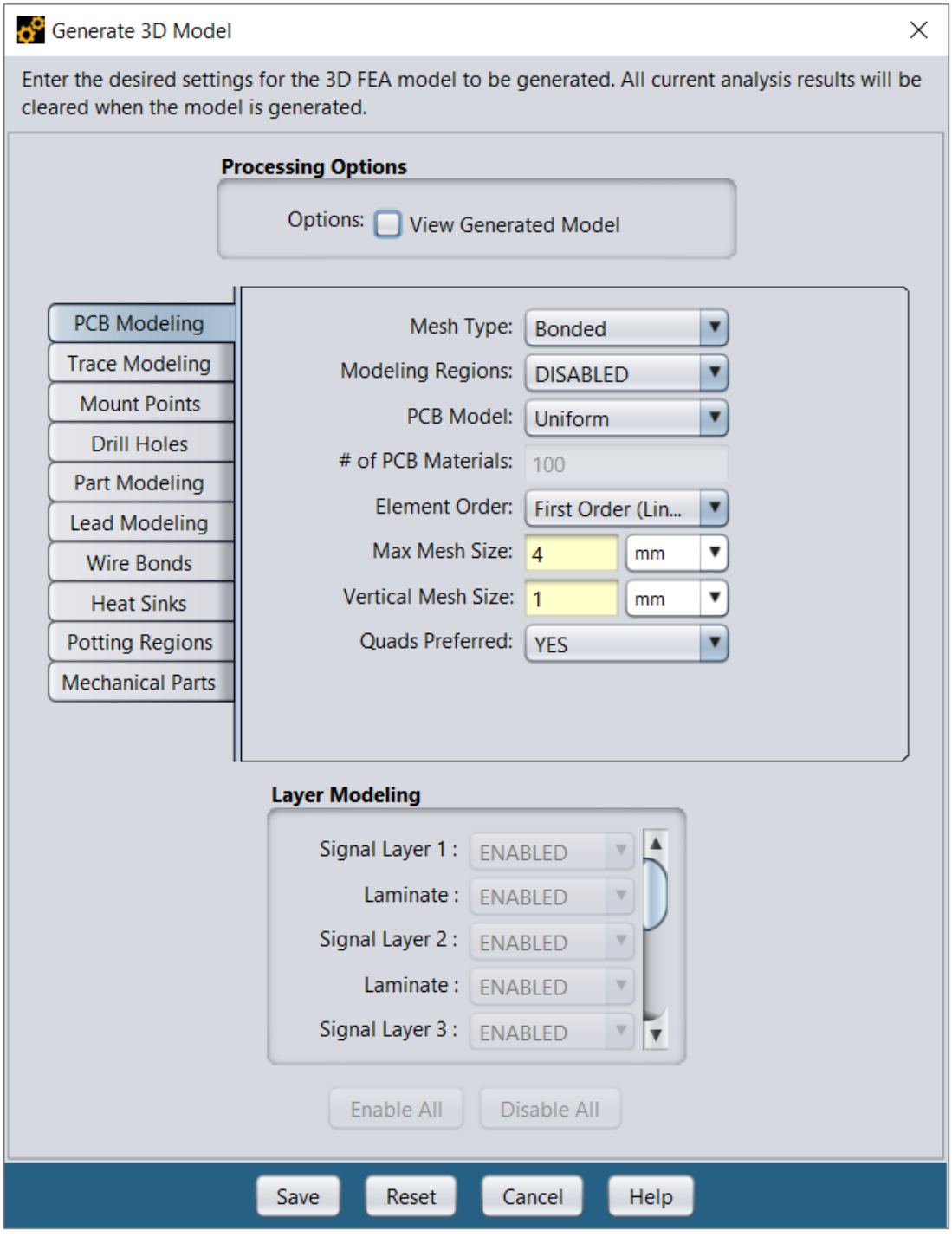
There are times when you just want to see how the 3D model will look when submitted for analysis. In those cases, simply right-click any FEA analysis task in the **Analysis** folder of the **Project Tree** and select the **Generate 3D Model** menu option. At that point, the **Generate 3D Model** dialog will appear, allowing you to specify the components to be included in the model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the **View Generated Model** option is enabled, the **Sherlock 3D Viewer** will be automatically launched after the model has been generated so that the model can be viewed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If a layered PCB model is generated (either **Layered** or **Layered Elements**), then the **Layer Modeling** panel can be used to selectively enable individual layers. This can be useful when testing the model for an individual layer or developing a model to be used for documentation or presentation purposes.

Note:

For each analysis task, the choices made in the **Generate 3D Model** dialog will be remembered from one invocation to the next.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.4. Export FEA Model

The **Generate 3D Model** feature generates elemental models that can be used for testing or documentation purposes. If you want to export a model for analysis purposes, it is usually best to export a geometric model specifically tailored for the FEA tool being used. The geometric model generated by Sherlock can be used as a starting point when building custom FEA models or when performing custom FEA processing. A good deal of time can be saved over a manual model building approach because Sherlock automatically:

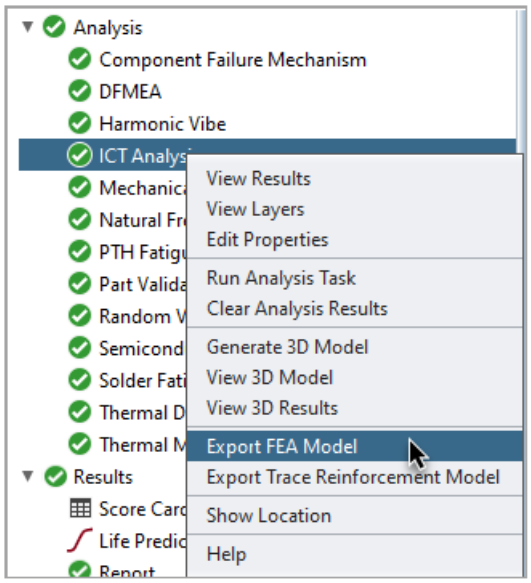
- Handles all component size and placement issues
- Assigns material properties to all PCB and component elements
- Defines all bonding surfaces/nodes
- And creates bonding constraints for all components and sub-assemblies

---

Note:

When exporting an FEA model (*.apdl, *.cdb and *.wbjn), the Sherlock application models the PCB as isotropic material with an in-plane material modulus and a coefficient of thermal expansion (CTE). Although the Stack-up provides both in-plane and out-of-plane material properties for the PCB, the Sherlock application always considers a printed circuit board as isotropic when exporting an FEA model.

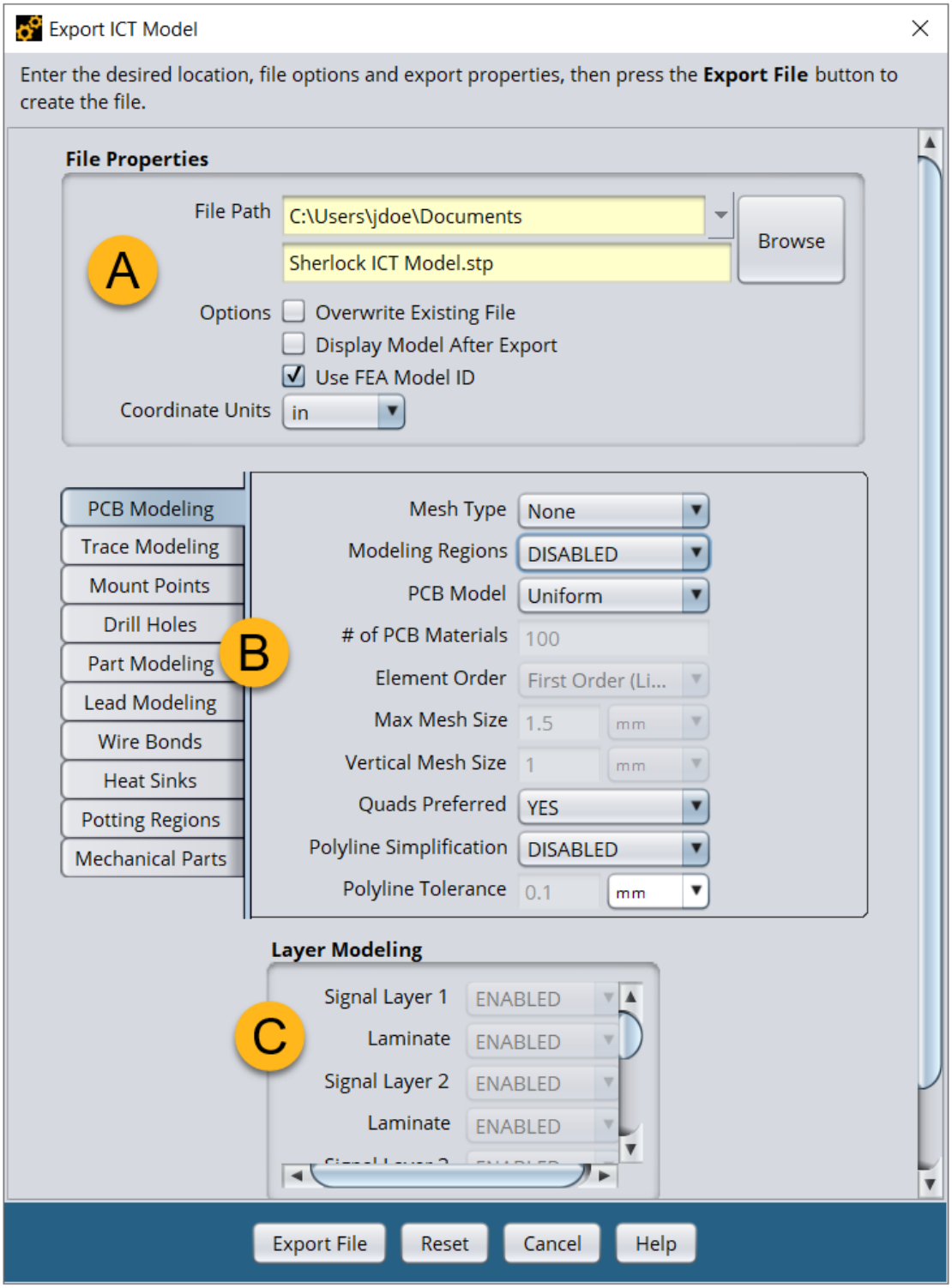
A geometric model can be exported for any FEA-based analysis task by right-clicking the analysis task icon in the **Project Tree** and selecting **Export FEA Model** from the context menu. At that point, the **Export Model** dialog will be displayed.



The **Export Model** dialog (shown below) is similar to the **Generate 3D Model** dialog in that you can specify which components are to be included in the model, as well as the individual modeling properties for each type of component. When generating a layered PCB model, you can also select the individual layers to be exported. Beginning with Sherlock 2023 R2, you can use the Modeling Regions

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

feature to define separate bodies for the purpose of exporting geometries. See the subsection below *Modeling Regions and STEP File Exports*.



Export <FEA> Model dialog options: See A in the image above.

- **File Properties**:

– **File Path:** Specify the location, name and type of the file to be exported. You can either enter the location and name of the file directly or use the **Browse** button to search for the appropriate export location in the file system. Sherlock supports the following types of export file formats based on the suffix of the export file name specified:

  → Abaqus Python Script (*.py)

  → Ansys APDL Script (*.apdl)

  → Ansys CDB Script (*.cdb)

  → Ansys Workbench Journal (*.wbjn) See note below regarding drill holes.

  → NX Nastran Bulk Data File (*.bdf)

  → STEP File (*.stp, *.step) The STEP file format can be used to export models from Sherlock to a wide range of 3D design and analysis packages. Also, see note below regarding drill holes.

  → STL (*.stl)

  Note:

  When exporting an FEA model to .wbjn or STEP file format, the Sherlock application exports the plating geometry for drill holes if the plating thickness is greater than zero and the plating material is copper.

– **Options**:

  → **Overwrite Existing File**: If enabled, Sherlock will automatically overwrite an existing file that has the same file name. Otherwise, Sherlock will ask for confirmation before overwriting the file.

  → **Display Model After Export**: If enabled, Sherlock will attempt to launch the application associated with the selected file format after the file has been exported.

  Sherlock uses the standard Windows file association mechanisms to launch the application, so you must define the associations (usually done by the application installer) for the **Display Model...** option to work.

  → **Use FEA Model ID:** This option enhances Sherlock-Workbench integration in situations where multiple circuit boards from different projects have the same name and would lead to name collisions in Ansys Mechanical. When enabled, Sherlock adds a unique prefix to all created Named Selections to uniquely identify each circuit board. A board's id can be found on the **Edit Circuit Card Details** properties form (right-click the circuit card in Sherlock's **Project Tree** and select **Circuit Card Properties**).

  Because the feature adds extra characters to the name and Ansys Mechanical has a 32-character limit for names, you should enable this option only when you are importing CCAs with identical names into Mechanical.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **Coordinate Units**: (in, mm, m, mil, micron, nm) This option appears when the file type selected is Ansys Workbench Journal (*.wbjn) or Abaqus Python Script (*.py). Use the dropdown menu to specify the coordinates to be used in the exported model. This feature replaces the **Scale to Microns** option for Abaqus exports in earlier versions of Sherlock.

→ **Clear FEA Data Base Before Defining Model**: This option only appears when the file type selected is Ansys APDL Script (*.apdl) or Ansys CDB Script (*.cdb). When selected, Sherlock adds `/clear` to the Mechanical APDL output.

- **Modeling Properties tabs:** See B in the image above.
  - **PCB Modeling** tab:
    → **Polyline Simplification**: When enabled, Sherlock attempts to eliminate close, adjacent points in the exported PCB outline. Use the **Polyline Tolerance** field to define "close". This feature is useful when arcs are present in the board outline that are comprised of many small segments.

    → **Polyline Tolerance**: This option is available when **Polyline Simplification** is enabled. Sherlock uses the distance you specify to determine if two points are deemed "close." For units, choose in, mm, m, mil, micron, or nm.

    → The other **PCB Modeling** options are explained in FEA- PCB Modeling (p. 473).
  - **Trace Modeling** tab: Refer to FEA- Trace Modeling (p. 495).
  - **Mount Points** tab: Refer to FEA- Mount Points and Fixtures (p. 479).
  - **Drill Holes** tab: Refer to FEA- Cutouts and Drill Holes (p. 489).
  - **Part Modeling** tab: Refer to FEA- Part Modeling (p. 552).
  - **Lead Modeling** tab: Refer to FEA- Lead Modeling (p. 563).
  - **Wire Bonds** tab: Refer to FEA- Wire Bonds (p. 607).
  - **Heat Sinks** tab: Refer to FEA- Heat Sink Modeling (p. 598).
  - **Potting Regions** tab: Refer to FEA- Potting & Staking (p. 616).
  - **Mechanical Parts** tab: Refer to FEA- Mechanical Parts (p. 629).
- **Layer Modeling panel**: See C in the image above.

  If a layered PCB model is generated (either **Layered** or **Layered Elements**), then the **Layer Modeling** panel can be used to selectively enable individual layers. This can be useful when testing the model for an individual layer or developing a model to be used for documentation or presentation purposes.

Note:

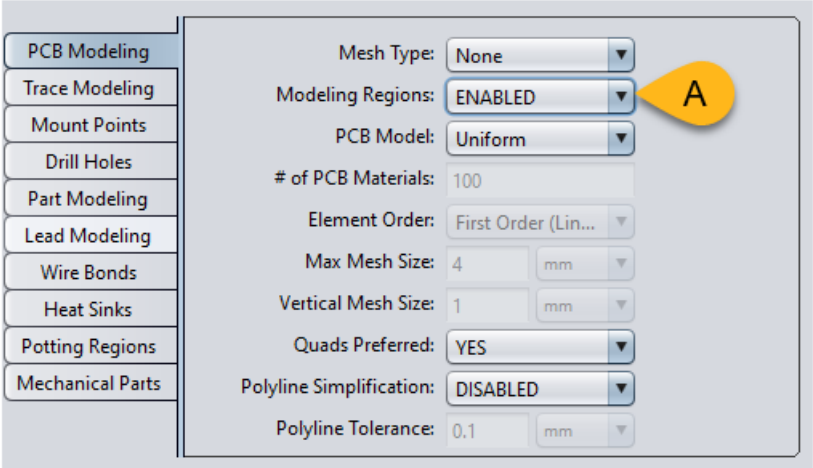
For each analysis task, the choices made in the **Generate 3D Model** dialog will be remembered from one invocation to the next.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeling Regions and STEP File Exports

You can now use the Modeling Regions (p. 523) feature to define distinct bodies within the geometry of a PCB for STEP file exports which include Workbench journal scripts.

If a Modeling Region is enabled and the **Export Model** dialog has **Modeling Regions** enabled (A, below), then each modeling region will be defined as a separate body in the export, and the area where the modeling region exists will be cut from the main body of the PCB.



Modeling Regions for geometry exports simply define the outline of the region and determine the modeling region state (**Enabled**, **Disabled**, or **Excluded**). The PCB Modeling Properties (p. 527) section of the **Modeling Region Editor** do not apply when exporting geometry models.

Similar to how the PCB is cut where the Modeling Region exists, if a Modeling Region itself contains another Modeling Region, the first region will be cut to make room for the body of the sub-region.

A Modeling Region which has a state of **Excluded** will result in a cutout in the export, and there will not be a body defined for the excluded region. Similar to how Modeling Regions in general are used, the Modeling Region can have traces enabled in the export even if the PCB has traces disabled.

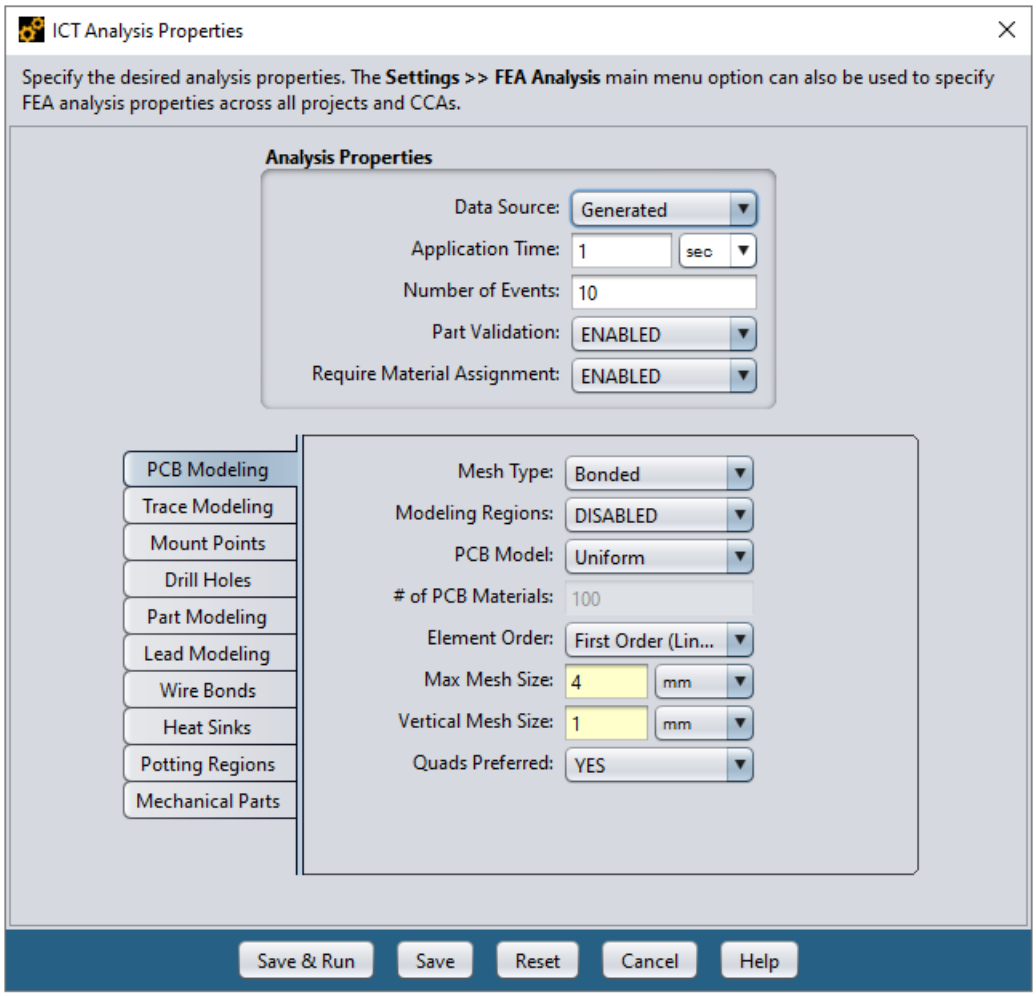
7.4.5. Run FEA Analysis

When running an FEA analysis task in Sherlock, the same basic approach is taken to define the components to be included in the FEA model as is done when generating the 3D model for review or exporting the 3D model to an FEA tool. The primary differences are:

- All PCB layers are automatically included in the model *and*
- Mount points are always included.

Other than those differences, all other aspects are the same and any input changes made in the **Analysis Properties** dialog will be remembered and used as the default values in all subsequent uses of the **Analysis Properties**, **Generate 3D Model** or **Export FEA Model dialogs.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7.5. FEA Processing

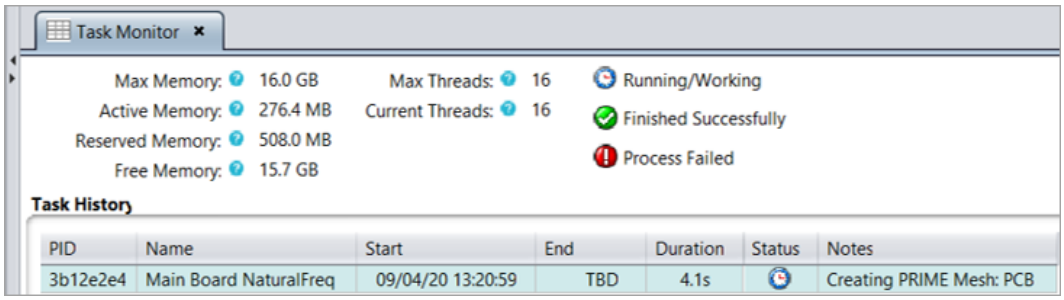
By their nature, FEA simulations may take a while to complete depending on the complexity of the model being analyzed, the type and number of constraints applied, and the analysis being performed. More importantly, FEA processing time is not a simple linear function, it is a non-linear function of the model complexity. That is, if you double the model complexity, the processing time could increase by a factor of four.

Sherlock is designed to help users manage such processing tasks by (a) launching separate FEA processes for each analysis task and (b) providing feedback for currently running tasks. Separate processes allow users to launch multiple analysis tasks at the same time, provided that their workstation has sufficient CPU and memory resources, and that their FEA engine license supports the number of instances required. The **Number of CPUs** property in the **FEA Settings** dialog controls the maximum number of CPUs to be used by each FEA engine.

To help manage background processing tasks, an hour-glass icon is displayed in the Project Tree next to the processing task node. When the FEA process has completed successfully, a GREEN check mark icon will be displayed next to the processing task node, indicating that the analysis results are ready for review.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also view the **Task Monitor** (accessed from the **Help** menu) to monitor all currently active processing tasks, as shown here:



The **Task Monitor** shows all background processing tasks and their current status. During FEA processing the **Notes** entry will change indicating which part of the analysis is being performed. The **Task Monitor** also shows the total amount of memory being consumed by Sherlock to process the results.

Memory Usage Fields in the **Sherlock Task Monitor**:

Max Memory is the maximum amount of memory available to Sherlock based on the Launcher Setting (p. 84) **Max Memory Size**. This number is usually slightly smaller than configured due to process overhead.

Active Memory is the total amount of memory Sherlock is currently using for processing.

Reserved Memory is the highwater mark of active memory that Sherlock has used for processing. Any reserved memory not being used is available to Sherlock for re-use before requesting additional memory from the operating system.

Free Memory is the total amount of unallocated memory and non-active reserved memory available to Sherlock.

When analyzing large projects, Sherlock may run slowly even if the computer has ample amounts of memory. This is likely a Java issue.

Note:

Since the FEA process itself is performed in a separate O/S process, the memory consumed by that process can only be viewed using O/S performance analysis tools, such as the standard Windows Task Manager.

7.6. FEA Results

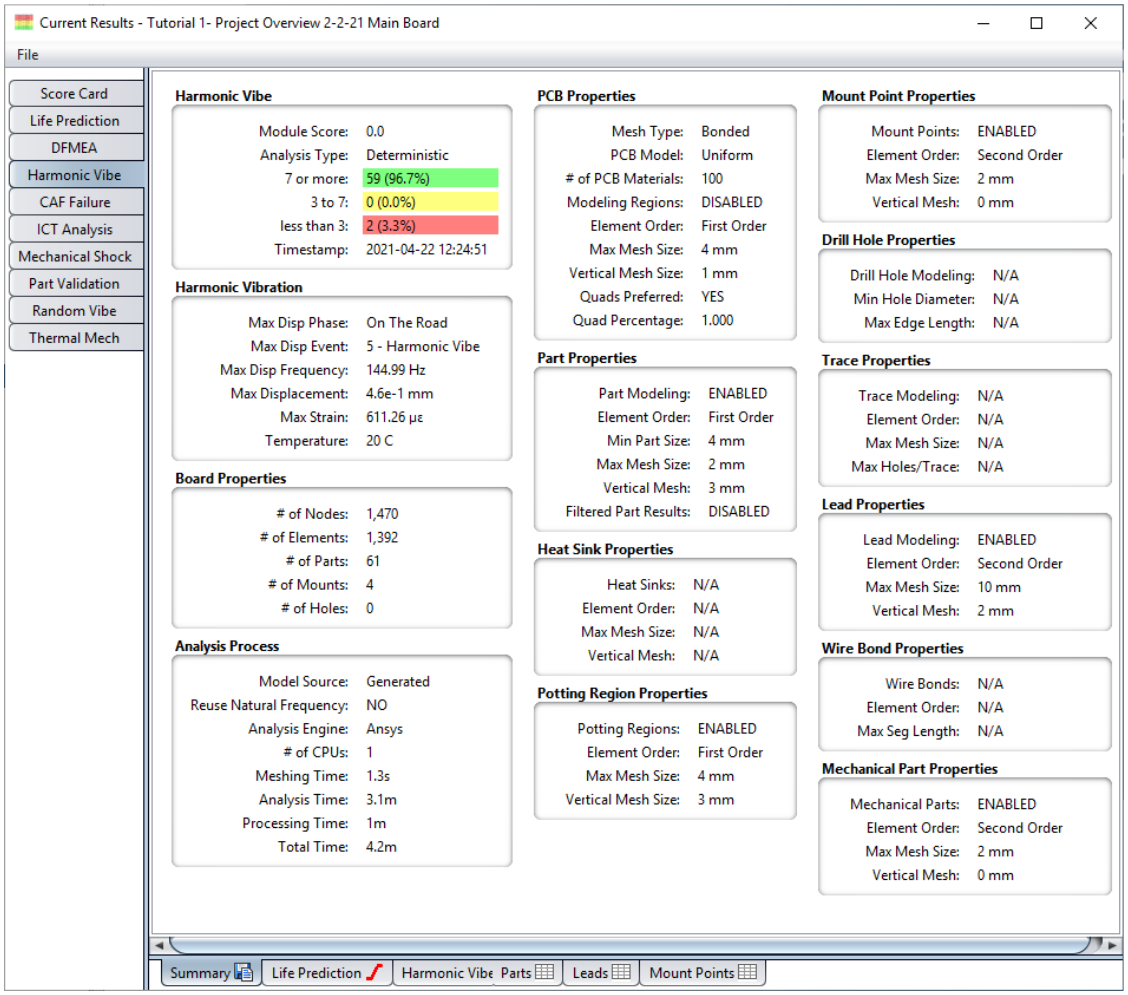
The analysis modules that use FEA techniques generate similar results, which usually includes a Summary Panel, one or more tabular results, one or more graphical results layers and a 3D results file. We now discuss some of the common features for these results.

7.6.1. Summary Panel
7.6.2. Tabular Results
7.6.3. Graphical Result Layers
7.6.4. 3D Results File
7.6.5. Log Files

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

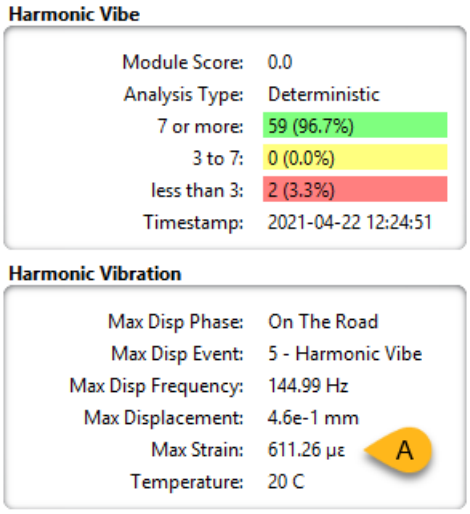
7.6.1. Summary Panel

In addition to module-specific summary results, each FEA module displays a summary of **Board Properties**, **PCB Properties** and **Analysis Process Times.** Such information is most useful when reviewing results at a later time to determine which model was being used during the analysis. Shown below is an example of a Harmonic Vibe result:



A closeup of the module-specific summary results is shown below. For Random Vibe, Harmonic Vibe, ICT, and Mechanical Shock, the overall Max Strain result (see A, below) is displayed. For Thermal Mech, max delta strain is displayed. Max strain the maximum PCB surface strain seen on either the top or bottom surface (applies to Ansys or Abaqus analysis only). For NX Nastran, the Max Strain represents the maximum nodal strain seen across the entire assembly.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Board Properties** section indicates the complexity of the FEA model used during the analysis. Processing time is generally related to the number of elements contained in the model. The number of parts, mount points, and holes are also provided for reference purposes. The number of nodes and elements here are for the PCB itself and does not include all other objects in the model. The numbers generated are specifically the number of nodes and elements generated for a single layer of the PCB. In this case, since the board is three layers, there are three times as many elements for the PCB as shown here.

The **PCB Properties** section indicates the settings used when the PCB model was created. In this case, a Bonded model was generated. The **Quads Preferred** option toggled to YES indicates that quad elements were desired in the mesh, as confirmed by the Quad Percentage value of 0.989.

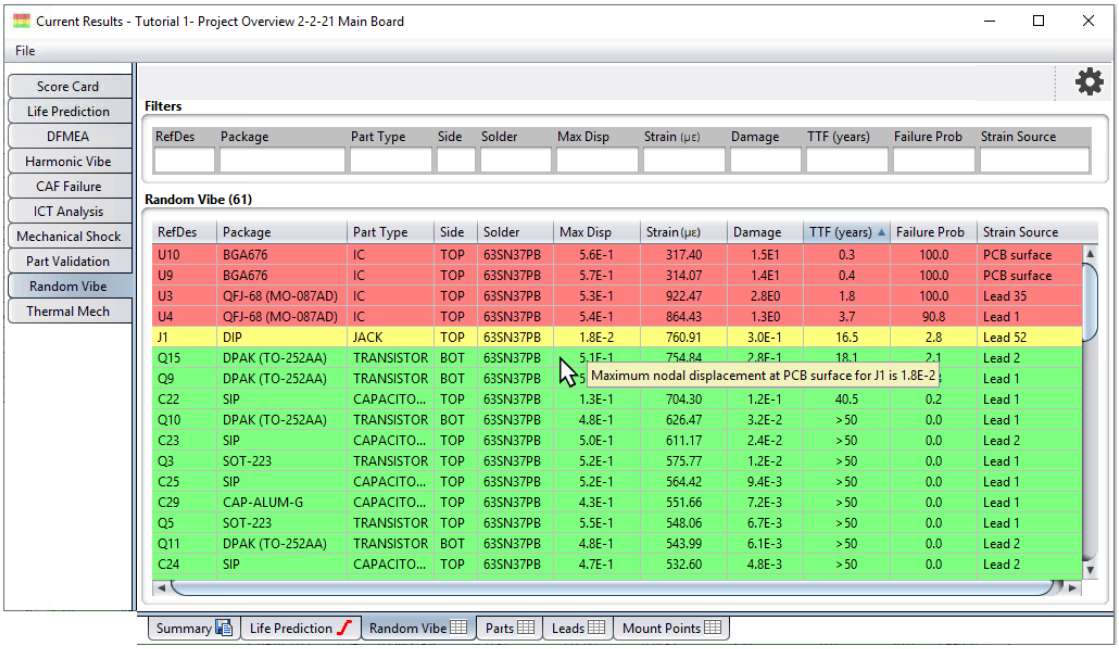
The **Part Properties** section indicates the setting used when the part models were created. In this case, the Min Part Size to be included in the analysis is 0 mm, indicating that all parts were included for analysis. You can also see how the part mesh settings differ from the PCB mesh settings, making both mesh settings customizable.

The **Analysis Process** section indicates which FEA engine was used for the analysis, the number of CPUs used and some of the key processing times. In this case, a majority of the time was spent analyzing the model, as opposed to processing the FEA results and generating the reliability results. As mentioned earlier, Ansys is the default analysis engine. If you choose to use NX Nastran or Abaqus, that engine will be shown in the analysis process panel when you run the analysis.

7.6.2. Tabular Results

If **Part Modeling** is enabled, you can filter the list to see only the parts that interest you. **Tabular FEA Results** are generated to show the maximum displacement and strain values experienced by each part during the simulation, as well as the score assigned to each part. For example, here's a tabular result for the Random Vibe module:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this case, most parts have been colored RED, indicating that they were assigned a score below 3.0. Such color coding provides a quick and easy way to determine which parts need to be examined in detail. By default, the following coloring scheme is used for FEA results:

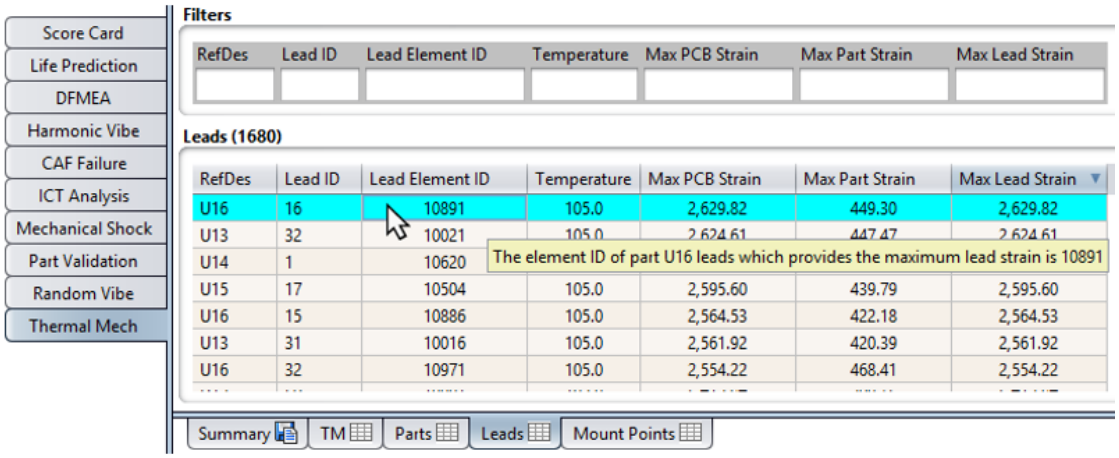
Score	Status / Color
7.0 or above	OK (green)
3.0 up to 7.0	Warning (yellow)
below 3.0	Error (red)

You can customize the score-color mapping by using the **Settings >Score** dialog to specify different cutoff values for each status color. The score-color settings will be used the next time that the analysis module is executed.

The Random Vibe, Harmonic Vibe, ICT, and Mechanical Shock results tables have the following additional features:

- The results tables include tooltips for most cells. For other modules, tooltips appear on some secondary results tables such as leads, mount points, and wire bonds.
- You can add or remove columns to customize the kind of results you want to see. Simply click the **gear icon** in the upper-right of the table to open the **Column Settings** dialog.
- The **Strain** column in the main results table indicates the maximum PCB surface strain for components without lead modeling. For leaded components, it is the maximum elemental mean strain in the lead.

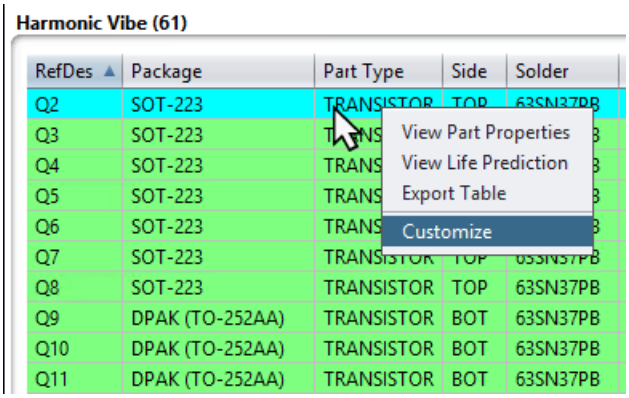
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The secondary results tables may be configured to include a **Lead Element ID** column. In the example above, the element reporting the Max Lead Strain for part U16 is element 10891.

- For tables that do not provide predictions, the **Color Settings** dialog (select **Settings > Color** from the main menu) allows you to set the table background color. Simply edit the colors for the **Result Table Row** and the **Result Table Alt Row**.

- The main results table may be configured to include the **Strain Source** column which is used to show the source of the **Strain** value for a given component. This will either be the PCB surface or the lead number which provides the largest strain for the component. To add or remove the **Strain Source** (or any) column, right-click any cell on the table and select **Customize**. This will open the **Column Settings** dialog.



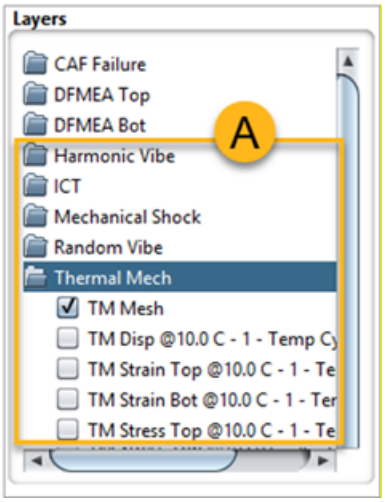
7.6.3. Graphical Result Layers

All FEA modules generate 2D graphical layers that show the scores assigned to each part and the displacement and strain values experienced by the PCB during the simulation.

Additionally, the 2D graphical results for Random Vibe, Harmonic Vibe, ICT, and Mechanical Shock have the following features:

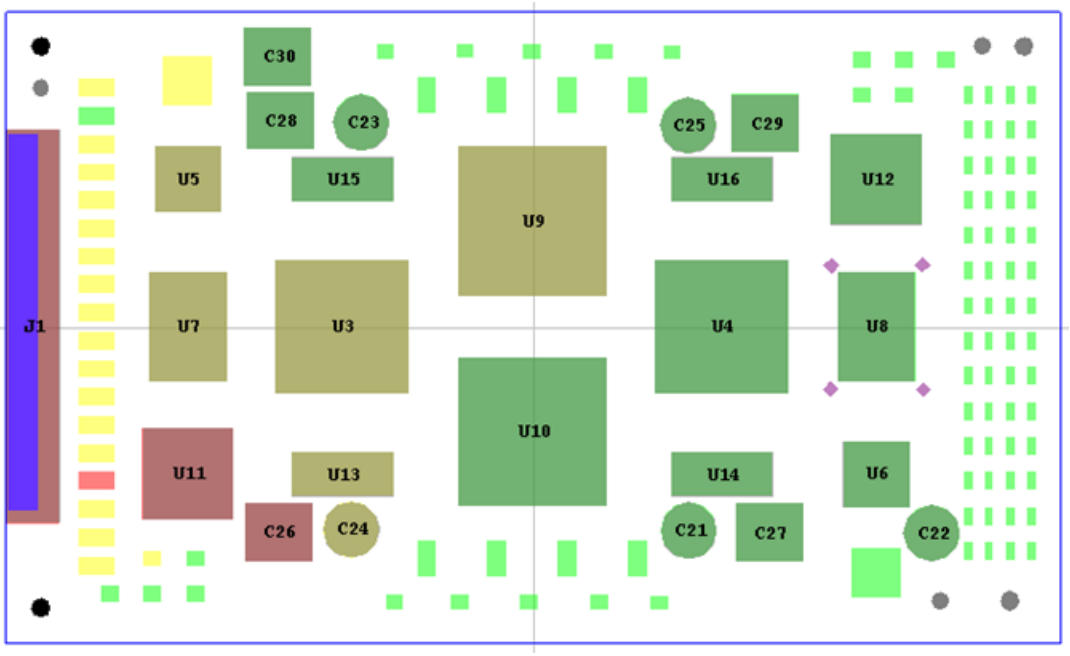
- When you run an FEA analysis, Sherlock creates a folder in the Layer Viewer for that analysis. (See A, below and the highlighted analysis folders: Harmonic Vibe, ICT, and so on.) These folders include items from FEA Scores, FEA Results, and other layers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



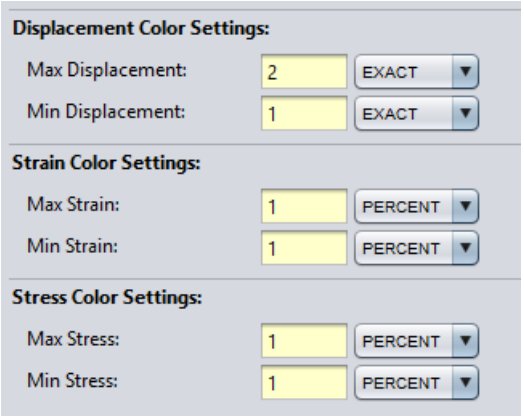
- FEA strain is reported in terms of microstrain. To change this setting, refer to the earlier section Analysis Properties (p. 448).
- FEA layer results contain both top and bottom layer strain results.
- FEA layer results contain top and bottom layer surface Von-Mises Equivalent Stress results.

Sherlock's 2D displays are useful for quickly determining which regions of the PCB need improvement. For example, the following layer shows the score assigned to each part as the result of an ICT analysis. The score-color mapping is the same as that used for the ICT tabular results.

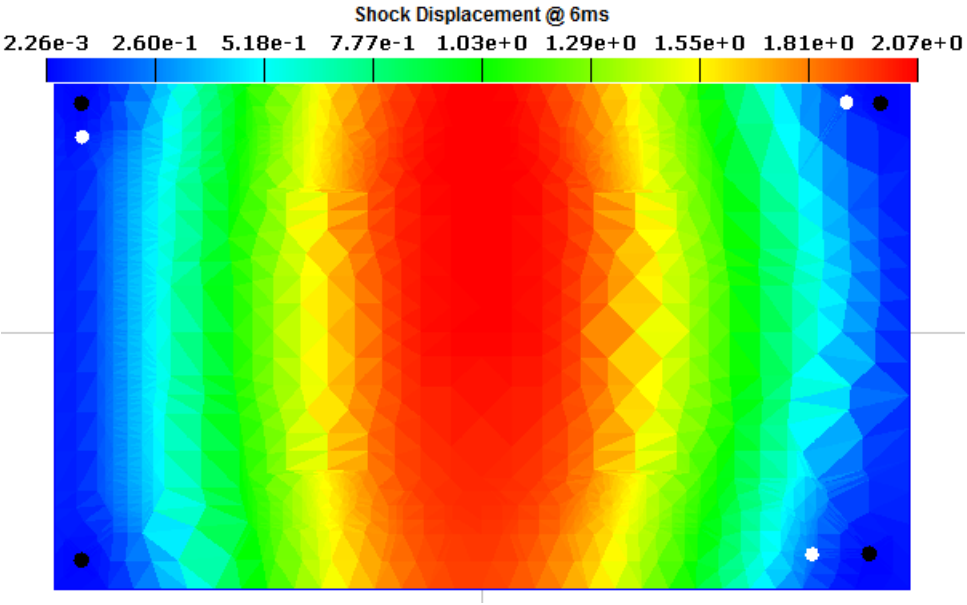


Displacement and strain results are displayed using a color spectrum to show the range of values across the entire PCB. The exact mapping of displacement values to the color range depends on the **Displacement Color Settings** defined in the **Settings > FEA Analysis** dialog, as shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specifically, all values above the **Max Displacement Value** will be colored RED, while all values below the **Min Displacement Value** will be colored BLUE. All values in between those values will be colored using the corresponding spectrum color. In this case, the min and max values are defined as percentages, indicating that color spectrum covers the middle 98% of the result values.



The **Display Quality** property found in the **Settings > Meshing** dialog can also be used to alter the display properties for 2D graphical result layers, supporting both **Coarse** and **Fine** display modes.

The smaller triangles with interpolated results not only smooth out the colored regions, but they provide better clarity. The fine layer clearly shows areas of high strain along the edge of each IC, which are not as apparent in the coarse rendering. All results are the same, just easier to see when using the fine setting.

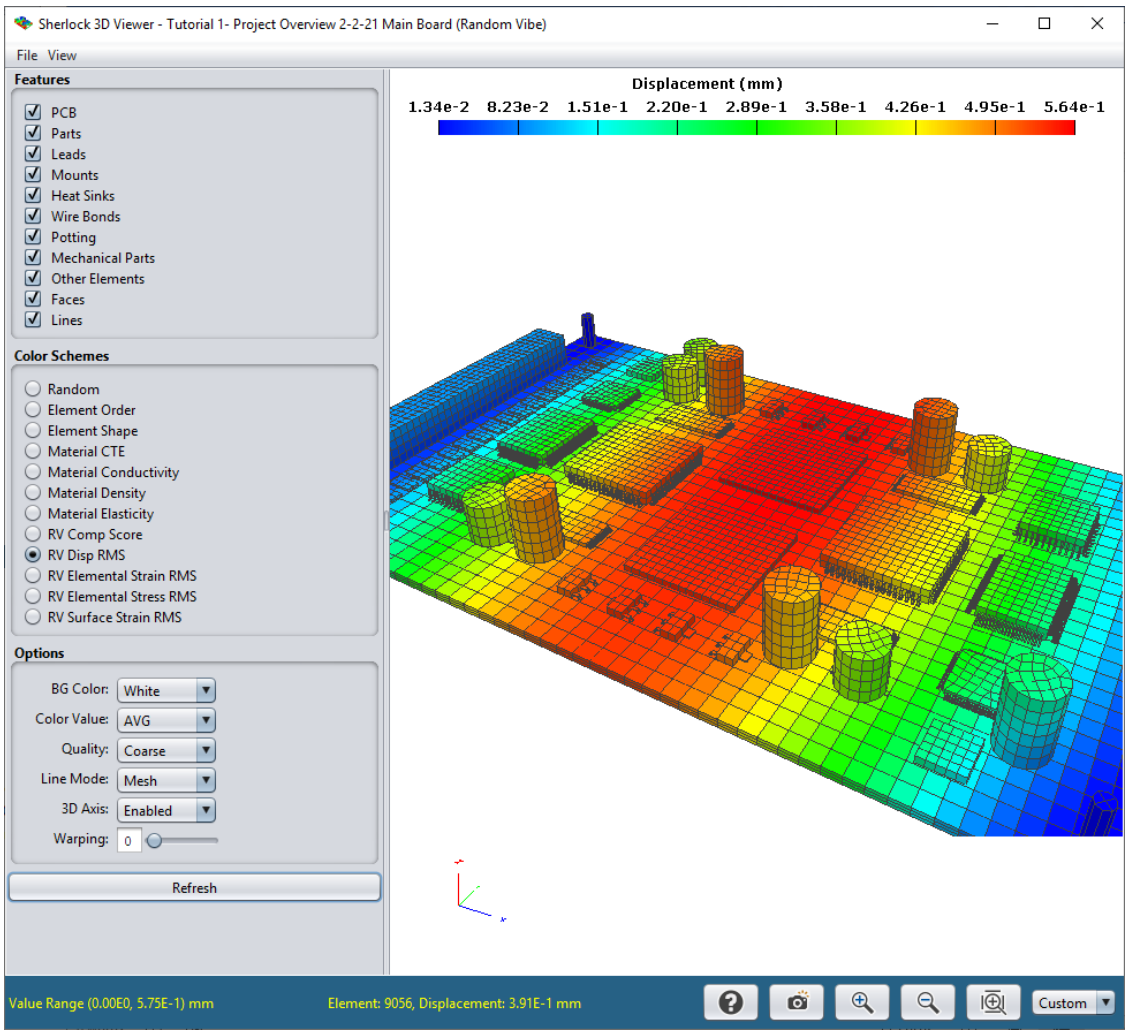
Note:

Coarse/Fine settings do not apply to the Random Vibe, Harmonic Vibe, ICT, Mechanical Shock, or Thermal Mech modules. The analysis results for these modules are based on elemental averaging and colored accordingly. Likewise, the Color Value option does not apply to Strain and Stress results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.4. 3D Results File

In addition to the 2D graphical results layers, FEA modules also generate a 3D results file containing the complete 3D elemental model, as well as color mappings for each of the analysis results. For example, here's a 3D representation of the tutorial board:



The 3D model and/or results can be viewed by right-clicking any FEA module entry in the **Project Tree** and selecting the **View 3D Model** or **View 3D Results** options in the context menu. See Viewing and Managing Analysis Results (p. 785) for more details.

7.6.5. Log Files

If the **Show FEA Logs** option is enabled in the **Settings > FEA Analysis** dialog under **Engine Properties**, then a **Log Panel** will be displayed along with the other analysis results engine during the analysis process. Advanced users can review such log data to determine if everything was processed as expected and/or to trouble-shoot FEA processing issues.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 8: FEA Modeling Guide

This chapter provides detailed instructions on how to model the various parts of the circuit board for a finite element analysis: the PCB, mount points, traces, leads, and so on. For an introduction to the basic concepts of FEA modeling in Sherlock, see the *FEA Models* section (p. 453) in the *FEA Overview* which covers helpful topics such as FEA modeling properties in Sherlock, generating a 3D model, and exporting an FEA model.

In this section, the following topics are covered:

8.1. FEA- PCB Modeling

8.2. FEA- Mount Points and Fixtures

8.3. FEA- Cutouts and Drill Holes

8.4. FEA- Trace Modeling

8.5. FEA- Image File Layers

8.6. FEA- PCB Modeling Regions

8.7. FEA- Part Modeling

8.8. FEA- Lead Modeling

8.9. FEA- Heat Sink Modeling

8.10. FEA- Wire Bonds

8.11. FEA- Potting & Staking

8.12. FEA- Mechanical Parts

8.13. FEA- Sub-Assembly Analysis

8.14. Advanced BGA Modeling

8.1. FEA- PCB Modeling

Ansys Sherlock creates a 3D Finite Element Analysis (FEA) model using the circuit card data extracted from 2D layer files, Pick & Place files, the Parts List and from the PCB stack up information. The type of model created depends on the tool that will be used to analyze the model.

If the PCB model is being used by Sherlock to automatically perform an FE analysis task, an Elemental Model will be generated. Elemental Models consist of a collection of 3D elements (either wedges or bricks) that fill the volume contained by the PCB outline, minus any holes or cutouts. The shape of the elements is determined automatically by Sherlock using an internal meshing algorithm. Sherlock can generate a Elemental Models that are Bonded to support a range of analysis approaches. In a Bonded Mesh Model (p. 453), Sherlock meshes the PCB independently of the parts, allowing both the PCB and part elements to be more uniformly shaped.

If the PCB model is being exported by Sherlock to an FEA tool for analysis outside of Sherlock, a geometric model can be generated. Geometric models consist of a collection of 3D volumes that are defined

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

using the input commands provided by a given FEA tool. Unlike Elemental Models, that define the actual elements used for the FE analysis, the volumes defined by the geometric models only define the shape of the objects to be modeled. Then, the native meshing algorithms provided by the FEA tool are used to generate the elements required for analysis.

You can choose to export the Bonded Elemental Models to Abaqus, Ansys Mechanical and NX Nastran FEA tools. These tools can then import the orphan mesh created by Sherlock and incorporate the elements into a larger model or simply perform custom analysis on the mesh model created by Sherlock. Also, you can choose to export a hybrid model that includes mesh elements (created by Sherlock) for some regions of the PCB and geometric volumes (meshed by the FEA tool) for other regions of the PCB. See the section on Modeling Regions (p. 523) for more details.

This section describes how the Bonded Models are generated by Sherlock and how you can control what is contained in those models.

Before you begin

Before getting into the details of how best to mesh your circuit board, observe the following tips.

- If you plan to export your model for analysis in Ansys Mechanical, be aware that Trace Reinforcement Model Export Automation (p. 509) currently supports only Layered PCB Models (p. 475).
- Sometimes the outline of an imported circuit board is defined by an excessive number of points—that is, many more points than are needed to accurately define the board outline. These extra vertices can lead to meshing issues. If you encounter this problem, use Sherlock's Polyline Simplification tool which is explained in Board Outline Editor (p. 51) in the *Layer Viewer Tutorial*.

In this section, the following topics are covered:

8.1.1. PCB Models

8.1.2. PCB Modeling Properties

8.1.3. PCB Material Management

8.1.4. PCB Model Tradeoffs

8.1.1. PCB Models

Regardless of the type of model being generated (Bonded or Geometric), Sherlock automatically assigns material properties to each of the PCB elements/volumes based on the percentage of metal, resin, and laminate materials contained in the PCB layers. That is, Sherlock determines the average material properties for each region of the PCB based on all the known design data, creates virtual materials in the FEA input deck, and then assigns those materials to the appropriate elements/volumes defined in the model. This eliminates one of the most time-consuming and error-prone manual tasks required for FE analysis of PCBs, making it one of Sherlock's most valuable features.

The exact method used to determine the average material properties for each PCB region depends on how the PCB regions are defined. Sherlock supports four different approaches for modeling the PCB, including the following PCB models:

8.1.1.1. Uniform Model

8.1.1.2. Layered Model

8.1.1.3. Uniform Elements Model

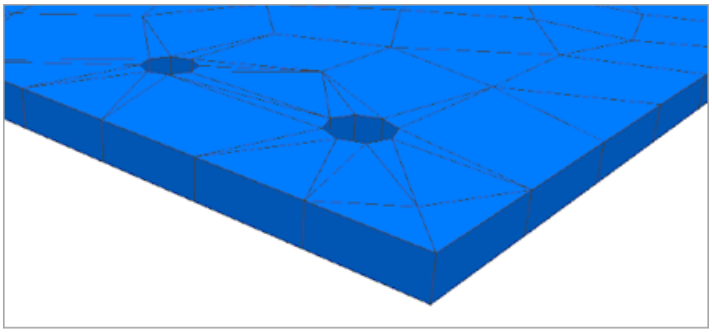
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.1.4. Layered Elements Model

Each PCB model defines both how the PCB elements are created and how material properties are assigned to each element. We now discuss the differences between these models.

8.1.1.1. Uniform Model

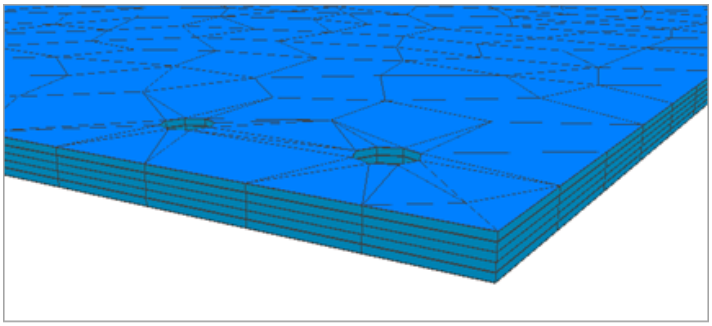
The Uniform Model creates a homogeneous set of elements to model the PCB. Each element spans the vertical space from the bottom face of the PCB to the top face of the PCB and the same material properties are assigned to each of the PCB elements, as shown in the following example.



The material properties are based on the percentage of metal, resin and laminate materials found throughout all layers of the PCB. Numeric values for the uniform material properties can be found in the Stackup panel.

8.1.1.2. Layered Model

The Layered Model creates individual elements for all PCB layers and assigns the same material properties to all elements on a given layer. As shown in the following example, which is color coded by elasticity, the thin signal layers are sandwiched between the relatively thicker laminate layers.

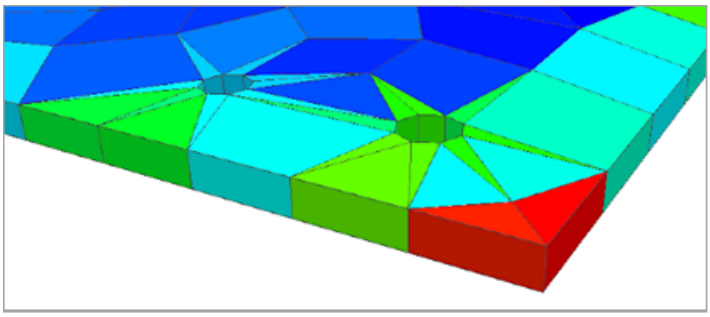


The material properties for each layer are based on the percentage of metal, resin and laminate materials found in each layer. The elements defined for each layer share nodes with the elements in the layers above and below, eliminating the need for surface ties in the FEA model.

8.1.1.3. Uniform Elements Model

The Uniform Elements Model creates elements like the Uniform Model that span the entire thickness of the PCB, but it assigns individual material properties to each element based on the percentage of materials covered by the element itself.

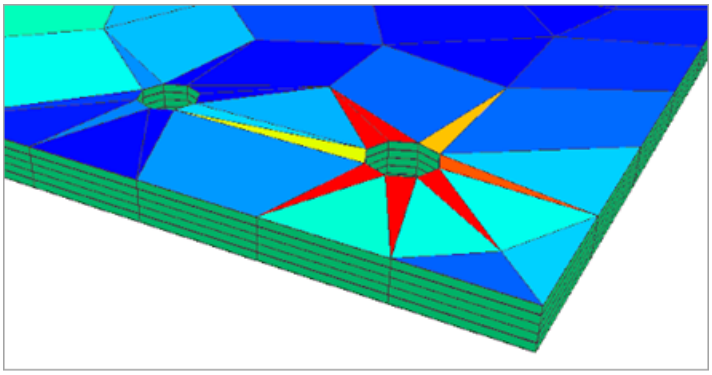
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



As shown in the example above, which is color coded by elasticity, the Uniform Elements Model generates the same number of elements as the Uniform Model but assigns more localized material properties.

8.1.1.4. Layered Elements Model

The Layered Elements Model creates elements like the Layered Model that only span a given PCB layer, but it assigns individual material properties to each element based on the percentage of materials covered by the element itself.

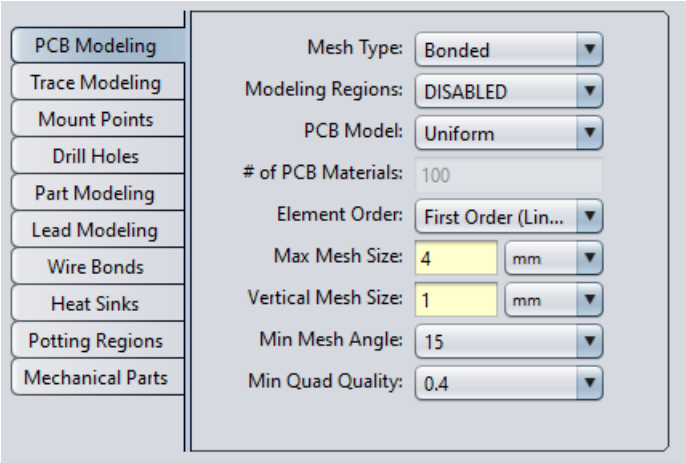


This model is the most detailed PCB model provided by Sherlock, generating the greatest number of PCB elements and the most localized material properties. The elements defined for each layer share nodes with the elements in the layers above and below, eliminating the need for surface ties in the FEA model.

8.1.2. PCB Modeling Properties

All the FEA models generated by Sherlock depend on a collection of PCB Modeling Properties that define how the elements/volumes are generated. Some of these properties are ignored when exporting a geometric model, depending on the specific FEA tool being targeted.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The **Mesh Type** property specifies the type of model being generated, **Bonded** or **None**.

  The **Bonded** choice is available when executing FE analysis directly from Sherlock or when exporting a Calculix Input Script using the **Export FEA Model** dialog.

  The **None** choice is available when exporting Geometric Models for specific FEA tools or when exporting for non-vendor specific formats such as STEP. The **None** choice means that just the Geometric Model, without meshing commands, will be imported.

- The **PCB Model** property specifies how the elements and material properties are defined for the PCB, based on the currently defined layer and stackup properties, as described above.

- The **Element Order** property indicates the type of FEA element to be used to model the PCB. First order elements (that is, 8 node bricks and 6 node wedges) result in reduced analysis time, but stiffer elements. Second order elements (that is, 20 node bricks and 15 node wedges) more accurately model bending and twisting of the PCB but result in longer analysis times. Shell elements provide the fastest analysis times, in exchange for a cruder approximation of the mechanical properties of the PCB.

- The **Max Mesh Size** property specifies the desired size of the elements to be generated. The smaller the Max Mesh Size, the more accurate the results, but the greater the amount of memory and CPU processing required during the processing. In general, you should use a large mesh size initially to generate approximate results using a relatively small amount of memory and CPU resources. When you need more accurate results, you can then decrease the mesh size and re-run the analysis.

- The **Vertical Mesh Size** property specifies the desired height of the elements used to model the entire PCB (for Uniform and Uniform Elements models) or for each layer (for Layered and Layered Elements models).

- The **Quads Preferred** property specifies whether the meshing engine should attempt to generate quad-shaped or hex elements when creating the mesh. If quad-shaped elements cannot be generated, then triangular elements will be used.

8.1.3. PCB Material Management

The Uniform Elements and Layered Elements Models can generate many different PCB materials depending on the metal, resin and laminate materials assigned to each layer, as well as the percentage

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of metal contained by each element on each layer. Large numbers of material definitions are generally not a problem for the FEA engines supported by Sherlock, but they make it more difficult for users to manually view and/or modify the resulting 3D models. Furthermore, the small differences between the generated material properties often have little or no impact on the FEA results. Therefore, users can control the number of PCB materials generated by specifying the **# of PCB Materials** property whenever generating a Bonded 3D model.

The **# of PCB Materials** value is used by Sherlock to quantize the conductor percentage used when computing the material properties of each element on each layer. For example, if a value of 10 is specified, then at most 11 unique percentages (0.0, 0.1, 0.2, ... 1.0) will be used to combine the conductor and dielectric material properties for each element on each layer. The actual percentages determined by processing each layer will be rounded to the nearest quantized percentage before computing the material properties.

The **# of PCB Materials** property only controls the number of unique conductor percentages used on each layer, it does NOT set the upper limit for the number of PCB materials generated across all layers. The actual number of PCB materials required is determined by the value (N) specified by the user, the specific conductor and dielectric materials assigned to each layer, and the PCB model being generated, as we now explain.

For the Layered Elements Model, the approach described above ensures that at most N+1 unique materials will be generated for a given layer. If the same conductor and dielectric materials are used across a number of layers, then all of the layers will share the same N+1 material definitions. Layers that use different conductor or dielectric materials will usually generate a different set of at most N+1 material definitions. Therefore, for a PCB with L layers, the maximum number of materials generated is L * (N+1) in the situation when all layers have different materials assigned.

For the Uniform Elements model, the total number of materials generated depends on the unique combinations of conductor percentages that are computed for each element and each layer. For example, consider a single mesh element for a two-layer board (2 signal layers and 1 laminate layer). The conductor percentages inside of that element might be 50% on top, 0% in the middle and 20% on the bottom. Since the Uniform Elements model averages material properties across all layers for each element, the resulting material can be represented by the tuple (50, 0, 20). Since each entry in the tuple can take on N+1 values, the total number of combinations can theoretically be (N+1) * (N+1) * (N+1). The exact number depends on the specific materials assigned to each layer and the number of unique tuples generated. Sherlock automatically combines resulting materials that are numerically similar, but you should always expect to see more materials generated for the Uniform Elements model than for the Layer Elements model because of combinatorial complexity.

8.1.4. PCB Model Tradeoffs

The choice of PCB model to be used comes down to a basic tradeoff between accuracy of results and analysis time. For example, the following table shows some results for running an ICT analysis of the relatively simple tutorial board with a 10 mm mesh when using all four PCB models.

Table 8.1: 10mm ICT Analysis

	Number of Elements	Meshing Time (sec)	Analysis Time (sec)	Maximum Displacement	Maximum Strain
Uniform	1,316	2.4	1.5	3.1 mm	6.6
Layered	10,029	2.4	30.5	3.2 mm	6.5

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Uniform Elements	1,319	13.3	1.6	3.1 mm	6.6
Layered Elements	10,008	13.4	29.4	3.2 mm	8.3

Of the two tradeoff properties, analysis time is the easiest to predict. In general, the fewer the number of PCB elements, the faster the FE analysis time will be. As such, the Uniform and Uniform Elements Models will almost always result in lower analysis times than the Layered and Layered Elements Models. In the example above, the Uniform models generate roughly 1,300 elements and take only 1.5 seconds to analyze, as compared to 10,000 elements and 30 seconds for the Layered models.

Although the analysis time is the most important time for real-world circuit card modeling, the meshing time is also a direct function of the type of PCB model used. Specifically, the Uniform Models take less time to mesh because only a few average material properties need to be determined, whereas the Elements Models need to compute individual material properties for all elements. Individual material properties are computed by examining all PCB layers and determining the percentage of metal, resin and laminate materials that fall inside of each element outline.

It is difficult to make a general conclusion about the accuracy of results provided by the various PCB models. Such a determination depends on the actual distribution of materials throughout a PCB, as well as the placement of all parts, mount points and loads. In the example above, based on the relatively simple tutorial board, the first three models yield basically the same maximum displacement and strain results, while the Layered Elements model yields a higher maximum strain value. Of course, maximum displacement and strain values don't tell the whole story either, because Sherlock uses localized strain values to predict component failures. In general, the models that assign localized material properties (Uniform Elements and Layered Elements) should yield the most realistic FEA results, but it is always prudent to compare the values from each model to real-world test results to see which correlates the best.

Such tradeoffs also exist for the Element Order and Element Shape characteristics of the generated model. It is sometimes wise to use second order elements for parts, while using only first order or shell elements for the PCB itself to reduce overall analysis type. Similarly, depending on the FEA tool being used, wedge elements may be accurate enough for preliminary testing purposes, while helping to reduce overall analysis time.

8.2. FEA- Mount Points and Fixtures

For Ansys Sherlock to analyze a circuit card using FEA simulations, the circuit card must be constrained in one or more places in order to ensure that the circuit card doesn't oscillate wildly during the simulation. In the real-world, mount points are represented by a wide variety of components, including mechanical screws, plastic stanchions, aluminum standoffs, edge-connectors, etc. In Sherlock, such constraints are defined by one or more **Mount Points** or **Fixtures** that are attached to the circuit card. Furthermore, to reduce data input and FEA processing complexity, Sherlock defines four different types of mount points, including:

- Mount Hole
- Mount Pad
- Standoff

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Support Pin

In most situations, these four types of mount points are enough to model real-world constraints on the circuit card. This document discusses how mount points are defined and used in Sherlock for FEA processing.

Note:

In Sherlock, the terms **Mount Point** and **Fixture** are essentially synonymous, except that Fixtures are used only by the ICT Analysis module, whereas Mount Points are used by all other FEA modules.

In this section, the following topics are covered:

8.2.1. Mount Point & Fixture Types

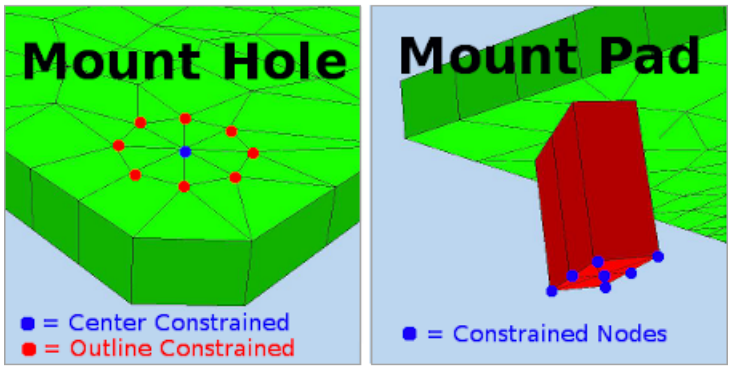
8.2.2. Mount Point Management

8.2.3. Fixture Management

8.2.1. Mount Point & Fixture Types

The following types of mount points and fixtures are supported in Sherlock:

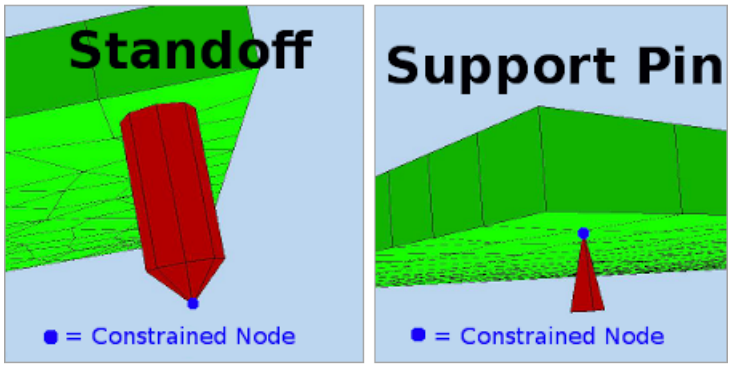
A **Mount Hole** is modeled as a collection of constrained points on the PCB surface. Mount Holes can be constrained by either the Center or Outline nodes used to define the hole. A rigid body constraint connects the Center node to the Outline nodes in the FEA model. As such, both approaches effectively prevent the PCB from rotating at the hole location.



A **Mount Pad** is modeled as a 3D solid with a given length, width, height and material. A Mount Pad is attached at one end to the top or bottom of the PCB and is constrained by all nodes on the face furthest from the PCB, thereby preventing the PCB from rotating around that face.

A **Standoff** is modeled the same as a Mount Pad, except that it is constrained by only the center point of the face furthest from the PCB. The PCB is free to rotate around the constrained center node.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A Support Pin is modeled as a single constrained point on the PCB surface. The PCB is free to rotate around the constrained node.

By combining one or more of these mount point types, with appropriate constraints, users can model a large number of circuit card constraint situations.

8.2.2. Mount Point Management

Mount points are viewed and edited using the circuit card Layer Viewer.

8.2.2.1. Viewing Mount Points
8.2.2.2. Adding Mount Points
8.2.2.3. Modifying Mount Points
8.2.2.4. Mount Point Properties
8.2.2.5. Mount Point Multi-Edit
8.2.2.6. Saving Mount Point Changes
8.2.2.7. Mount Point Results

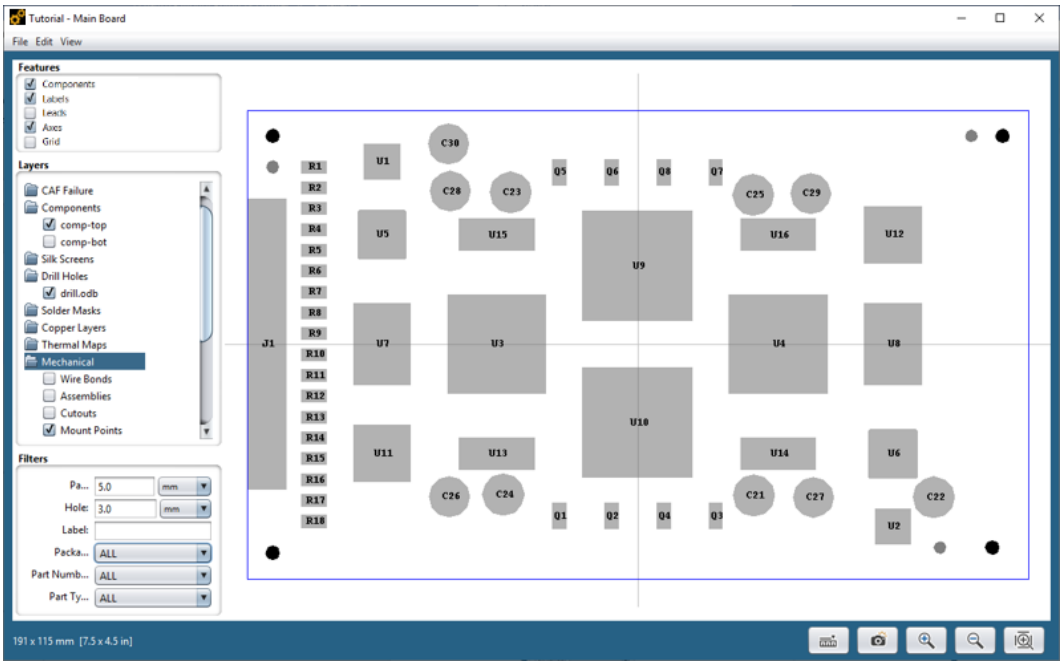
8.2.2.1. Viewing Mount Points

To display all currently defined mount points:

- Double-click the **Layers** node in the project tree to display the Layer Viewer.
- Expand the **Mechanical** folder in the **Layers** list as necessary.
- Enable the **Mount Points** layer.

By default, mount points are displayed as solid black elements in the **Layer Viewer**. As shown in this example, mount points can be seen in each of the four corners of the circuit card.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The gray circles next to some of the mount points are large diameter drill holes that are not currently designated as mount points.

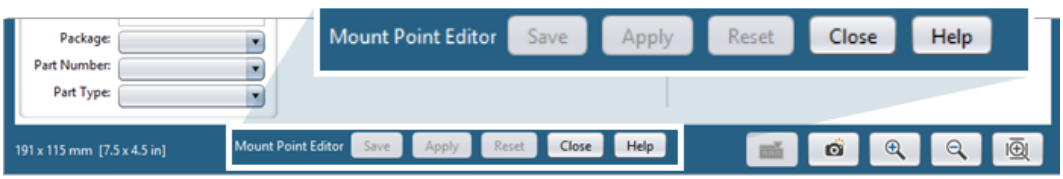
Note:

By default, mount points are displayed as black regions in the Layer Viewer. You can customize the color using the **Color Settings** dialog. Under **Settings** in main menu, click the **Color** menu option.

Sherlock automatically designates large drill holes as mount points whenever a drill file is processed AND no mount points are currently defined. If you have to re-parse the drill file to correct a problem, you should delete all mount holes prior to re-parsing the drill file so that the newly parsed drill holes will be used.

8.2.2.2. Adding Mount Points

To add a mount point to the circuit card, select the **Edit > Edit Mount Points** option in the **Layer Viewer** menu. At that point, the **Mount Point Editor** buttons will appear at the bottom of the **Layer Viewer** as shown here:



A mount point can be added either by:

- Left-clicking an existing drill hole while holding down the SHIFT key
- Or right-clicking in the circuit card and selecting **Add Mount Point** from the context menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When adding a drill hole as a mount point, a black circle will be displayed and the mount point will be automatically selected, allowing you to edit properties.

When using the Add Mount Point menu option, the mount point settings dialog will be displayed immediately, with the current mouse location automatically entered as the center point for the new mount point. You are free to alter any of the mount point properties, including the center location, as discussed in the next section. When the **Save** button is pressed, the new mount point will be created.

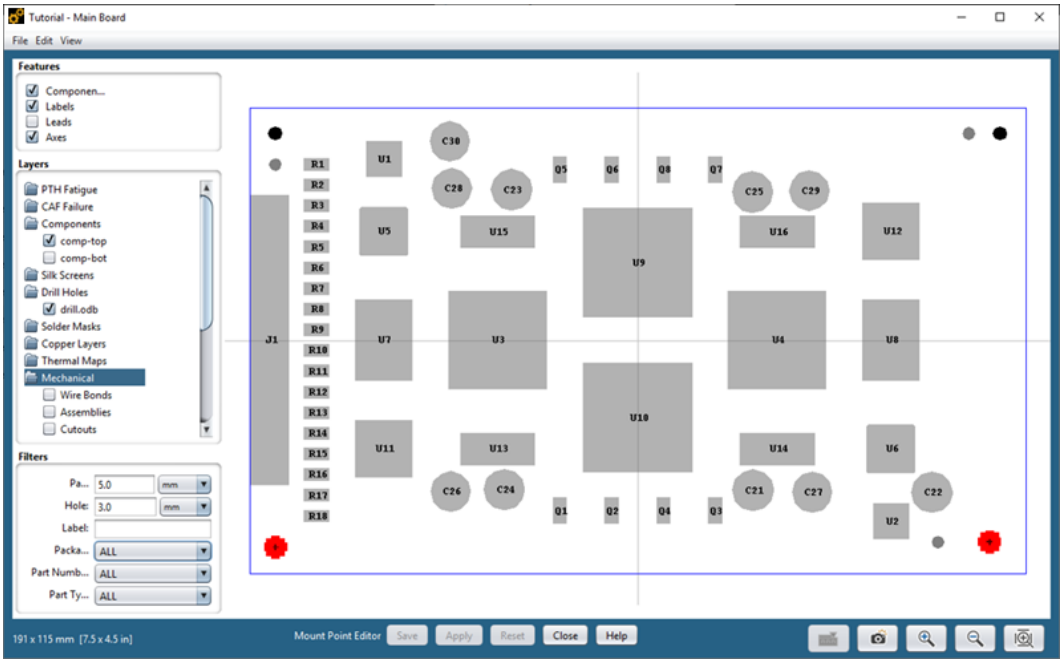
When adding a mount point with shape **Polygonal** after clicking the **Save** button in the dialog, use Shift-Left-click to define each of the nodes that make up the shape of the mount point. When the mount point is complete, click the **Done** button in the editor and the mount point will be created.

8.2.2.3. Modifying Mount Points

When the **Mount Point Editor** mode is enabled, the following mouse controls can be used to select or modify mount points and fixtures:

- Shift-Left-click any mount point to select only that mount point
- Shift-Left-click and drag to select one or more mount points within a rectangular region
- Control-Left-click to add/remove a mount point from the selection set

Selected mount points and fixtures will be displayed with a red border and control points. Once selected, mount points and fixtures can be moved to a new location by left-clicking inside any of the selected mount points or fixtures and dragging them to the desired location. When more than one mount point or fixture is selected, all objects will be translated by the same amount, thereby staying in the same relative positions.



The red control points can be used to graphically scale an individual object. The exact transformation performed depends on the shape of the object being scaled. For example, with a rectangular mount point, the point opposite that selected will remain fixed, while the selected control point and the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

two adjacent control points are scaled to match the current mouse location. On the other hand, with a circular mount point, when any control point is dragged, the center of the mount point will remain fixed and the diameter of the mount point will be scaled to match the current mouse location. Such an approach makes it easier to translate and scale mount points as needed to graphically place them in a desired location and orientation relative to other PCB features.

In addition to the mouse actions used to select mount points and fixtures, the following mouse actions can be used to add new mount points or edit properties for existing mount points:

- Right-click anywhere in the PCB to add a new mount point.
- Right-click a mount point to display the context menu.

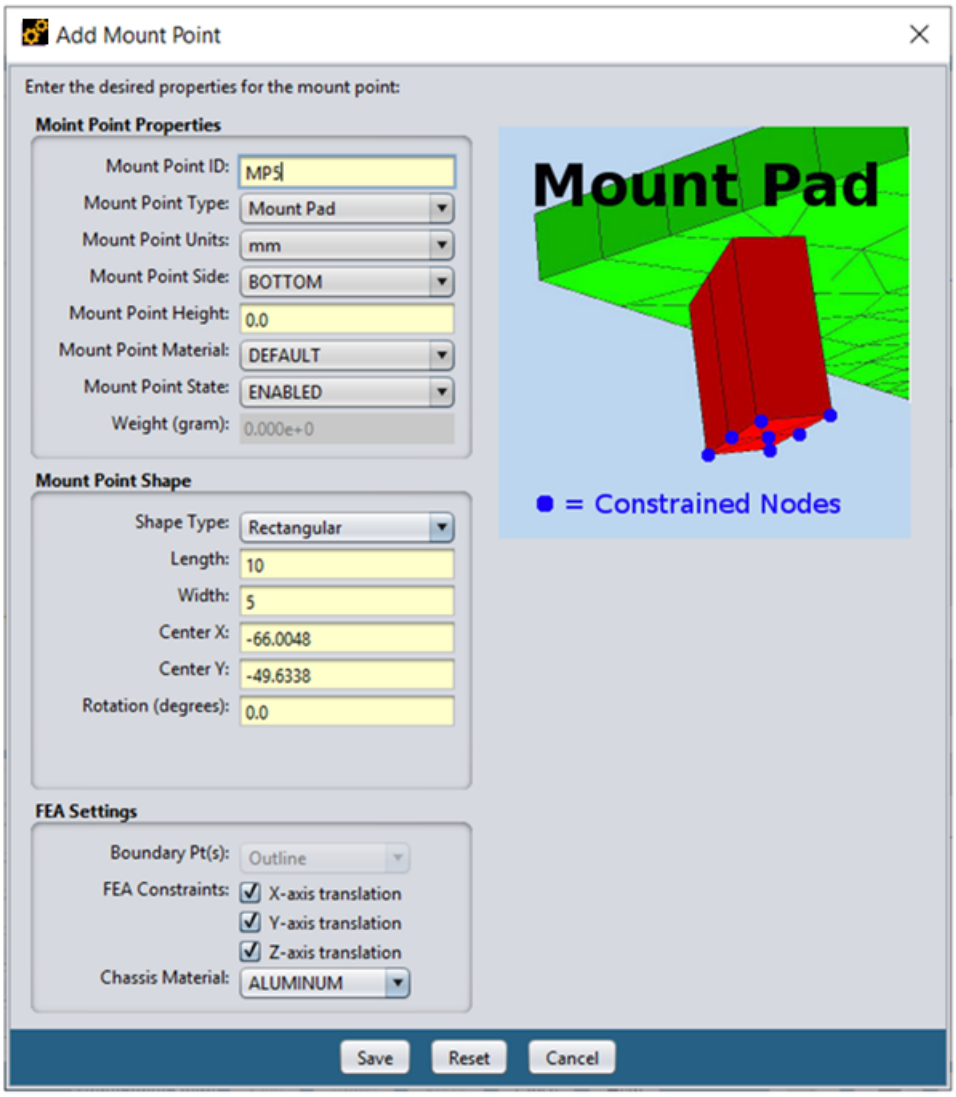
The mount point context menu provides options for:

- Editing mount point properties
- Moving, scaling and rotating mount points
- Merging mount points
- Adding, copying and deleting mount points
- Suppressing mount points from FEA models

8.2.2.4. Mount Point Properties

The **Mount Point Properties** dialog is displayed whenever the **Add Mount Point** or **Edit Properties** options are selected in the mount point context menu, allowing users to specify exact shape and location properties, as well as the FEA constraints associated with the mount point.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The exact forms and fields displayed in the property dialog depend on the type of mount point being viewed.

The **Mount Point State** property allows a mount point to remain defined on the circuit card, but not be included in any FEA models.

The **Shape Type** property defines the geometric shape of the mount point, which controls how it can be edited. The following shape types are supported:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 8.2: Shape Types

Rectangular	Defined by length and width dimensions, a center location and an angle of rotation.
Circular	Defined by a diameter, number of nodes, a center location and an angle of rotation.
Slot	Defined by length and width dimensions, number of nodes, a center location and an angle of rotation.
Polygonal	Defined by a set of node locations and an angle of rotation.
PCB	Defined by the outline of the PCB. Only available for mount pads and standoffs.

The Circular and Slot shapes allow you to specify the number of nodes to use when approximating arcs used to form those shapes. For circular shapes, the nodes are evenly spaced along the circumference of the circle. For slot shapes, 4 nodes are used to define the rectangular part of the slot and the remaining nodes are evenly spaced along the semi-circles on both ends of the slot.

For all shapes, an angle of rotation can be specified to rotate the shape in the XY plane. When scaling shapes graphically using the control points, the angle of rotation is preserved, and the shape dimensions are automatically updated to reflect the scale operation.

The **FEA Settings** form lists the types of constraints that can be applied to the mount point. As discussed above, the application of the constraints in the FEA model depends on the **Type** of mount point specified. Rotational constraints are applied by selecting the appropriate type.

For mount pads and standoffs, the estimated weight is displayed when viewing mount point properties. This is a read-only property only intended for reference.

8.2.2.5. Mount Point Multi-Edit

Multiple mount points may be selected, and the properties of those mount points updated at the same time. When the properties for more than one mount point are edited at the same time, any mount point property which varies between the selected mount points will display the value <*VARIOUS*>. Any field which is left as the value <*VARIOUS*> when saving the form will leave the original mount point value specified for that given field. Any other field values will be assigned to all selected mount points.

When editing multiple mount points, if the original mount points had different FEA constraints assigned, then the FEA constraints for all the mount points being modified will not be able to be modified. This field will be disabled.

Multiple mount points may only be edited at the same time if the mount points are of the same type and shape type.

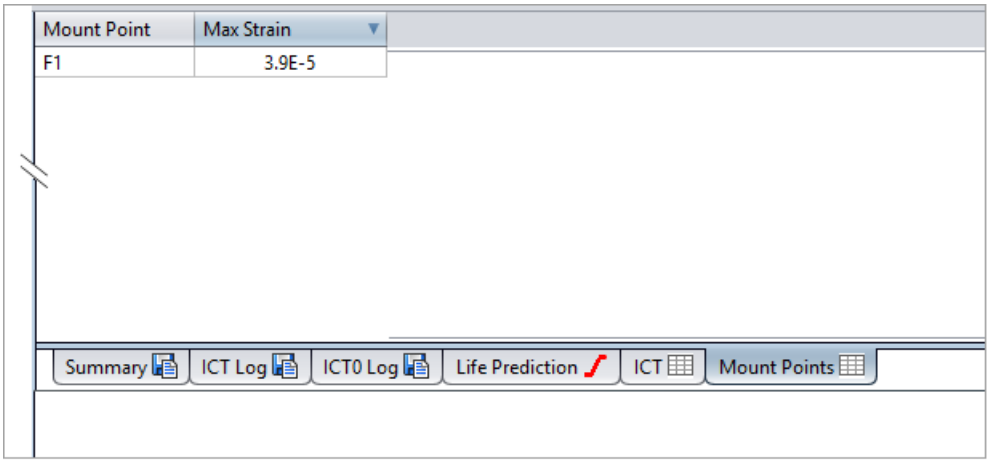
8.2.2.6. Saving Mount Point Changes

After adding, modifying or deleting mount points, you should press the **Save** or **Apply** buttons at the bottom of the **Layer Viewer** to save your changes. The **Apply** button will save the changes but keep the editor open, allowing you to make additional changes. The **Save** button saves the changes and automatically closes the editor.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.2.2.7. Mount Point Results

When mount pads or standoffs are enabled for FE analysis, the corresponding analysis results for mechanical shock, vibration, ICT, and thermal mechanical will each include a results table for each mount point or fixture.



8.2.3. Fixture Management

ICT Fixtures are viewed and edited using the circuit card **Layer Viewer**. As noted at the beginning of the section, since **ICT Fixtures** and **Mount Points** are essentially synonymous, the **Fixture Editor** works the same as the **Mount Point Editor** as previously described.

8.2.3.1. Viewing Fixtures

8.2.3.2. Adding Fixtures

8.2.3.3. Modifying Fixtures

8.2.3.4. Using Mount Points as Fixtures

8.2.3.5. Saving Fixture Changes

8.2.3.1. Viewing Fixtures

To display all currently defined fixtures:

1. Double-click the **Layers** node in the project tree to display the **Layer Viewer.**
2. Expand the **Mechanical** folder in the **Layers** list as necessary.
3. Enable the **Fixtures** layer.

Note:

By default, fixtures are displayed as solid purple elements in the **Layer Viewer**. When selected, a fixture is highlighted in red. In the example below, one fixture can be seen on the left side of the circuit card:

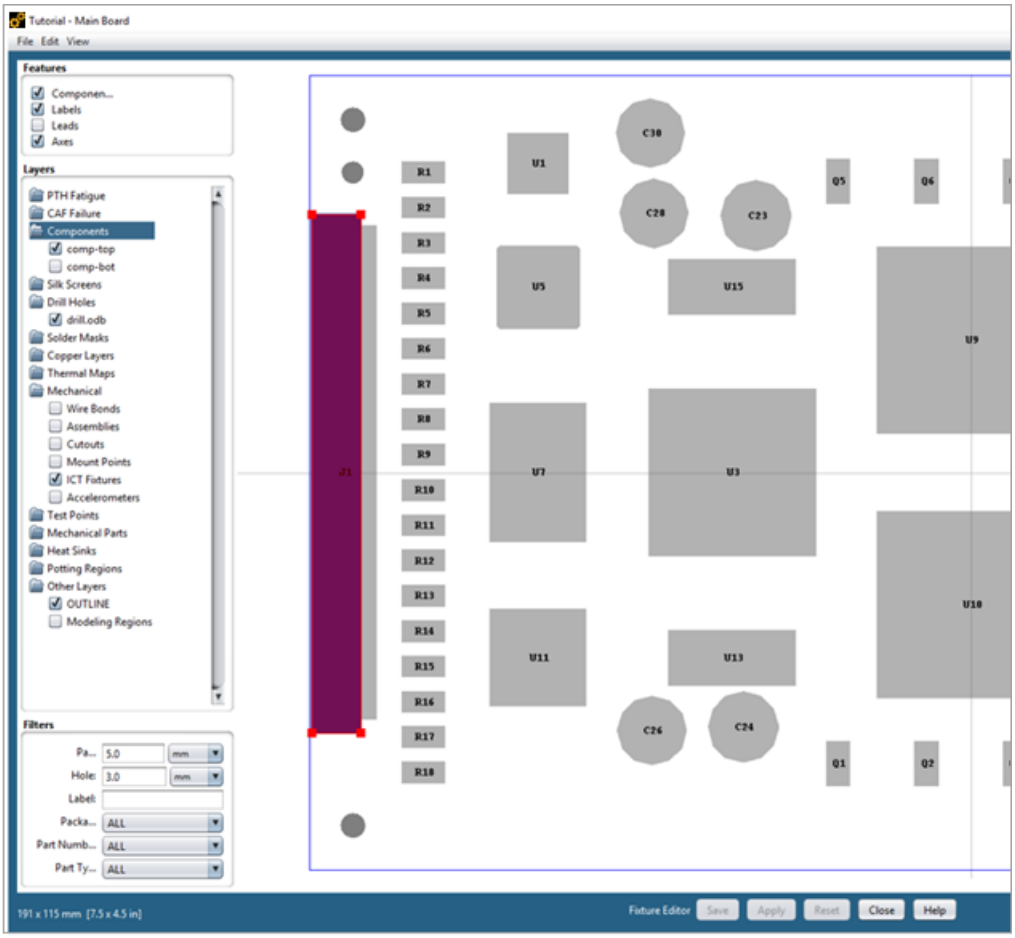
By default, fixtures are displayed as purple regions in the Layer Viewer. You can customize the color using the **Settings > Color Settings** option in the **Main Menu.**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.2.3.2. Adding Fixtures

To add a fixture to the circuit card, select the **Edit > Edit ICT Fixtures** option in the Layer Viewer menu. At that point, the **Fixture Editor** buttons will appear at the bottom of the **Layer Viewer.**

To add a fixture, simply right-click the circuit card at the approximate location for the fixture and select **Add Fixture** from the context menu. At that point, the **Add Fixture** dialog will appear, indicating the default properties to be assigned to the new fixture. You may define a fixture similar to how a mount point is defined.



8.2.3.3. Modifying Fixtures

As detailed previously for **Modifying Mount Points**, the fixture editor behaves the same way with all the same controls and properties as defined for mount points.

8.2.3.4. Using Mount Points as Fixtures

Fixtures may also be created by copying data from the existing mount points. You may select one or more mount points in the fixture editor, use the right-click menu to select **Copy as ICT Fixture.** It will use the selected mount points data to create new fixtures. In the fixture editor you may only select a set of mount points or a set of fixtures, may not have mix selections.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.2.3.5. Saving Fixture Changes

After adding, modifying or deleting fixtures, you should press the **Save** or **Apply** buttons at the bottom of the **Layer Viewer** to save your changes. The **Apply** button will save the changes but keep the editor open, allowing you to make additional changes. The **Save** button saves the changes and automatically closes the editor.

Note:

When importing ICT results, *at least one* fixture must be defined for Sherlock to post-process these results and provide time-to-failure predictions.

8.3. FEA- Cutouts and Drill Holes

Ansys Sherlock models two types of PCB voids: namely cutouts and drill holes. In this section, we describe how these modeling components are defined and how they can be used when generating 3D FEA models.

In this section, the following topics are covered:

8.3.1. Cutouts

8.3.2. Drill Holes

8.3.1. Cutouts

Cutouts may be circular, rectangular or any polygonal shape. Cutouts may overlap the PCB outline, thereby modifying the PCB outline itself, and may be edited manually and/or loaded from a cutout definition file. Cutouts are always included by Sherlock when generating an FEA model.

8.3.1.1. Cutout Viewing

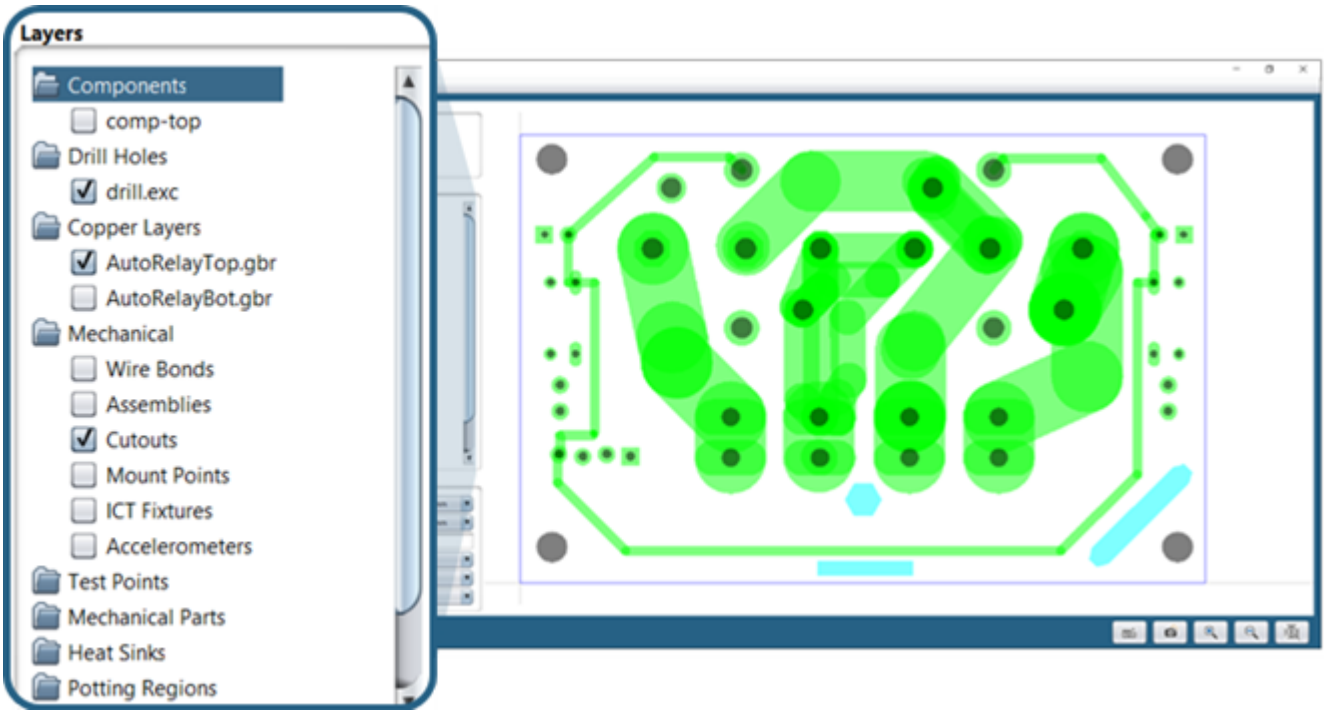
8.3.1.2. Cutout Editing

8.3.1.3. Cutout Properties

8.3.1.1. Cutout Viewing

For this user guide, we will be working on the Auto Relay project. To open the Auto Relay project, import the Auto Relay Project.zip file located in the Sherlock tutorial folder. Cutouts can be viewed in the Layer Viewer by enabling the **Mechanical > Cutouts** selection in the **Layers** control panel. Shown in this example, cutouts are displayed using the light cyan color and they can take on a variety of shapes and orientations.

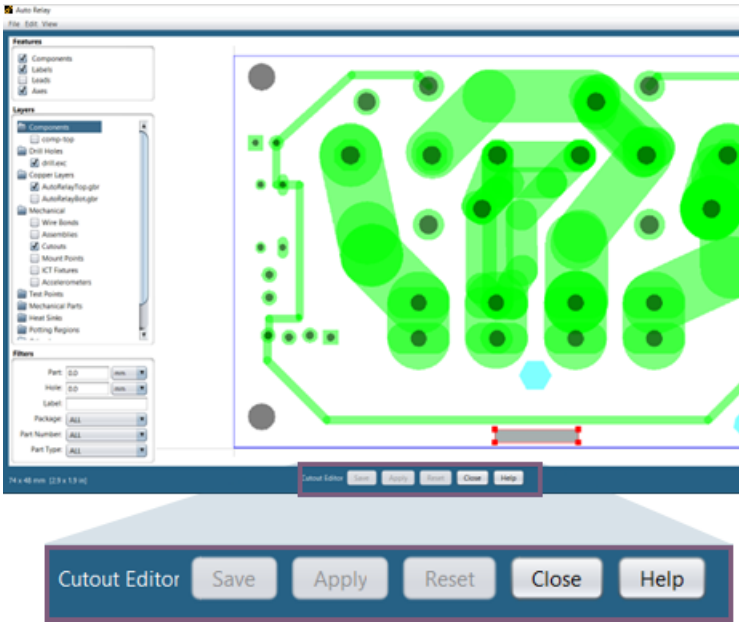
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this example, there is a rectangular cutout at bottom-center, a circular cutout just above it and a slot shaped cutout rotated 45 degrees in the lower-right section of the PCB.

8.3.1.2. Cutout Editing

Cutouts can be added, deleted, and edited using the **Edit > Edit Cutouts** menu option available in the **Layer Viewer** to enter the **Cutout Editor** mode. As shown in this example, cutouts are displayed using the light cyan color and they can take on a variety of shapes and orientations.



When the **Cutout Editor** mode is enabled, the following mouse controls can be used to select or modify cutouts:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Shift-Left-click any cutout to select only that cutout.
- Shift-Left-click and drag to select one or more cutouts within a rectangular region.
- Control-Left-click to add/remove a cutout from the selection set.

Selected cutouts will be displayed with a red border and control points. Once selected, cutouts can be moved to a new location by left-clicking inside any of the selected cutouts and dragging them to the desired location. When more than one cutout is selected, all cutouts will be translated by the same amount, thereby staying in the same relative positions.

The red control points can be used to graphically scale an individual cutout. The exact transformation performed depends on the shape of the cutout being scaled. For example, with a rectangular cutout, the point opposite that selected will remain fixed, while the selected control point and the two adjacent control points are scaled to match the current mouse location. On the other hand, with a circular cutout, when any control point is dragged, the center of the cutout will remain fixed and the diameter of the cutout will be scaled to match the current mouse location. Such an approach makes it easier to translate and scale cutouts as needed to graphically place them in a desired location and orientation relative to other PCB features.

In addition to the mouse actions used to select cutouts, the following mouse actions can be used to add new cutouts or edit properties for existing cutouts:

- Right-click anywhere in the PCB to add a new cutout
- Right-click a cutout to display the context menu

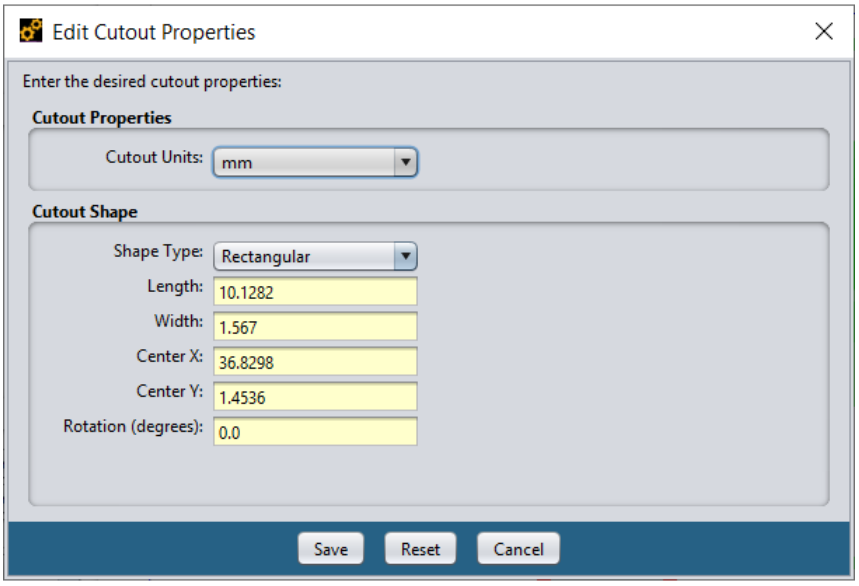
The cutout context menu provides options for:

- Editing cutout properties
- Moving, scaling and rotating cutouts
- Merging cutouts
- Adding, copying and deleting cutouts
- Converting a cutout to a mount point

8.3.1.3. Cutout Properties

The **Cutout Properties** dialog is displayed whenever the **Add Cutout** or **Edit Properties** options are selected in the cutout context menu, allowing users to specify exact shape and location properties as needed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Shape Type** property defines the geometric shape of the cutout, which controls how it can be edited. The following shape types are supported:

Table 8.3: Shape Types

Rectangular	Defined by length and width dimensions, a center location and an angle of rotation.
Circular	Defined by a diameter, number of nodes, a center location and an angle of rotation.
Slot	Defined by length and width dimensions, number of nodes, a center location and an angle of rotation.
Polygonal	Defined by a set of node locations and an angle of rotation.

The Circular and Slot shapes allow you to specify the number of nodes to use when approximating arcs used to form those shapes. For circular shapes, the nodes are evenly spaced along the circumference of the circle. For slot shapes, 4 nodes are used to define the rectangular part of the slot and the remaining nodes are evenly spaced along the semi-circles on both ends of the slot.

For all shapes, an angle of rotation can be specified to rotate the shape in the XY plane. When scaling shapes graphically using the control points, the angle of rotation is preserved, and the shape dimensions are automatically updated to reflect the scale operation.

8.3.2. Drill Holes

Drill holes are always circular and are defined in one or more drill hole definition files. Drill holes may not be entered or edited manually, but they may be included or excluded from an FEA model as needed or filtered based on their size.

8.3.2.1. Drill Hole Files

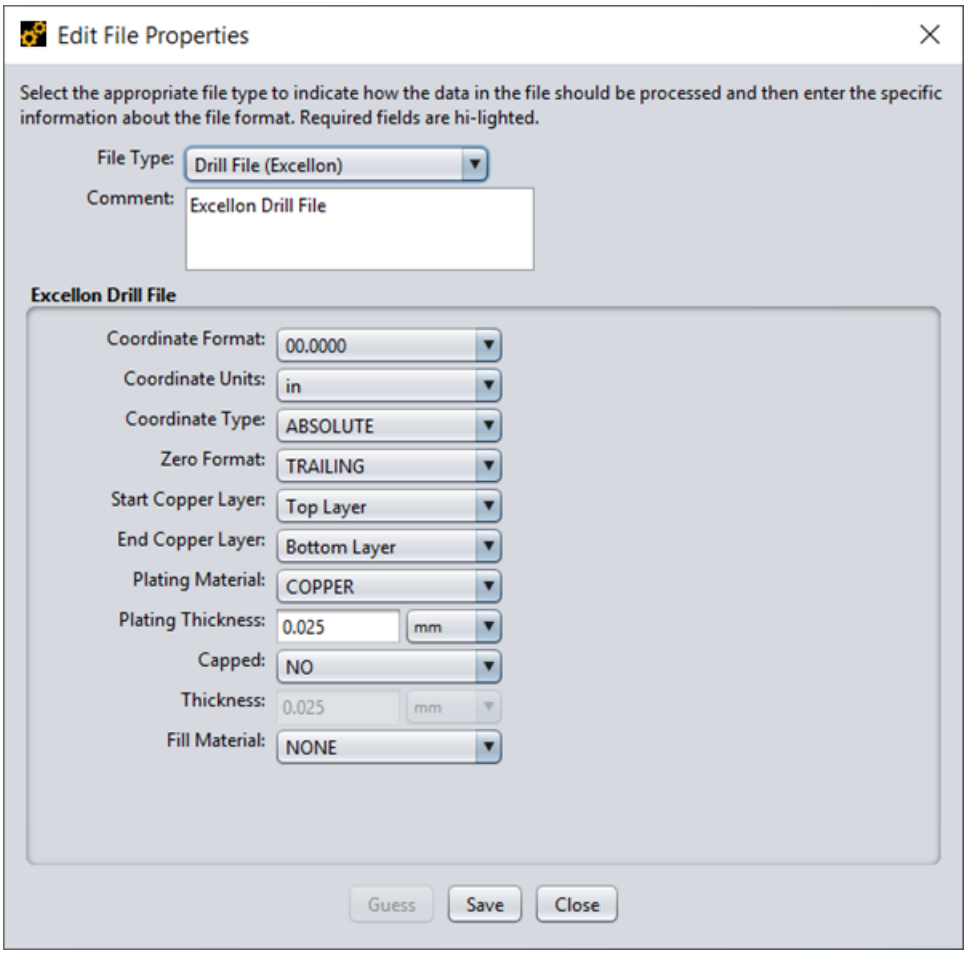
8.3.2.2. Drill Hole Modeling

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.2.1. Drill Hole Files

Drill holes are defined in Sherlock by adding one or more drill hole definition files to a CCA. When importing archive files like ODB++ and ODB XML, the drill hole files are automatically pulled from the archive and added by Sherlock during the import process. Holes that are not through the entire board are only supported by the Trace Reinforcement Export (p. 506) feature, not for FEA model export or analysis.

To control how a drill hole file is parsed, right-click the file name, in this case **drill.exc** in the **Files** folder found in the Project Tree and select the **Edit Properties** menu option. At that point, the **Edit File Properties** dialog will appear, and you should select the appropriate File Type for the file being parsed. In the example shown here, an Excellon Drill File is being parsed.



The **Coordinate Format** and **Coordinate Units** properties specify the format of the units used in the file. Typically, the file contains commands that are used by Sherlock to automatically override the values specified in this dialog. In some cases, however, the drill file may not contain any indication of the coordinate format being used. Typically, the EDA tool that generated the drill file generates a human-readable text file that includes the required information.

The **Coordinate Type** property indicates if the movement commands contained within the drill file use absolute or relative coordinates when moving from one hole to another. In almost all cases, absolute coordinates are used, which is the default property value.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drill files save disk space by dropping either leading or trailing zeros from coordinates, which is specified by the **Zero Format** property. In most cases, trailing zeros are dropped, which is the default property value.

Start Copper Layer and **End Copper Layer** are the layers where the copper begins and ends respectively. This information may be contained in the ODB++ archive or manually entered by the user. Sherlock will automatically detect the start and end layer of each drill hole for ODB ++ and IPC-2581 archives only. For all other supported hole formats, the user must specify this information in the **Drill Hole File Properties** dialog.

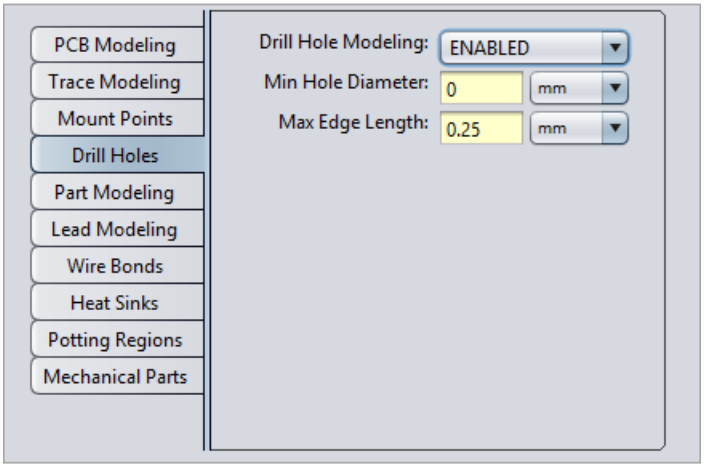
Plating Material and **Plating Thickness** specify the material and its thickness used to plate the hole.

The **Capped** property indicates whether the drill hole is effectively open or closed. If **Yes** is selected, a layer of plating material is added on each end.

Fill Material, which is valid only for buried/core vias, sets the material used to fill the via. Sherlock does not model this part of the via, but it uses this data to compute the effective material properties of the beam created to simulate the via.

8.3.2.2. Drill Hole Modeling

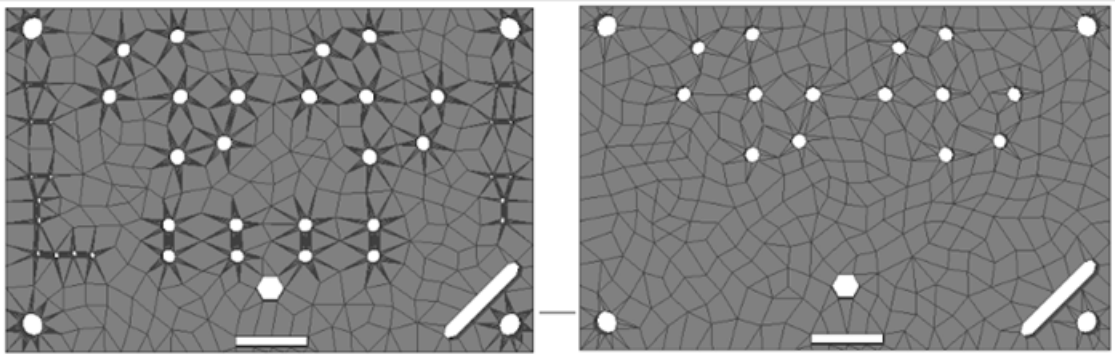
Drill holes can be enabled or disabled in a given PCB model using the Drill Holes properties tab in any of the FEA dialogs. When enabled, drill holes will be included in all PCB layers, using the specified properties to form each hole.



The **Min Hole Diameter** property specifies the smallest hole to be included in the FEA model. This allows users to ignore small holes that increase the complexity of the PCB model but may have little or no impact on the FEA results.

The **Max Edge Length** property specifies the maximum segment length that can be used when converting the circular drill holes to polygons in the mesh. Smaller segment lengths result in an increase in the number of elements generated around each hole, which could result in less accurate elements. For example, the left image below shows drill holes modeled using a value of 0.25 mm for **Max Edge Length**, while the right image shows drill holes modeled using a value of 1.0 mm for **Max Edge Length** and a **Min Hole Diameter** of 2.0 mm. The dark star patterns around the holes on the left consist of a large number of skinny triangles forced by the small circular segment length specified.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.4. FEA- Trace Modeling

Ansys Sherlock supports the modeling of signal traces defined for one or more signal layers and/or vias defined for one or more laminate layers. Such modeling provides a more accurate representation of the material properties across a given layer, usually at the cost of increasing the number and complexity of elements required. Trace models can be generated automatically by Sherlock directly from either an EDA file or image file used to define a given layer. Once generated, trace models can be viewed in Sherlock, exported as an image file or exported as a 3D FEA model suitable for analysis and/or combining with other models.

Note:

Because of the added complexity, trace modeling lengthens analysis time, making it impractical in some cases to apply trace modeling to an entire circuit board. As an alternative, Sherlock lets you convert a modeling region (p. 523) into a standalone circuit board, allowing you to create a trace model of a smaller CCA region. See Creating a CCA from a Modeling Region (p. 550).

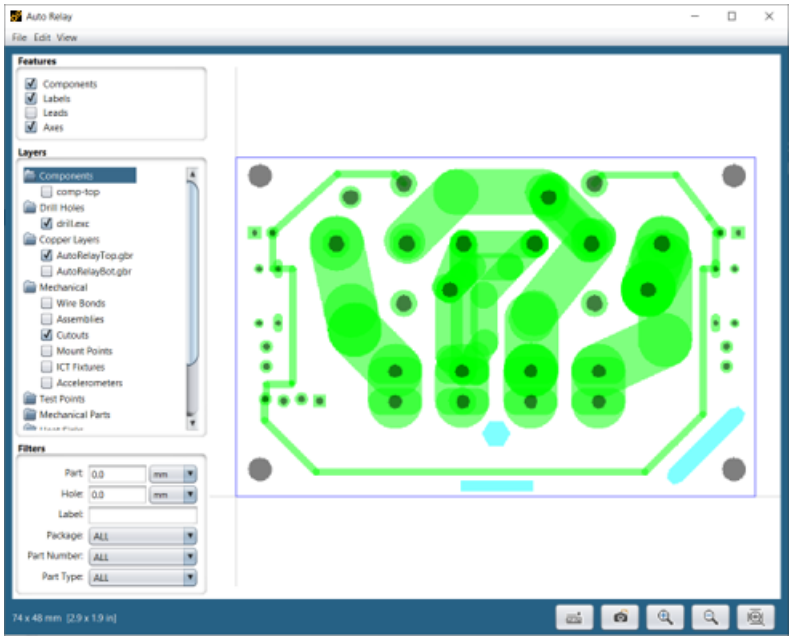
In this section, the following topics are covered:

8.4.1. Tutorial Project
8.4.2. Accessing the Trace Modeling Context Menu
8.4.3. Trace Model Generation
8.4.4. Advanced Trace Modeling Settings
8.4.5. Trace Model Viewing
8.4.6. Trace Model Import
8.4.7. Trace Model and Sketch Export
8.4.8. Trace Reinforcement Export
8.4.9. Importing the STEP File into Ansys Workbench or Ansys Discovery
8.4.10. Trace Model Resolution
8.4.11. Trace Model Analysis

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.1. Tutorial Project

Throughout this section, we will use a relatively simple, but real-world, PCB example to show how trace modeling is done. All the design files for the PCB can be found in the **Auto Relay Project.zip** file located in the **tutorial** folder found in the Sherlock Installation Directory. That ZIP file can be imported as a Sherlock Project for training purposes. As shown here, the top layer of the PCB consists of a collection of both narrow and wide traces that connect a number of through hole parts.



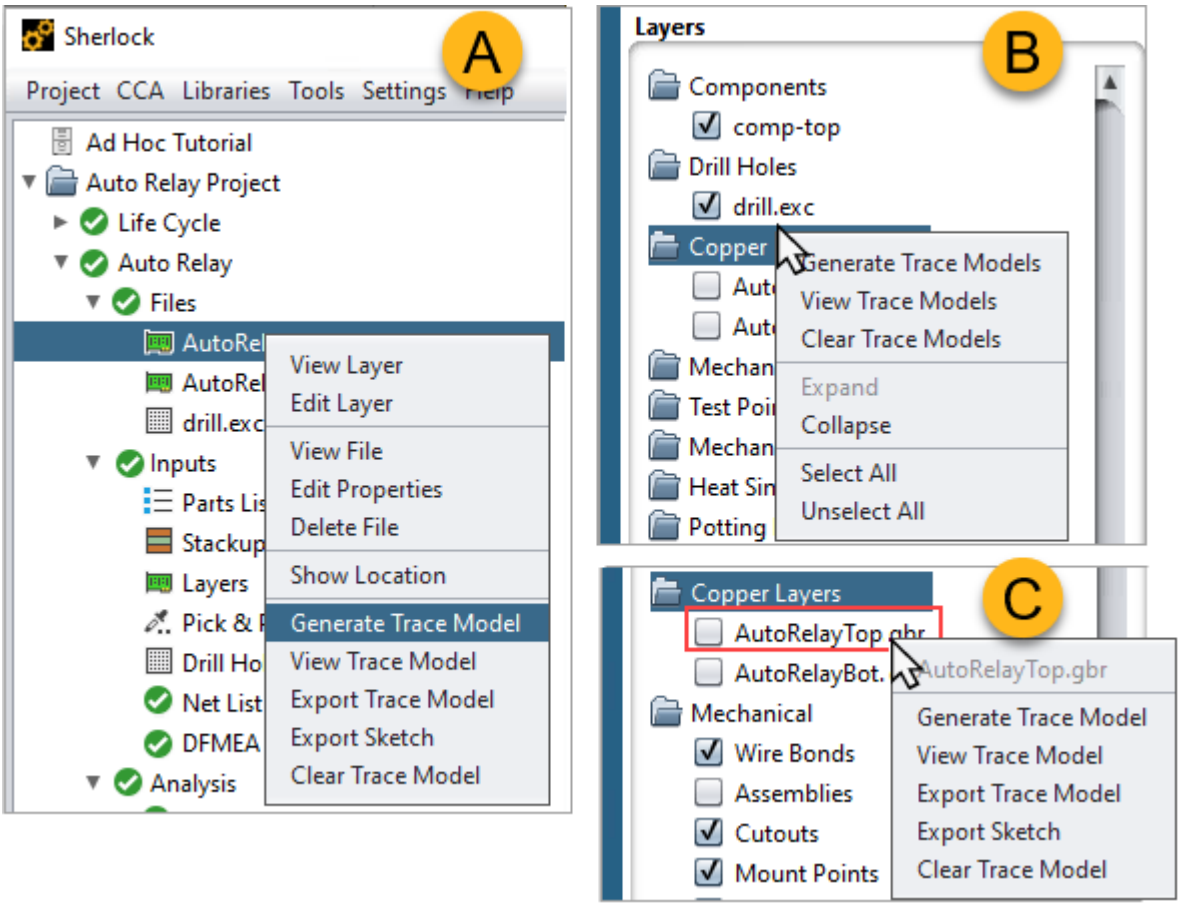
As is typical, the copper traces are formed by multiple drawing commands (in this case Gerber drawing commands) that overlap to form the continuous trace.

8.4.2. Accessing the Trace Modeling Context Menu

You can access the trace modeling features a number of places within Sherlock:

- In the **Sherlock Project Tree**, you can right-click one of the Trace Layers listed under a **Files** (A, below). The options listed in the context menu apply only to the layer you selected.
- In the **Layer Viewer** (p. 43), you can right-click the **Copper Layers** folder (B). The options listed in the context menu apply to *all* the layers shown under the **Copper Layers** folder.
- In the **Layer Viewer**, you can right-click one of the individual layers listed under the **Copper Layers** folder (C). The options listed in the context menu apply only to the selected layer.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Certain options (such as **View Trace Model**) only appear in the menu if a trace model has already been generated. Trace model menu options:

- **Generate Trace Model(s)**: Opens the **Generate Trace Model** dialog. See Trace Model Generation (p. 497) below.
- **View Trace Model(s)**: Displays the model in the **Trace Viewer.** If multiple layers have been selected for viewing, each layer is displayed in its own instance of the **Trace Viewer**. See Trace Model Viewing (p. 500).
- **Export Trace Model**: Opens the **Export Trace Model** dialog. See Trace Model and Sketch Export (p. 503).
- **Export Sketch**: Opens the **Export Sketch** dialog. See Trace Model and Sketch Export (p. 503).
- **Clear Trace Model**: This deletes any trace model and image file created during Trace Model generation for that layer (that is, images created by the **Create Image File** option as explained below (p. 499)).

8.4.3. Trace Model Generation

A trace model consists of one or more closed polygons that represent the signal traces in a signal layer or the vias in a laminate layer. When generated from an EDA file, Sherlock processes the drawing commands defined in the EDA file to determine the individual traces in the layer. When generated

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from an image file, Sherlock automatically detects all the edges in the raster image and converts them into polygons to represent the individual traces.

Tip:

Should you generate the trace model from an image or from the raw data contained in the EDA file?

The choice depends on the complexity of the layer being modeled. The drawing commands defined in EDA files usually provide the most accurate trace representations, but sometimes the feature sizes are too small for practical FEA modeling. Image files, on the other hand, allow you to:

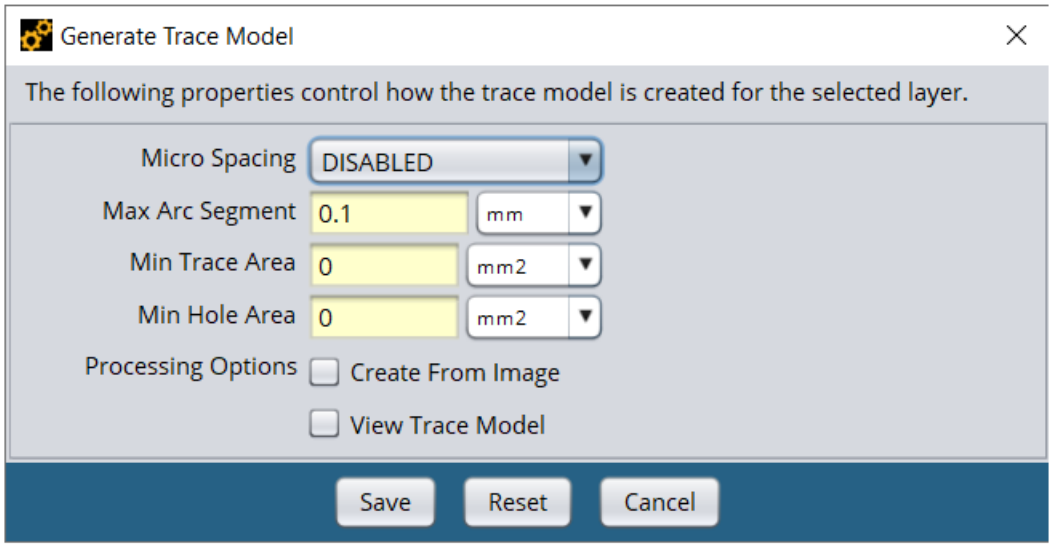
- Filter out or smooth over trace features
- Edit the image file by hand to correct problems
- Eliminate features you don't want in the FEA model

Regardless of how the trace model is generated, the resulting collection of traces can be exported to an FEA tool as a single layer, or they can be combined with other layers and exported as a multi-layer PCB model.

Tip:

The **Generate Trace Model** feature must be applied to the entire board, not an individual Modeling Region. However, Sherlock lets you convert a Modeling Region to a circuit board which is added to the **Project Tree** and handled like any other CCA. You can then generate Trace Models on the smaller circuit board. See Creating a CCA from a Modeling Region (p. 550).

To begin the trace model generation process, right-click the Trace Layer in the **Sherlock Project Tree** and select the **Generate Trace Model** menu option. At that point, a dialog will appear allowing you to specify a number of key processing properties.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options in the **Generate Trace Model** dialog:

- **Micro Spacing**: If the layer you are modeling contains very small features which Sherlock does not capture in the trace model, you can set **Micro Spacing** to ENABLED. This prompts Sherlock to use a finer grid when setting node locations, permitting smaller features and segments to be modeled. Enabling this feature might increase the time required for Sherlock to generate the model.
- **Max Arc Segment**: This property specifies the maximum segment-length Sherlock will generate when converting EDA arc drawing commands to line segments. Smaller values result in smoother arc representations on the FEA model but at the cost of generating a larger number of shorter segments. This, in turn, requires the FEA tool to generate a larger number of smaller elements to represent the curved solid, leading to increased processing time.
- **Min Trace Area**: Use this parameter to filter out smaller traces (based on area) from being generated in the model. Sherlock will not model any trace with an area less than the value specified. To include all traces in the model, set the value to zero.
- **Min Hole Area**: Use this parameter to filter out smaller holes (based on area) from being generated in the model. Sherlock will not model any hole with an area less than the value specified. To include all traces in the model, set the value to zero.
- **Processing Options:**
  - **Create from Image**: If the layer has a snapshot, Sherlock uses the snapshot to generate the trace model. Otherwise, Sherlock generates a temporary image which is used for the trace model generations. See the tip above, Should you generate the trace model from an image or from the raw data contained in the EDA file? (p. 498)
  - **View Trace Model:** When selected, Sherlock will automatically display the model in the **Trace Viewer** after it is generated.

When you click the **Save** button, Sherlock automatically converts the drawing commands to polygons and then combines and splits the polygons as needed to determine the outline of each individual trace. Any traces or portions of traces that extend beyond the board outline are clipped at the board outline. (The clipping feature, however, is not supported when exported to Ansys Electronics Desktop (.aedt (p. 109)).) During this process, Sherlock displays a progress dialog, showing the tasks being performed.

After all traces are found, Sherlock creates holes in each trace corresponding to any drill holes, cutouts, or gaps that are contained by each trace. The resulting set of non-intersecting polygons is called the **Trace Model** for the selected layer.

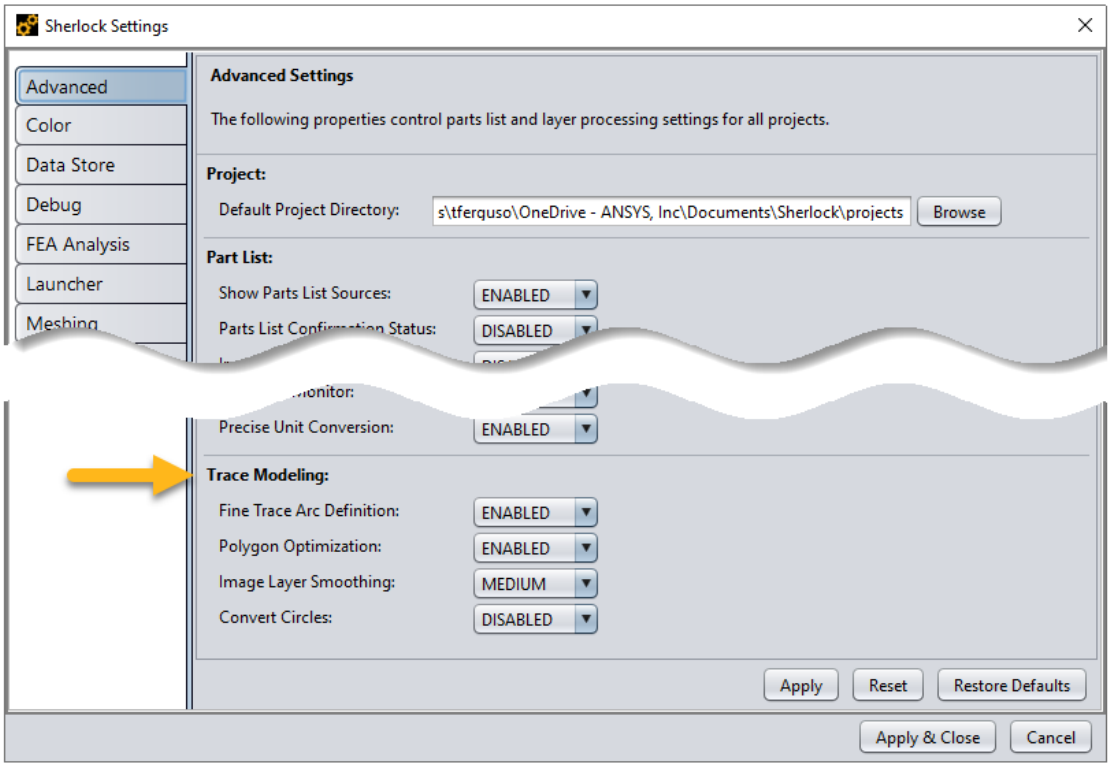
Note:

If you **Cancel** a trace model generation in progress, there might be a delay before Sherlock registers the **Cancel** command, depending on the complexity of the trace model.

8.4.4. Advanced Trace Modeling Settings

Aside from the properties detailed above, advanced trace modeling settings can be accessed by selecting **Settings > Advanced** from the main Sherlock menu. At the bottom of the **Advanced Settings** tab are **Trace Modeling** settings which can be enabled or disabled.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

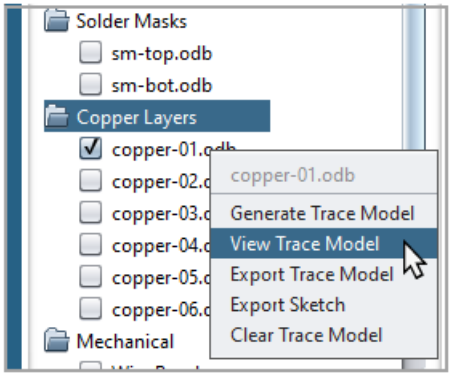


- **Fine Trace Arc Definition**: When ENABLED, Sherlock generates refined arc definitions when modeling traces.
- **Polygon Optimization**: When ENABLED, this setting optimizes and cleans up polygons. Each individual trace is defined by a series of non-intersecting polygons. **Polygon Optimization** enforces the minimum edge length regardless of other settings affecting the polygon. This creates a better trace but takes more time.
- **Image Layer Smoothing**: (OFF, LOW, MEDIUM, or HIGH) When generating trace models, this setting specifies the level of smoothing applied to polygons detected in the copper image layers. Less smoothing may generate more pixelated trace areas if they are present in the source image. Use the HIGH setting with caution, however. When set to HIGH, the smoothing algorithm becomes aggressive and may distort some objects. Experiment to determine the optimal setting for your project.
- **Convert Circles**: When ENABLED, Sherlock converts circular bodies to an octagonal approximation (an eight-sided shape).

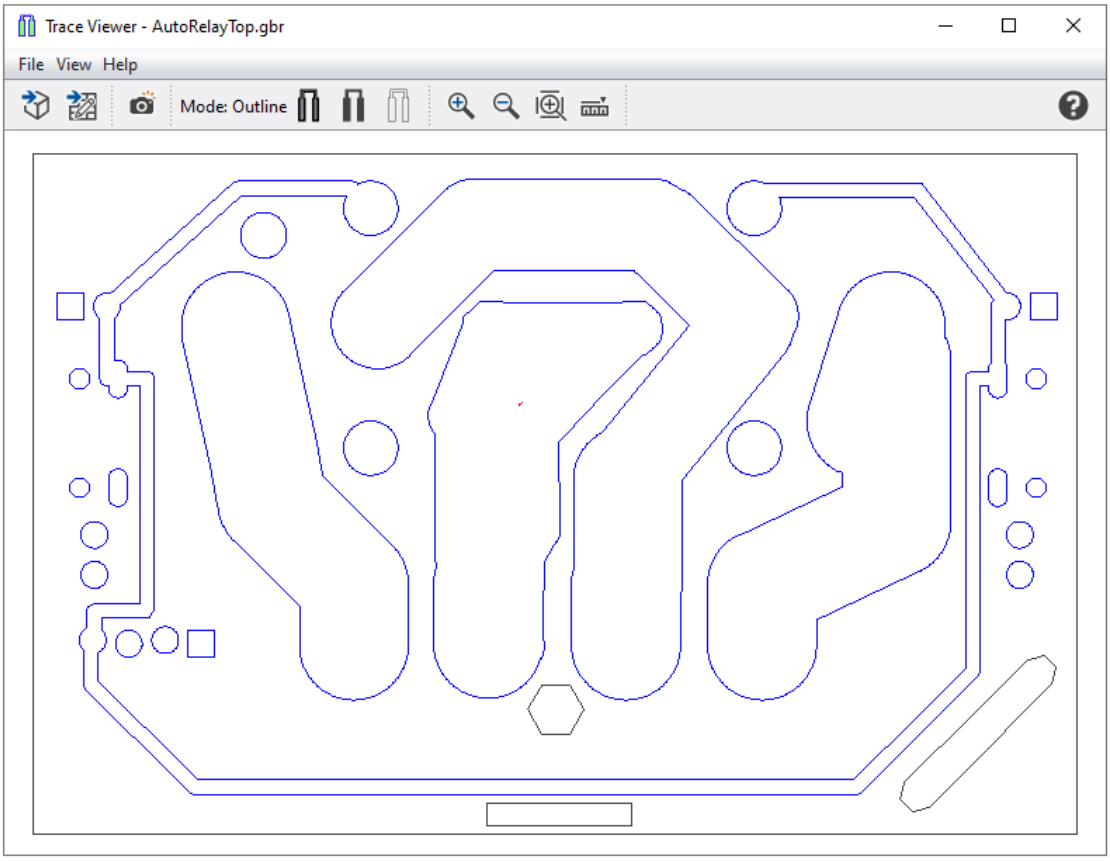
8.4.5. Trace Model Viewing

You can launch the **Trace Model Viewer** from Sherlock's **Layer Viewer** or from the **Project Tree**. From the **Layer Viewer**, right-click a copper layer and select **View Trace Model** from the context menu as shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Project Tree**, right-click the layer name and select the **View Trace Model** menu option. Remember, you can also enable the **View Trace Model** processing option when using the **Generate Trace Model** dialog. In either case, the **Trace Viewer** will appear, displaying either the trace outlines, trace model image, or both. For example, the following image shows the trace outlines for the top copper layer:



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blue lines represent the exterior outline of each trace, while red lines (not shown in this example) represent the interior outline of a hole in a given trace. Black lines are used to indicate the PCB outline and the outline of any cutouts in the PCB.

Note:

It is important to note that drill holes are not shown in the **Trace Model Viewer** for an individual layer. Drill holes are merged with the trace model of each layer when exporting to an FEA tool, allowing you to include/exclude drill holes as needed for analysis purposes.

Trace Viewer viewing options:



A. **Export trace model as an FEA model**: Clicking the icon opens the **Export Trace Model** dialog. See Trace Model Export (p. 503) below.

B. **Export trace model as a sketch**: See Trace Sketch Export (p. 506) below.

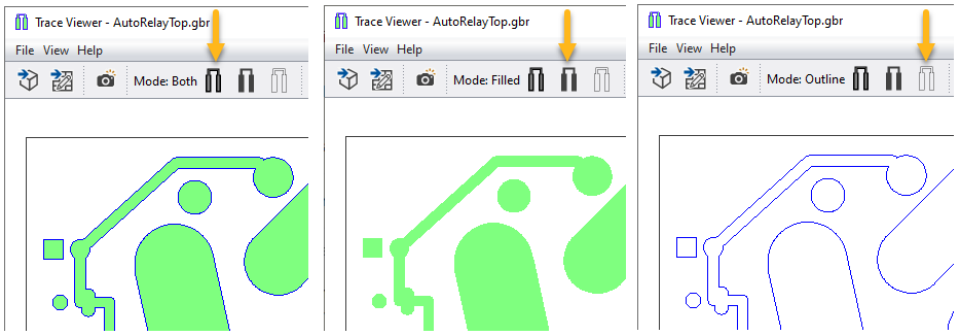
C. **Export an image of the model**: Opens the **Export Trace Image** dialog, allowing you to export a PNG image of the trace model.

D. **Display mode indicator**: Indicates which display mode has been selected: **Filled**, **Outline**, or **Both** (outlined and filled). Refer to the images below.

E. **Outlined and filled**: Displays the trace model with copper regions and holes outlined and filled.

F. **Filled**: Displays the trace mode with copper regions filled.

G. **Outline**: View the trace model with copper regions and holes outlined.



H. **Zoom in.**

I. **Zoom out.**

J. **Fit the model to the display size.**

K. **Distance Tool**: Select the **Distance Tool** to measure between two points on the model: Shift+left-click to set the starting location, and then left-click a second location to specify the endpoint. You may also shift+left-click to set the starting location, drag the mouse to the second location, and

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then release the mouse. To clear the current points, right-click anywhere on the display or press the **Close the Distance Tool** icon (✕).

If you did not select the **Create Image File** processing option in the **Generate Trace Model** dialog, then the image file will be automatically generated the first time it is referenced.

Note:

The image file may take a minute or two to be rendered depending on the complexity of the trace model. Once rendered, however, the image file will be retained and re-used as needed until the trace model is re-generated.

Much like the 2D **Layer Viewer**, the **Trace Viewer** provides convenient controls for panning, zooming, measuring distances, and saving snapshots.

8.4.6. Trace Model Import

You can import trace model files back into Sherlock as copper layer files. Refer to Copper Layer (Trace) (p. 106).

8.4.7. Trace Model and Sketch Export

Topics covered in this section:

8.4.7.1. Trace Model Export

8.4.7.2. Trace Sketch Export

8.4.7.1. Trace Model Export

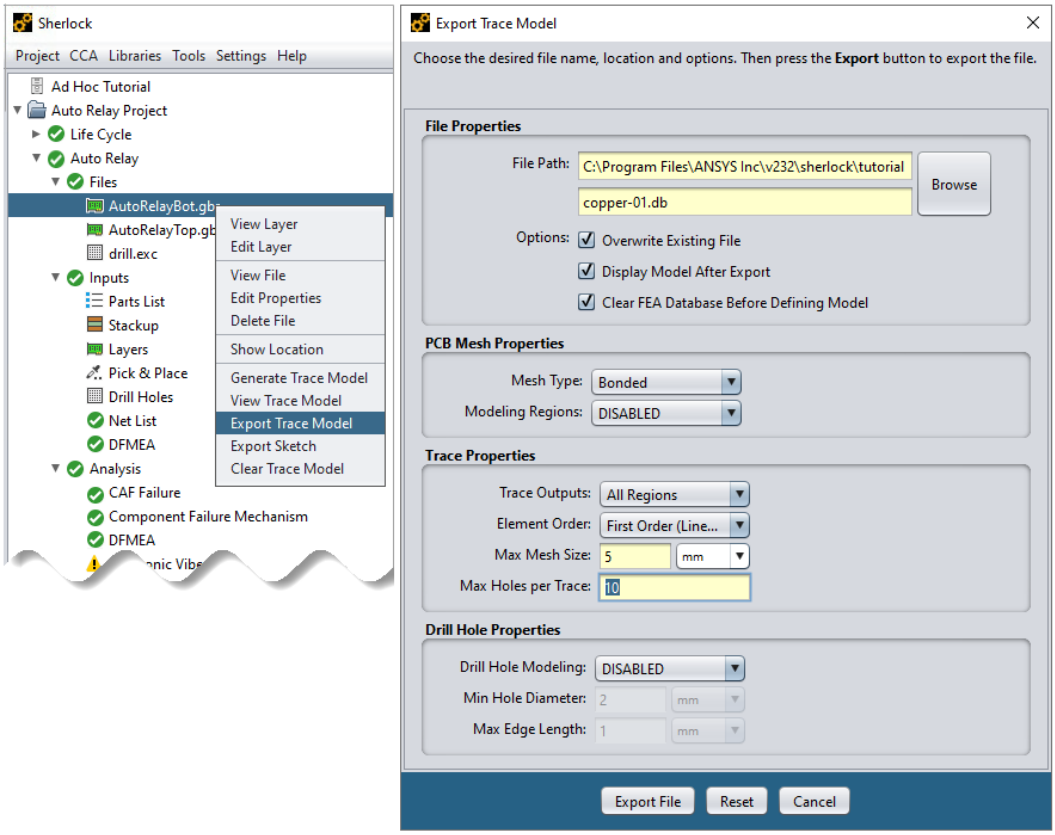
Trace models can be exported as individual layer models or can be exported as part of a multi-layer PCB model. In this section, we describe how to export an individual layer model. The Trace Model Analysis (p. 513) section below describes how to export trace models as part of a multi-layer PCB model.

You can access the **Export Trace Model** feature a couple of ways:

- From the **Trace Viewer** main menu, select **File > Export Trace Model**.
- To export the trace model for an individual layer, right-click the layer name in the Sherlock **Project Tree** and choose the **Export Trace Model** menu option.

In either case, the **Export Trace Model** dialog will be displayed, allowing you to specify which features will be included in the 3D FEA model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Export Trace Model dialog:

- **File Properties**
  - **File Path:** Use the **Browse** button to navigate to the folder where you want to export the model.
  - **[File Name]**: Enter the file name for the exported model and the appropriate suffix. The following file types are supported.
    - → Abaqus Python Script (*.py)
    - → Ansys APDL Script (*.apdl)
    - → Ansys CDB Script (*.cdb)
    - → Ansys Workbench Journal (*.wbjn)
    - → NX Nastran Bulk Data File (*.bdf)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ STEP File (*.stp, *.step) The STEP file format can be used to export models from Sherlock to a wide range of 3D design and analysis packages.

Note:

If you are exporting to a STEP file (*.stp), you will find additional information in the section below, Importing the STEP File into Ansys Workbench or Ansys Discovery (p. 510).

– **Overwrite Existing File**: If checked, Sherlock will automatically overwrite an existing file that has the same file name. If unchecked, Sherlock will ask for confirmation before overwriting the file.

– **Display Model after Export**: If selected, the relevant application (such as Workbench or Mechanical APDL) launches and displays the exported model.

– **Coordinate Units**: (in, mm, m, mil, micron, nm) This option appears when the file type selected is Ansys Workbench Journal (*.wbjn) or Abaqus Python Script (*.py). Use the dropdown menu to specify the coordinates to be used in the exported model. This feature replaces the **Scale to Microns** option for Abaqus exports in earlier versions of Sherlock.

– **Clear FEA Data Base Before Defining Model**: This option only appears when the file type selected is Ansys APDL Script (*.apdl) or Ansys CDB Script (*.cdb). When selected, Sherlock adds `/clear` to the Mechanical APDL output.

• **PCB Mesh Properties**

– **Mesh Types:** (Bonded, Sweep, or None) The options available depend on the export file-type.

– **Modeling Regions**: Select **ENABLED** to apply defined modeling regions to the exported model. See FEA- PCB Modeling Regions (p. 523).

• **Trace Properties**: Use the options in this section to select which trace regions to include in the 3D model.

– **Trace Outputs**: Options are listed below. By exporting one or both types of regions, you can build customized models, test the models themselves, or generate 3D images for documentation purposes.

→ **Conductor Regions**: The traces themsleves.

→ **Dielectric Regions**: Regions that surround the conductor regions (for example, areas filled by resin).

→ **All Regions**: Include both conductor and dielectric regions.

– **Element Order**: Select **First Order**, **Second Order**, or **Solid Shell**. Refer to FEA Model Properties (p. 562) for an explanation of these terms. All three element types are supported in both Abaqus and Ansys products.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Max Mesh Size**: This value is passed along to the FEA meshing tool to indicate the desired element sizes. In some cases, that value is simply a target size. The meshing tool may generate smaller or larger elements as needed to complete the meshing process.

  - **Max Holes per Trace**: In general, you should set **Max Holes Per Trace** to a large number like 1,000,000 so there is no limitation to the number of holes generated per trace. If you enter a small number, Sherlock will partition the trace regions into multiple, small geometries which leads to a significant increase in processing time.

- **Drill Hole Properties**: In this section you can specify whether to include drill holes in the layer model and, if so, the dimensions of the holes to be modeled.

  - **Drill Hole Modeling**: To model drill holes, select **ENABLED**.

  - **Min Hole Diameter**: Sherlock will ignore all holes smaller than the size you specify here. A setting of zero includes all holes. While smaller holes usually do not affect the overall mechanical properties of the layer, they can make meshing more complicated.

  - **Max Edge Length**: Specifies the size of the segments used to represent round drill holes by a polygon.

Note:

Ansys Sherlock generates meshing commands for Abaqus and Ansys that use the designated meshing algorithm and element size, with reasonable seed values. However, both FEA tools have a library of multiple meshing algorithms and approaches that can be used. User interaction may be required for complex trace models.

8.4.7.2. Trace Sketch Export

A sketch (2D surfaces) can also be exported of a given trace model using the same options as the **Export Trace Model options** (p. 503). A sketch may only be exported as a STEP (*.stp) file. PCB Mesh Properties, Trace Properties, and Drill Hole Properties are not included in the export.

8.4.8. Trace Reinforcement Export

Traces and coated holes can be modeled as reinforcement elements in Ansys Mechanical. Sherlock provides a Trace Reinforcement Export option based on generated traces for different layers. (See Accessing the Trace Modeling Context Menu (p. 496).) The copper (conductor) features of traces and holes along with the PCB stackup will be exported. When the exported model is opened in Workbench Mechanical, the following properties will automatically be applied based on the settings in Sherlock: the thickness of the trace models, the shell reinforcement elements, the cross-sections of the vias for beam reinforcement elements, and all respective material properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When exporting to Workbench, Sherlock automates the process of setting up a Workbench schematic. See the section below, Trace Reinforcement Model Export Automation (*.wbjn) (p. 509), for details.

Note:

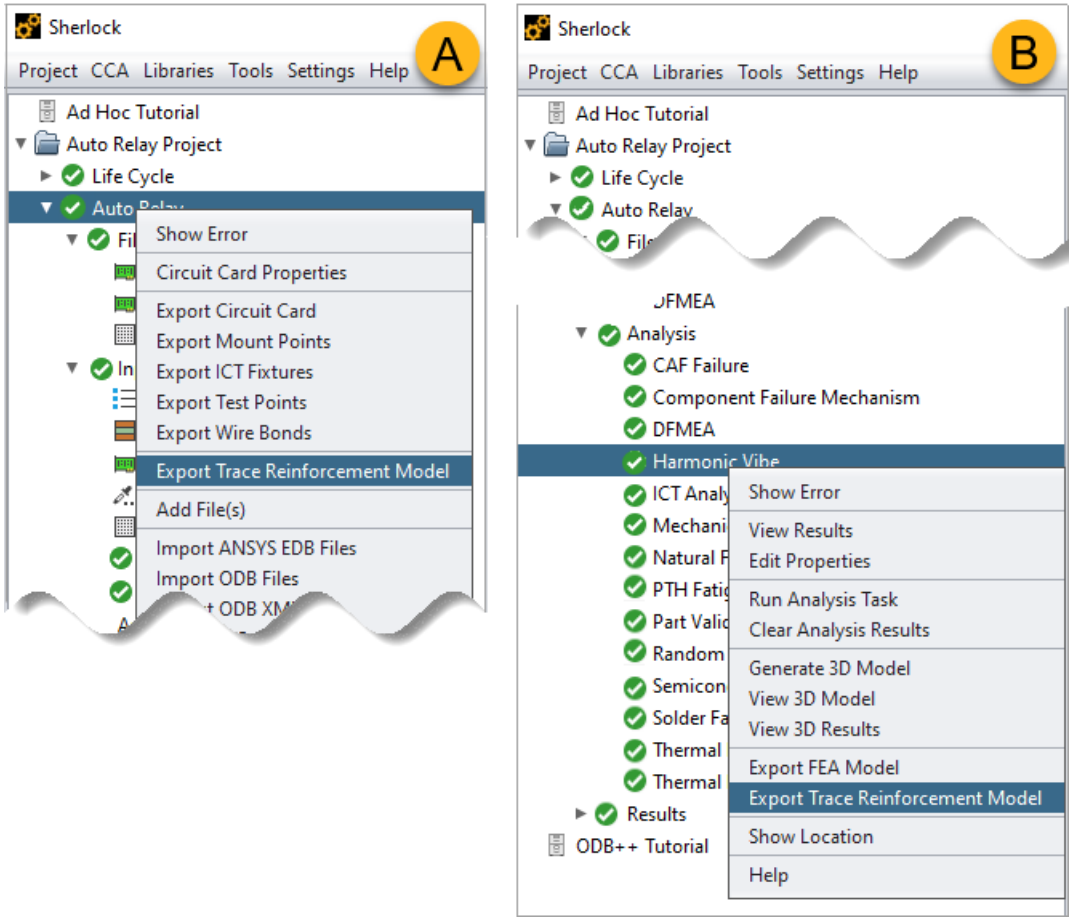
The Trace Reinforcement Export feature exports only the trace reinforcement model of the selected board. Subassemblies and electronic components are not exported.

Note:

For projects created before 2020 R2, the **Export Trace Reinforcement Model** feature has known limitations. A known workaround is as follows: Under the **Files** node in the **Project Tree**, right-click the drill file and select **Edit Properties**. Confirm that the **Plating Material** and **Fill Material** properties are set correctly. Make any necessary edits and then click the **Save** button.

To open the **Export Trace Reinforcement Model** dialog:

- Right-click the circuit card in the **Project Tree** and select **Export Trace Reinforcement Model** in the context menu (A, below).
- Right-click one of the FE analysis tasks in the **Project Tree** and select **Export Trace Reinforcement Model** in the context menu (B, below).



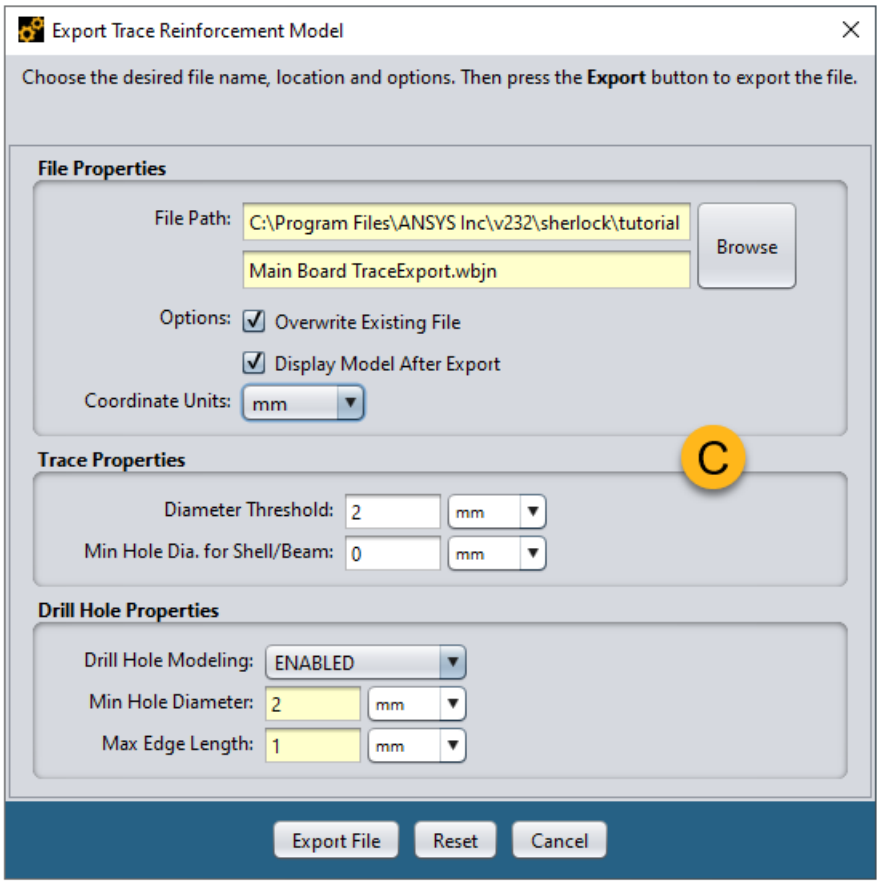
Once the **Export Trace Reinforcement Model** dialog (C, below) has opened, configure the export options:

- **File Properties**:
  - **[File Name]**: Options are Ansys Workbench Journal (*.wbjn) or STEP file (.stp).

    When you select the .wbjn file type, Sherlock automatically creates a schematic in Ansys Workbench. See Trace Reinforcement Model Export Automation (*.wbjn) (p. 509) for details.

    Tip:

    For STEP files, refer to the section below: Importing the STEP File into Ansys Workbench or Ansys Discovery (p. 510) where you will find setup information required for a successful export.



  - **Overwrite Existing File**: If checked, Sherlock will automatically overwrite an existing file that has the same file name. If unchecked, Sherlock will ask for confirmation before overwriting the file.
  - **Display Model after Export**: If selected, the relevant application (such as Workbench or Mechanical APDL) launches and displays the exported model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Coordinate Units**: (in, mm, m, mil, micron, nm) This option appears when the file type selected is Ansys Workbench Journal (*.wbjn). Use the dropdown menu to specify the coordinates to be used in the exported model.

- **Trace Properties:**

  - **Diameter Threshold:** This value determines whether a hole is modeled with beams or shell reinforcement elements. Holes equal to or greater than the **Diameter Threshold** value are modeled with shell reinforcement elements. Smaller holes are modeled with beam elements. (A hole buried inside the board is always modeled as a beam.)

  - **Min Hole Dia. for Shell/Beam**: Vias smaller than the specified diameter will not be exported. A value of zero will export all vias.

- **Drill Hole Properties**

  - **Drill Hole Modeling**: Set to **ENABLED** to model drill holes in the export. (The properties below are grayed out if **Drill Hole Modeling** is **DISABLED**.)

    Note:

    Drill Hole Modelling under **Drill Hole Properties** is used solely to determine whether geometric hole definitions are created in the exported PCB. The option does not control the drill holes (modeled as beams/shell) in the exported trace model.

  - **Min Hole Diameter**: Drill holes smaller than the specified diameter will not be exported. A value of zero will export all drill holes.

  - **Max Edge Length**: This value specifies the maximum segment size used when representing round drill holes by a polygon.

8.4.8.1. Trace Reinforcement Model Export Automation (*.wbjn)

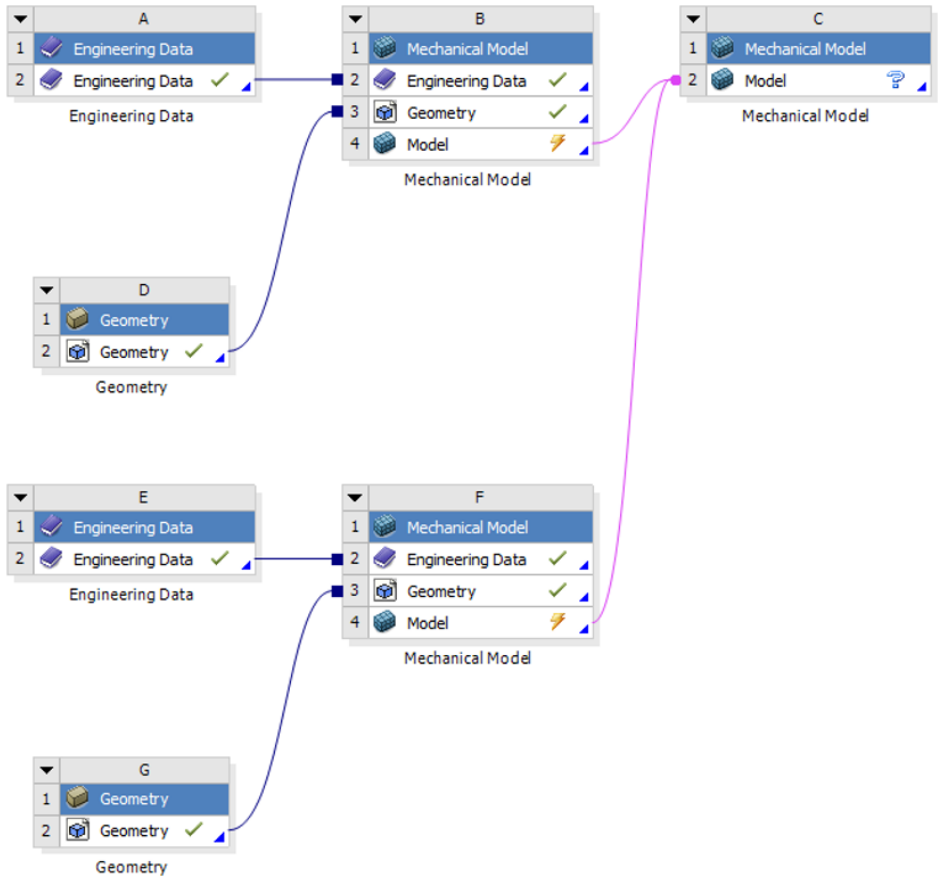
Note:

Trace Reinforcement Model Export Automation in Sherlock currently supports only Layered PCB Models (p. 475).

When exporting a trace reinforcement model to an Ansys Workbench Journal file (*.wbjn), Sherlock automates the following steps:

- Sherlock exports both the FEA model and the trace reinforcement model to the Workbench Journal file. If you selected **Display Model after Export**, Workbench will launch after the export is complete and display the **Project Schematic** (shown below) with both models represented (system **B** and system **F**).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



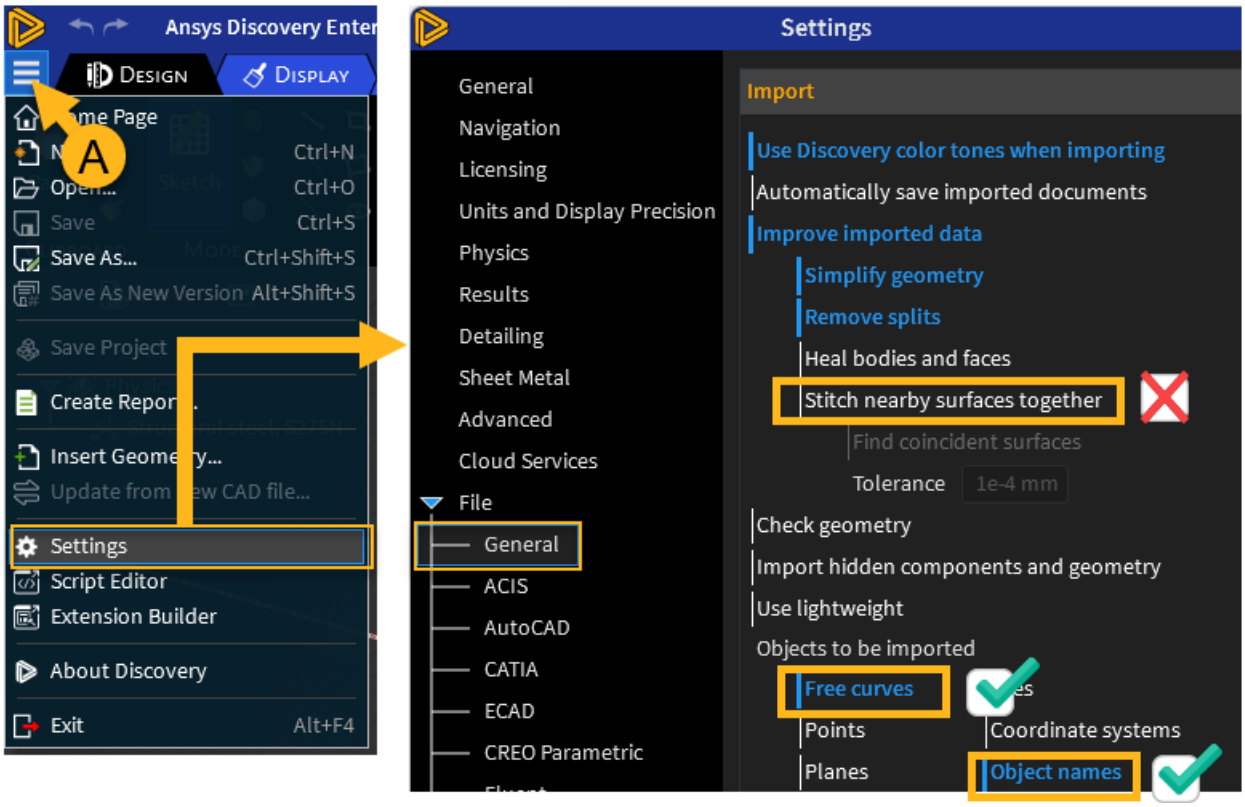
- Material Properties assigned to resin (signal) layers in the FEA model are not affected by the conductor material. It will contain 100% of the resin material defined in the stackup in Sherlock. The conductor traces are embedded as reinforcement elements with conductor material properties within the resin layer mesh.
- Circular beam cross sections are created and assigned to the corresponding reinforcement beams that represent holes as per the **Diameter Threshold** input (p. 509)in Sherlock.
- The material properties for reinforcement beams are calculated based on the plating material, plating thickness, and the fill material defined for the drill holes in Sherlock. If the fill material is set to **NONE**, plating material will be used instead.
- Vias exported as reinforcement shells get their thicknesses assigned using the plating thickness as defined in Sherlock.
- Traces exported as reinforcement shells get their thicknesses automatically assigned using the layer thickness as defined in the Sherlock Stackup
- Separate Named Selections are created for the vias exported as reinforcement shells and beams. Also, layer-wise Named Selections are generated for traces exported as reinforcement shells.

8.4.9. Importing the STEP File into Ansys Workbench or Ansys Discovery

When importing a Sherlock-generated STEP file into the Ansys Workbench or Discovery, you must select specific import options so that trace reinforcements appear.

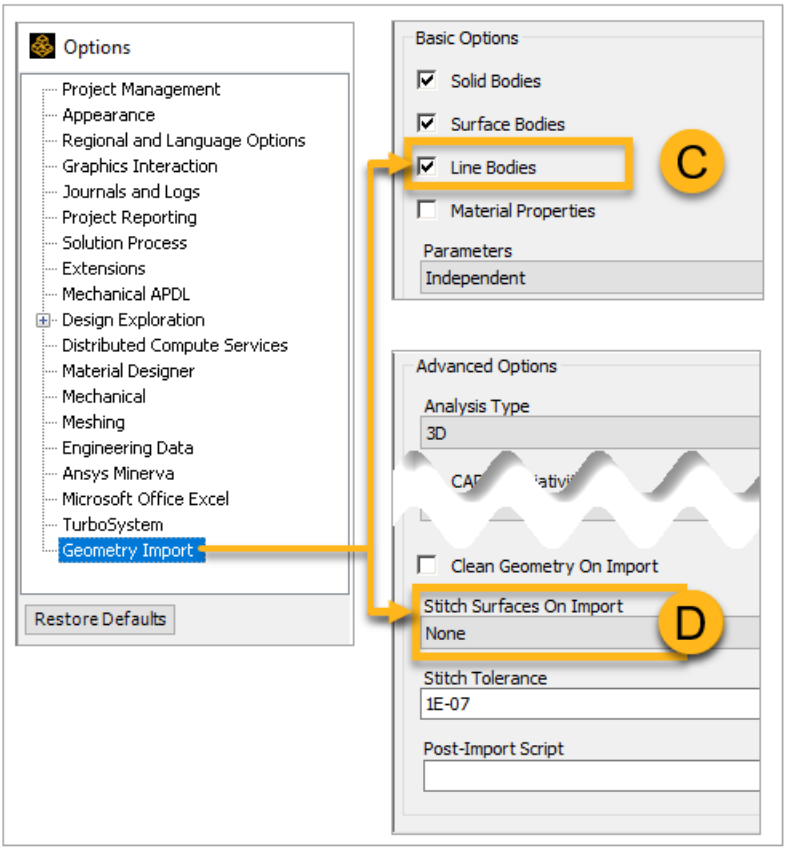
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In Discovery, navigate to the **Import Options** by clicking the **File Menu** icon (A in the image below) and then clicking **Settings**. In the **Settings** window, click **File > General**. Verify the following settings:
  - **Stitch nearby surfaces together** must be *unselected* as shown in the image below.
  - Under **Objects to be imported**, select **Free Curves** and **Object Names**.



- In the Workbench application, select **Tools > Options... > Geometry Import**. **Line Bodies** in the **Geometry Import Options. Stitch Surfaces On Import** should be set to **None** as shown in the image below.
  - Select **Line Bodies** (C) under **Basic Options**.
  - Under **Advanced Options**, **Stitch Surfaces On Import** should be set to **None** (D).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Currently, plating material, plating thickness, capped holes, and filled holes are not modeled in Sherlock. These features are planned for a future release.

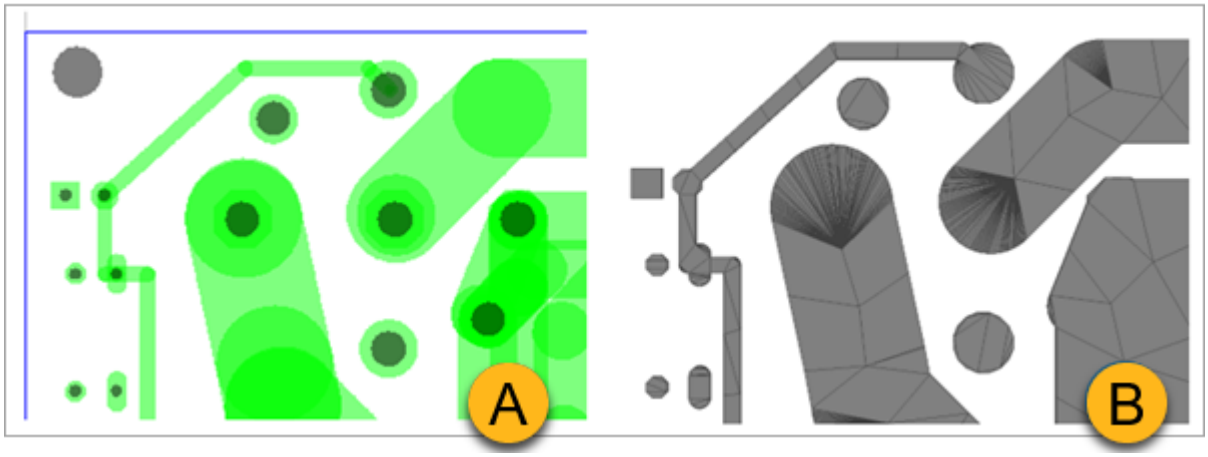
Holes that do not go all the way through the board are only supported with **Trace Reinforcement Export**, not regular **FEA Export** (p. 458) and analysis.

8.4.10. Trace Model Resolution

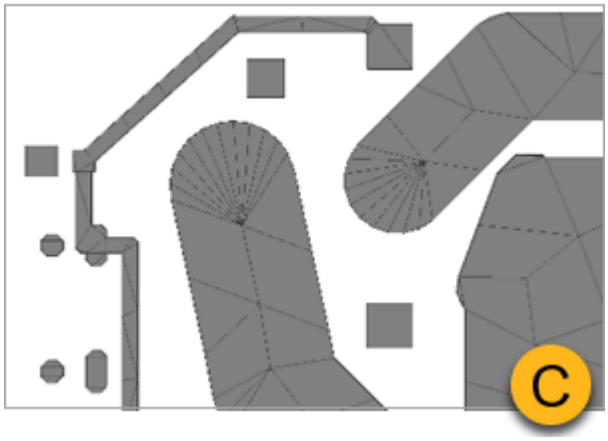
As discussed above, a number of properties are provided to allow users to control the size of features represented by the trace model. The most important property is the **Max Arc Segment** property specified in the **Generate Trace Model** dialog which controls the length of the segments used to represent arcs in the trace model. The smaller the segment length, the smoother the arc will be, at the cost of increasing the complexity of the resulting 3D model.

For example, consider the upper left corner of the top copper layer that we've been using (below, A) and the trace model generated using a **Max Arc Segment** value of **0.1mm** (B).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



We can see that the small round pads on the left side of the images are modeled by hexagons with sides that are no more than 0.1 mm in length. Similarly, the large round pads and arcs are modeled as a larger number of short segments, which result in a large number of wedges generated by the meshing algorithm. Although some meshing algorithms are better than others for such shapes, the short segments would always cause a large number of small elements to be generated, be they wedge or brick elements.

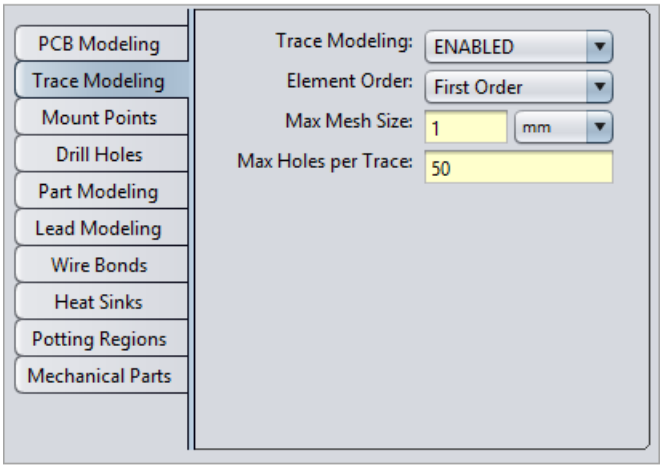


If we increase **Max Arc Segment** to **1mm**, then a simpler, but less accurate, model is generated. As shown above (C), the oblong pads on the left side of the image now have mitered, instead of rounded, ends. Similarly, the round pad at the top of the image is now modeled by a square, as is the elbow pad on the left side. The most important difference is the reduced number of segments and mesh elements needed to model the rounded ends of the large signal traces, thereby reducing import and analysis times.

8.4.11. Trace Model Analysis

Whenever an FE analysis task is performed (for example, Natural Freq, ICT, Harmonic Vibe, Random Vibe, Mechanical Shock) or a 3D model is generated (for example, Generate 3D Model, Export FEA Model), users can enable or disable the use of trace models in the PCB model using the **Trace Modeling** tab.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When Trace Modeling is enabled, all currently defined trace models are used to generate the 3D elements in the corresponding layer, instead of using elements based on the stackup properties for that layer. If you don't want a specific layer to be modeled using traces, simply clear the trace model before exporting the FEA model or performing the FE analysis task. The **Element Order**, **Max Mesh Size** and **Max Holes per Trace** properties control the same modeling aspects as described previously for single layer models.

Note:

Trace Modeling is only supported when **PCB Model** is set to **Layered** or **Layered Elements** in the **PCB Modeling** tab.

The resulting multi-layer PCB model is defined so that all the conductor and dielectric regions on a given layer are bound together. In the Abaqus model, all the conductor and dielectric regions on a given layer are merged together to form a single part, which is then meshed. In the Ansys and Calculix models, all the conductor and dielectric regions on a given layer share the same set of nodes, thereby binding them together. In the STEP model, conductor and dielectric regions are modeled in separate parts and the FEA tool (for example, Ansys Workbench) or user is responsible for bonding surfaces that touch.

In all FEA models, except the STEP model, adjacent layers are bound together by surface-to-surface constraints. In the STEP model, the FEA tool (for example, Ansys Workbench) or user is responsible for bonding adjacent layers.

8.5. FEA- Image File Layers

In some cases, it is not possible to obtain the EDA files for one or more layers that need to be modeled in detail. To help solve such problems, Ansys Sherlock provides the following file types to allow layers to be defined using image files instead of EDA design files:

- Copper Layer (Image)
- Laminate Layer (Image)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copper layers define the conductor regions formed by signal traces, surrounded by dielectric regions usually consisting of a resin material. Laminate layers define the conductor regions formed by vias, surrounded by dielectric regions usually consisting of fiber and resin materials. In both cases, even when EDA files are not available, image files showing the conductor regions can be used to:

- Display the layer in the 2D Layer Viewer
- Determine the percent of metal contained by the layer *or*
- Generate a trace model for the layer.

Note:

Image file layers allow you to perform custom layer edits using any standard image editor or EDA file viewer before the layer is processed by Sherlock. You can, for example, reduce unwanted details or add missing features. In fact, Sherlock can convert any copper layer file into an image file if needed. See the section below, Converting a Copper Layer File to an Image File (p. 520).

In this section, the following topics are covered:

8.5.1. Preparing Image Files
8.5.2. Adding Image File Layers
8.5.3. Image File Alignment
8.5.4. Converting a Copper Layer File to an Image File
8.5.5. Image Layer Stackup Data
8.5.6. Image Layer Trace Model

8.5.1. Preparing Image Files

When using image files for trace modeling purposes, the following properties are desired:

- The image should be the highest resolution possible.
- The image must have either a single background color or a single foreground color.
- The image should NOT show the board outline or corner reference points.
- The image should be tightly cropped to fit inside the board outline.

The image resolution determines the accuracy of the trace model generated by Sherlock for the layer. The higher the resolution, the more likely that the trace model will contain all the detailed features. In the examples below, we use image files with resolutions that are roughly 3000x2000 pixels, which is sufficient in most cases.

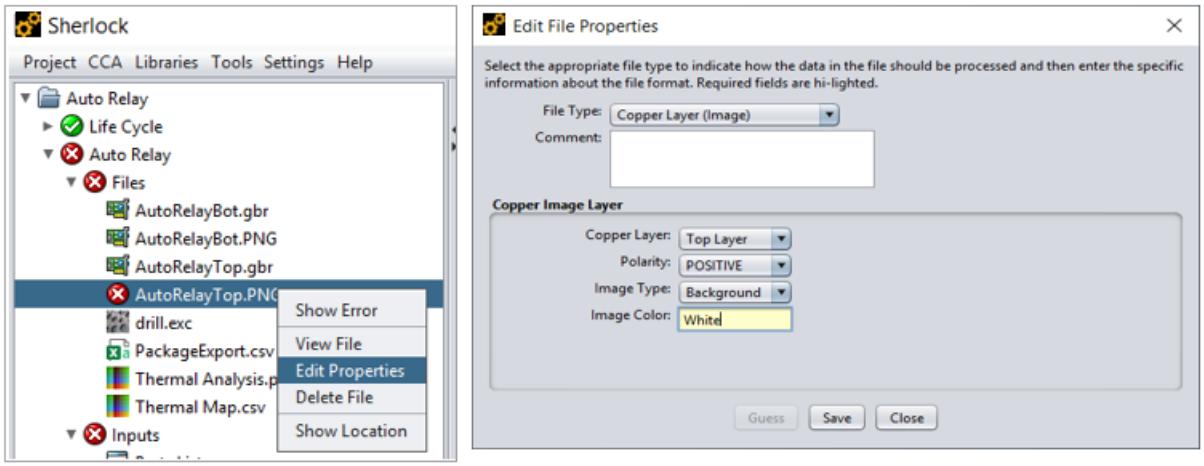
The image conversion process used by Sherlock is capable of processing (a) image files with a single background color or (b) image files with a single foreground color. For example, the image might have a white, black or clear background color, as long the background is not shaded. Alternatively, the image might have a single foreground color, with lots of different background colors. In either case, the image is essentially converted to a black-and-white image and then processed to extract the percentage of metal and, optionally, the trace model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As will be demonstrated shortly, after adding an image file, users must then align the image to circuit card outline. You might think that images containing the board outline or corner reference points would make that process easier, but such features would then be treated as conductor regions, making the trace model and stackup calculations inaccurate. To prevent such extraneous regions while accommodating the alignment process, the best approach is to use an image editor to crop the image so that it fits inside of the board outline.

8.5.2. Adding Image File Layers

Image file layers are added to Sherlock just like EDA design files by using the **CCA > Add File(s)** menu option from the Sherlock **Main Menu** or **Project Tree**. Once added, right-click the file name in the **Project Tree** and select the **Edit Properties** menu option to display the following dialog.



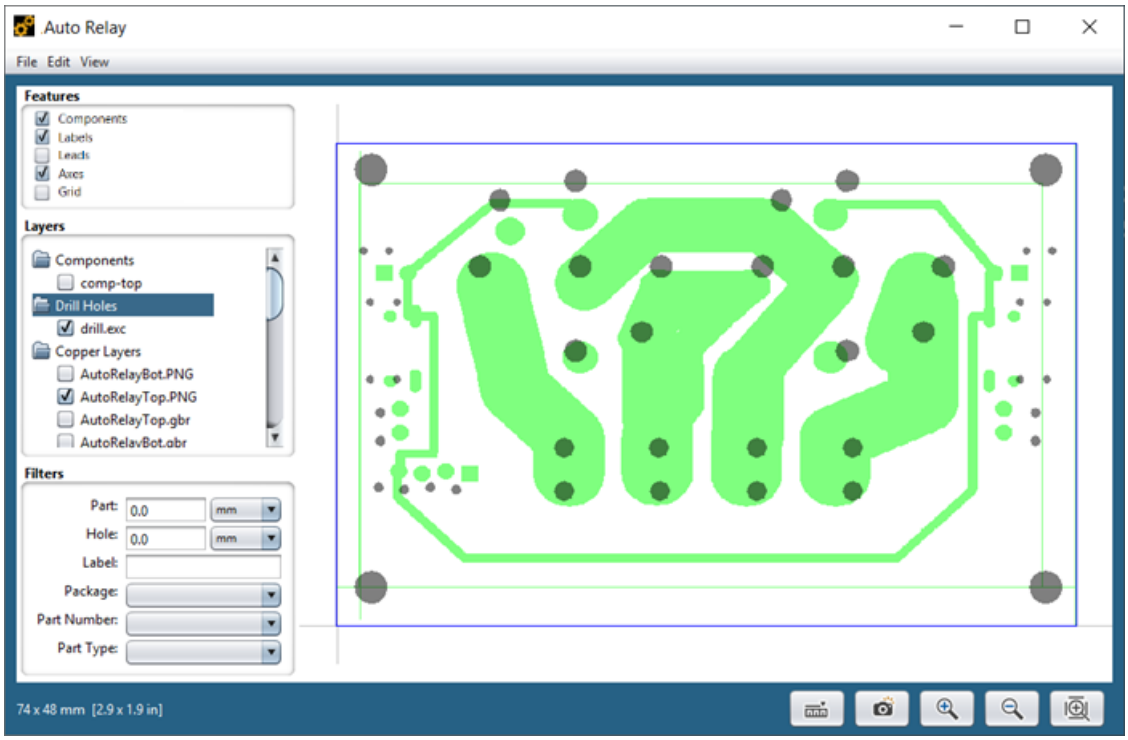
Tip:

In Sherlock, you can convert any copper layer file to an image file. This allows you to edit the traces using an image editor. See below, Converting a Copper Layer File to an Image File (p. 520).

Simply specify the proper file type and indicate the layer that it represents. The **Image Type** and **Image Color** fields indicate whether the image contains a constant background color or a constant foreground color. In this example, the image has a WHITE background color, with various shades of green as the foreground colors. By specifying the background color, all the foreground colors will be considered conductor material.

When the **Save** button is pressed, Sherlock will prepare the image file layer for display in the Sherlock **2D Layer Viewer**, as shown here along with the drill holes:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



It's important to note that the top copper layer does NOT initially line up with the drill holes. This is because there is no way for Sherlock to automatically determine how many pixels per inch are represented by the image, nor where the image is located relative to the other circuit card coordinates. Therefore, Sherlock scales the image to 80% of the original and places it in the center of the circuit card, leaving it up to the user to make the final placement.

8.5.3. Image File Alignment

In Sherlock, there are two methods for aligning an image file with the circuit board:

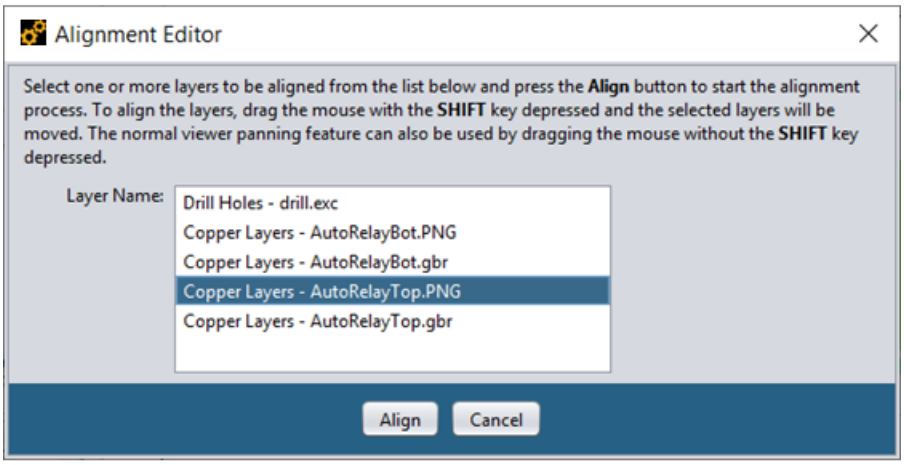
8.5.3.1. Aligning Images in the Layer Viewer

8.5.3.2. Aligning Images Using the Mapping Editor

8.5.3.1. Aligning Images in the Layer Viewer

Copper and laminate layers defined using image files can be easily aligned to other circuit card features using the **Edit > Align Layers** menu option available from the Layer Viewer menu bar. When that tool is chosen, the **Alignment Editor** dialog will appear allowing you to select one or more layers to be aligned. Typically, it is easier to align each image file layer separately, although experienced users may prefer to align all layers at the same time.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

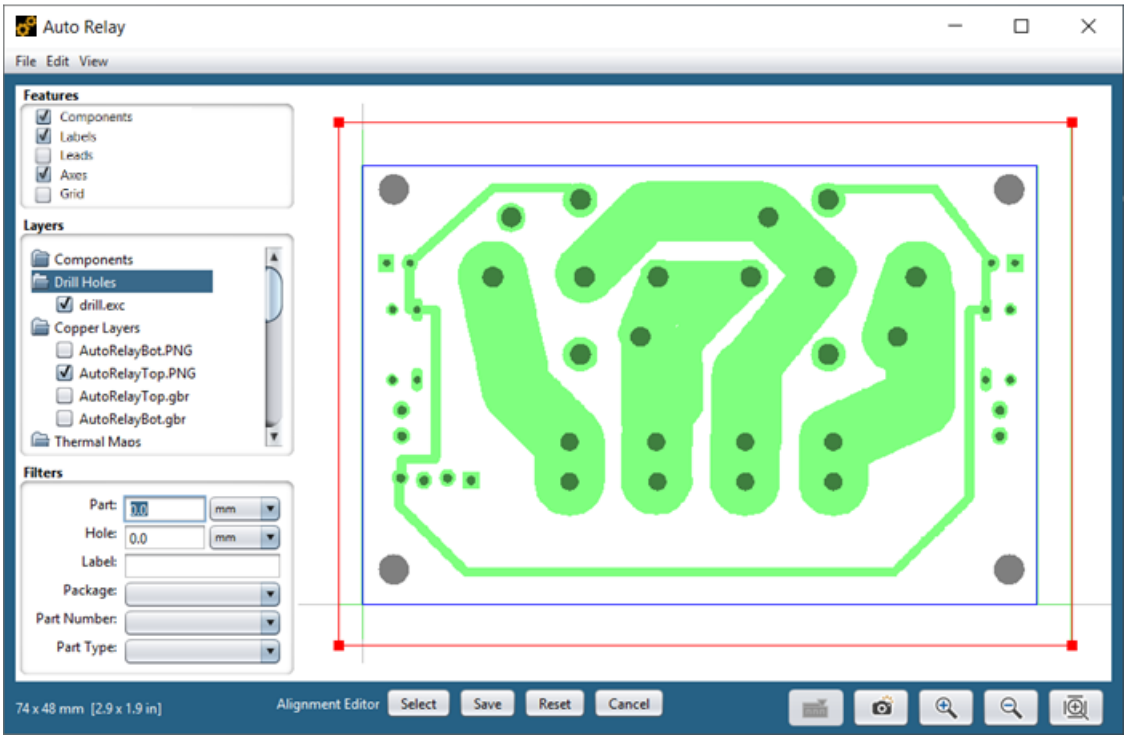


When the **Align** button is pressed, Sherlock displays each of the selected layers in the Layer Viewer, with a red outline displayed for each layer being aligned. The following user interactions can be used while in the alignment editor:

- Left-click and drag the mouse while holding the SHIFT key to drag a layer to a different location.
- Left-click and drag the mouse in any red corner icon while holding the SHIFT key to scale the image with the opposite corner fixed in place.
- Left-click and drag the mouse (without holding the SHIFT key) to pan the entire viewer display.
- Use the mouse wheel or the zoom buttons at the bottom of the viewer panel to zoom in or out.
- Enable/disable any layer in the **Layers Panel** to help align the image layers.

Using those controls, it is relatively easy to align the image file layer with the other layers. Press the **Save** button to save the alignment values.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.5.3.2. Aligning Images Using the Mapping Editor

1. In the **Project Tree**, right-click the image file and select **Edit Layer** from the context menu to open the **Mapping** tab.

2. To align the image to the board, click the **Align Board** button. Adjust the boundaries of the box by grabbing and dragging the nodes. Clicking **Max Bounds** expands the blue box to the edges of the image. You may click **Reset** to restore the nodes to their last saved or default configuration. Click **Done** when you are finished sizing the blue border.

Tip:

You may also manually enter the coordinates of each node that defines the box. Right-click any node of the blue box and select **Edit Location**. This opens the **Edit Vertex** dialog where you can enter the X and Y coordinates of the node. While the dialog allows you to enter the coordinates in the units of your choice, Sherlock converts the coordinates to inches when you save and close the dialog. If you reopen the **Edit Vertex** dialog later, Sherlock displays the coordinates in inches.

If your image has a legend, this tip also applies to aligning the box around the legend. You will find an example of this in the section Adding Strain Map Image Files (p. 770).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

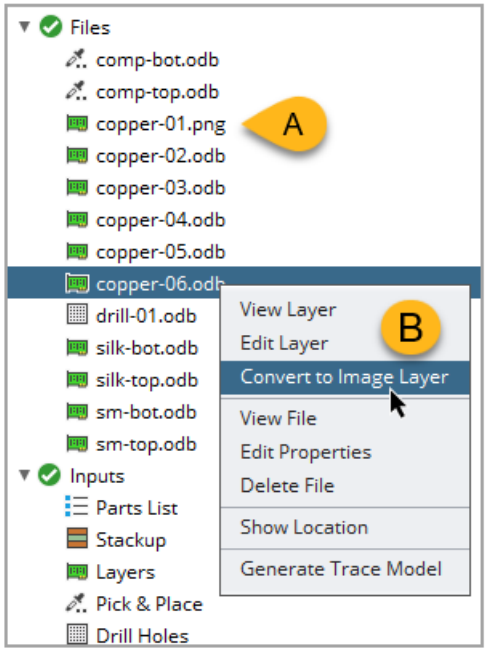
3. To preserve your changes, you must click **Save** before exiting the **Mapping** tab.

Tip:

For an illustrated example of using the Mapping Editor, see Adding Strain Map Files (Image or CSV) to Sherlock (p. 770).

8.5.4. Converting a Copper Layer File to an Image File

Sherlock allows you to convert a copper layer file into an image file. This feature is helpful because, for example, you can use an image editor to make corrections to the copper traces in a file. As shown in the image below (A), the `copper-01` file has been converted from an `.odb` file to an image (`.png`) file.



Warning:

This conversion cannot be undone. The original file is replaced by the new image file.

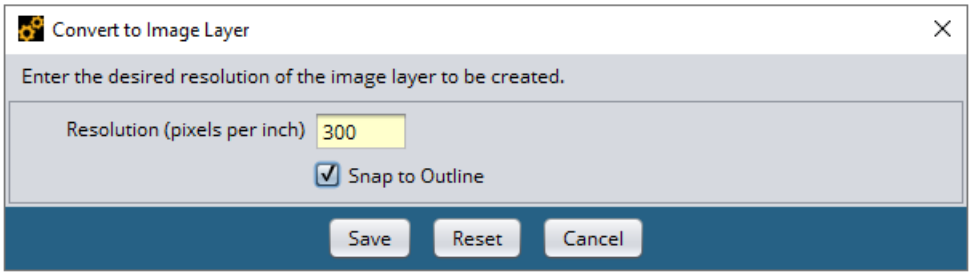
If you wish to preserve a copy of the original, do the following: Right-click the file in the **Project Tree** and select **Show Location**. This opens the file location in Windows File Explorer. Make a copy of the file and store it in another folder for safe keeping. Later, if you wish to restore the original file to your project, follow the procedure described in Adding Files to a Circuit Card Assembly (p. 100).

To convert a copper layer to an image file, do the following:

1. Right-click the file in the **Project Tree** and select **Convert to Image Layer** (B, above).
2. In the **Convert to Image Layer** dialog, set the following options:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The default **Resolution (pixels per inch)** is 300. Higher resolutions might be required to capture very fine features.
- Enabling the **Snap to Outline** option is recommended. This feature resolves most scaling issues caused by certain image editors. When enabled, traces outside the board outline are cropped.

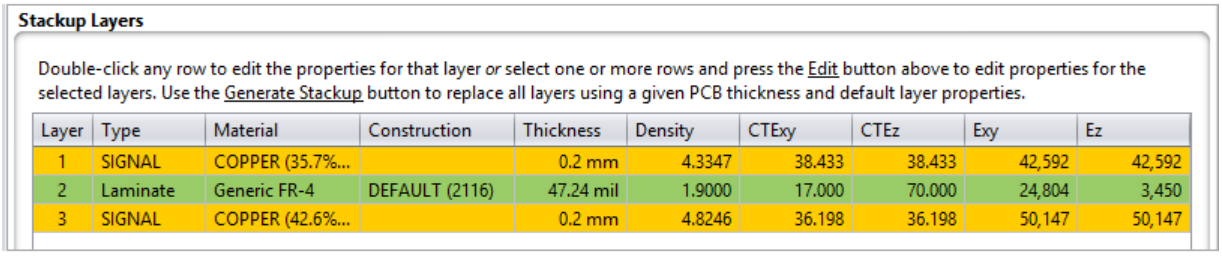


To edit the new image file, do the following:

1. In the Sherlock **Project Tree**, right-click the new image file and select **Show Location** in the context menu. This opens the folder location of the image file in Windows File Explorer. Open the image file in the image editor of your choice.
2. Once you have completed the edits in the image editor, save the file.
3. In order for Sherlock to recognize the file change, you must edit the file properties: Right-click the new file in the **Project Tree** and select **Edit Properties** from the context menu. Set the file properties as explained above in Adding Image File Layers (p. 516).
4. Now open the image file in the Layer Viewer by right-clicking the file in the **Project Tree** and select **View Layer**.
5. If the image requires alignment, follow the steps in the section above, Image File Alignment (p. 517).

8.5.5. Image Layer Stackup Data

After an image layer is aligned, Sherlock automatically computes the percentage of conductor material for the layer, which is displayed in the Stackup data table, as shown here:



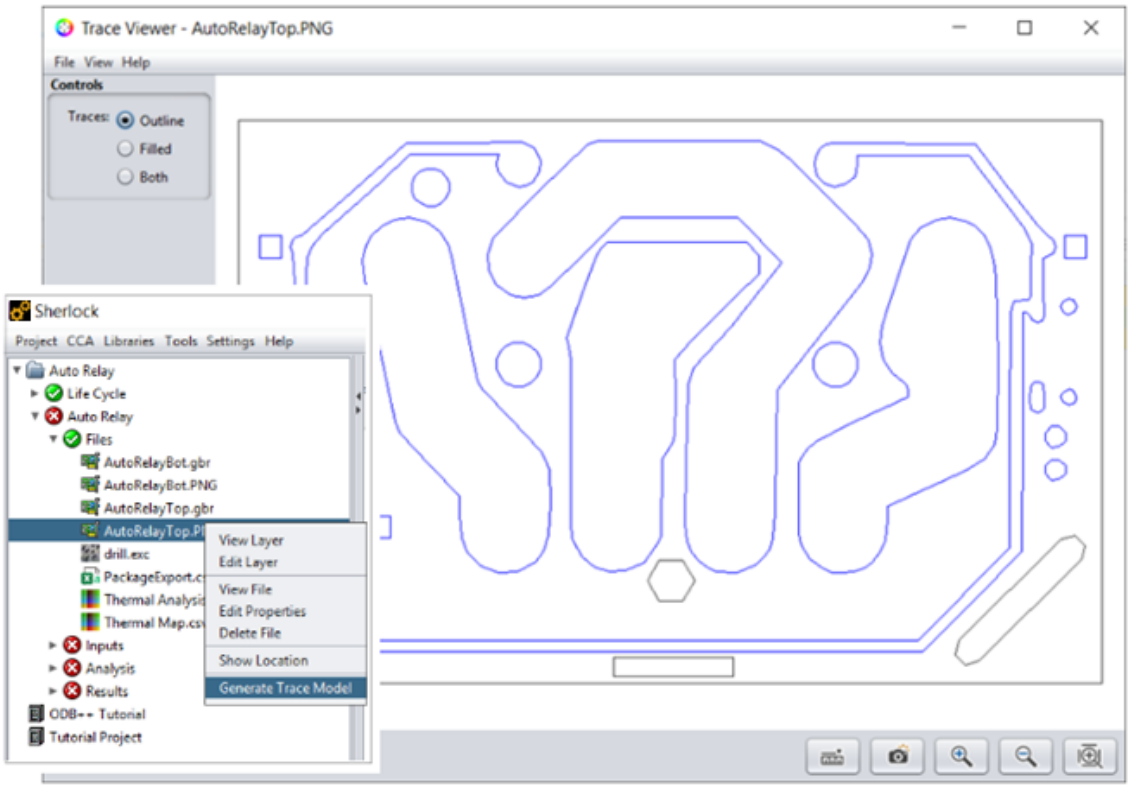
Layer	Type	Material	Construction	Thickness	Density	CTExy	CTEz	Exy	Ez
1	SIGNAL	COPPER (35.7%...		0.2 mm	4.3347	38.433	38.433	42,592	42,592
2	Laminate	Generic FR-4	DEFAULT (2116)	47.24 mil	1.9000	17.000	70.000	24,804	3,450
3	SIGNAL	COPPER (42.6%...		0.2 mm	4.8246	36.198	36.198	50,147	50,147

At that point, you are free to update the conductor and dielectric materials as needed. (See sections Additional Stackup Tools (p. 69) and Importing Copper Layers (p. 65) in Tutorial: Ad Hoc Project Creation (p. 56) for guides to editing the Stackup.) If any future alignment operations are performed, Sherlock will automatically update the conductor percentage without changing the assigned materials.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.5.6. Image Layer Trace Model

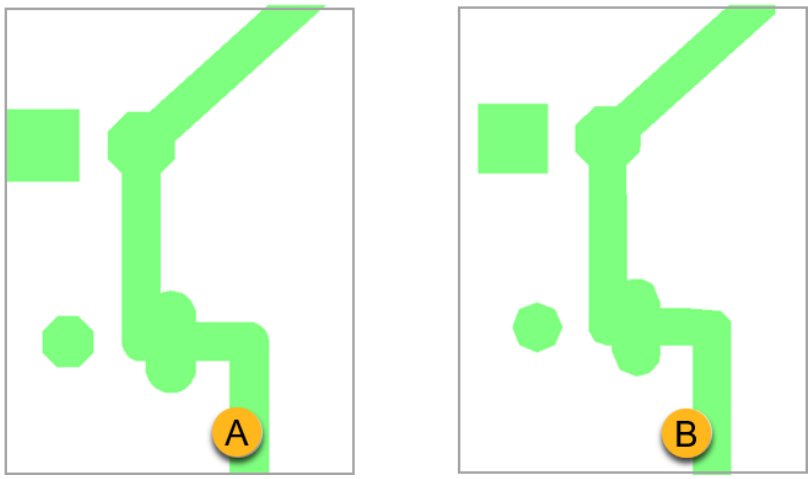
Just like a copper or laminate layer defined by an EDA file, you can generate a trace model for an image file layer by right-clicking the layer name in the **Project Tree** and selecting the **Generate Trace Model** menu option. After entering the desired trace model properties, Sherlock will scan-convert the image file to determine the polygonal trace outlines.



In the example above, the **Max Arc Segment** property was set to 0.1mm, to show most of the detail found in the image file.

Unfortunately, unlike EDA drawing commands, that provide precise straight-line and arc definitions, the scan conversion process can only guess which segments are straight and which are curved. Although it does a good job, that process can't generate the exact same trace model that would be found using the EDA drawing commands. For example, image 1 below shows a portion of the trace model generated directly from EDA drawing command, while the image 2 shows the trace model generated from the image file. Overall, the traces look the same, but there are subtle differences in some areas. Most notably, the oblong pad that overlaps the trace has perfectly rounded ends in the left model, but not in the right model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A = Trace Model Generated from EDA File. B = Trace Model Generated from Image File.

Nonetheless, the trace models generated from image files are usually close enough for FEA modeling purposes, as long as the image resolution is high enough to find the details. Furthermore, because you can freely edit the image file to add, remove or clean-up features, you can refine the trace model over time if necessary.

8.6. FEA- PCB Modeling Regions

The inherent accuracy-vs.-performance trade-off associated with the number of elements in a 3D FEA model leads many FEA modelers to a local/global approach in which a larger collection of smaller elements are placed near interesting locations on the PCB and a smaller collection of larger elements are used everywhere else. Basically, the total number of elements is reduced by using smaller elements only in places where needed. Sherlock supports such an approach using one or more **PCB Modeling Regions**, which allow users to control meshing properties (including trace modeling) in a given region for both analysis and export purposes.

When exporting models, modeling regions can be used to create Hybrid models that contain both mesh elements generated by Sherlock for certain PCB regions and mesh elements generated by the FEA tool for other PCB regions. Such an approach is useful when users want to create custom meshes for key areas of the PCB but want the convenience of using mesh elements generated by Sherlock for everything else. Modeling regions can also be used to export only a subset of the PCB or to exclude one or more regions from the PCB model. This allows users to more easily integrate the models generated by Sherlock into larger custom created models.

Our discussion of modeling regions begins with some simple examples of how they can be used. Following the examples, we show how modeling regions are defined and more complex uses of them.

Note:

Sherlock lets you convert a modeling region into a standalone circuit board. This feature can be helpful when, for example, you wish to create a trace model only for a specific modeling region. See Creating a CCA from a Modeling Region (p. 550).

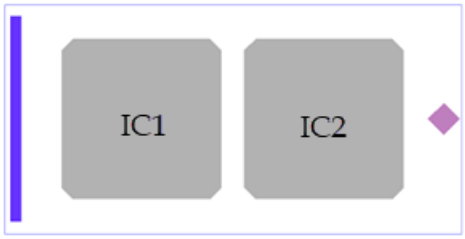
In this section, the following topics are covered:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.1. Modeling Region Example
8.6.2. Modeling Region Editor
8.6.3. Using Modeling Regions
8.6.4. Common Examples
8.6.5. Hybrid Modeling
8.6.6. Hybrid Models with Sub-Assemblies
8.6.7. Creating a CCA from a Modeling Region

8.6.1. Modeling Region Example

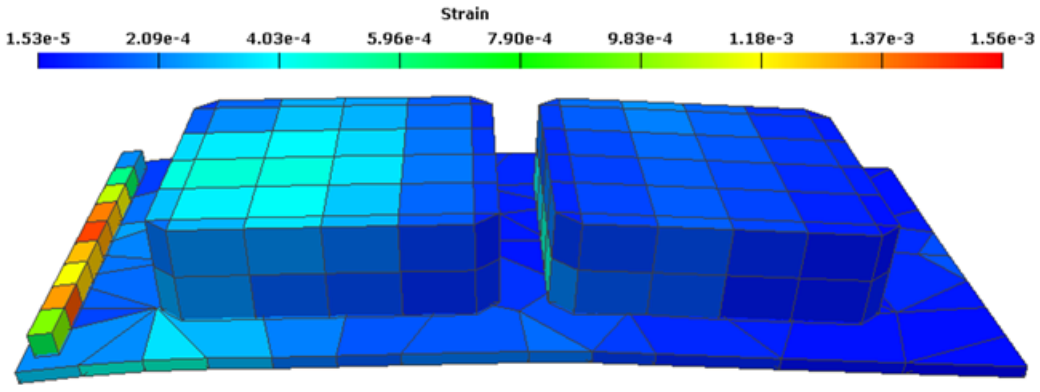
Consider a simple rectangular PCB with two large components (gray boxes), a single ICT fixture (blue rectangle on left side) and one test point (purple diamond on right side).



When the 20N load represented by the test point presses down on the right side of the board, the board will flex to the right and a distribution of strain values will be seen by the PCB, component and fixture elements. If we hold all other modeling variables constant and change the element sizes, we should see different displacement and strain values generated by the FE analysis. The following examples show how the analysis results vary depending on how many small elements we use.

8.6.1.1. Model 1 - Large Elements

If we use only large elements for the PCB, the analysis will run faster, but we expect the results to be less accurate. For example, the following strain map is generated if we use a 5mm mesh size for the PCB.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As shown in the strain map, the highest strain values are seen by the ICT fixture, with little or no strain seen across the PCB or components. The specific results for this example are as follows:

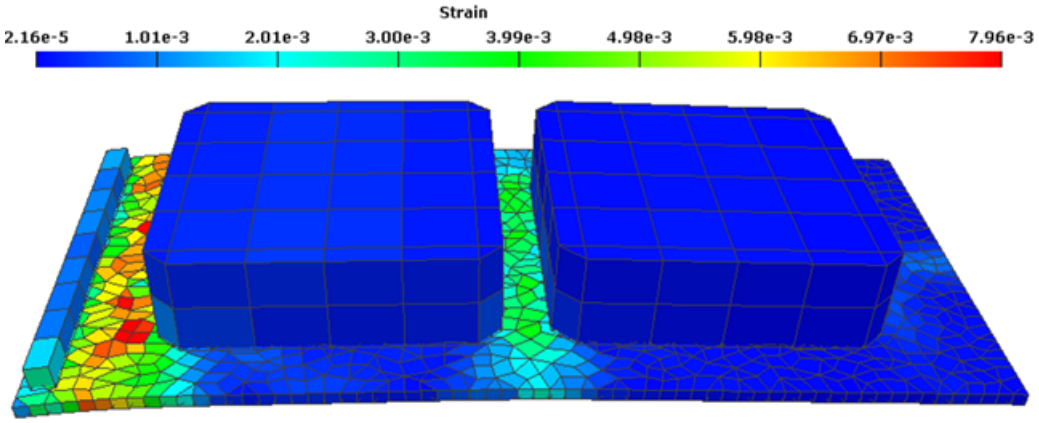
Table 8.4: Strain Map Results (5mm Mesh)

Mesh Size	5mm	**Max Displacement**	0.51mm
# of PCB Elements	71	**Max Strain**	3.90E-004

In this overly simple model, only 71 elements are generated for the PCB. More importantly, the elements are large enough that a single element can span the space between the ICT fixture and the left-most component, as well as the space between the two components, thereby creating a relatively stiff structure across the PCB. This is not a very accurate result, based on what we know about the structure of this simple example, because the mesh size is too large for the PCB elements.

8.6.1.2. Model 2 - Small Elements

If we decrease the mesh size from 5mm to 1mm, a much larger number of smaller elements will be created across the entire PCB. The smaller elements will allow the model to generate more accurate results, but the analysis time will also increase. In this case, the following strain map is generated using the same test point load:



With a more detailed PCB model, we can now see that the maximum strain occurs between the ICT fixture and the left-most component, as well as increased strain between the two components. The specific results for this example are as follows:

Table 8.5: Strain Map Results (1mm Mesh)

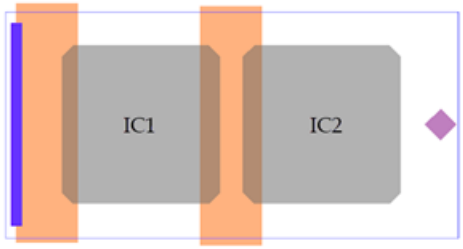
Mesh Size	1mm	**Max Displacement**	6.0mm
# of PCB Elements	1520	**Max Strain**	8.60E-003

The more accurate results come with a price of a significant increase in the number of elements, a 21-fold increase from 71 to 1520 elements (which is consistent with the 5-fold decrease in the mesh size). Analysis time is determined by the number of elements in the 3D model, so any unnecessary increase in the number of elements can be very expensive in terms of analysis time. Although in this simple case the analysis time is still reasonable, if a complex, real-world model is being generated, the analysis time could easily run into many hours or days depending on the number of elements used.

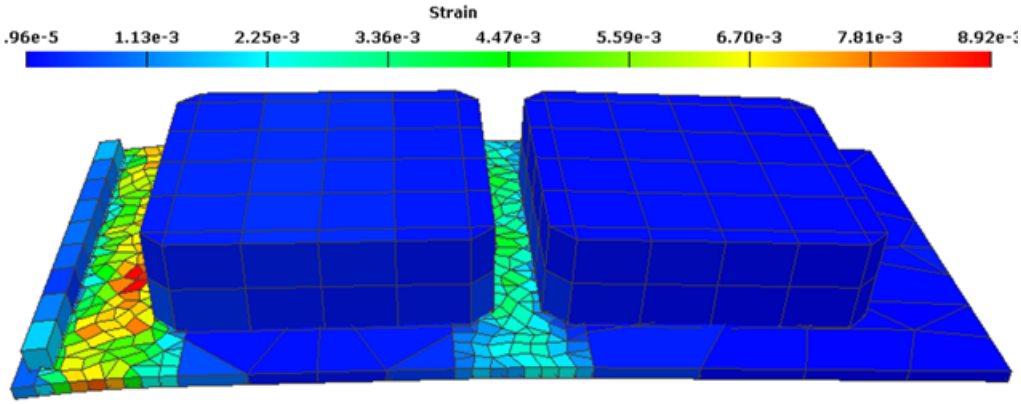
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.1.3. Model 3 - Large and Small Elements

Knowing that the PCB will bend the most where the structure is weakest, we can budget the number of elements needed by using small elements only in the weakest areas. For example, we can target specific PCB regions for special treatment, as shown here. The orange rectangles mark the PCB areas where we expect to see the most bending and strain because those areas have the least amount of structure.



To reduce the total number of elements generated in the model, we can use a 1mm mesh in the orange regions and a 5mm mesh everywhere else in the PCB. The resulting hybrid model generates the following strain results:



The combination of small and large mesh elements yield the following analysis results:

Table 8.6: Strain Map Results (1mm and 5mm Mesh)

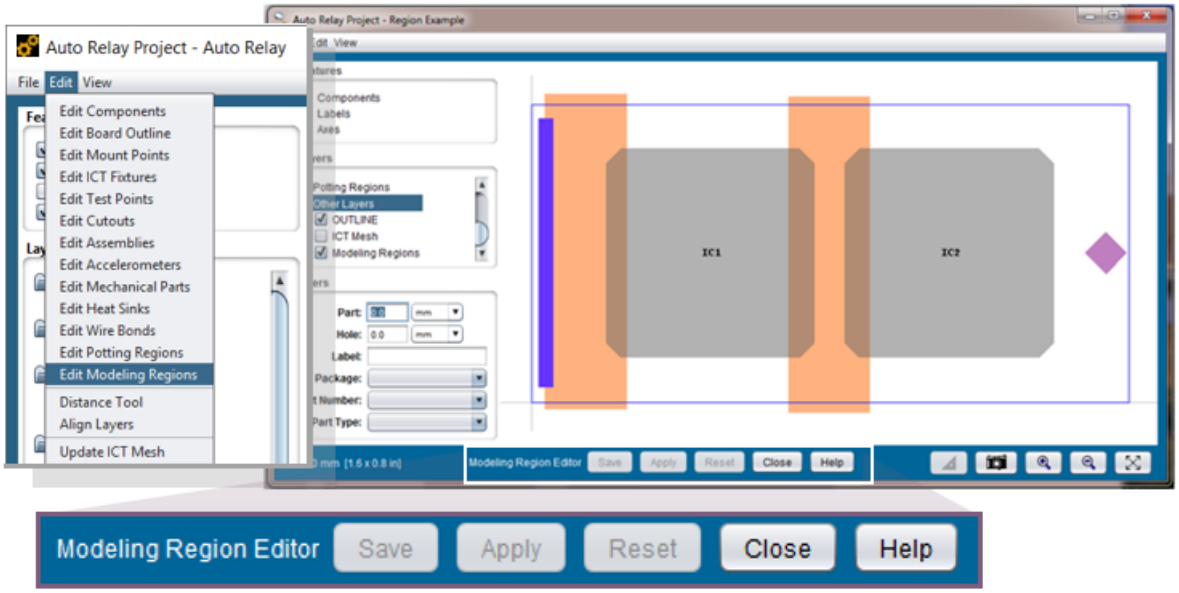
Mesh Size	1mm & 5mm	**Max Displacement**	6.1mm
# of PCB Elements	506	**Max Strain**	8.90E-003

When compared to the previous results, we see that the maximum displacement and strain values are within 3% of each other, but the total number of elements has been decreased by 66%. That is, we are generating basically the same results, but the model size is one-third the size, which will speed up the analysis time.

8.6.2. Modeling Region Editor

Modeling regions are managed in Sherlock using the **Edit > Edit Modeling Regions** menu option available in the **2D Layer Viewer**. When that menu option is selected, the **Modeling Region Editor** buttons will appear at the bottom of the **Layer Viewer** and all currently defined modeling regions will be displayed as orange polygons as shown.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Modeling regions can be selectively filtered from the **Layer Viewer** display by toggling the **Modeling Region** filter located in the **Other Layers** folder in the **Layers** panel on the left-side of the Layer Viewer. As a convenience, modeling regions are automatically displayed whenever the **Modeling Region Editor** is started.

Additional topics in this section:

8.6.2.1. Modeling Region Properties

8.6.2.2. PCB Modeling Properties

8.6.2.3. Trace Modeling Properties

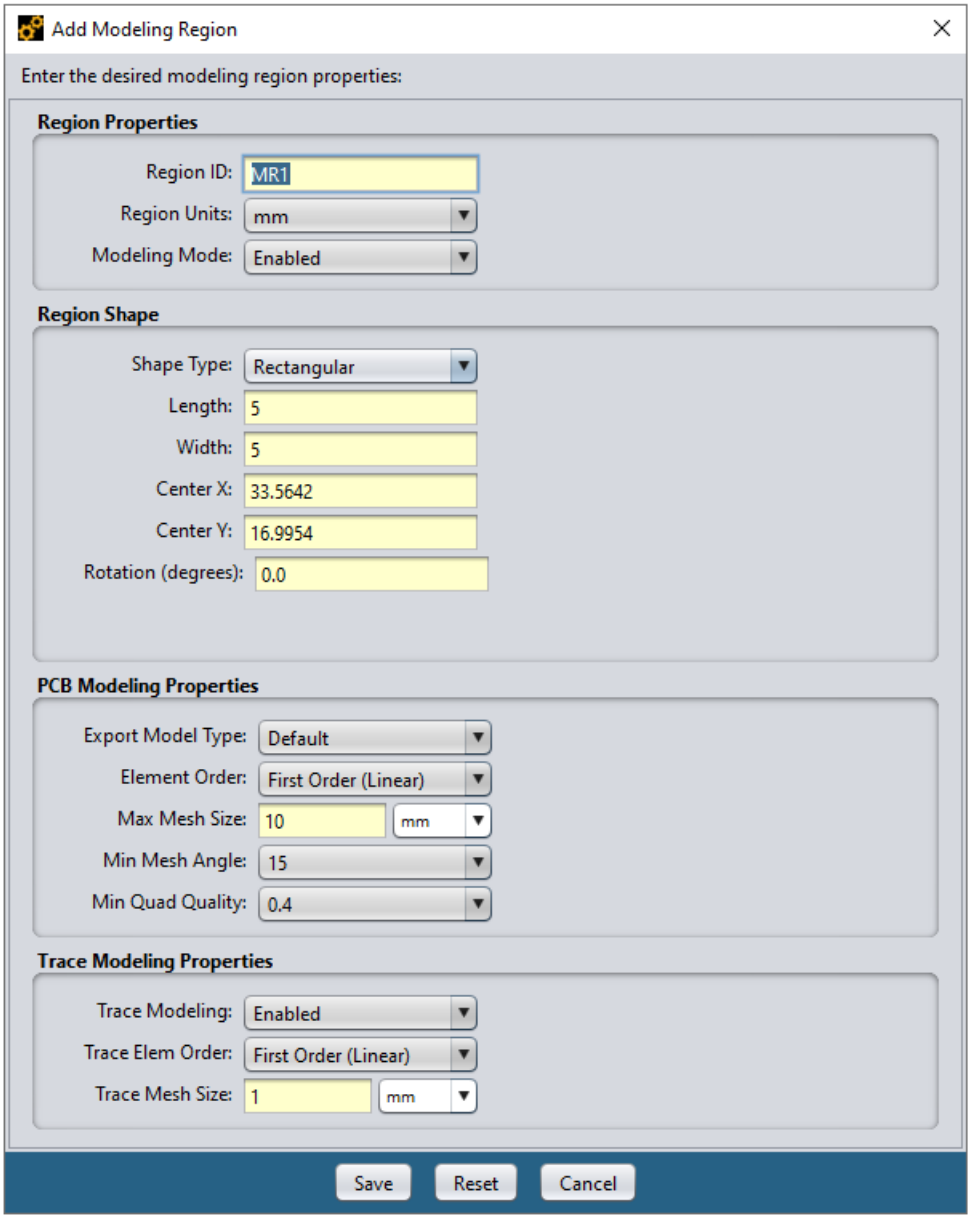
8.6.2.4. Graphical Editing

8.6.2.5. Modeling Region Restrictions

8.6.2.1. Modeling Region Properties

A modeling region can be added by right-clicking the mouse in the Layer Viewer outside of all existing modeling regions and selecting the **Add Modeling Region** option from the context menu. At that point, the **Add Modeling Region** dialog will be displayed, allowing you to edit the properties for the new modeling region.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An existing modeling region can be edited by right-clicking the mouse inside of the modeling region and selecting the **Edit Properties** option from the context menu. At that point, the **Edit Region Properties** dialog will be displayed, allowing you to edit the region properties. The layout and use of the **Add Modeling Region** and **Edit Region Properties** dialogs are the same, as we now describe.

The **Region Properties** panel defines the unique ID assigned to the modeling region (Region ID) and the units to be used for all dimensional properties (Region Units). The **Modeling Mode** property specifies how the region is used:

- **Enabled** = Use the modeling region
- **Disabled** = Ignore the modeling region
- **Excluded** = Don't generate any elements for the region

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Region Shape** panel specifies the shape of the region, with the following types supported:

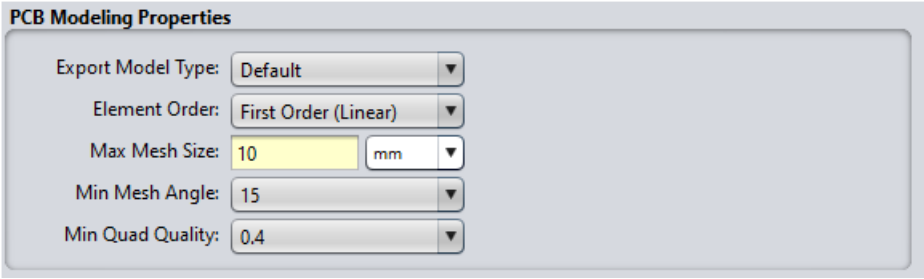
Table 8.7: Region Shapes

Polygonal	Defined by N points that form a convex polygon
Rectangular	Defined by 4 points that form a rectangle >
Slot	Defined by 8 points that form a rectangle with rounded ends
Circular	Defined by N points that form a circle

The shape specified for a region determines how the region can be graphically edited in the Layer Viewer. For example, the length and width of rectangular, slot and circular shapes are automatically updated when any of the corners are moved, regardless of the rotation specified. Polygonal shapes allow you to move/add/remove nodes as needed to surround a desired region.

8.6.2.2. PCB Modeling Properties

The primary purpose of modeling regions is to allow different sections of the PCB to be modeled using different mesh properties. As such, each modeling region allows the specification of the following PCB mesh properties:



Export Model Type - The type of model generated when exporting to Abaqus or Ansys Mechanical. See the Hybrid Modeling (p. 538) section below for more details.

Element Order - The type of 3D elements (First Order, Second Order, Shell) to be created for the PCB in the region.

Max Mesh Size - The maximum size of the mesh to be used in the region.

Min Mesh Angle - The smallest internal angle allowed for mesh triangles generated in the region.

Min Quad Quality - The minimum quality allowed for mesh quads generated in the region.

All the **PCB Modeling Properties** defined for a given region will override any similar properties defined in the FE analysis and/or export dialogs.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.2.3. Trace Modeling Properties

Much like the **PCB Modeling Properties**, each region can be used to override certain trace modeling properties specified in the FE analysis and/or export dialogs. The following trace modeling properties can be defined for each modeling region:



Trace Modeling - Specifies if trace modeling should be performed inside of the region. This can be used to reduce model complexity in one section of the PCB, while accurately modeling traces in another section of the PCB.

Trace Elem Order - The type of 3D elements (First Order, Second Order, Shell) to be created for traces in the region when trace modeling is enabled.

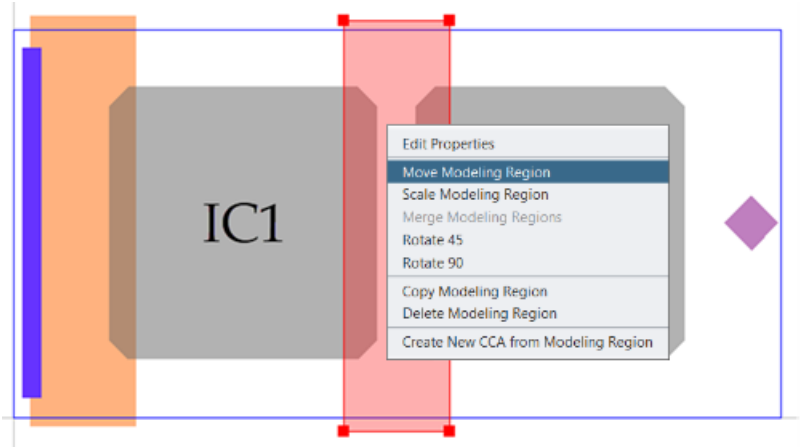
Trace Mesh Size - The maximum mesh size to be used in the region when trace modeling is enabled.

Note:

Traces must be generated before enabling them in Modeling regions.

8.6.2.4. Graphical Editing

Modeling regions can be graphically edited in the **Modeling Region Editor** by right-clicking the mouse inside of the modeling region.



At that point, the selected region will be high-lighted in red and the pop-up edit menu will be displayed, with the following options:

Edit Properties - Display the edit dialog.

Move Modeling Region - Move the modeling region to a new location, preserving the existing size and shape.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scale Modeling Region - Scale the modeling region around the current center location, preserving the existing shape.

Merge Modeling Regions - Merge two or more selected and intersecting regions into a single region.

Rotate 45/90 - Rotate the region by 45/90 degrees.

Copy Modeling Region - Create a copy of the selected region with the same properties

Delete Modeling Region - Remove the modeling region completely.

In addition to the edit options available in the context menu, you can also move any region by left-clicking in the region to select it and then drag the region to the new location. You may also left-click any of the highlighted region nodes and drag them to reshape the region based on the current **Region Shape** property.

Multiple regions can be selected by left-clicking and dragging a box around the regions to be selected. You can also left-click individual regions while pressing the CONTROL key to toggle the selection of individual regions, adding or removing them from the list of selected regions.

After editing one or more regions, the **Save**, **Apply**, **Reset** and **Cancel** buttons will be enabled in the Modeling Region Editor panel at the bottom of the Layer Viewer. Press the **Save** button to make all changes permanent and exit the editor. Press the **Apply** button to make all changes permanent, but remain in the editor. Press the **Reset** button to cancel all changes, but remain in the editor. Press the **Cancel** button to cancel all changes and exit the editor. If no changes are pending, the **Close** button can be pressed to exit the editor.

8.6.2.5. Modeling Region Restrictions

Since modeling regions are used to define PCB areas to be modeled in specific ways, there are several limitations on how multiple regions can be defined.

1. Modeling regions **may not** intersect each other. Region intersection will be checked whenever the **Save** or **Apply** buttons are pressed.

2. Modeling regions *are* allowed to contain one or more sub-regions as needed, but each sub-region must be fully contained by the region and they may not intersect each other. When graphically selecting regions with sub-regions, Sherlock automatically selects the smallest region containing the current mouse location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.3. Using Modeling Regions

Modeling regions can be used when performing FE analysis in Sherlock via one of the FE analysis dialogs or when exporting an FEA model to Abaqus or Ansys Mechanical using the FEA Export dialog. In both cases, the **Modeling Regions** PCB modeling property can be selectively **ENABLED** or **DISABLED** to control use of modeling regions. When enabled in the analysis or export dialog, all modeling regions that are not individually disabled will be used during the PCB modeling process. When disabled, all modeling regions, regardless of their individual settings will be ignored. This approach allows users to easily compare models that use modeling regions with those that do not. Users can further determine how specific modeling regions affect the PCB model by selectively enabling or disabling individual modeling regions.

When modeling regions are enabled, Sherlock automatically defines a **Default** region that consists of all PCB areas, if any, that are not covered by any modeling region. The **PCB Modeling** and **Trace Modeling** properties specified in the FE analysis or FEA Export dialogs are used to determine how the default region is modeled. For both analysis and export purposes, Sherlock automatically defines the necessary surfaces and tie constraints to bond the sides of adjacent and/or embedded modeling regions to each other.

It's important to note that individual modeling region properties only control how the PCB is modeled in each modeling region. The modeling of parts, mount points, test points, heat sinks, etc. is controlled solely by the modeling properties defined in the **FE analysis** or **FEA Export** dialogs.

8.6.4. Common Examples

Modeling regions can be used in a variety of ways to handle a wide range of modeling needs. The simplest approach uses one or more regions to mark PCB areas that need different element sizes and/or types. More complex approaches might use multiple nested regions to disable trace modeling in one region, while forcing detailed modeling in another. Regions may also be used to create precise holes in the PCB model that can be manually filled using a 3D model created in another application or even generate a 3D mesh only for a specific PCB region. In this section, we show by example some of the ways that modeling regions can be used in Sherlock.

8.6.4.1. Simple Regions

8.6.4.2. Nested Regions

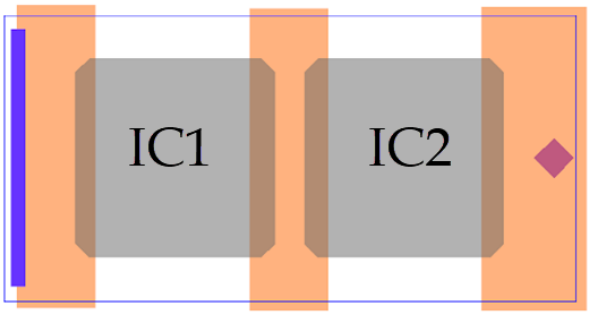
8.6.4.3. Regional Trace Modeling

8.6.4.4. Excluded Regions

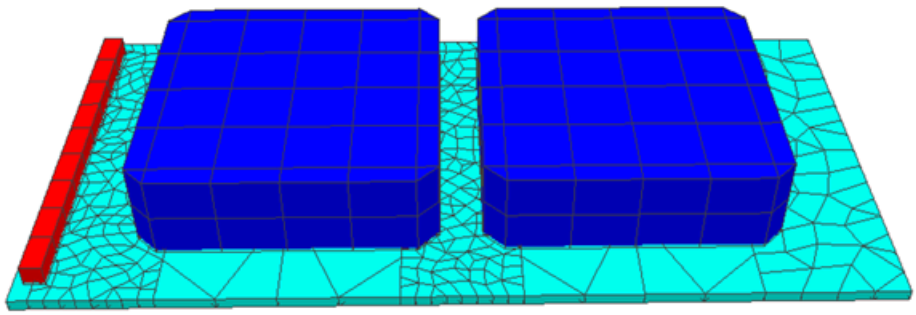
8.6.4.1. Simple Regions

The simplest way to use modeling regions is to define one or more distinct regions in key PCB areas, with appropriately defined mesh properties. For example, the following set of 3 modeling regions can be defined to cover the PCB areas where we expect to see the most flexing. The default mesh size for the PCB (as defined in the analysis or export dialogs) is 5mm.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the following model, a 1mm mesh has been defined for the left and center regions, while a 2mm mesh has been assigned to the right-most region. The model has small sized elements in the left and center regions, medium sized elements in the right region and large sized elements everywhere else in the PCB. Sherlock automatically generates TIE constraints to bond elements between different regions.



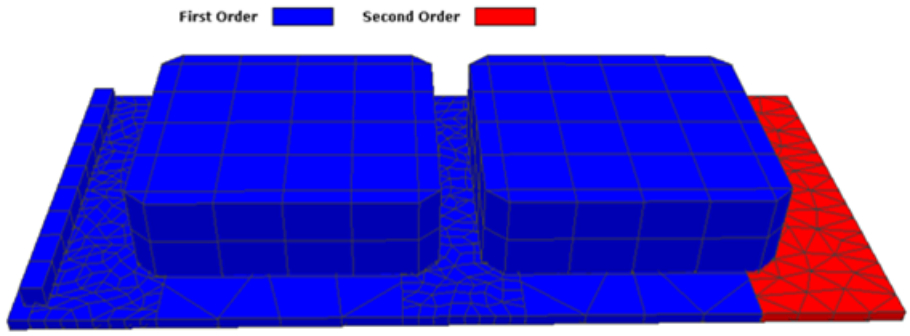
In addition to the mesh size, we can also specify different element type and shape properties for each of the modeling regions. For example, assume that First Order elements are used by default across all PCB regions and that we specify the following **PCB Modeling Properties** for the right-most region:

Table 8.8: PCB Modeling Properties

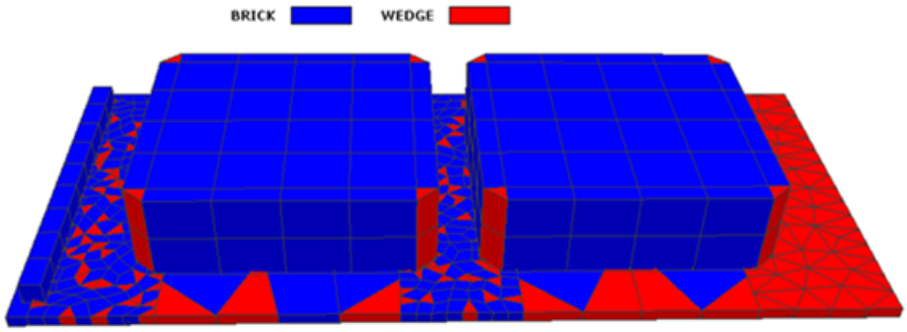
Element Order	Second Order
Max Mesh Size	2mm
Min Quad Quality	1.0

The resulting 3D model will not only have different size elements in the right-most region, but also those elements will all be **Second Order Elements**, as shown here:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



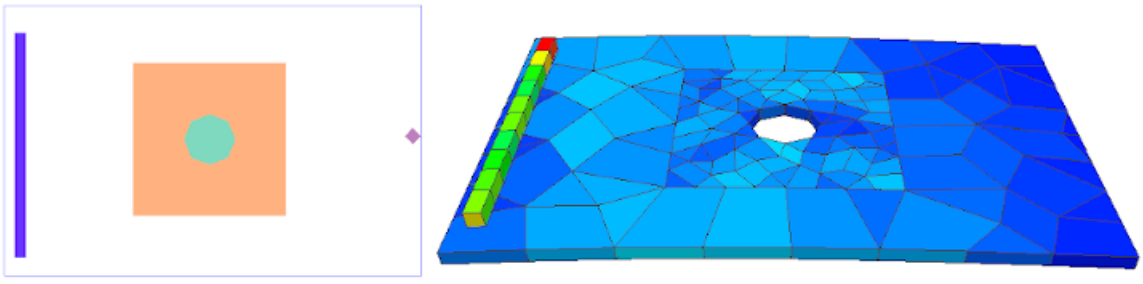
Furthermore, all the elements in the right-most region will be wedges, as shown here:



The trade-off between element types and shapes is similar to that of mesh size in that accuracy can be improved by selecting the appropriate element type/shape, but sometimes the cost is increased analysis time. Sherlock modeling regions allow users to make those tradeoffs as needed across different PCB areas.

8.6.4.2. Nested Regions

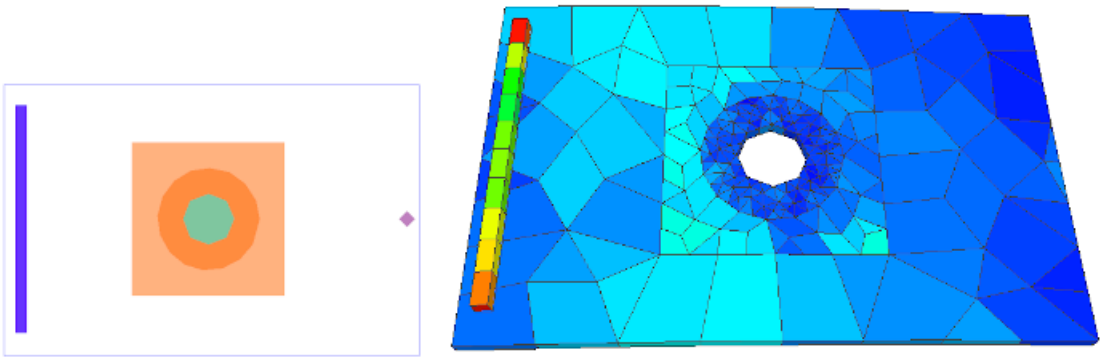
Although the previous examples have all shown rectangular regions that overlap the PCB outline, there are no restrictions on shape and/or location of modeling regions. Furthermore, modeling regions may be nested inside of each other to handle a variety of modeling needs. For example, consider the PCB shown here that has a circular cutout in the middle of the PCB and a single modeling region that surrounds the cutout. If we use a 4mm mesh for the modeling region and a 10mm mesh everywhere else, the following 3D strain map would be generated.



The square region around the hole has smaller elements, with a mixture of wedges and bricks, as specified by the modeling region properties.

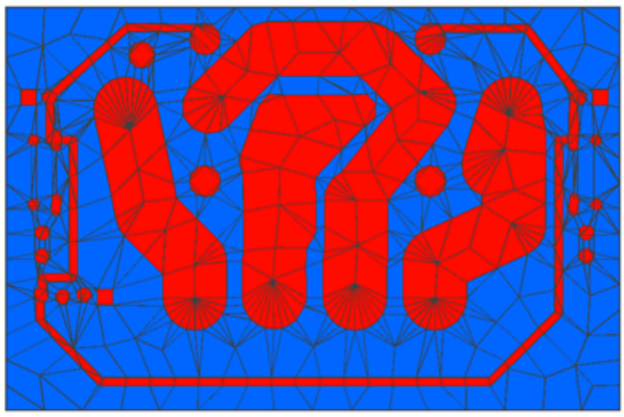
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

We can refine the model even more by creating a circular region inside of the square region, as shown here. The circular region can be defined to generate only 2mm wedge elements, resulting in the following 3D model.



8.6.4.3. Regional Trace Modeling

Consider the PCB model shown below with trace modeling enabled using a 5mm mesh size. In general, trace modeling results in a significant increase in the number of elements for a given mesh size because of the relatively small trace features. Furthermore, wedges with high aspect ratios are common if the mesh size is too large. Modeling regions can be used to help control how trace models are generated.



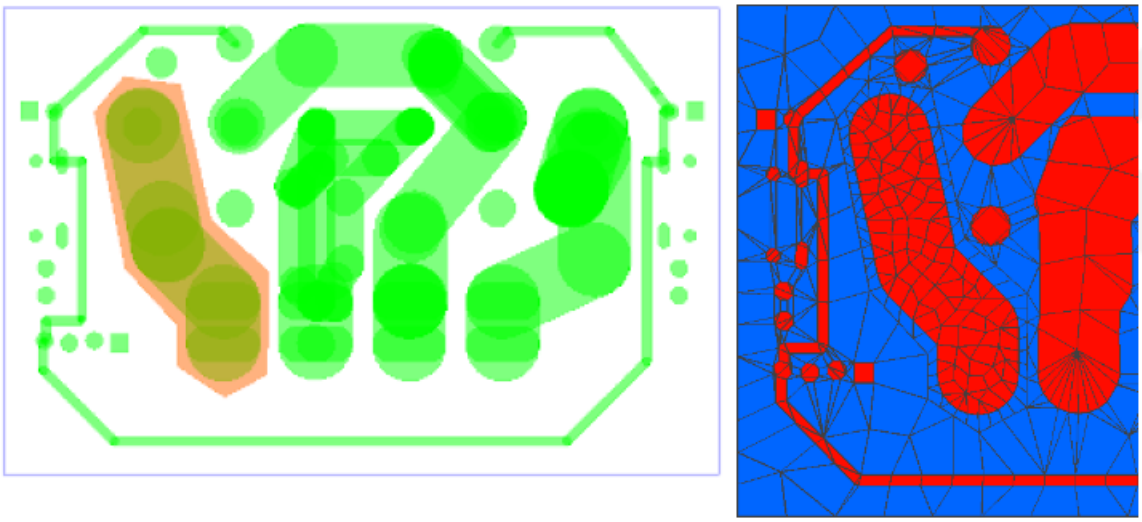
For example, suppose we want a more detailed model for the left-most large trace. A polygonal modeling region can be defined that surrounds the trace, as shown below (left). If we define the following **Trace Modeling Properties** in that region:

Table 8.9: Trace Modeling Properties

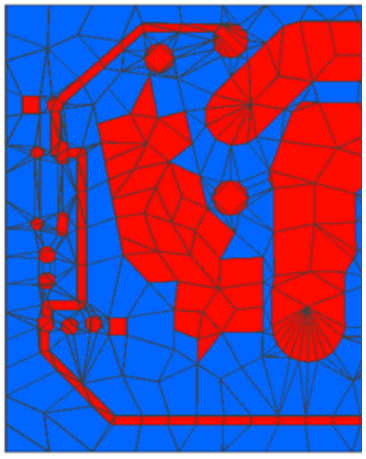
Trace Modeling	Enabled
Trace Mesh Size	2 mm

Then the resulting 3D model (below, right) will include smaller elements for the copper and resin areas inside of the modeling region and larger elements everywhere else.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



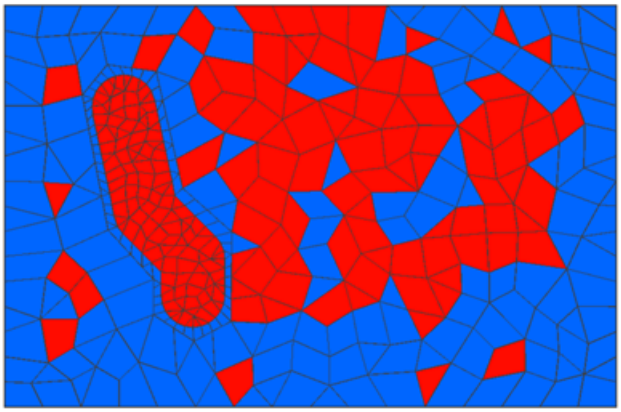
Instead of just changing element properties, we may want to enable or disable trace modeling selectively in different PCB areas. For example, if we set the **Trace Modeling** property in the modeling region defined above to **DISABLED**, then the 3D model shown below will be generated.



By default, trace modeling is performed across the entire PCB, but disabled inside of the modeling region. Since trace modeling is disabled inside of the region, the **PCB Mesh Size** (4mm) is used to create the elements in that region and the material properties are determined by averaging the copper and resin properties in each element. Although the resulting region doesn't look as pretty, the elements in that region fairly accurately represent the trace and the number of elements is significantly reduced.

We can achieve the opposite approach (trace modeling only inside a specific region) by disabling trace modeling in the **Default** region and enabling trace modeling in the desired region. The 3D model shown below would then be generated. In this case, trace modeling is only done inside of the modeling region using a 2mm mesh size, while the rest of the PCB is modeled without traces using a 5mm mesh size. Multiple regions can be used to create other PCB areas with trace modeling as needed for analysis purposes.

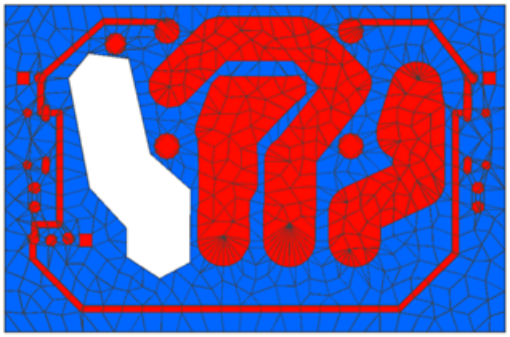
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.6.4.4. Excluded Regions

In the previous examples, all the modeling regions have been **Enabled**, allowing them to override the default PCB and Trace modeling properties within specified PCB areas. Modeling regions can also be used to selectively exclude PCB areas from the 3D model that is generated. Such an approach may be needed, for example, to create a void in the PCB where the user intends to insert a custom-built mesh that was created using another FEA modeling tool. On the other hand, you may want to exclude everything except a given region so that the resulting 3D mesh can be inserted into another custom-built model. Both approaches are supported by defining modeling regions in which the **Modeling Mode** property is set to **Excluded**.

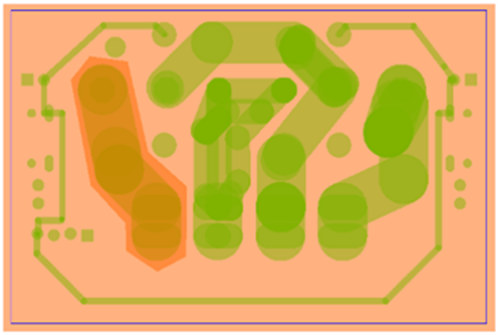
For example, using the same region described in the previous example, if we enable trace modeling in the default region, but mark the inner region as **Excluded**, then the 3D model shown below will be generated.

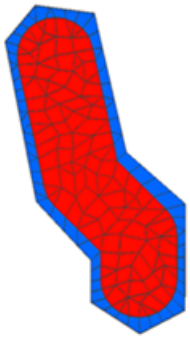


To achieve the opposite effect, we need to create an outer region that covers the entire PCB, as shown below (left). We can then mark the outer region as **Excluded** and the inner region as **Enabled** to generate the 3D model shown on the right.

Note, if **Modeling Mode** is set to **Disabled** for a given region, then that region will be ignored when generating the 3D model. This allows users to selectively enabled or disable regions when trying to construct the perfect model for their purposes. If you don't want a given region included in the model at all, use the **Excluded** setting.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





8.6.5. Hybrid Modeling

Modeling regions can be used not only to control how specific PCB regions are meshed by Sherlock, but they can also be used to export **Hybrid Models** that contain both mesh elements generated by Sherlock for some regions and mesh elements generated by Abaqus or Ansys Mechanical for other regions. This flexibility allows users to customize key PCB areas, while still using Sherlock to automatically generate models for everything else.

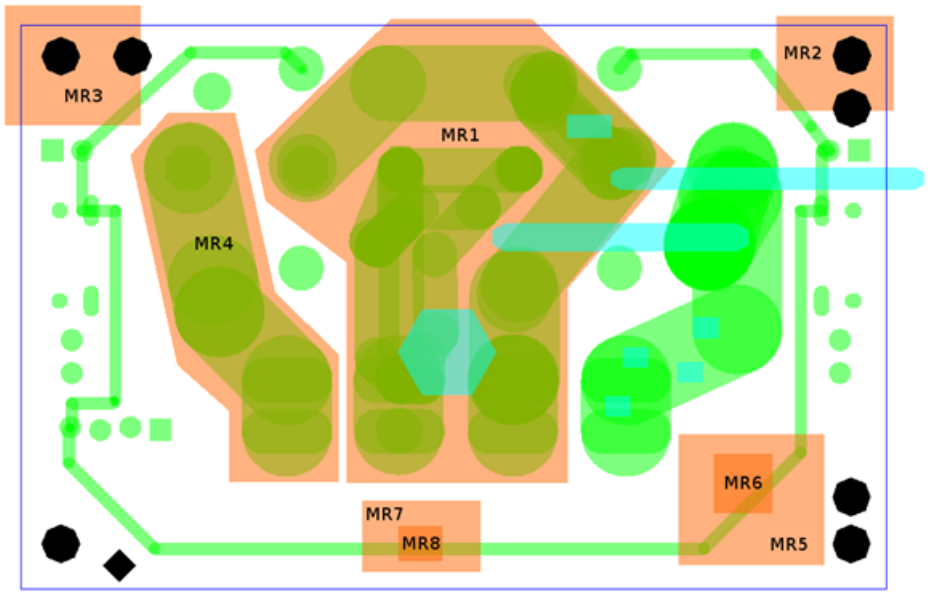
The **Export Model Type** modeling region property allows users to specify the type of model to be generated for a given modeling region, with the following options:

- **Default -** use the model type specified in the Export FEA Dialog
- **Sherlock -** use mesh elements generated by Sherlock to model the region
- **Sweep -** use mesh elements generated by the FEA tool to model the region
- **None -** define only geometric volumes for the region, without generating mesh elements

The **Default** model type is used to indicate that a given region should be modeled based on the **Mesh Type** property defined in the FEA Export Dialog. For example, if a **Bonded** mesh type is specified in the FEA Export Dialog, then all **Default** modeling regions will be modeled by elements generated by Sherlock. On the other hand, if a **Sweep** mesh type is specified in the FEA Export Dialog, then all **Default** modeling regions will be modeled by elements generated by the FEA tool.

For demonstration purposes, consider the following set of eight modeling regions:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The modeling regions encompass different trace and mounting areas of the PCB, including two pairs of embedded areas. The modeling regions are defined with the following properties:

Table 8.10: Modeling Region Properties

Region	Export Model Type	Max Mesh Size	Trace Modeling	Trace Mesh Size
MR1	Sherlock	2 mm	Enabled	1 mm
MR2	Sweep	2 mm	Disabled	N/A
MR3	Sherlock	2 mm	Disabled	N/A
MR4	Default	3 mm	Enabled	1 mm
MR5	Default	1 mm	Default	1 mm
MR6	Default	2 mm	Default	2 mm
MR7	Sherlock	5 mm	Disabled	N/A
MR8	Sherlock	5 mm	Disabled	N/A

The following sub-sections show the exported models based on these modeling regions for different settings in the FEA Export dialog.

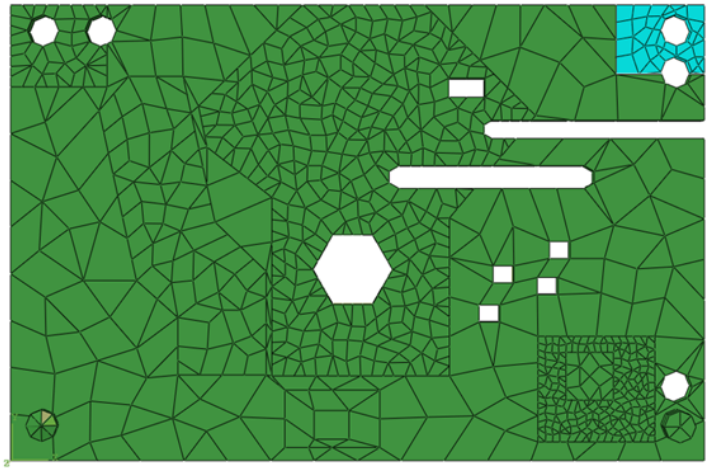
8.6.5.1. Bonded Uniform
8.6.5.2. Bonded Layered
8.6.5.3. Bonded Trace Layered
8.6.5.4. Bonded Layered Elements
8.6.5.5. Bonded Trace Elements
8.6.5.6. Sweep Uniform
8.6.5.7. Sweep Layered
8.6.5.8. Sweep Trace Layered
8.6.5.9. Sweep Layered Elements

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.5.10. Sweep Trace Elements

8.6.5.1. Bonded Uniform

When the default mesh type is set to **Bonded** (that is, Sherlock meshing), with a **Uniform** PCB material model, the following **Hybrid Model** is exported:

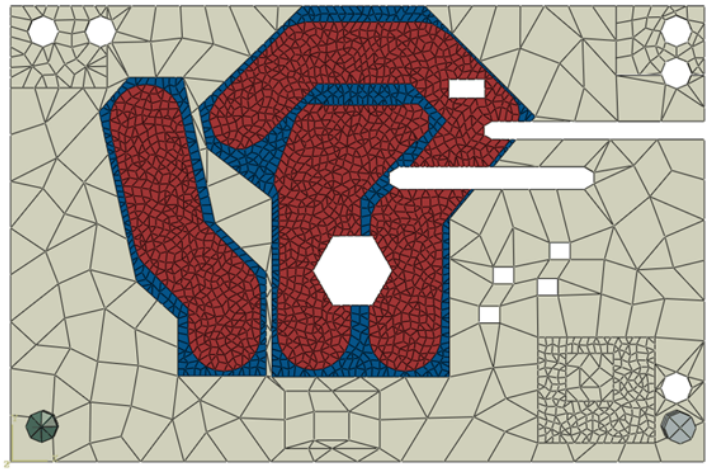


In this case, all regions are meshed by Sherlock, except for region MR2 (upper right corner), which is meshed by the FEA tool using its native sweep meshing algorithm. Note how the mesh size varies depending on the region, with the default mesh size set to 5 mm. Sherlock automatically defines constraints to tie the regions together. Since a **Uniform** PCB material model was specified, a single uniform PCB layer is generated, without any trace modeling.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.5.2. Bonded Layered

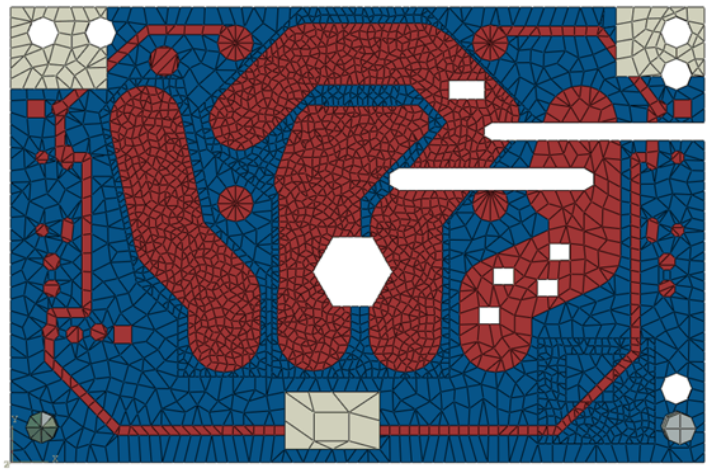
When the default mesh type is set to **Bonded** (that is, Sherlock meshing), with a **Layered** PCB material model, the following **Hybrid Model** is exported:



In this case, the **Trace Modeling** property is disabled for the default region, resulting in uniform material properties for most of the PCB layer being shown. However, since **Trace Modeling** is enabled for the MR1 and MR4 regions, the traces contained by those regions are modeled using the 1 mm mesh size specified for both regions. The appropriate conductor and dielectric material properties are assigned to the elements in those regions, while the average layer material properties are assigned to all other layer elements.

8.6.5.3. Bonded Trace Layered

The following model differs from the previous one by enabling trace modeling in the FEA Export dialog, with a trace mesh size of 2 mm, for the default region:

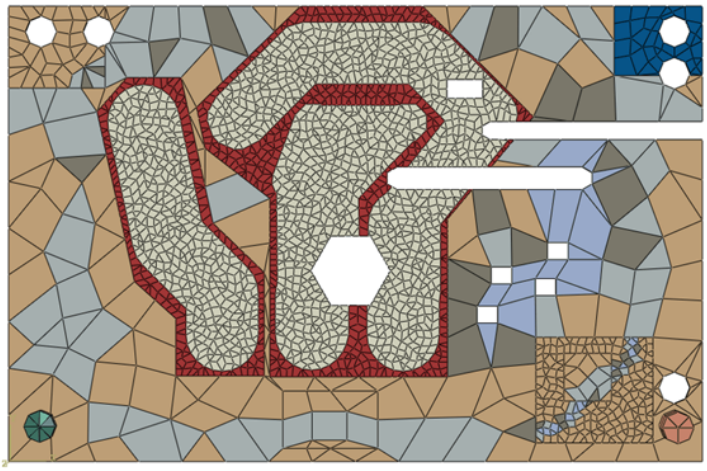


In this case, most of the PCB traces are modeled with different trace mesh sizes in each of the modeling regions. However, since **Trace Modeling** is specifically disabled in the MR2, MR3, MR7 and MR8 regions, no trace modeling is visible in those regions and the resulting elements are assigned the average layer material properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.5.4. Bonded Layered Elements

Unlike the **Layered** PCB material model which computes a single average material across the entire layer, the **Layered Elements** PCB material model computes a range of materials based on the ratio of conductor and dielectric materials WITHIN EACH ELEMENT. By combining the **Layered Elements** material model with trace modeling in selected regions, users can fine-tune models to suit their specific needs. For example, the following model enables trace modeling in two regions (MR1 and MR4), while using the **Layered Elements** material model for all other regions.



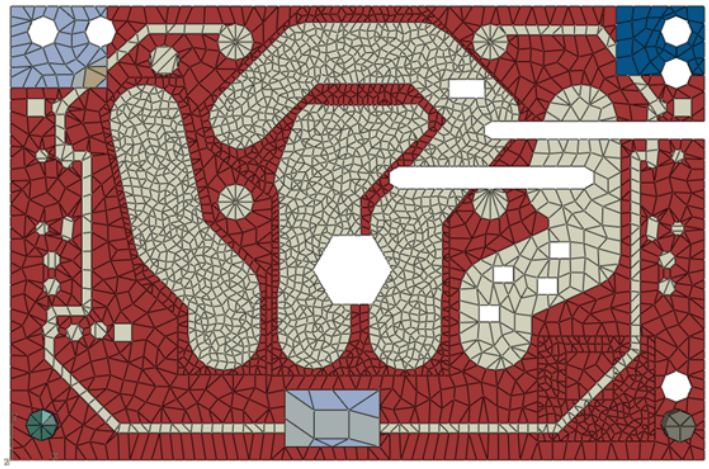
The conductor and dielectric materials in the trace regions are clearly evident. We can also see a range of materials across the other regions, with the exception of region MR2, that show the ratio of conductor and dielectric materials in each element. The small mesh sizes defined for regions MR5 and MR6 (lower right corner) creates a rough, but effective, approximation of the trace running through those regions.

It is important to note that all the elements in the MR2 region (upper right corner) are assigned the same material properties. This occurs because the MR2 region is defined as a single volume that is meshed by the FEA sweep algorithm. As such, Sherlock can only assign a single average material to that volume when it is defined. The average material properties are based on the ratio of conductor and dielectric materials CONTAINED INSIDE THE MR2 REGION ITSELF.

8.6.5.5. Bonded Trace Elements

This model differs from the previous one by enabling trace modeling in the **FEA Export** dialog:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

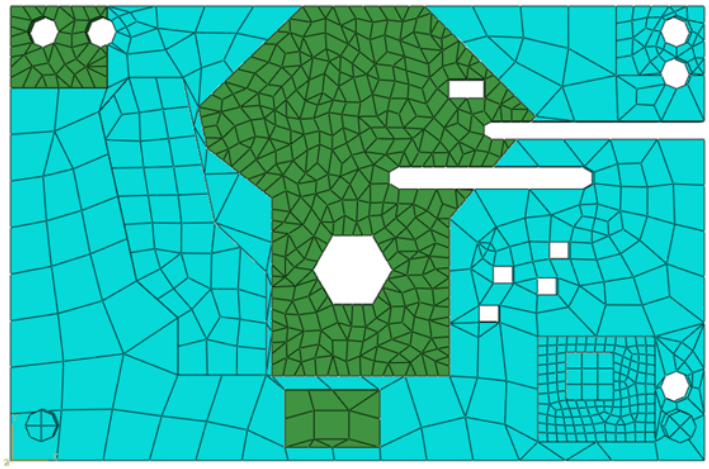


Since **Trace Modeling** is disabled in the MR3 (upper left corner) and MR7/MR8 regions (lower center), but those regions are meshed by Sherlock, the elements in those regions are assigned material properties based on the ratio of conductor and dielectric materials contained by each element. In region MR2 (upper right corner), trace modeling is disabled, the region is meshed by the FEA tool, and all elements are assigned the same material properties based on the ratio of conductor and dielectric materials contained by that region.

It is also interesting to note how the elements in the MR5/MR6 regions (lower right corner) differ from the previous example. Since **Trace Modeling** is set to **Default** in regions MR5 and MR6, the **Trace Modeling** setting from the **FEA Export** dialog is used to determine if traces should be generated in those regions. In this case, **Trace Modeling** was enabled in the **FEA Export** dialog, so it is also automatically enabled in those regions, but each region has a different trace mesh size than the default region.

8.6.5.6. Sweep Uniform

The previous examples used a **Mesh Type** of **Bonded** to generate models in which Sherlock meshed the default PCB region and all regions where **Export Model Type** was set to **Default**. In this example, the default mesh type is set to **Sweep** to generate a model in which the FEA tool meshes the default PCB region, using a **Uniform** material model.



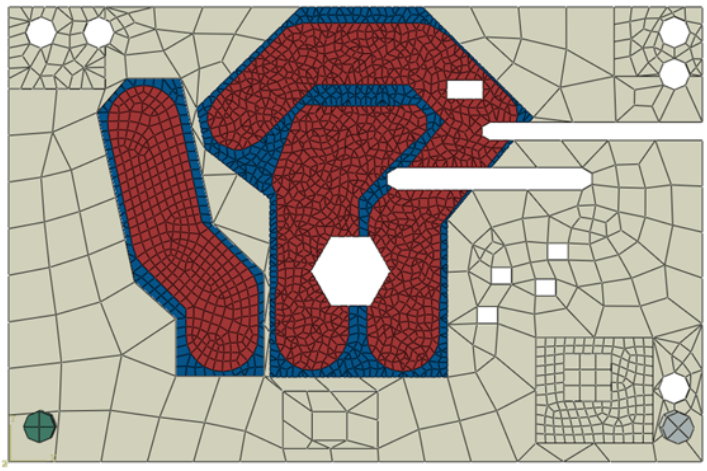
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, most of the PCB is meshed by the FEA tool, except for the MR1, MR3, MR7 and MR8 regions, which specifically set the **Export Model Type** to **Sherlock**. In those regions, Sherlock generates the mesh elements and exports them to the FEA as an "orphan mesh". Sherlock also automatically generates the tie constraints required to bond adjacent modeling regions to each other.

It is important to note the mount pads located in the lower left and lower right corners. Since the **Sweep** mesh type is specified in the FEA Export dialog, all PCB components, including mount points are meshed by the FEA tool. In the previous **Bonded** examples, the mount pads were meshed by Sherlock.

8.6.5.7. Sweep Layered

The following example differs from the previous **Bonded Layered** example by setting the default **Mesh Type** to **Sweep** in the FEA Export dialog.

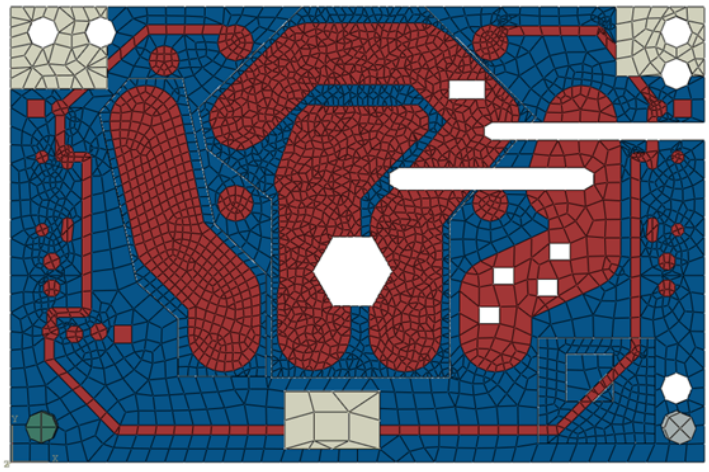


In this case, most of the PCB is meshed by the FEA tool and assigned an average material based on the composition of the entire layer. The MR1 region (upper center) is meshed by Sherlock, with trace modeling enabled. The MR4 region (left center) is meshed by the FEA tool because the **Export Model Type** is set to **Default** for that region and traces are modeled in the MR4 region because **Trace Modeling** is specifically enabled for that region. The elements generated for the MR1 and MR4 regions are assigned the appropriate conductor and dielectric material properties because trace modeling is enabled.

8.6.5.8. Sweep Trace Layered

The following example differs from the previous **Bonded Trace Layered** example by setting the default **Mesh Type** to **Sweep**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

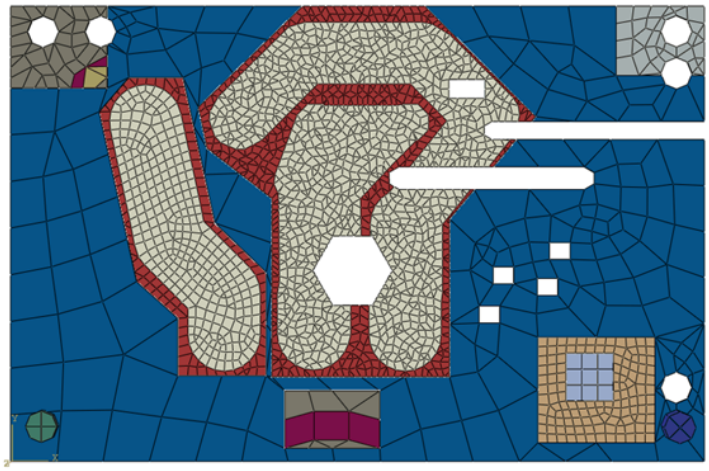


In this case, most of the PCB traces are modeled, with different trace mesh sizes in each of the modeling regions. However, since **Trace Modeling** is specifically disabled in the MR2, MR3, MR7 and MR8 regions, no trace modeling is performed in those regions and the resulting elements are assigned the **average** layer material properties.

By comparing this model to the previous **Bonded Trace Layered** model, it is interesting to note that the Sherlock meshing algorithm is best in some areas, while the FEA tool meshing algorithm is best in others. In general, the FEA tool sweep algorithm works best for mostly rectangular areas. For round areas, like circular pads, the sweep algorithm tends to generate a larger number of smaller elements around the area than the Sherlock meshing algorithm.

8.6.5.9. Sweep Layered Elements

The following model was generated by setting **Mesh Type** to **Sweep** and **PCB Model** to **Layered Elements** in the FEA Export dialog.



This is a good example of how Sherlock automatically computes and assigns material properties to elements.

In the MR1 region (upper center), Sherlock meshing is used with trace modeling enabled. In the adjacent MR4 region (left center), FEA sweep meshing is used with trace modeling enabled. In both cases, the resulting elements are assigned the appropriate conductor and dielectric properties.

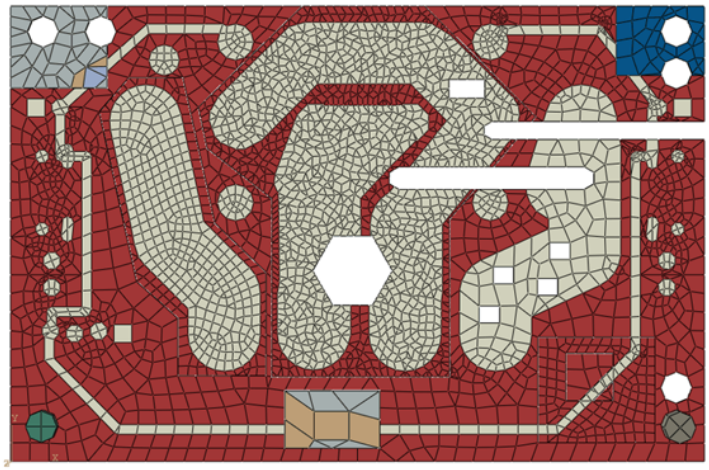
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the MR2 region (upper right corner) and the MR5/MR6 regions (lower right corner), FEA sweep meshing is used with trace modeling disabled, resulting in single volume definitions for each region. Since the **Layered Elements** material model is specified in the FEA Export dialog, the elements created by the FEA tool are assigned the same material properties, based on the ratio of conductor and dielectric materials contained in each individual region.

In the MR3 region (upper left corner) and the MR7/MR8 regions (lower center), Sherlock meshing is performed with trace modeling disabled. In such cases, Sherlock can automatically compute the average material properties for each individual element, resulting in crude approximations of the traces that are contained by those regions.

8.6.5.10. Sweep Trace Elements

This model differs from the previous one by enabling trace modeling for the default region:

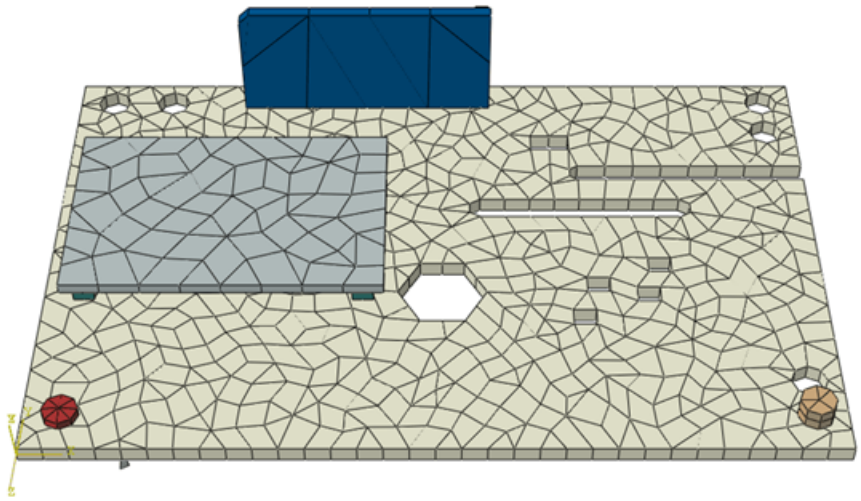


With trace modeling enabled in the default region, trace modeling is automatically enabled in the MR5/MR6 regions (lower right corner) as well because **Trace Modeling** is set to **Default** in those regions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.6. Hybrid Models with Sub-Assemblies

When exporting hybrid models with sub-assemblies, the modeling of each sub-assembly depends on the FEA settings that were last saved for the individual sub-assembly, either for analysis or export purposes. That allows users to selectively enable/disable modeling regions and/or use different meshing algorithms for individual sub-assemblies. For example, consider the following PCB (tan elements) that has a mezzanine card (gray elements) attached by mount points and an edge-connected card (blue elements) attached along the bottom edge.



In this case, all boards use the same modeling properties, except for the **Max Mesh Size**, as shown in the following table:

Table 8.11: Board Modeling Properties

Board	Mesh Type	Modeling Regions	PCB Model	Max Mesh Size	Part Modeling
PCB	Bonded	Disabled	Uniform	3	Disabled
Mezzanine	Bonded	Disabled	Uniform	4	Disabled
Edge-Connected	Bonded	Disabled	Uniform	10	Disabled

Since the Bonded mesh type is specified for all boards, Sherlock generates the mesh elements for the entire model, using the given mesh size for each board.

Note:

Ansys Sherlock saves different sets of modeling properties for each FE analysis task. As such, be sure to modify properties for sub-assemblies under the same FE analysis task. For example, when exporting a model from the **Natural Freq** analysis task, you should modify the **Natural Freq** modeling properties of each sub-assembly as needed.

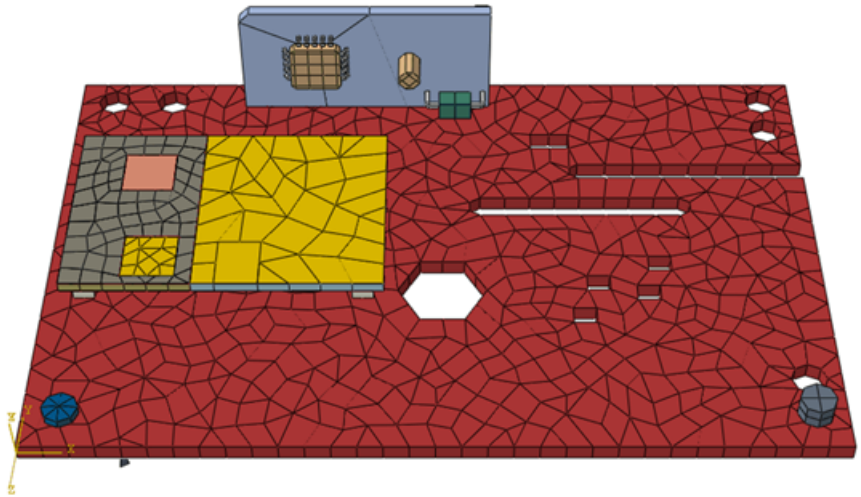
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

We can generate detailed models of the sub-assemblies using the following modeling properties (changes from previous example are **CAPITALIZED**):

Table 8.12: Sub-Assembly Modeling Properties

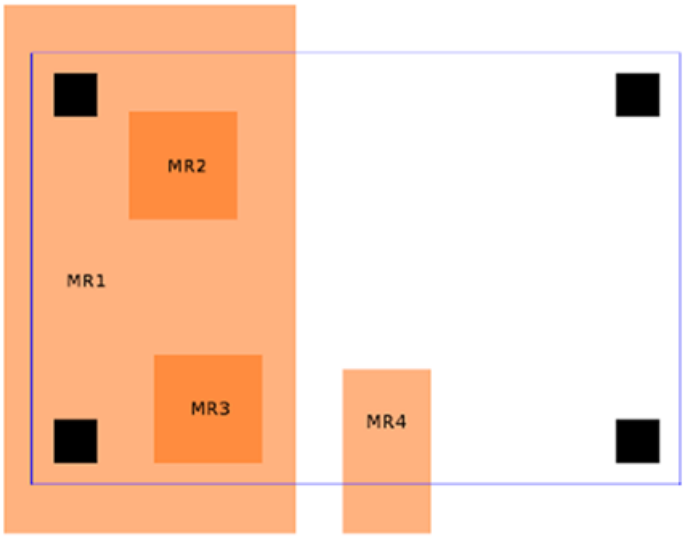
Board	Mesh Type	Modeling Regions	PCB Model	Max Mesh Size	Part Modeling
PCB	Bonded	Disabled	Uniform	3	Disabled
Mezzanine	Bonded	**ENABLED**	**LAYERED ELEMENTS**	4	Disabled
Edge-Connected	**SWEEP**	Disabled	Uniform	10	**ENABLED**

In this case, we are enabling modeling regions and using a Layered Elements material model for the mezzanine card, while enabling part modeling and using the FEA tool sweep meshing algorithm for the edge-connected card, resulting in the following model:



There are four modeling regions defined for the mezzanine card as shown in the diagram to below. The MR1 and MR2 regions are meshed using the FEA tool sweep algorithm, while the MR3, MR4 and default regions are meshed by Sherlock. Since the **Layered Elements** material model is specified for the mezzanine card each of the regions meshed by the FEA tool are assigned different material properties based on the region and all of the regions meshed by Sherlock are assigned the same material properties based on the layer.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

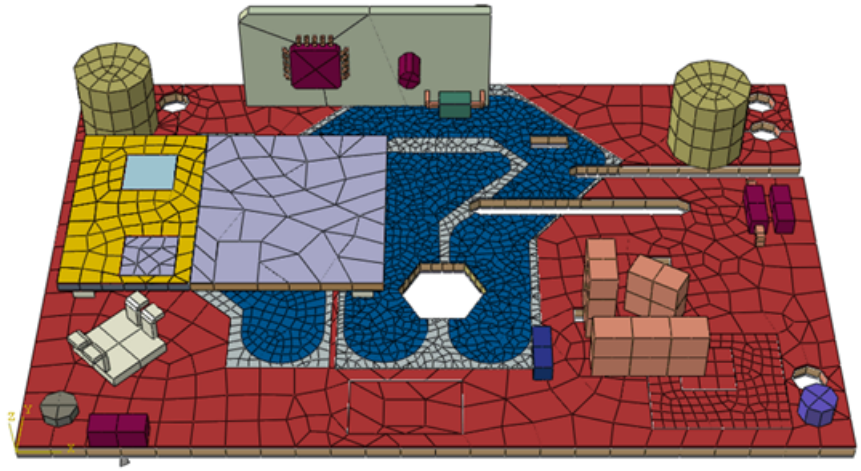


The edge-connected card is meshed by the FEA tool and includes part models that are meshed by Sherlock. The part models are meshed by Sherlock because the **Mesh Type** property in the FEA Export dialog FOR THE MAIN BOARD was set to **Bonded**.

Note:

All sub-assembly components (for example, parts, leads, mount points, wire bonds, etc.) are meshed using the same method as that used for the PCB components, which is determined by the **Mesh Type** property in the FEA Export dialog.

The following example differs from the previous example by changing various properties in the **FEA Export** dialog, without changing any sub-assembly properties.



Specifically, the following modeling properties were used (changes are **CAPITALIZED**):

Table 8.13: Modeling Properties

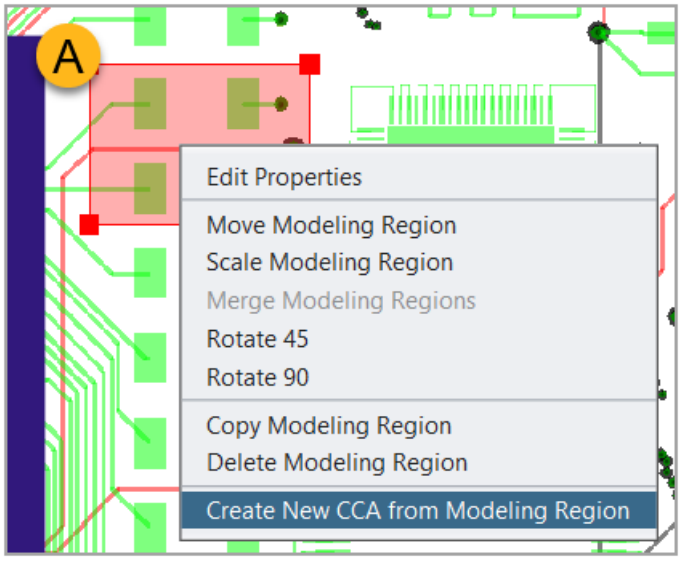
Board	Mesh Type	Modeling Regions	PCB Model	Max Mesh Size	Part Modeling
PCB	**SWEEP**	**ENABLED**	**UNIFORM**	3	**ENABLED**
Mezzanine	Bonded	Enabled	Layered Elements	4	Disabled
Edge-Connected	Sweep	Disabled	Uniform	10	Enabled

This example shows how users can effectively use a wide range of different modeling capabilities in a multi-card assembly to increase modeling accuracy while maintaining a reasonable model size.

8.6.7. Creating a CCA from a Modeling Region

Sherlock lets you convert a modeling region into a standalone circuit board. This feature can be helpful when, for example, you wish to create a trace model only for a specific modeling region. By default, Sherlock converts the original CCA's copper layer files to trace model layer files for the new CCA. The resolution of the copper layer files in the new CCA are controlled by the **Trace Mesh Size** property of the modeling region. See Modeling Region Properties (p. 527). A finer mesh results in higher resolution.

In the Layer Viewer, when the Modeling Region Editor is active (see Modeling Region Editor (p. 526)), right-click the modeling region and select **Create New CCA from Modeling Region** from the context menu (A, below). This opens the **Add Circuit Card** dialog.

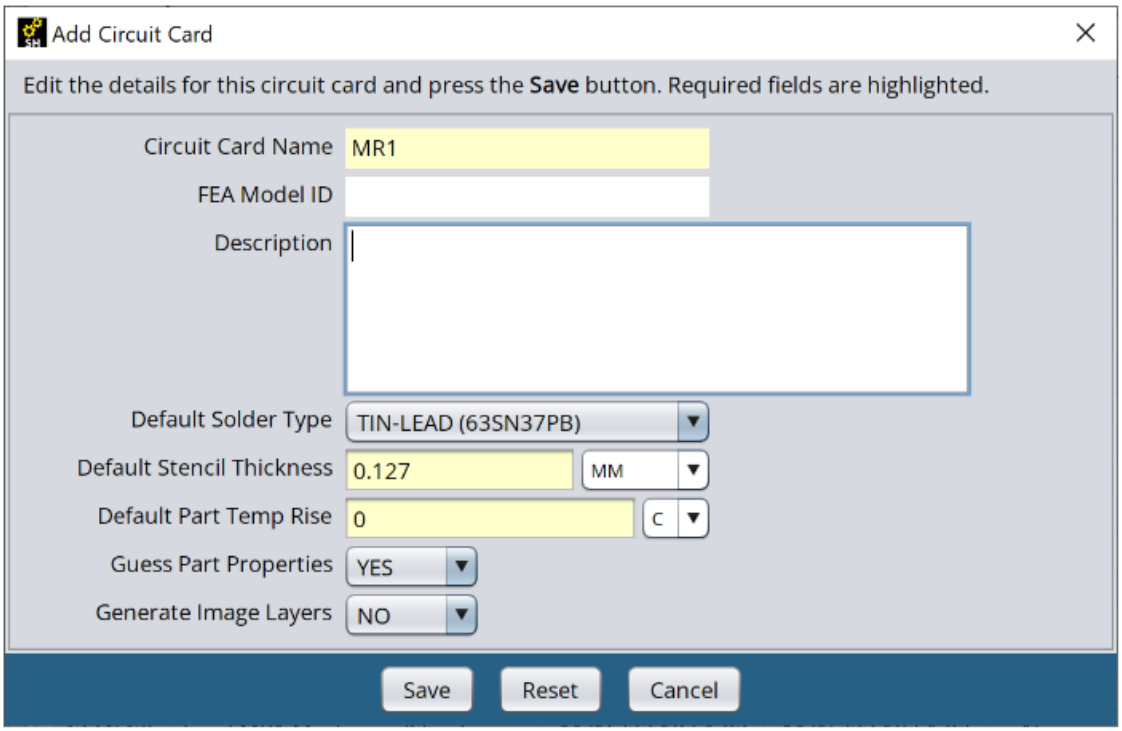


In the **Add Circuit Card** dialog (B, below) the **Circuit Card Name** field is prefilled with the Modeling Region ID. By default, the circuit card options, such as **Default Solder Type**, are set to the same values as the parent board. While most of the dialog options are self-explanatory, a couple reminders might be helpful:

- For **Default Part Temp Rise**, see *Analysis Options (p. 436)* in the *Thermal Derating* section.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When **Guess Part Properties** is ENABLED, Sherlock will add initial values for parts added to the CCA.
- **Generate Image Layers**: If you prefer to convert the copper layer files to image files, set **Generate Image Layers** to YES. Copper and laminate layers copied to the new CCA are converted to image layers regardless of the source of the original layer.

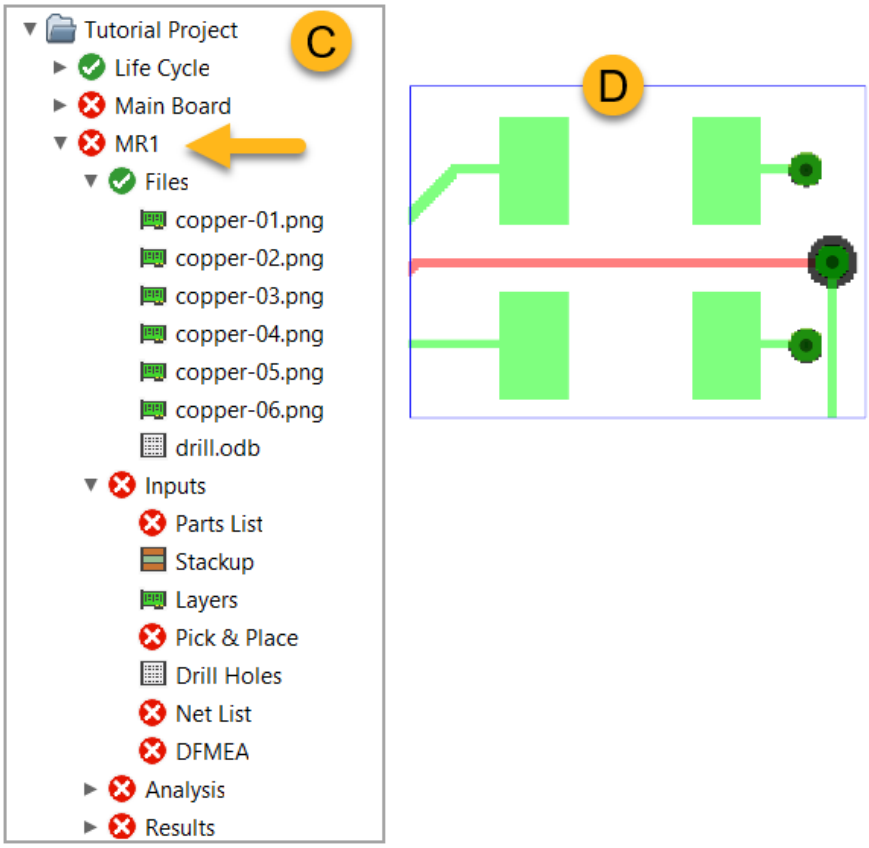


Click **Save** to create the new CCA. The new CCA will appear in the **Project Tree** (C, below) and will have the following characteristics:

- It will contain a board outline based on the selected modeling region. If needed, Sherlock will clip the new outline so it does not extend past the original board outline.
- It will have the same stackup as the parent board, with adjusted copper percentages.
- Modified copper, laminate, drill, and mount point layers will be clipped to the modeling region.
- The board location will be the same as the coordinates in the original model.
- Drill layers copied to the new CCA are converted to ODB++ Drill layers regardless of the source of the original layer.
- The board will have no components, but you can add them manually.

The new board created from the example above (image A) is shown below (D) as it would appear in the Layer Viewer.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.7. FEA- Part Modeling

For Finite Element Analysis (FEA) purposes, Ansys Sherlock can create two different 3D models to represent a given component on the PCB. A simple block model represents the component as a monolithic block of one or more elements that sit directly on the PCB. A more complex leaded model extends the block model, which represents the body of the component, by creating elements that represent each of the leads associated with the component. When the leaded model is used, the component body is *not* attached to the PCB. Rather, the body is attached to the leads, which are then attached to the PCB.

In this section, we discuss how the body of a component is edited and modeled in Sherlock. The **Lead Modeling** user guide (p. 563) section discusses how to model leads.

Note:

When reviewing part models, the following component-types may need extra attention:

- **Through Hole Parts**: Through-hole parts require special scrutiny in the **Parts List** because of the lack of standard package designations, their different shapes, and the different ways that CAD tools designate their location. Whenever you create a Sherlock project, you should always examine each of the through-hole parts.
- **BGA Components**: The following applies to ODB++, Eagle, and ODB XML imports. When importing a BGA package, Sherlock calculates the package dimensions from the ball pattern rather than using the dimensions specified for the package.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this section, the following topics are covered:

8.7.1. Layer Viewer Component Selection
8.7.2. Component Size, Location, and Orientation
8.7.3. Editing Part Properties
8.7.4. 2D Component Outline
8.7.5. 3D Block Model
8.7.6. Corner Orientation
8.7.7. Corner Face Restrictions
8.7.8. 3D Part Model Viewer
8.7.9. FEA Model Properties

8.7.1. Layer Viewer Component Selection

Part properties can be edited by selecting one or more parts in the **Parts List** (p. 183) and using the **Part Editor** (p. 200) to edit the various properties associated with those parts. In addition to those capabilities, users can also edit components directly from the **Layer Viewer** (p. 43), providing a visual, point-and-click alternative that is useful for many types of editing chores.

To make changes to one or more components, select **Edit > Edit Components** from the main menu in the **Layer Viewer**. At that point, the **Component Editor** mode will be indicated along the bottom of the **Layer Viewer** and five buttons will be displayed. As long as the **Component Editor** is active, you can select one or more components, edit them graphically or edit their part properties, and then update the **Parts List.**

In order to make changes in the **Component Editor** you must first select one or more parts to be changed. Part selection is done graphically using the following set of mouse/key operations:

- Shift-Left-click any part to select only that part
- Shift-Left-click and drag the mouse to select all parts within a rectangular region
- Control-Left-click any part to toggle selection of that part
- Control-Left-click and drag the mouse to toggle selection of all parts within a region
- Shift-Double-Left-click any part to select only that part and display part properties
- Right-click outside all selected parts to clear the current selections

Part selection takes a little practice, but you'll soon become comfortable with the selection process.

8.7.2. Component Size, Location, and Orientation

One of the most common part editing chores is that of adjusting the size, location and/or orientation of one or more parts. This is often necessary for small surface mount parts, because there is no standard regarding the orientation of certain packages, like an 0804. Some EDA tools assume the standard orientation is horizontal, while others assume it is vertical. You might also want to reduce or increase the size of a given part, move it slightly to eliminate overlaps with other parts, or visually align parts with underlying PCB features.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

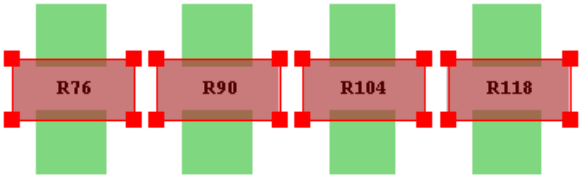
The location of one or more selected parts can be visually adjusted simply by left-clicking inside of any selected part and dragging the selected parts to the new location. The coordinates of all selected parts will be adjusted by the same amount. After moving one or more parts, you can press the **Apply** button to save the changes or the **Reset** button to return the parts to their previous locations.

The size of one or more selected parts can be visually adjusted by left-clicking inside of any red control points displayed at the corners of each selected part and dragging the mouse to change the size of the component. All selected components will be scaled by the same amount, with their center locations remaining fixed in place.

To rotate a single part, right-click anywhere inside of the part outline (which will automatically select that part) and select **Rotate 90** from the context menu.



To change the orientation of multiple parts, left-click and drag the mouse over the part outlines, while holding down the SHIFT key. After all desired parts are selected, right-click in any of the selected parts to display the **Edit Menu** and select **Rotate 90** to rotate all selected parts.



When you're done editing the parts, right-click anywhere in the layer viewer to clear the selections and then press the **Apply** button at the bottom of the **Layer Viewer** to update the **Parts List** and continue editing parts.

8.7.3. Editing Part Properties

Part properties can be edited directly from the **Layer Viewer** by right-clicking any component and selecting the **Edit Properties** option from the context menu to display the **Part Editor**. Unlike the **Parts List,** however, you can only edit a single part at a time in the **Layer Viewer.** Nonetheless, it is a useful feature to have when visually inspecting all the parts on a PCB. See The Part Editor (p. 200) for a detailed discussion of how part properties can be modified using the **Part Editor**.

8.7.4. 2D Component Outline

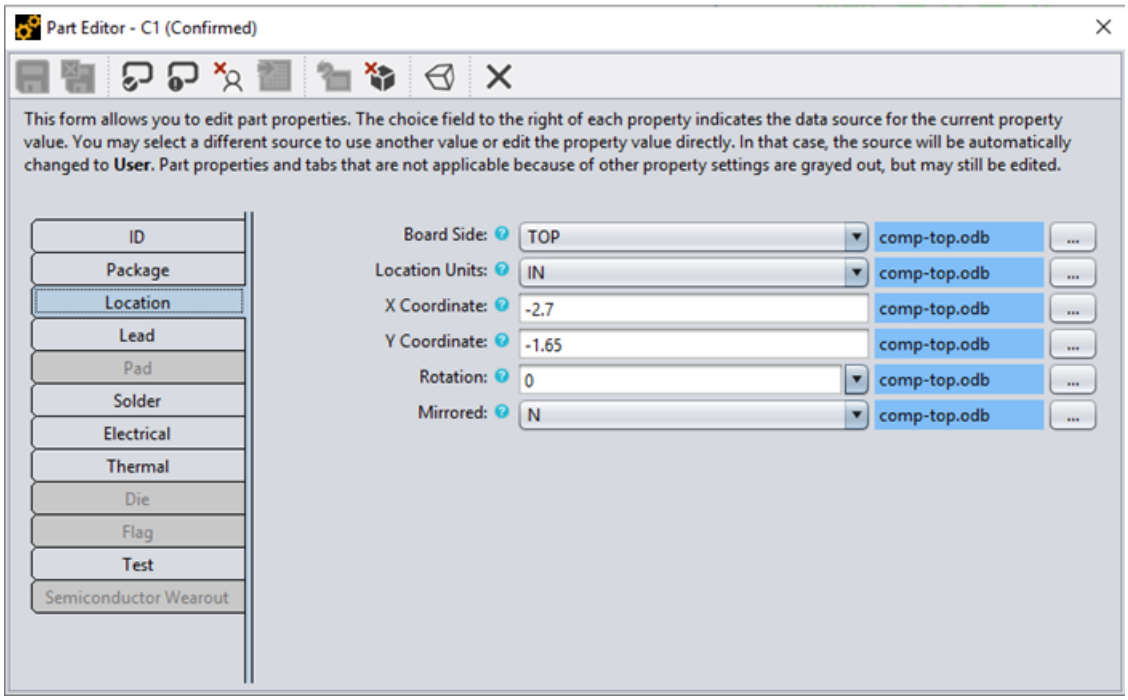
Components are displayed in the **Layer Viewer** using a simple 2D outline to show their size, location and orientation, while still allowing users to see underlying PCB details, such as pads, traces, drill holes, etc. The location and orientation of the component is specified by the following properties found in the **Location** tab of the **Part Editor:**

- Board Side
- Location Units

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- X Coordinate
- Y Coordinate
- Rotation
- Mirrored

The X,Y coordinates denote the center of the component and the center of rotation. The **Mirrored** property indicates if the component shape is to be mirrored across the Y-Axis.

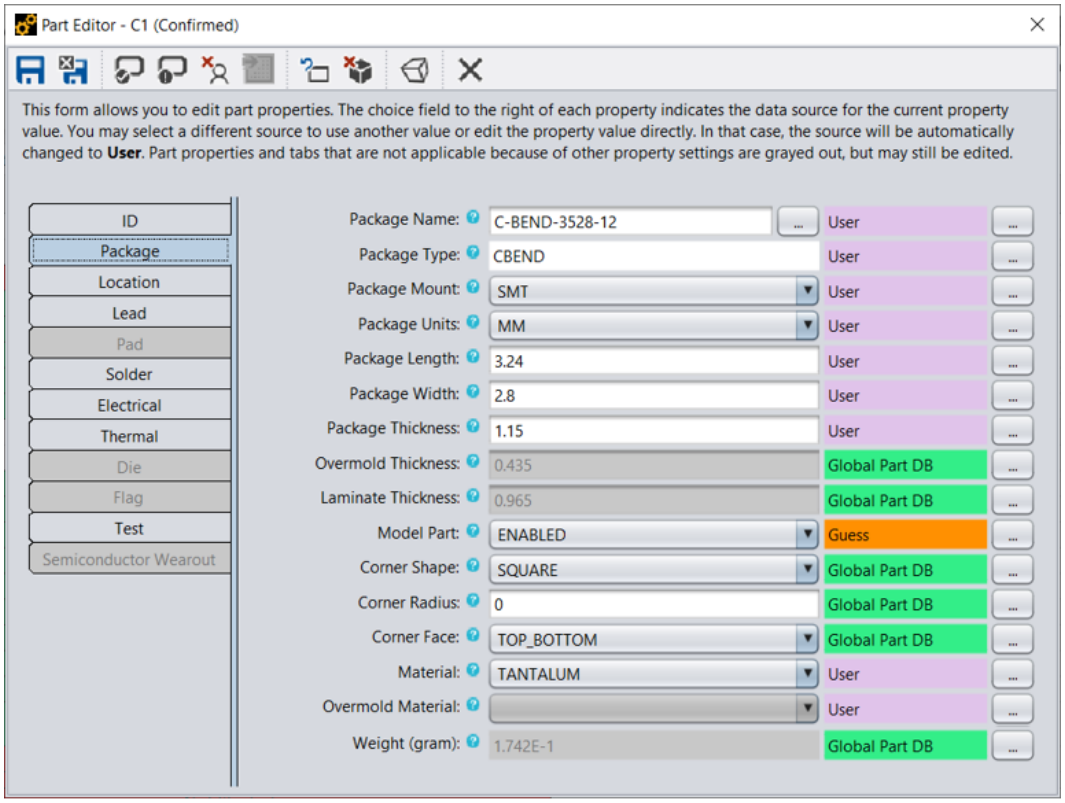


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The shape of the 2D outline is specified by the following part properties found in the **Package** tab of the Part Editor:

- Package Units
- Package Length
- Package Width
- Corner Shape
- Corner Radius

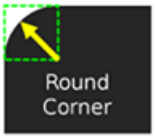
In canonical orientation (Rotation = 0), the Package Length is the dimension along the X-axis and the Package Width is the dimension along the Y-axis.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Corner Shape** and **Corner Radius** properties define how the component corners should be represented, with the following corner shapes supported:

Table 8.14: Corner Shapes

Square	Corner displayed as a 90 degree angle. Corner Radius is ignored.	
Miter	Corner represented by a single diagonal segment. Corner Radius defines the distance from each end of the diagonal segment to the location where the Square corner would have been located.	
Round	Corner represented by multiple arc segments. Corner Radius defines the arc radius.	

For the Miter and Round corner shapes, if the **Corner Radius** is greater than 1/2 the **Package Length** or **Package Width** (whichever is shortest) Sherlock will automatically set it to the maximum value suitable for the package. A circular component outline (for example, can capacitor) can be defined by setting the following properties, where D is the diameter of the component:

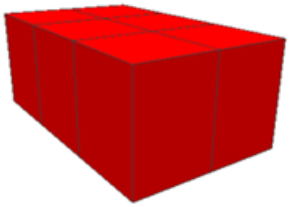
Table 8.15: Circular Component Properties

Property	Value
Package Length	D
Package Width	D
Corner Shape	Round
Corner Radius	D

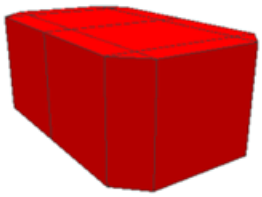
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.7.5. 3D Block Model

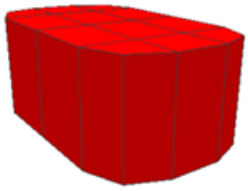
For FEA purposes, a 3D block model is created for a component by generating a mesh for the 2D component outline as described above and then extruding the 2D mesh elements in the Z-dimension based on the **Package Thickness** property defined in the **Package** tab of the **Part Editor**. This process yields one or more wedge and/or quad elements that are bound together by shared nodes. The exact number and shape of the elements depends on the shape of the 2D outline, as well as the **Max Mesh Size** and **Vertical Mesh Size** properties used when generating the 3D model. For example, the following images show the 3D models generated for a component that is 10mm long, 6 mm wide and 4 mm tall, using a 4 mm **Max Mesh Size** and 4 mm **Vertical Mesh Size.**



Square Corner (4mm mesh)



Miter Corner (4mm mesh)



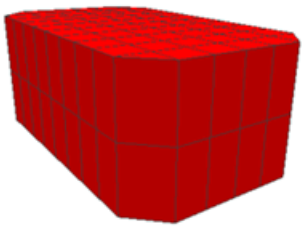
Round Corner (4mm mesh)

In this case, all the elements are 4mm tall. For the miter corner, a 1mm wedge is created to fill the corner, while a non-regular brick is created to fill the round corner.

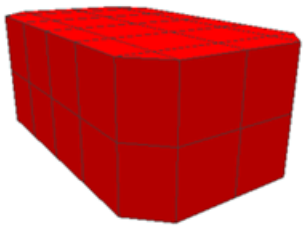
Note:

The simple block shape and corner types supported by Sherlock allow it to generate 3D component models that are formed from mostly regular bricks, with a minimal number of wedges.

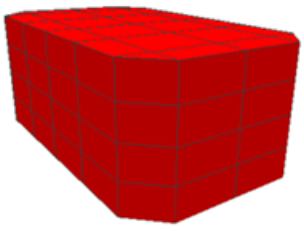
The **Max Mesh Size** and **Vertical Mesh Size** properties specified when generating the model (described below) also affect the number of elements created. For example, the following images show the same part with mitered corners using different mesh size properties as indicated.



Max Mesh Size = 1mm
Vertical Mesh Size = 2mm



Max Mesh Size = 2mm
Vertical Mesh Size = 2mm

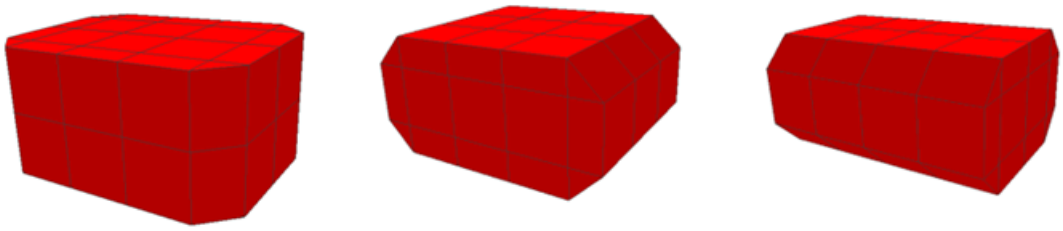


Max Mesh Size = 2mm
Vertical Mesh Size = 1mm

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

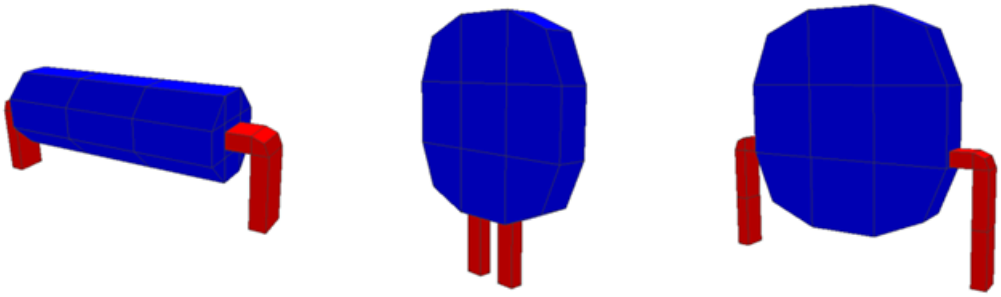
8.7.6. Corner Orientation

The package corners shown in the previous section were all defined in the XY plane and then extruded into 3D shapes along the Z dimension. Such an approach supports a large percentage of SMT and Through Hole parts used today. However, certain package types, such as resistors with axial leads or disc capacitors, are best modeled by a rounded or mitered outline in the XZ or YZ planes that is then extruded across the remaining dimension. Starting in Version 5.1, Sherlock supports such package modeling with the **Corner Face** part/package property. The Corner Face property supports the following orientations:



Corner Face = TOP_BOTTOM **Corner Face = FRONT_BACK** **Corner Face = LEFT_RIGHT**

In all the examples above, the **Package Length**, **Package Width** and **Package Thickness** values are the same, as well as the **Corner Shape** and **Corner Radius** values. The only difference between the examples is the **Corner Face** property. When combined with **Lead Modeling**, the **Corner Face** property allows for a wide range of package types to be modeled, such as the following examples:



Corner Face	LEFT_RIGHT	**Corner Face**	FRONT_BACK	**Corner Face**	LEFT_RIGHT
Corner Shape	MITER	**Corner Shape**	ROUND	**Corner Shape**	ROUND
Lead Layout	AXIAL	**Lead Layout**	SIP	**Lead Layout**	DIP
Lead Bend	90	**Lead Bend**	0	**Lead Bend**	90

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.7.7. Corner Face Restrictions

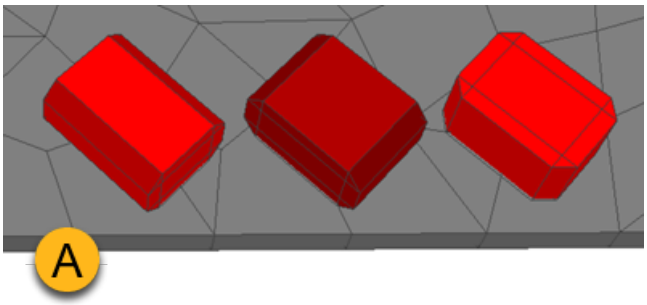
Topics:

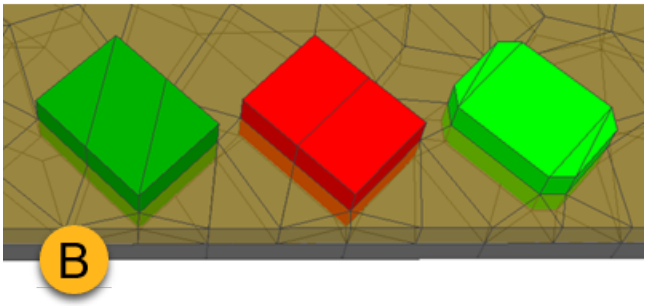
8.7.7.1. Merged PCB Models and Potting Regions
8.7.7.2. Round Part Corners and PCB Contact

8.7.7.1. Merged PCB Models and Potting Regions

There are a couple restrictions regarding how the **Corner Face** property is used when modeling parts in a **Merged** PCB model or when modeling parts within a potting region. In both cases, if the **Corner Face** property is NOT **TOP_BOTTOM** and leads are not being modeled, then the **Corner Face** and **Corner Shape** properties will be ignored, and the part will be modeled as a SQUARE shape. This is necessary when generating a **Merged** PCB model because the PCB and parts are meshed at the same time in the XY plane and extruded in the Z dimension. Similarly, when parts are contained by a potting region, the potting region and parts are meshed at the same time in the XY plane and extruded in the Z dimension.

For example, the image below (A) shows a group of parts with different **Corner Face** values modeled on a **Bonded** PCB model. Image B shows the same parts modeled inside of a potting region. Note how the LEFT_RIGHT part (left-most) and FRONT_BACK part (center) are modeled using SQUARE sides because of the potting region.





See the PCB Modeling (p. 473) and Potting & Staking (p. 616) user guide sections for more details concerning those modeling features.

8.7.7.2. Round Part Corners and PCB Contact

In previous versions of the Sherlock application, a part with a round corner shape and a left/right or front/back circular corner face sometimes had issues during the FEA analysis: Because there was no complete element face parallel to the PCB surface, the part might not remain in contact with

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the PCB surface. Beginning with release 2025 R1, the Sherlock application now creates a connection between the PCB surface and the part nodes in contact with the PCB surface, even when no face is parallel to the PCB surface.

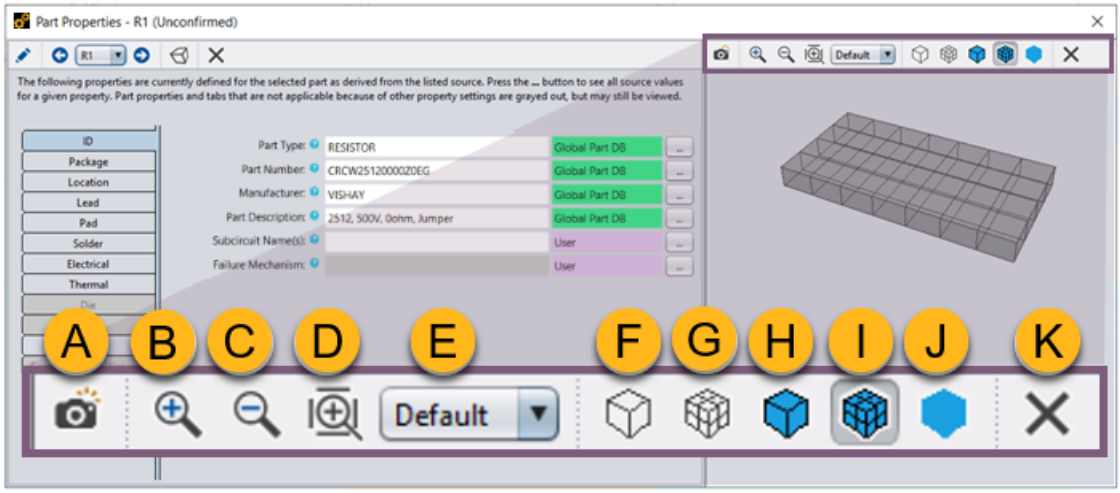
For circular parts, the Sherlock application rotates the part so that the faces of the elements closest to the PCB are in contact with the PCB surface. With or without the rotation, if there is a round corner with a front/back or left/right corner side, then there will be at least nodal connections between the PCB surface and part nodes on the PCB surface. However, these connections might not be fully constrained, possibly resulting in an FEA error.

8.7.8. 3D Part Model Viewer

The 3D block model generated by Sherlock for a given component can be previewed by pressing the **View the 3D Model of the Part** button provided in the **Part Editor** ribbon. On the right of the part properties, a 3D model of the part will appear.

 View the 3D Model of the part

The **Part Viewer** provides the same 3D viewing features as the **Sherlock 3D Viewer**, allowing you to pan, zoom and rotate the model. You can also define a custom viewing angle that is used by default whenever the viewer is launched and adjust the shading and outline properties of the 3D Model.



3D Viewer Controls

A. Export an image or a 3D PDF of the model

B. Zoom in 3D model

C. Zoom out 3D model

D. Zoom fit 3D model

E. Orientation of the model. Includes: Default (isometric), Top, Bottom, Front, Back, Left, Right, Custom

F. View model outline

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

G. View model with mesh outline

H. View model with solid body and outline

I. View model with solid body and mesh outline

J. View model as solid body and no outline

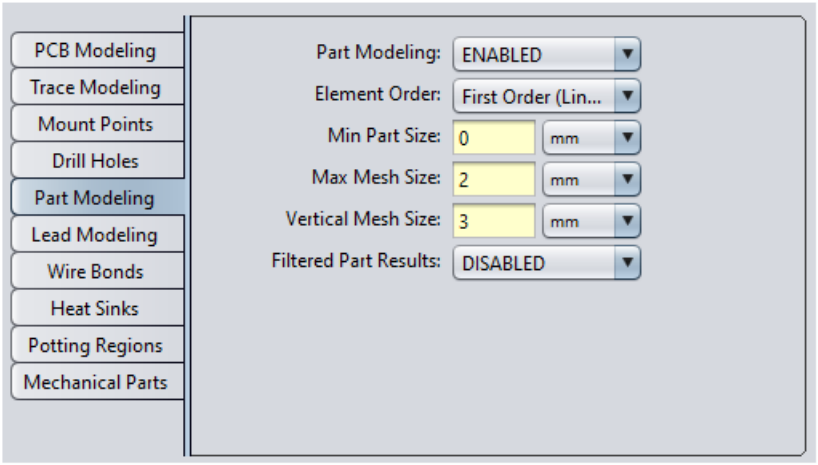
K. Close the 3D model

8.7.9. FEA Model Properties

Component models can be included in any FEA model generated by using any of the following analysis menu options:

- **Generate 3D Model**
- **Export FEA Model**
- **Edit Analysis Properties**

In each of these cases, the **Part Modeling** tab defines the properties used to generate 3D component models.



If the **Part Modeling** property is **DISABLED**, then no component models will be included in the FEA model. This provides a convenient way to test a PCB by itself, without modifying the Parts List.

The **Element Order** property specifies the type of FEA element Sherlock uses when modelling components. **First Order** elements are 3D elements (eight-node hex elements and six-node prism elements) that provide a faster analysis. **Second Order** elements are 3D elements (20-node hex and 15-Node prism) that use midpoint nodes to improve result accuracy but require a longer analysis time. **First Order** elements, also called linear elements, are stiffer compared to **Second Order**, also called quadratic elements.

The **Min Part Size** property is used to filter out small components. When that property is non-zero, any component which has a length and width less than that value will be ignored when creating part models.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Max Mesh Size** property specifies the approximate size of the mesh elements generated by Sherlock or the FEA tool (when using the **Export FEA Model** option). Mesh elements will usually have dimensions that are no greater than the value specified.

The **Vertical Mesh Size** property specifies the maximum height of the mesh elements generated by Sherlock or the FEA tool (when using the **Export FEA Model** option). If a value of 0 is specified, no limitation is used, and each element will be as tall as the part itself.

When ENABLED, the **Filtered Part Results** property will include predictions for parts that are not modeled. The predictions are based on data related to the part's location.

Note:

Part Modeling property values are automatically saved by Sherlock each time an FEA model is generated for a given analysis module and used as the default property values the next time the **Part Modeling** tab is displayed for that analysis module.

8.8. FEA- Lead Modeling

As explained in Part Modeling, (p. 552) you can choose to include 3D component models when generating an FEA model. By default, Ansys Sherlock creates a simple block model of each component, based on the package length, width, and thickness. For surface mount packages, such block models are usually enough for reliability analysis because the analysis models used by Sherlock already take the package type into account. For example, Sherlock considers the differences between a BGA and a QFN package when predicting failure rates based on the maximum strain on the PCB surface experienced by such parts. There are situations, however, when you might want to also model the leads of a surface mount device to better understand the stresses on those leads or how they affect the overall mechanical properties of the PCB. Similarly, for through-hole parts, analysis results can be improved in some situations by modeling the individual leads. For that reason, Sherlock allows you to selectively enable/disable lead modeling for individual parts as needed. This section describes how lead models are defined using various lead properties and the modeling properties that can be used to control the modeling process.

Note:

Lead modeling may significantly increase the amount of time required to perform analysis. If you wish to model component leads in your analysis, it is currently recommended you export the project file to Ansys Workbench. Generate the FEA model, run the analysis in Workbench and import the results back into Sherlock. See FEA- Ansys Workbench™ Integration (p. 747) for details.

In this section, the following topics are covered:

8.8.1. Lead Properties

8.8.2. Lead Modeling Analysis Properties

8.8.3. Advanced Lead Meshing

8.8.4. BGA Solder Ball Properties

8.8.5. Surface Mount Lead Modeling

8.8.6. Through-Hole Lead Modeling

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.7. Solder Ball Modeling
8.8.8. 2D Lead Viewing

8.8.1. Lead Properties

Sherlock uses a collection of properties specified in the **Lead** tab of the **Part / Package Editors** to construct the lead models for a given part or package. The most important of these are the **Lead Count** and **Lead Geometry** properties, that specify the total number of leads attached to the part (not counting any sink) and the shape of those leads. All other lead properties are used to determine the exact shape and placement of the leads, depending on the **Lead Count** and **Lead Geometry** property values.

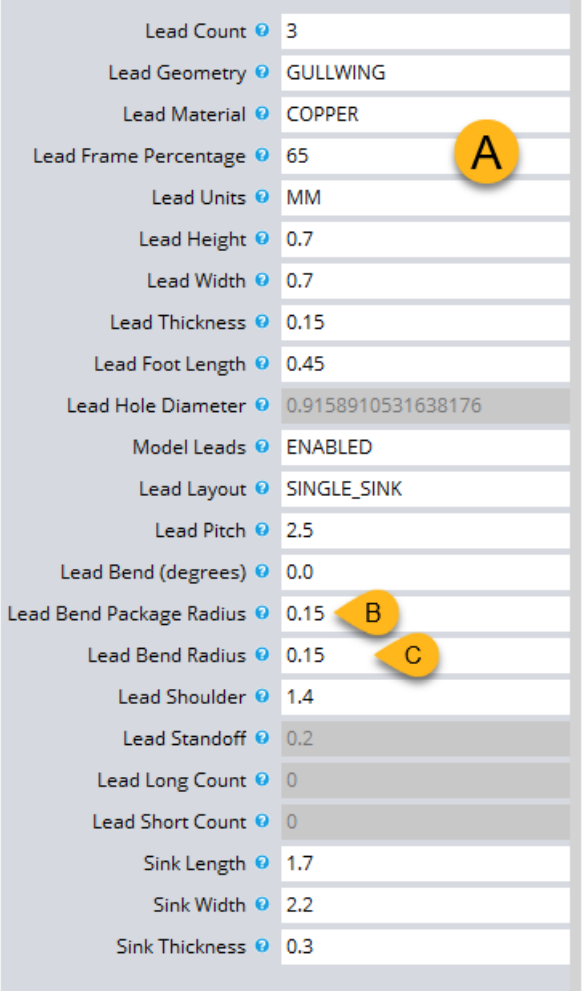
Lead modeling can be enabled/disabled for an individual part using the **Lead Modeling** property. This allows users to construct detailed models for only specific parts to reduce analysis time.

Note:

Some lead properties require additional explanation:

- The **Lead Frame Percentage** property (A, in the image below) does not affect FE analyses. It is used when exporting a project to Ansys Electronics Desktop. See Icepak Thermal Map (TMAP) (p. 109).
- Starting with Sherlock version 2024 R2, two new properties have been added to the **Lead** tab in the **Part Properties Window** to support the Advanced Lead Meshing feature: **Lead Bend Package Radius** B, below) and **Lead Bend Radius** (C). These two properties appear in the **Part Properties** window only. They are not tracked in the Part Library nor the Package Library. See Advanced Lead Meshing (p. 566) below.

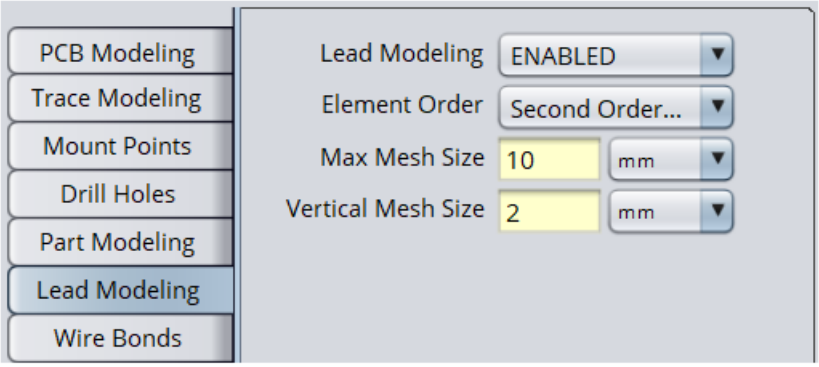
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.8.2. Lead Modeling Analysis Properties

When setting up your FE analysis, click the **Lead Modeling** tab in the **Analysis Properties** window to specify how you want Sherlock to handle lead modeling in the analysis. The image below shows the settings for default lead modeling. For Advanced Lead Meshing (p. 566), see the section which follows.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tip:

If the options on the **Lead Modeling** tab are grayed out, click the **Part Modeling** tab and verify that the **Part Modeling** option is set to ENABLED.

Lead Modeling Analysis Properties:

- **Lead Modeling**: When ENABLED, Sherlock models the leads for all components. Lead modeling may significantly increase the amount of time required to perform the analysis.

Note:

Remember, each component on your circuit board has its own property called **Model Part**. You can find it in the **Package** tab in the **Part Properties Viewer/Editor** (p. 197). If the **Model Part** property has been disabled for an individual part, Sherlock will never model that component when generating an FEA model or running an FE analysis—even if **Lead Modeling** or **Part Modeling** is enabled in the **FE analysis Properties** window.

- **Element Order**: This property specifies the type of FEA element to be used when modelling leads. **First Order** elements are 3D elements (eight-Node hex elements) that provide a faster analysis. **Second Order** elements are 3D elements (20-node hex) that use mid-point nodes to improve result accuracy at the cost of requiring longer analysis time. **First Order** elements, also called linear elements, are stiffer compared to **Second Order**, also called quadratic elements.
- **Max Mesh Size**: This property specifies the desired size of the elements to be generated. The smaller the Max Mesh Size, the more accurate the results, but the greater the amount of memory and CPU processing required during the processing. In general, you should use a large mesh size initially to generate approximate results using a relatively small amount of memory and CPU resources. When you need more accurate results, you can then decrease the mesh size and re-run the analysis.
- **Vertical Mesh Size**: This property specifies the desired height of the elements used to model the lead.

8.8.3. Advanced Lead Meshing

Starting with version 24.2, Sherlock offers the Advanced Lead Meshing (ALM) option which you can enable in the **Meshing** tab of the **Sherlock Settings Window** (p. 85). Advanced Lead Meshing allows

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you to specify lead meshing parameters in greater detail rather than being limited to a fixed mesh size. This section provides first a basic overview of ALM and is followed by more detailed explanations to help you understand how the ALM settings affect your analysis.

8.8.3.1. Overview of Advanced Lead Meshing

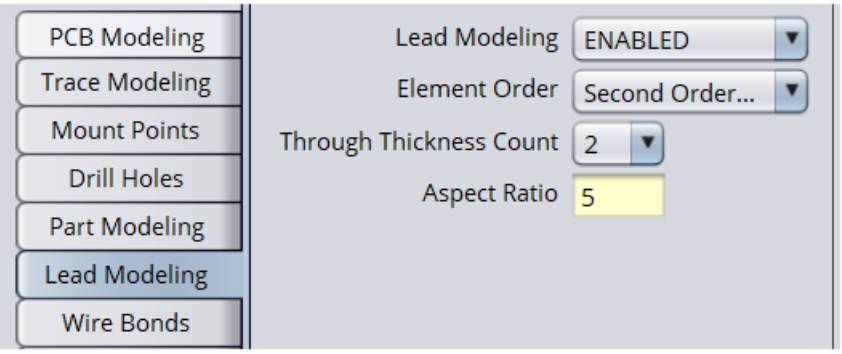
8.8.3.2. Through Thickness and Aspect Ratio

8.8.3.3. Lead Bend Radius and Lead Bend Package Radius

8.8.3.4. Frequently Asked Questions About Advanced Lead Meshing (ALM)

8.8.3.1. Overview of Advanced Lead Meshing

When Advanced Lead Meshing (ALM) is enabled, the **FE Analysis Lead Modeling** form presents the options shown in the image below. Until you run your first analysis with ALM, the settings default to what you selected in the **Sherlock Settings Window** (p. 85).



Advanced Lead Meshing Settings:

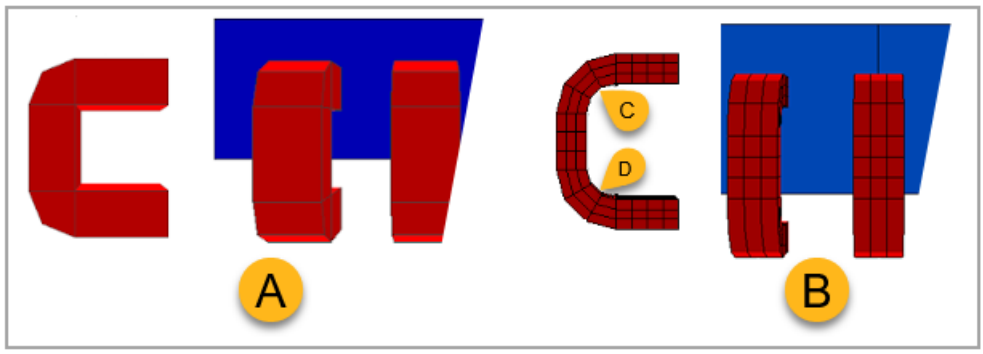
- For **Lead Modeling** and **Element Order**, see the section above Lead Modeling Analysis Properties (p. 565).
- **Through Thickness Count** specifies the number of elements through the thickness and width of the lead. The default value is three, and the maximum is five. Notice in the image below (B) how the width and thickness of the lead are comprised of multiple elements. See Through Thickness and Aspect Ratio (p. 568) below for additional information.

> **Tip:**
>
> Ansys recommends setting **Through Thickness Count** to three when **Element Order** is set to **First Order**.

- **Lead Aspect Ratio** determines the ratio between the elements' thickness and height. The default value is two and the maximum is ten. See Through Thickness and Aspect Ratio (p. 568) below for additional information.

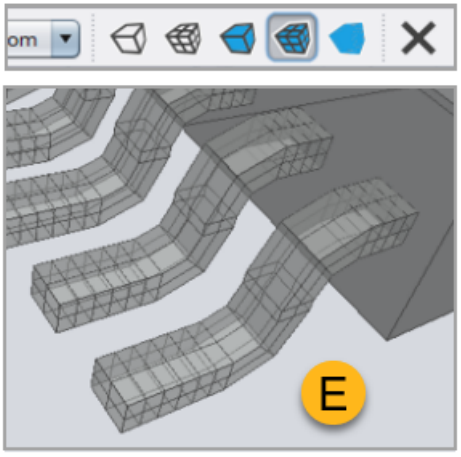
Advanced Lead Meshing also changes how lead bends are meshed. In place of a single element, Sherlock meshes the lead bend based on the **Through-Thickness Count** while the bend radius is defined by the following Part Properties: **Lead Bend Radius** and **Lead Bend Package Radius**. These properties are only used when the Advanced Lead Meshing is enabled. While they appear in the Part Properties List, they are *not* tracked in Sherlock's Part Library.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The image above illustrates the differences between Sherlock's default lead meshing (model A) and Advanced Lead Meshing (model B). Advanced Lead Meshing (ALM) allows you to define lead thickness and width with multiple elements. Also, ALM uses the **Lead Bend Package Radius** (C) and the **Lead Bend Radius** (D) properties. The **Lead Bend Package Radius** (C) is the bend closest to the package. The **Lead Bend Radius** (D) is the bend closest to the PCB surface. If the lead has only one bend, only the **Lead Bend Radius** is used. See Lead Bend Radius and Lead Bend Package Radius (p. 569) below for additional information.

Once enabled, Sherlock applies the ALM settings to the model when generating the 3D FEA model, when running an FE analysis, or when generating a 3D model of a component in the **Part Viewer/Editor** as shown below (E).



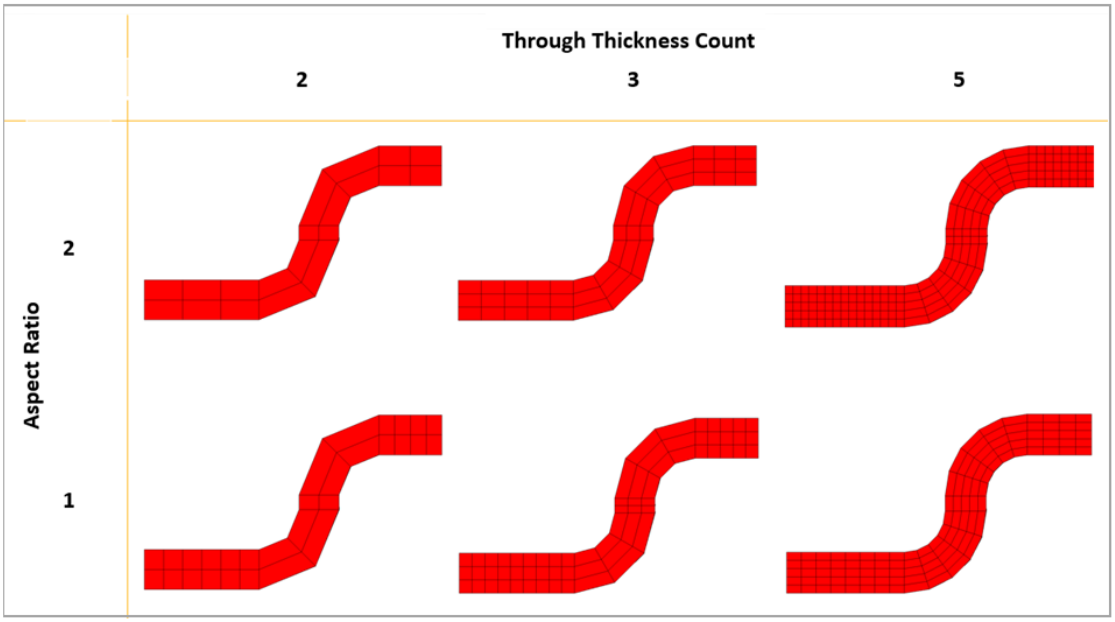
Note:

Using Advanced Lead Meshing increases the complexity of the FEA models and lengthens processing time when Sherlock runs an analysis.

8.8.3.2. Through Thickness and Aspect Ratio

As a reference, the image below illustrates how **Through Thickness Count** and **Aspect Ratio** affect the lead model. Computer processing time increases as the number of elements through the thickness increases and/or as Aspect Ratio decreases—all which increase the number of elements and nodes. Ansys recommends using three elements through the thickness when meshing with First Order elements. Note as well that the number of elements along the bend is the same as the number of elements through the thickness.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.8.3.3. Lead Bend Radius and Lead Bend Package Radius

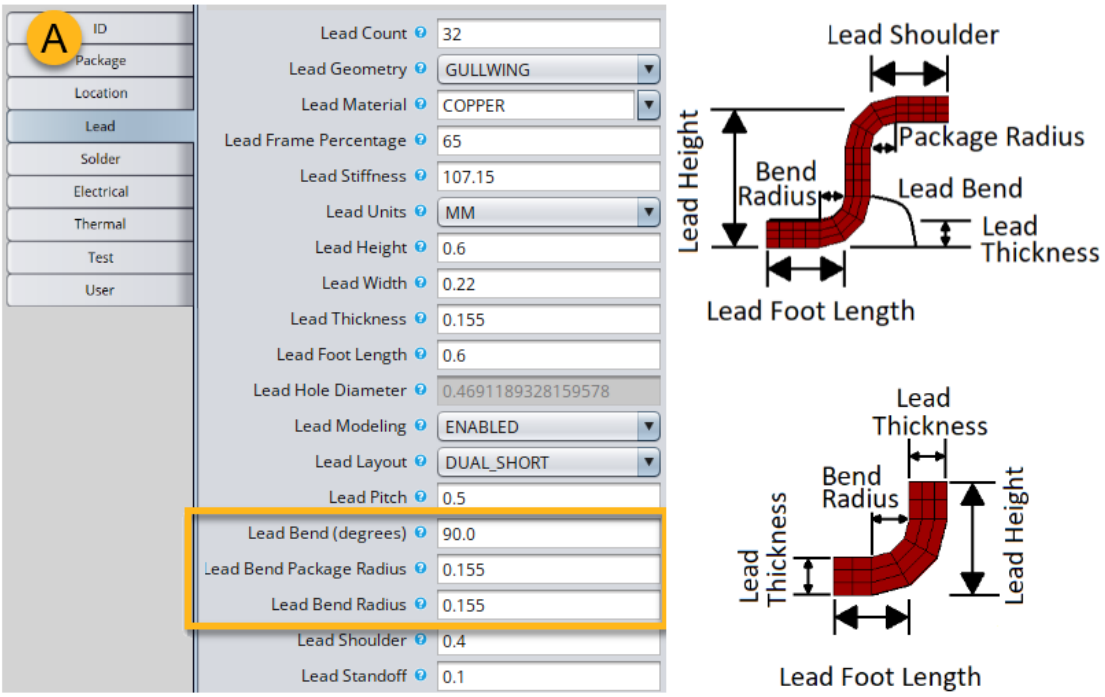
Starting with release 2024 R2, part properties which track lead bend radii have been added to Sherlock to support Advanced Lead Meshing (ALM). These properties appear in the **Part Properties Viewer/Editor** as shown in the image (A) below and apply only when the ALM feature is enabled.

The radii values default to **Lead Thickness**, but you may manually edit the fields if desired. Package manufacturers rarely provide this information, requiring you to estimate the values. When estimating, keep in mind the following:

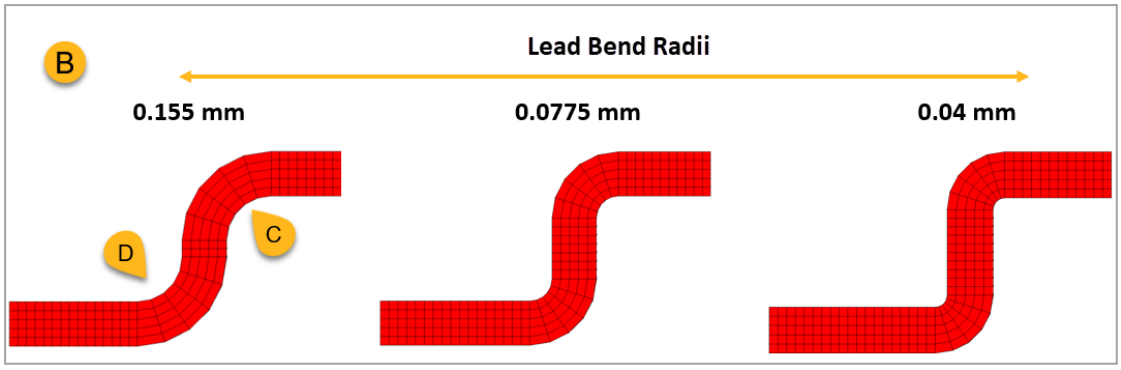
- A radius that is too small can cause stress concentration regions in the leads, leading to failure.
- Sherlock sets the following limitations on bend radii: The bend must be no smaller than 10% of **Lead Thickness**. It must be no larger than 50% of the *smallest* of **Lead Shoulder** and **Lead Height**.

  In the **Lead** properties tab of a Gullwing component shown below (A), the **Lead Thickness** is 0.155 mm, so the bend radii cannot be smaller than 10% of that value—or 0.0155 mm. The bend can be no larger than 50% of the smallest of **Lead Height** (0.6 mm) and **Lead Shoulder** (0.4 mm). Since **Lead Shoulder** is smallest (0.4 mm), the bend radius can be no larger than half that value—or 0.2 mm.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

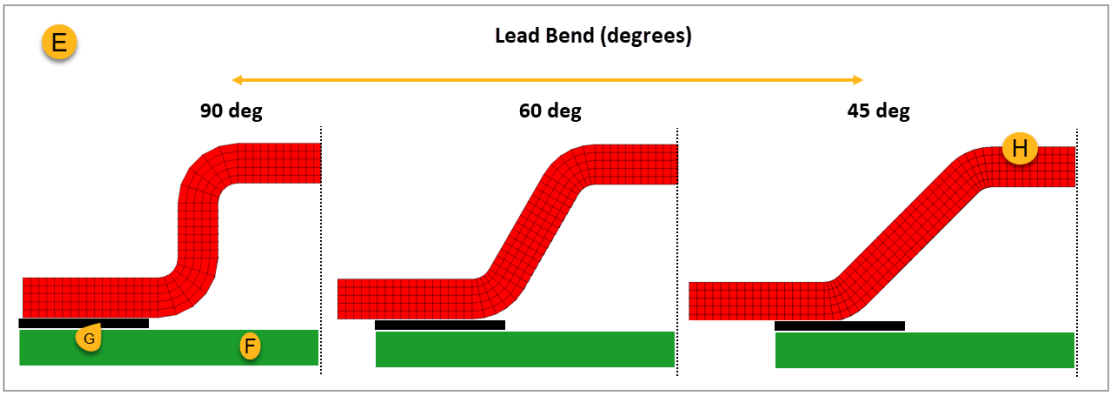


Gullwing leads with different bend radii are shown below (B). When a lead has two bends as the Gullwing does, the bend closest to the package is the **Lead Bend Package Radius** (C), and the one closest to the PCB is the **Lead Bend Radius** (D). If the lead has only one bend, it is termed the **Lead Bend Radius**. The radius specified in the **Part Editor** is the inner radius of the lead. The outer radius is calculated based on the **Lead Thickness**.



Version 2024 R2 of Sherlock introduces a new property to the Gullwing lead geometry: **Lead Bend (degrees)** as shown in image A above. Previously, this option was active only for Thru Hole geometries. Now this property is active for Gullwings whenever the Advanced Mesh Settings is ENABLED. The parameter defines the lead angle with respect to the PCB surface. The default for all Gullwing geometry is 90 degrees and the minimum angle you can define is 45 degrees. Lead bend variation is shown in the image below (E).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

As seen in the image above (E), when the lead bend changes, the distance between the lead footprint and package changes. (G represents the copper pad. F represents the PCB.) Sherlock does not compensate to account for the additional distance. If you change the **Lead Bend** from the default, the lead foot will move outside the copper pad. You must adjust the **Shoulder Length** (H) as needed to realign the lead foot with the pad.

8.8.3.4. Frequently Asked Questions About Advanced Lead Meshing (ALM)

1. **After I enable ALM, can I switch back to simple lead meshing?**

   Yes. Set the ALM to DISABLED, and Sherlock returns to the simple lead meshing.

2. **What happens when I use Sherlock version 2024 R2 or later to open projects created in earlier versions that do not have the ALM feature?**

   The first time you open a project created in an earlier version of Sherlock, Sherlock sets ALM to DISABLED. Once you set the option to ENABLED, Sherlock sets **Lead Bend (degrees)** to 90 and sets **Lead Bend Radii** equal to **Lead Thickness**.

3. **How does enabling ALM affect my package reliability?**

   Sherlock generates lead failure prediction for its vibration modules (Harmonic and Random Vibration Analyses), so the setting will have a significant impact in those two modules. The values you enter for the **Package Bend Radius** and the angle will affect reliability.

   In earlier versions of Sherlock, the leads were modelled with single element through thickness and suffered from sharp corners for lead bends. ALM provides fillets for relaxing the stress concentration region. Also, having multiple elements through the thickness makes lead compliance more realistic.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.4. BGA Solder Ball Properties

Sherlock uses a collection of properties specified in the **Ball** tab of the **Part / Package Editors** to construct the ball models for a given BGA part or package. The most important of these is the **Ball Modeling** property that allows users to enable/disable solder ball modeling for a specific part. When enabled, the exact size, composition, and layout of the solder balls in the 3D model is determined by the other solder ball properties as defined in the Solder Ball Modeling (p. 590) section.



Note:

The **Lead Modeling** property found in the **Lead Modeling** panel in the analysis input dialog is used to enable/disable lead and solder ball modeling for the entire circuit card during an analysis run.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.5. Surface Mount Lead Modeling

Sherlock can generate models for the following types of SMT leads:

Table 8.16: SMT Lead Types



C_Lead	J_Lead	Gullwing	Stub	L_Lead

Note: The models are shown with Sherlock's default lead meshing. See the sections below for examples where Advanced Lead Meshing is enabled.

For each model, the exact shape and placement of each lead depends on various lead properties, as explained in the sections below.

8.8.5.1. C-Lead / J-Lead Model

8.8.5.2. Gullwing Model

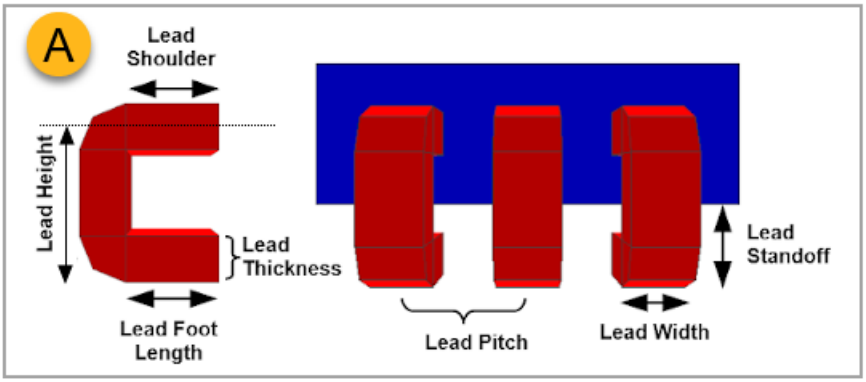
8.8.5.3. Stub Model

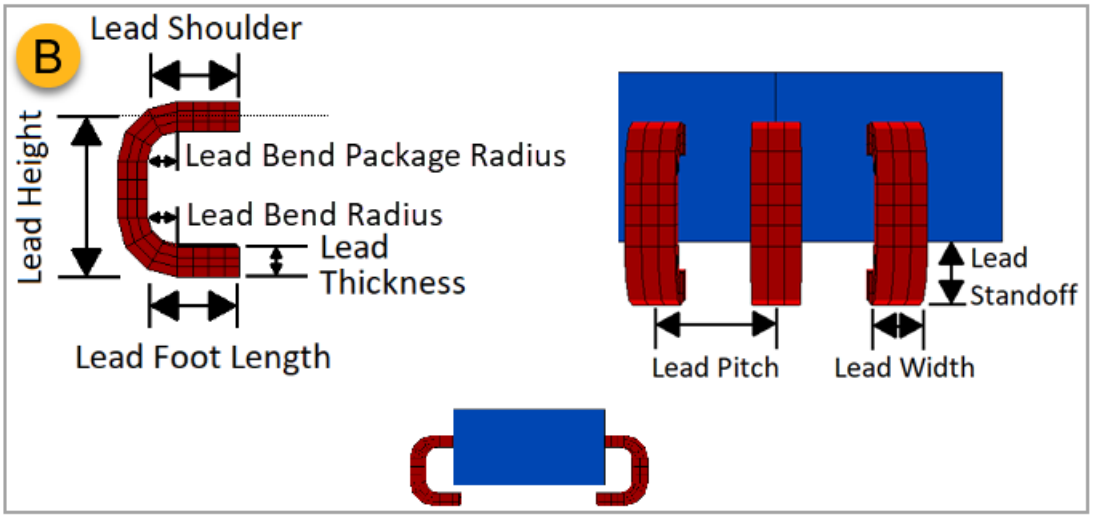
8.8.5.4. L Lead Model

8.8.5.5. Lead Placement

8.8.5.1. C-Lead / J-Lead Model

Sherlock models C- and J-Leads using the same basic elements (for example, shoulder, leg, and foot) in the same relative arrangement, with the foot tucked back under the part. In practice, C-Leads usually have equal shoulder and foot lengths, whereas the foot is usually longer than the shoulder in J-Leads. In the real world, J-Leads also have a toe that curls toward the bottom of the part, but for simplicity and ease of modeling, Sherlock ignores the toe when modeling the J-Lead. The following diagrams depicts how each of the lead properties are used by Sherlock to construct a C-Lead or J-Lead model. Model A below represents Sherlock's default lead modeling. Model B shows Advanced Lead Meshing (p. 566) (ALM) enabled.





When ALM is enabled (Model B above), you must specify the **Lead Bend Radius** and **Lead Bend Package Radius** properties. See Lead Bend Radius and Lead Bend Package Radius (p. 569) above for a detailed discussion of these properties..

Note:

With ALM enabled, the vertical leg may be eliminated when the **Lead Height** is equal to the sum of the **Lead Thickness**, the **Lead Bend Radius**, the **Lead Package Radius**, and half the **Lead Thickness**. Likewise, the horizontal portion of the shoulder may be eliminated if the **Lead Shoulder** is equal to the **Lead Package Radius**.

When incorporated into the FEA model, each C or J-Lead is bonded to the part using the side shoulder face and bonded to the PCB using the bottom foot face.

Lead Height for Gull Wing leads is measured from the foot of the lead to the center of the lead thickness at the shoulder.

8.8.5.2. Gullwing Model

The Gullwing model generated by Sherlock is similar to the C-Lead and J-Lead models except that the foot points away from the part body. In the image below, model A shows Sherlock's default lead modeling. Model B shows Advanced Lead Meshing (p. 566) (ALM). ALM enables the **Lead Bend Radius** and the **Lead Bend Package Radius** properties. The **Lead Bend Package Radius** defines the inside radius of the bend between the leg and the shoulder of the lead. The **Lead Bend Radius** defines the inside radius of the bend between the leg and the foot of the lead.

Note:

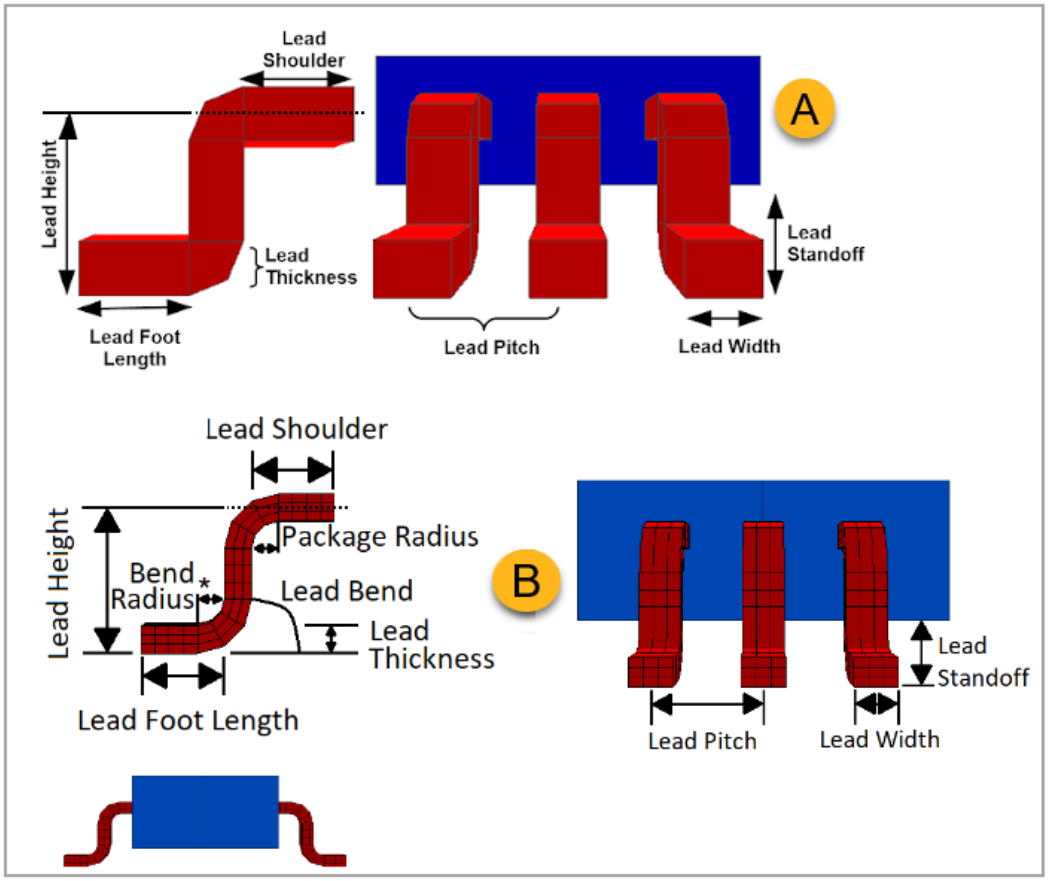
When ALM is enabled, the vertical leg may be eliminated when the **Lead Height** is equal to the sum of the **Lead Thickness**, the **Lead Bend Radius**, the **Lead Package Radius**, and half the **Lead Thickness**. Likewise, the horizontal portion of the shoulder may be eliminated if the **Lead Shoulder** is equal to the **Lead Package Radius**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Gullwing model, with ALM enabled, allows you to specify the **Lead Bend (degrees)**. This indicates the angle of the Gullwing leg, relative to the PCB surface. The default is 90 degrees, meaning the lead leg is entirely vertical.

Tip:

Gullwing leads require special care when setting **Lead Bend (degrees)**. If you have enabled Advanced Lead Meshing, refer to Lead Bend Radius and Lead Bend Package Radius (p. 569) above for a detailed discussion on Gullwings.

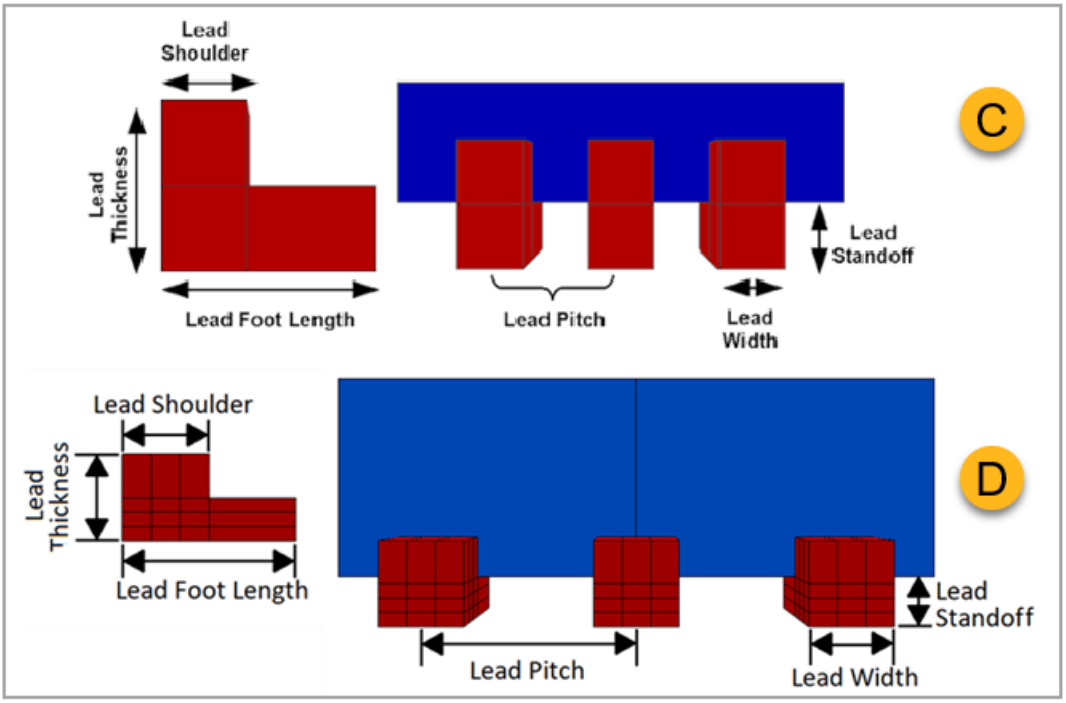


When incorporated into the FEA model, each Gullwing lead is bonded to the part using the side shoulder face and bonded to the PCB using the bottom foot face.

8.8.5.3. Stub Model

Stub leads in Sherlock can be used to represent a variety of different types of simple leads or pads in real-world packages, based on the following block model properties. The models below show Sherlock's default lead modeling (C) as well as a model with Advanced Lead Meshing (p. 566) enabled (D).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



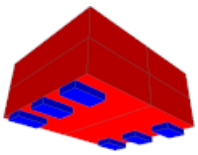
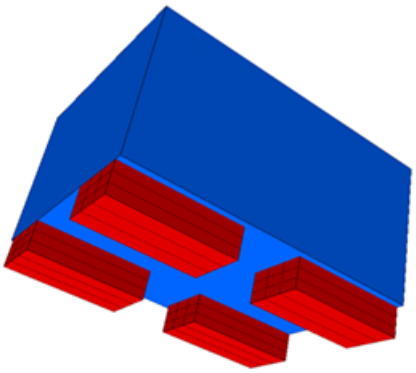
When incorporated into the FEA model, each Stub lead is bonded to the part using the top foot face (if any) and the side shoulder face (if any) and is bonded to the PCB using the bottom foot faces. By setting these properties in various ways, multiple configurations can be created. Specifically, the following stub lead configurations can be modeled by Sherlock.

Note:

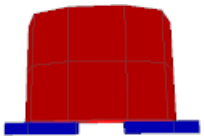
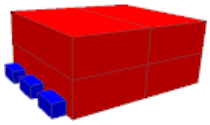
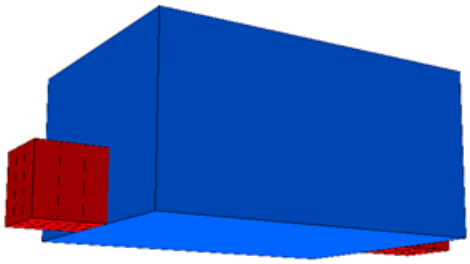
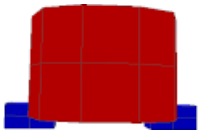
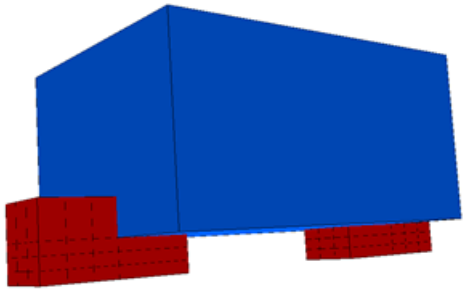
In all configurations, **Lead Foot Length** and **Lead Thickness** are always greater than 0.

The table below shows the different types of stub models. Advanced Lead Meshing models shown provide examples of how the leads are meshed. Note the difference in meshing between the extended foot section of the lead versus the foot and shoulder.

Table 8.17: Stub Model Lead Configurations

	Standard Lead Modeling	Advanced Lead Meshing
Stub Foot Only Shoulder = 0 Standoff = Thickness		

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stub Extended Foot Shoulder < Foot Length Standoff = Thickness		
Stub Shoulder Only Shoulder = Foot Length Standoff = 0		
Stub Foot & Shoulder Shoulder > 0 Shoulder < Foot Length Standoff > 0 Standoff < Thickness		

8.8.5.4. L Lead Model

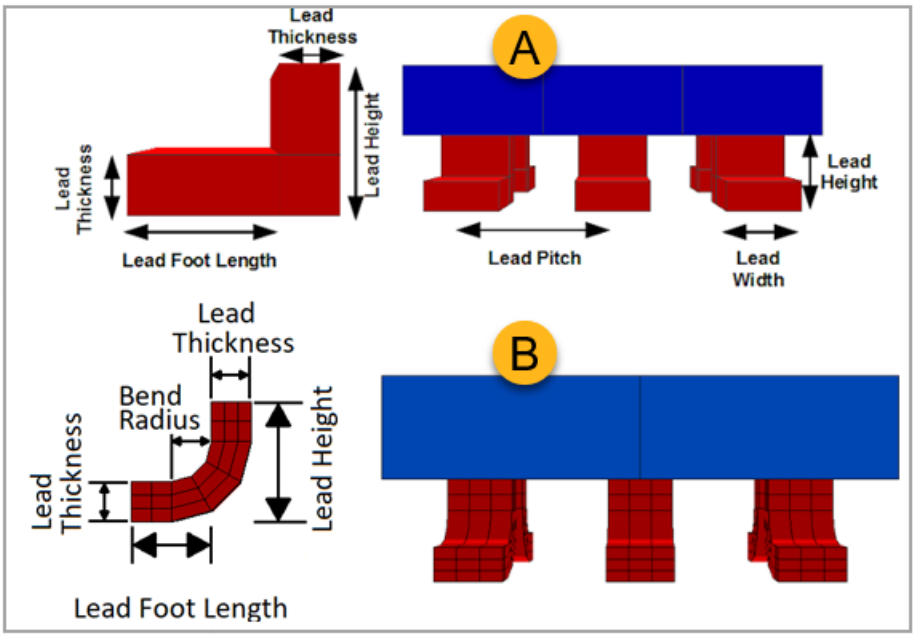
The L-Lead model consists of three brick elements, called the foot, elbow and leg. The top of the vertical leg element is always bonded to the bottom of the part body. When using the Bonded model, the bottom of the horizontal foot and elbow elements are bonded to the PCB.

The image below illustrates the properties used to define L-Leads. Model A shows Sherlock's default lead modeling. Model B shows Advanced Lead Meshing (p. 566) (ALM) which models the bend based on the **Lead Bend Radius** instead of modeling it as a rectangular brick. The **Lead Bend Radius** defines the inside radius of the bend between the leg and the foot of the lead.

Note:

When ALM is enabled, the vertical leg may be eliminated when the **Lead Height** is equal to the **Lead Thickness** plus the **Lead Bend Radius**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



L-Leads can be arranged using the SIP, DIP and CIRCULAR layouts.

SIP

When the SIP layout (C, below) is specified, the **Lead Count** must be exactly two. The leads will be located lengthwise along the part body centerline and the lead feet will point to the left and right sides of the part body. The **Lead Pitch** property is used to determine the location of each lead along the part centerline. D, below, shows the same model with Advanced Lead Meshing (p. 566) enabled with default settings.

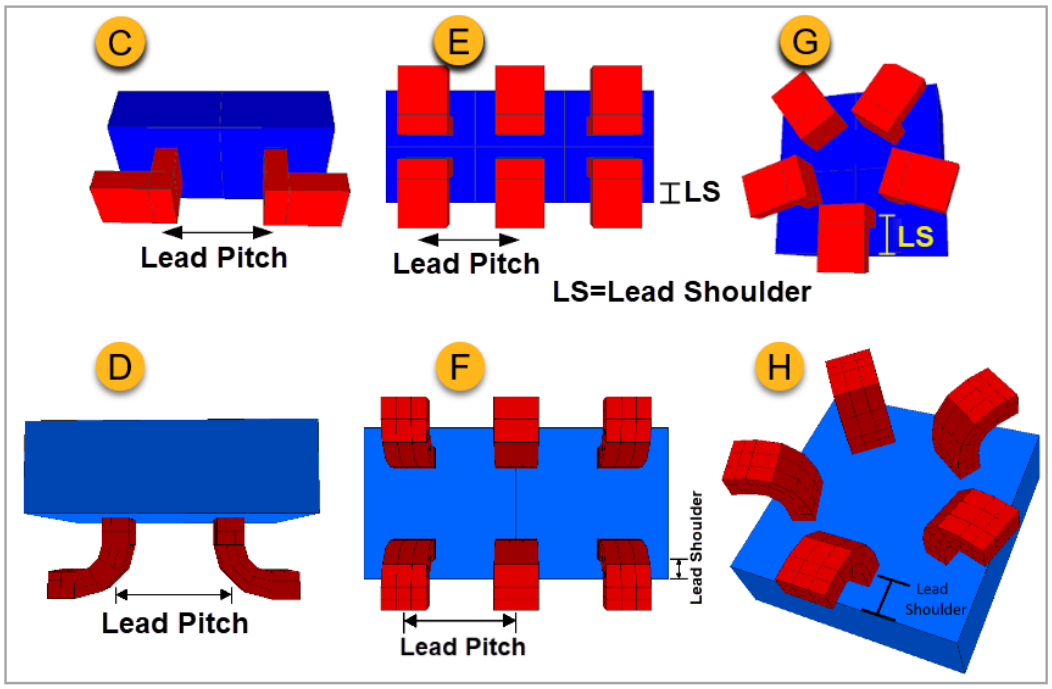
DIP

When the DIP layout (E) is specified, the **Lead Count** must be at least two and the lead feet will point to the front and back sides of the part body. The **Lead Pitch** property determines the spacing of leads in each row. The **Lead Shoulder** property determines the set-back of the vertical leg from the part body side. F, below, shows the same model with Advanced Lead Meshing (p. 566) enabled with default settings.

Circular

When the CIRCULAR layout (G) is specified, the **Lead Count** must be at least two and the lead feet will point outwards from the center point. The **Lead Shoulder** property determines the distance from the edge of the part body to the circle of lead centers. H, below, shows the same model with Advanced Lead Meshing (p. 566) enabled with default settings.

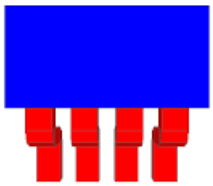
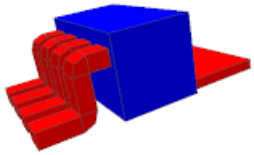
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.8.5.5. Lead Placement

In addition to the properties used to define the individual lead models, the **Lead Layout** property is provided to allow users to easily modify the placement of leads around a part. Although some real-world packages have some unique lead placements, the generalized layouts provided by Sherlock cover most lead patterns and are usually sufficient for FEA purposes. The following **Lead Layout** choices are supported, where N is the number of leads specified in the **Lead Count** property:

Table 8.18: Lead Layouts

Single N leads are placed along the length of the part on the same side.	**Single_Sink** N leads are placed on the same side of the part and a sink is placed on the opposite side of the part. The dimensions of the sink are specified by the **Sink Length**, **Sink Width** and **Sink Thickness** properties.
 Lead Count = 4	 Lead Count = 4
Single_1 N-1 leads are placed along the length of the part and a single lead is	**Single_2** N-2 leads are placed along the length of the part and 2 leads are placed on

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

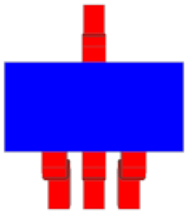
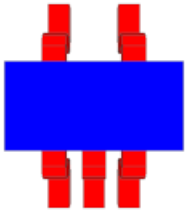
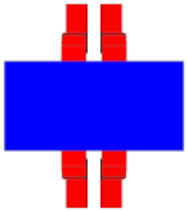
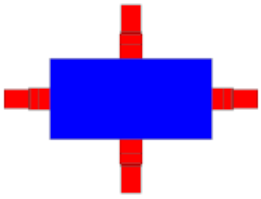
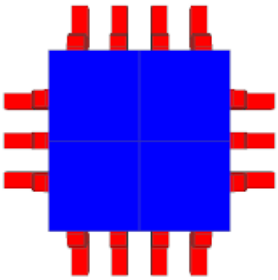
centered on the opposite side of the part.	the opposite side such that the outer each of the leads are aligned with the outer edges of the leads on the other side.
 Lead Count = 4	 Lead Count = 5

Table 8.19: Lead Layouts (continued)

Dual_Long	**Dual_Short**
N/2 leads are placed along the longest side of the part on opposite sides. If an odd number of leads is specified, one side will have an extra lead.	N/2 leads are placed along the shortest side of the part on opposite sides. If an odd number of leads is specified, one side will have an extra lead.
 Lead Count = 4	 Lead Count = 4
Quad	**Rectangular**
N/4 leads are placed along each side of the part. If N is not divisible by 4, the extra leads will be ignored.	The number of leads placed along the long side of the part determined by **Lead Long Count** and the number placed along the short side determined by **Lead Short Count.**
 Lead Count = 4	

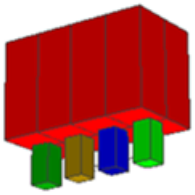
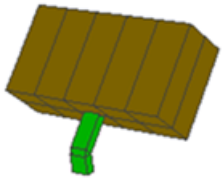
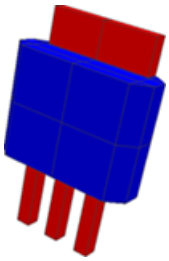
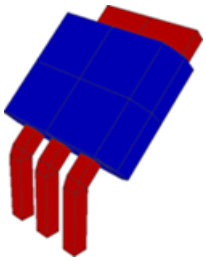
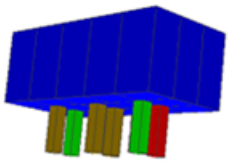
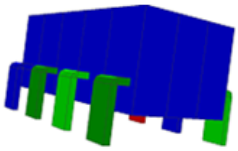
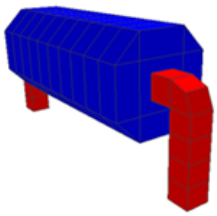
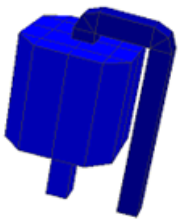
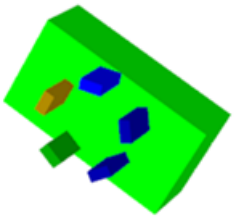
In all cases, leads are centered along each part side, spaced by the **Lead Pitch** property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

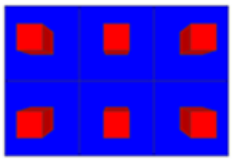
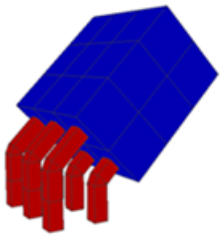
8.8.6. Through-Hole Lead Modeling

Through-hole leads can be modeled for a given part (or package) by setting **Lead Geometry** to **THRUHOLE** in the **Part Editor** (or **Package Editor**). At that point, the following **Lead Layout** property values are available to arrange the through-hole leads:

Table 8.20: Lead Layout Properties

Note: The models are shown with Sherlock's default lead meshing. See the sections below for examples where Advanced Lead Meshing is enabled.		
SIP Layout - (p. 582) Single rows of leads attached to the bottom of the component, running along the length of the component.		
SIP_SINK Layout - (p. 583) Same as SIP layout, with the addition of a heat sink on the top of the part.		
DIP Layout - (p. 584) Two rows of leads attached to the bottom or side of the component, running along the length of the component.		
AXIAL Layout - (p. 585) Two leads that are attached to opposite ends of the component.		
CIRCULAR Layout - (p. 588) Two or more leads arranged in a circle and attached to the bottom of the component.		

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GRID Layout - (p. 588) Similar to the SIP layout, with the exception that the leads are arranged in a rectangular grid pattern on the bottom of the part body.		

We now discuss how the lead properties are used by each through-hole layout type.

8.8.6.1. SIP Layout

8.8.6.2. SIP-SINK Layout

8.8.6.3. DIP Layout

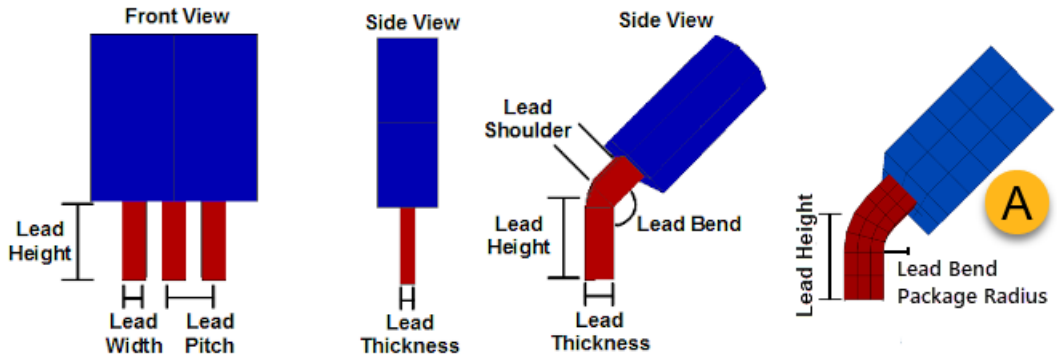
8.8.6.4. AXIAL Layout

8.8.6.5. CIRCULAR Layout

8.8.6.6. GRID Layout

8.8.6.1. SIP Layout

The SIP layout consists of a single row of leads attached to the bottom of the component, running along the length of the part on the longitudinal centerline of the part, as shown in the following diagrams:



The **Lead Count** property specifies the number of leads in the row and the **Lead Pitch** property determines the center-to-center spacing of the leads along the row. The **Lead Width** and **Lead Thickness** properties specify the cross-section dimensions of the lead. The Lead Width is measured along the length of the part, while the Lead Thickness is measured along the width of the part.

SIP leads can be straight or bent, depending on the value of the **Lead Bend** property.

For straight leads (that is, Lead Bend = 0), the **Lead Height** property specifies the distance from the PCB to the bottom of the part. The **Lead Shoulder** property is ignored for straight leads.

For bent leads (that is, 0 < Lead Bend <= 90), the **Lead Height** property specifies the distance from the PCB to the mid-point of the bend, and the **Lead Shoulder** property determines the distance from the inside corner of the bend to the part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

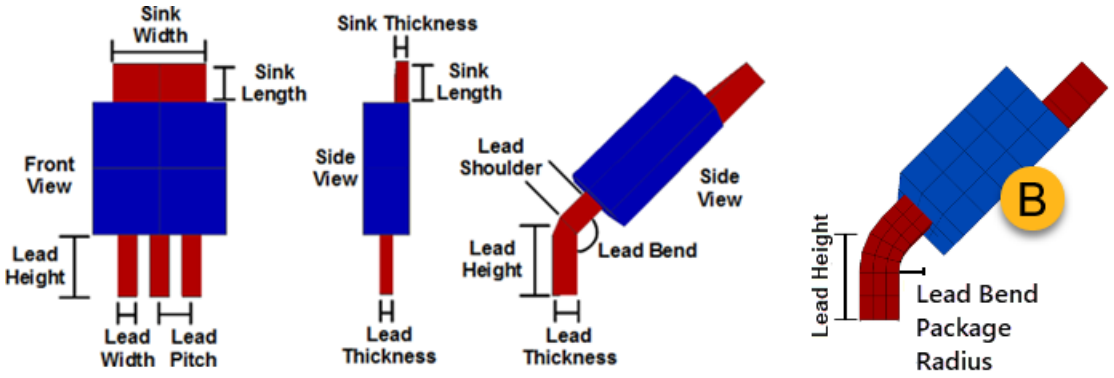
Model A in the image above shows the Advanced Lead Meshing (p. 566) (ALM) enabled which uses the **Lead Bend Package Radius** property—the inside radius of the bend between the leg and the shoulder of the lead. The unit used (such as mm) is set by the **Lead Units** property.

Note:

With ALM enabled, the vertical leg and shoulder may be eliminated when the **Lead Height** is equal to the **Lead Bend Package Radius**.

8.8.6.2. SIP-SINK Layout

The SIP-SINK layout consists of a single row of leads attached to the bottom of the component, running along the length of the component on the longitudinal centerline of the part and a single sink attached to the top of the part, as shown in the following diagrams:



The **Lead Count** property specifies the number of leads in the row and the **Lead Pitch** property determines the center-to-center spacing of the leads along the row. The **Lead Width** and **Lead Thickness** properties specify the cross-section dimensions of the lead. The Lead Width is measured along the length of the part, while the Lead Thickness is measured along the width of the part.

SIP_SINK leads can be straight or bent, depending on the value of the **Lead Bend** property.

For straight leads (that is, Lead Bend = 0), the **Lead Height** property specifies the distance from the PCB to the bottom of the part. The **Lead Shoulder** property is ignored for straight leads.

For bent leads (that is, 0 < Lead Bend <= 90), the **Lead Height** property specifies the distance from the PCB to the mid-point of the bend and the **Lead Shoulder** property determines the distance from the inside corner of the bend to the part.

The dimensions of the sink on top of the part are specified by the **Sink Length**, **Sink Width** and **Sink Thickness** properties, as depicted in the diagrams above. The sink is always centered along the length of the part and aligned with the back face of the part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model B in the image above shows the Advanced Lead Meshing (p. 566) (ALM) enabled which uses the **Lead Package Radius** property— the inside radius of the bend between the leg and the shoulder of the lead. The unit used (such as mm) is set by the **Lead Units** property.

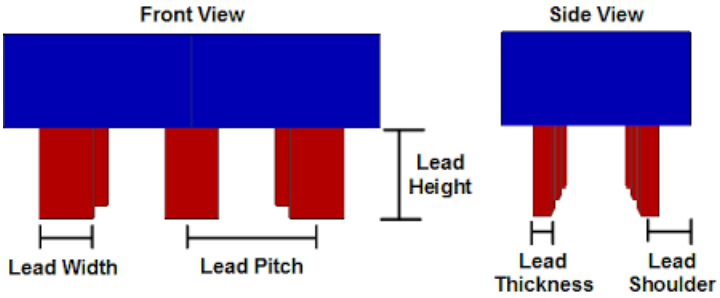
Note:

When ALM is enabled, the vertical leg and shoulder may be eliminated when the **Lead Height** is equal to the **Lead Bend Package Radius**.

8.8.6.3. DIP Layout

The DIP layout consists of two rows of leads attached either to the bottom or side of the component, running along the length of the component. The **Lead Count** property specifies the total number of leads, which should be an even number. The **Lead Pitch** property determines the center-to-center spacing of the leads along each row. The **Lead Width** and **Lead Thickness** properties determine the cross-section dimensions of the lead. DIP leads can be straight or bent by 90 degrees as specified by the **Lead Bend** property.

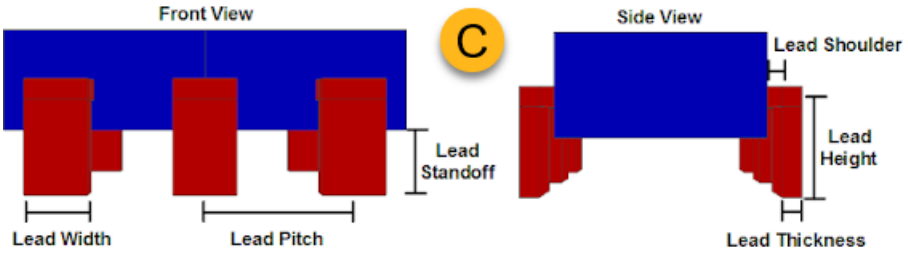
Straight leads (that is, Lead Bend = 0) are attached to the bottom of the part, as shown in the following diagrams:

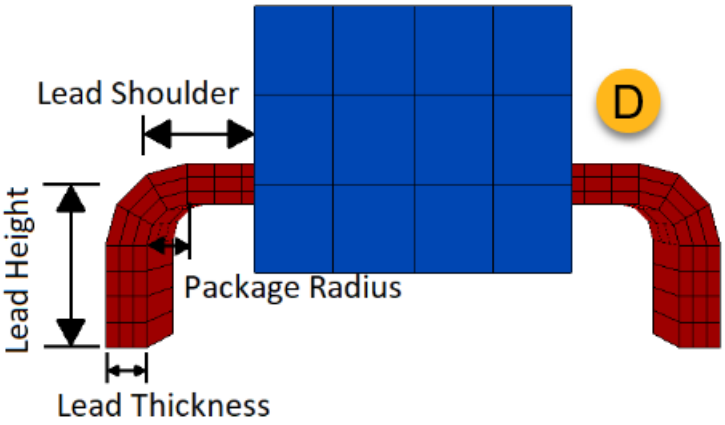


For straight leads, the **Lead Height** property specifies the distance from the PCB to the bottom of the part and the **Lead Shoulder** property specifies the distance from the length-wise sides of the part to the center of the lead.

Bent leads (that is, **Lead Bend** = 90) are attached to the length-wise sides of the part, as shown in the following diagrams.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





For bent leads, the **Lead Height** property specifies the distance from the PCB to the mid-point of the bend and the **Lead Shoulder** property specifies the distance from the inside corner of the bend to the side of the part. The **Lead Standoff** property specifies the distance from the PCB to the bottom of the part, which also determines the vertical position where the leads attach to the side of the part. The **Lead Height** should be greater than Lead Standoff + Lead Thickness / 2 and should be less than Lead Standoff + Part Thickness - Lead Thickness / 2.

Model C above shows Sherlock's default lead modeling. Model D shows Advanced Lead Meshing (p. 566) which uses the **Lead Bend Package Radius** to define the inside radius of the bend between the leg and the shoulder of the lead. The unit used (such as mm) is set by the **Lead Units** property.

Note:

When Advanced Lead Meshing is enabled, the vertical leg and shoulder may be eliminated when the **Lead Height** is equal to the **Lead Bend Package Radius**. Likewise, the horizontal portion of the shoulder may be eliminated if the **Lead Shoulder** is equal to the **Lead Package Radius**.

8.8.6.4. AXIAL Layout

The AXIAL layout consists of two leads that are attached to opposite ends of the component. The **Lead Width** and **Lead Thickness** properties determine the cross-section dimensions of the leads. AXIAL-Leads can be bent either 90° or 180° as specified by the **Lead Bend** property.

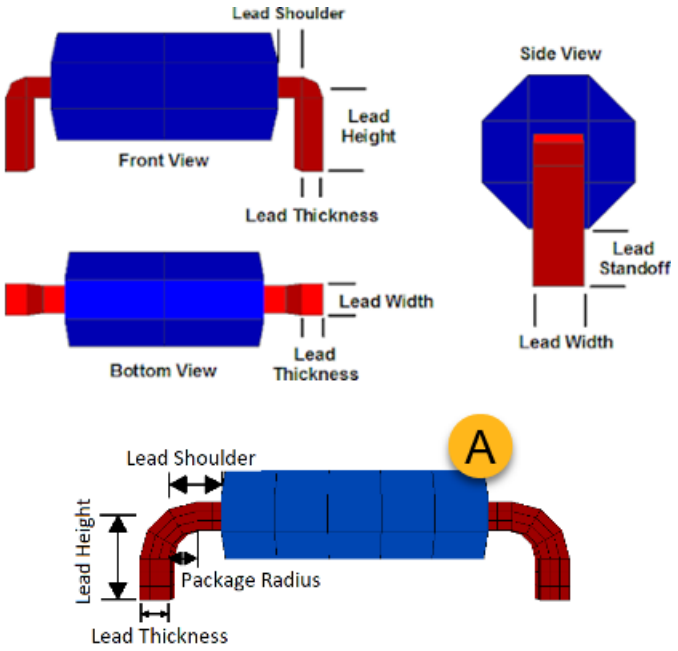
AXIAL 90 leads (that is, **Lead Bend** = 90) are attached to the left and right sides of the component, as shown in the following diagrams. Model A is shown with Advanced Lead Meshing (p. 566) enabled,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

requiring you to define the **Lead Bend Package Radius**—the inside radius of the bend between the leg and the shoulder of the lead. The unit used (such as mm) is set by the **Lead Units** property.

Note:

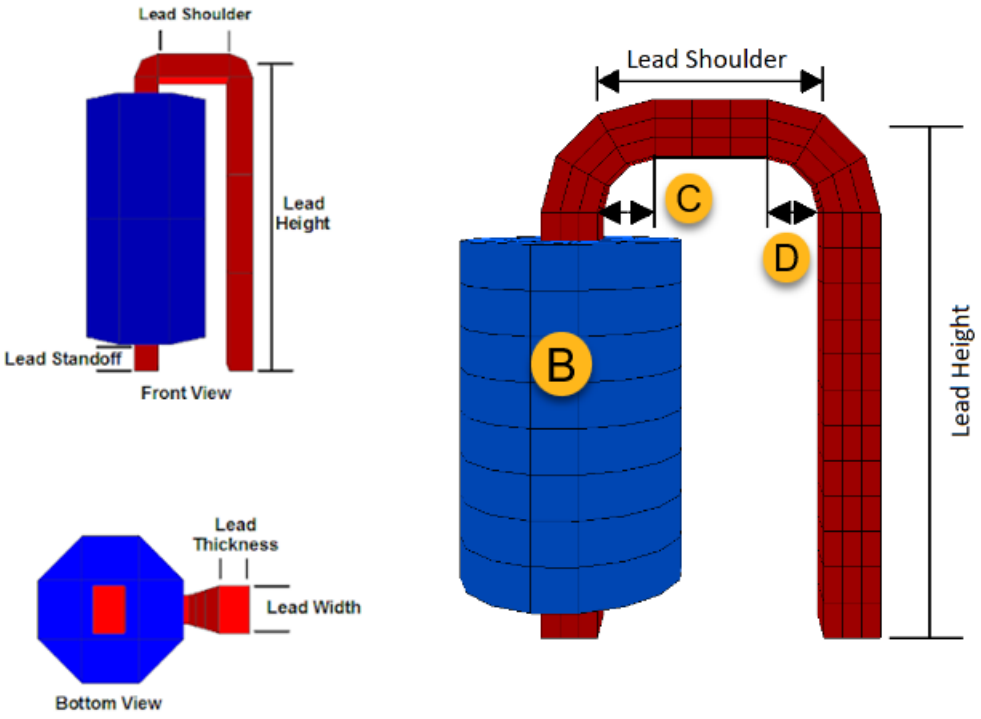
When Advanced Lead Modeling is enabled, the vertical leg may be eliminated when the **Lead Height** is equal to the **Lead Bend Package Radius** and the **Lead Standoff** is zero. Likewise, the horizontal portion of the shoulder may be eliminated if the **Lead Shoulder** is equal to the **Lead Bend Package Radius**.



For AXIAL 90 leads, the **Lead Height** property specifies the distance from the PCB to the mid-point of the bend and the **Lead Shoulder** property specifies the distance from the inside corner of the bend to the side of the part. The **Lead Standoff** property specifies the distance from the PCB to the bottom of the part, which also determines the vertical position where the leads attach to the side of the part.

AXIAL 180 leads (that is, **Lead Bend** = 180) are attached to the top and bottom of the component, as shown in the following diagrams:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For AXIAL 180 leads, the **Lead Height** property specifies the distance from the PCB to the mid-point of the bend and the **Lead Shoulder** property specifies the distance between the two vertical leads. The **Lead Standoff** property specifies the distance from the PCB to the bottom of the part, which also determines the distance between the top of the part and the bottom of top horizontal lead segment.

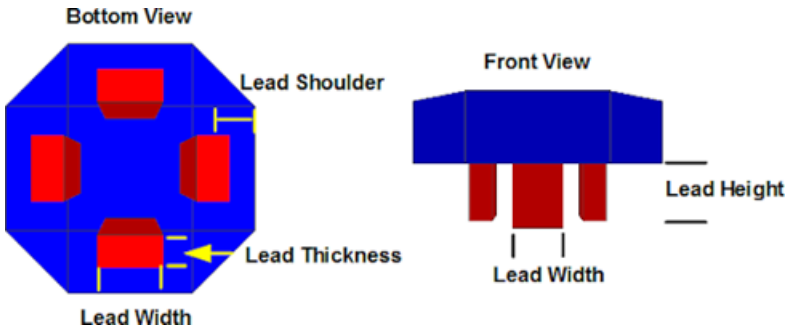
The Lead Height should be greater than Lead Standoff + Lead Thickness / 2 and should be less than Lead Standoff + Part Thickness - Lead Thickness / 2.

Model B is shown with Advanced Lead Meshing (p. 566) enabled which uses the **Lead Bend Package Radius** property (C) and the **Lead Bend Radius** property (D). The **Lead Bend Package Radius** defines the inside radius of the bend between the shoulder and the top of the part. The **Lead Bend Radius** defines the inside radius of the of the bend between the leg and the shoulder of the lead. The unit used (such as mm) is set by the **Lead Units** property.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

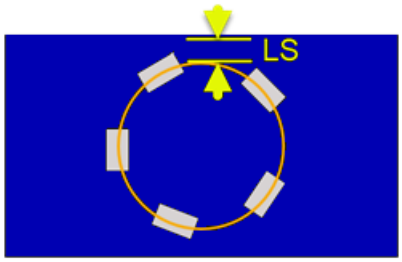
8.8.6.5. CIRCULAR Layout

The CIRCULAR layout consists of two or more straight leads arranged in a circle and attached to the bottom of the part, as shown in the following diagrams:



The **Lead Count** property specifies the total number of leads. The **Lead Width** and **Lead Thickness** properties determine the cross-section dimensions of the leads. The **Lead Height** property specifies the distance between the PCB and the bottom of the part.

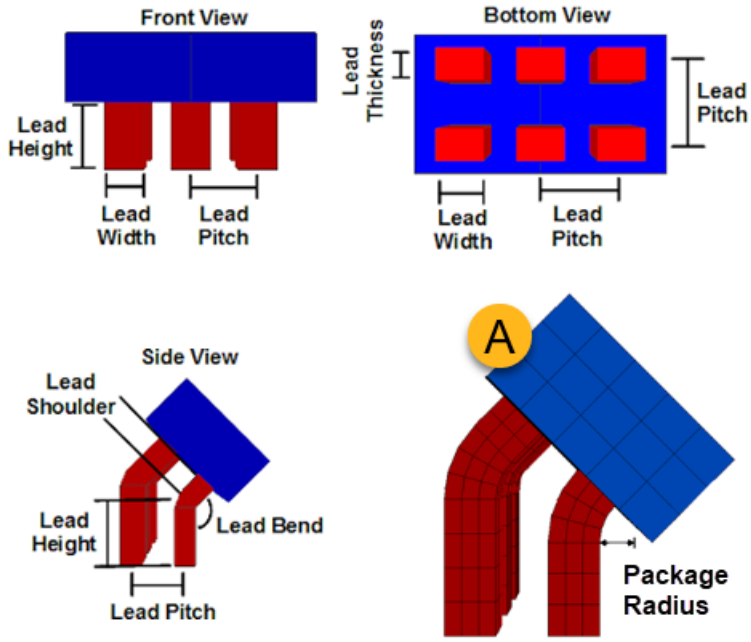
The **Lead Shoulder** property specifies the distance from the edge of the part to the circle along which the leads are placed. If the part length and part width are different, the **Lead Shoulder** distance will be measured along the shorter dimension to determine the layout circle, as shown in the following diagram:



8.8.6.6. GRID Layout

The GRID layout is similar to the SIP layout, with the exception that the leads are arranged in a rectangular grid pattern on the bottom of the part body, as shown in the following diagrams:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Lead Long Count** and **Lead Short Count** properties specify the number of leads along the long and short sides of the part, respectively. In the examples shown above, Lead Long Count is 3 and Lead Short Count is 2. The **Lead Count** property specifies the total number of leads and should always be equal to Lead Long Count multiplied by Lead Short Count.

The **Lead Pitch** property determines the center-to-center spacing of the leads along each row and column. The **Lead Width** and **Lead Thickness** properties determine the cross-section dimensions of the leads.

GRID leads can be straight or bent, depending on the value of the **Lead Bend** property.

For straight leads (that is, Lead Bend = 0), the **Lead Height** property specifies the distance from the PCB to the bottom of the part. The **Lead Shoulder** property is ignored for straight leads.

For bent leads (that is, 0 < Lead Bend <= 90), the **Lead Height** property specifies the distance from the PCB to the mid-point of the bend and the **Lead Shoulder** property specifies the distance from the inside corner of the bend to the part.

Advanced Lead Meshing (p. 566) (A in the image above) enables the **Lead Bend Package Radius** property which defines the inside radius of the bend between the leg and the shoulder of the lead. The unit used (such as mm) is set by the **Lead Units** property.

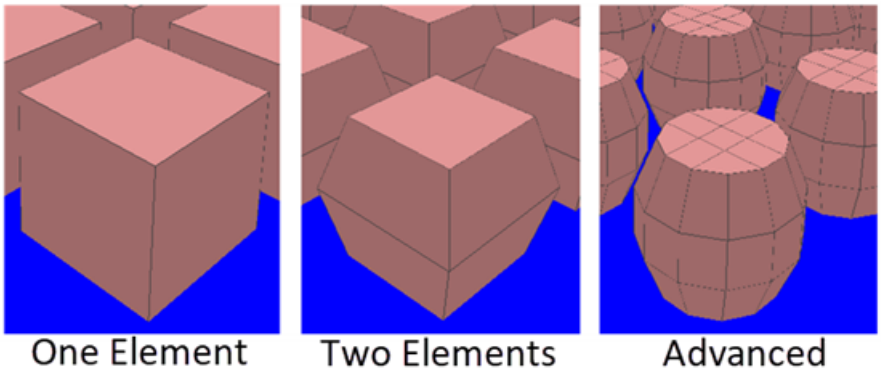
Note:

The vertical leg may be eliminated when the **Lead Height** is equal to the **Lead Bend Package Radius** and the lead **Standoff** is set to zero. Likewise, the horizontal portion of the shoulder may be eliminated if the **Lead Shoulder** is equal to the **Lead Bend Package Radius**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.7. Solder Ball Modeling

BGA solder balls can be modeled for a given part (or package) by setting the **Ball Modeling** property to either **One Element**, **Two Elements**, or **Advanced** which determines the number of elements used to model each solder ball, as shown below. (See the **Ball Modeling** property in BGA Solder Ball Properties (p. 572).)



- **One Element** solder balls are modeled by a single brick element that can be a first order, second order or shell element, depending on the **Element Order** property defined in the **Lead** tab of the analysis / export input dialog. One element solder balls reduce the number of elements generated for BGA parts, thereby decreasing analysis time.
- **Two Element** solder balls are modeled by a pair of brick elements that share common nodes. As with one element solder balls, the elements can be first order, second order or shell elements based on the analysis input properties. Two element solder ball models more closely represent the actual shape of a solder ball, at the cost of increased model complexity and analysis times.
- **Advanced** solder balls consist of 3x3x3 elements.

Note:

– When exported to AEDT Icepak, **One-Element** solder balls are exported as cylinders. **Two-Element** and **Advanced** solder balls are exported as spheres.

– When exported as a STEP file (see Ansys Workbench Geometry and Materials Import (p. 752)), a **One-Element**, **Two-Element**, or **Advanced** solder ball is treated as a cubical body.

The **Ball Material** property is used to specify the material properties assigned to each solder ball. The Ball Material must reference one of the solder materials specified in the **solder.csv** file. See the section on User Data Files and Documents (p. 166) for more details.

Note:

Solder balls are always modeled as square vertical leads when exporting an FEA model to Abaqus and Ansys Mechanical to reduce the possibility of the FEA engines generating extremely small or irregular elements when meshing the solder ball shapes.

Links to subsections:

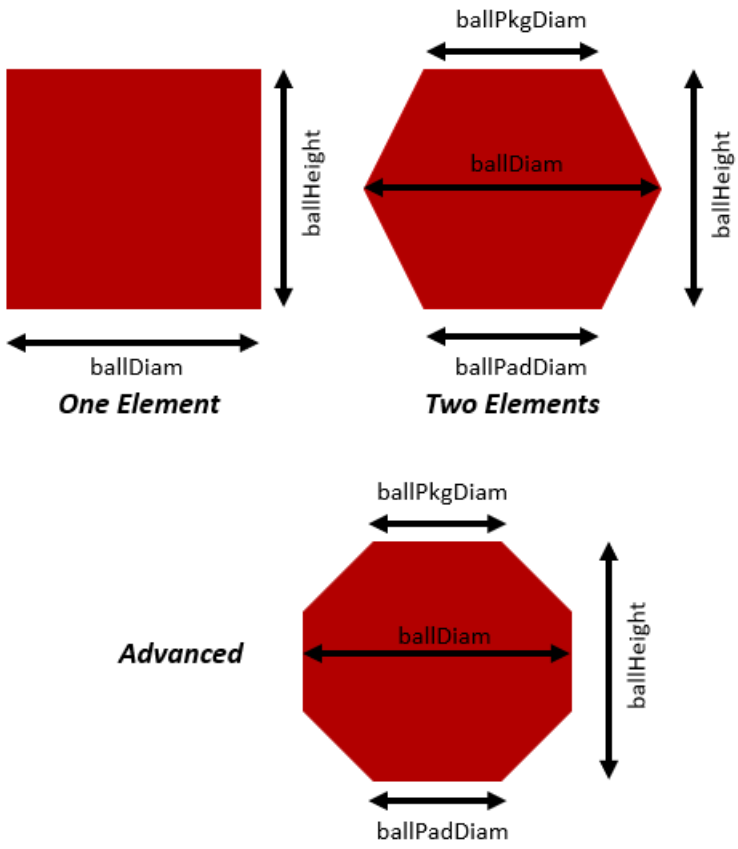
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.7.1. Solder Ball Shape Properties
8.8.7.2. Solder Ball Layout Properties

8.8.7.1. Solder Ball Shape Properties

The properties used to specify the shape of a solder ball are listed and illustrated below.

Table 8.21: Ball Shape Properties

• Ball Diameter • Ball Height • Ball Package Diameter • Ball Units • Ball Pad Diameter	

The **Ball Diameter** property (ballDiam) specifies the size of the square faces at the top and bottom of a one element solder ball and the size of the square middle face of a two-element or advanced solder ball.

The **Ball Package Diameter** property (ballPkgDiam) specifies the size of the square face at the top of a two-element or advanced solder ball. The property is ignored for one element solder balls.

The **Ball Pad Diameter** property (ballPadDiam) specifies the size of the square face at the bottom of a two-element or advanced solder ball. The property is ignored for one element solder balls.

The **Ball Height** property (ballHeight) specifies the overall height of the solder ball.

The **Ball Units** property specifies the type of units entered for all shape properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.7.2. Solder Ball Layout Properties

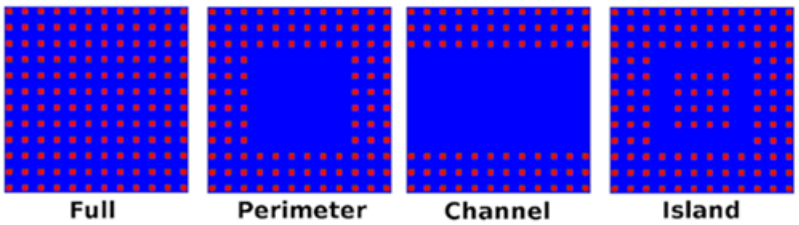
The following properties are used to specify the layout of all solder balls for a given part:

Table 8.22: Solder Ball Layout Properties

Ball Pattern	Ball Count	Ball Units
Ball Pitch	Ball Chan Width	Ball Perimeter Rows
Ball Perimeter Cols	Ball Island Rows	Ball Island Cols

By defining these properties in various ways, a large variety of common and non-common BGA solder ball patterns can be modeled in Sherlock.

The **Ball Pattern** property specifies the overall layout pattern for the BGA solder balls, with the following patterns supported:



Additionally, a pattern named **NETLIST** exists which will use the ball locations defined in netlist for the part instead of the parameterized values here. The **NETLIST** pattern only makes use of the Solder ball shape properties.

For each supported pattern, one or more of the layout properties are used to specify the exact layout of solder balls, as we now discuss.

8.8.7.2.1. Full Ball Pattern

8.8.7.2.2. Perimeter Ball Pattern

8.8.7.2.3. Channel Ball Pattern

8.8.7.2.4. Island Ball Pattern

8.8.7.2.5. Corner Solder Balls

8.8.7.2.6. Automatic Consistency Checks

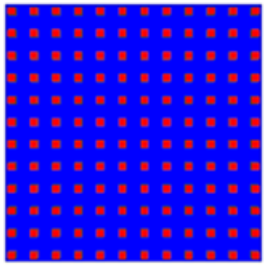
8.8.7.2.1. Full Ball Pattern

The table below lists the properties used to specify the **Full** solder pattern. The Full pattern consists of a solder ball grid that contains a given number of rows and columns specified by the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ball Perimeter Rows and **Ball Perimeter Cols** properties, respectively. The balls are spaced, both horizontally and vertically, based on the **Ball Pitch** property.

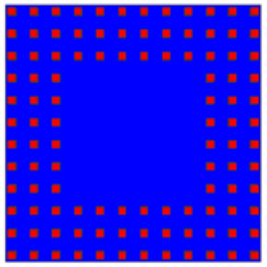
Table 8.23: Full Ball Pattern Properties

• Ball Count • Ball Perimeter Rows • Ball Pitch • Ball Perimeter Cols	

8.8.7.2.2. Perimeter Ball Pattern

The following properties are used to specify the **Perimeter** solder pattern:

Table 8.24: Perimeter Solder Pattern Properties

• Ball Count • Ball Perimeter Rows • Ball Chan Width • Ball Pitch • Ball Perimeter Cols	

The Perimeter pattern consists of one or more rows/cols of solder balls that run along the perimeter of the part. The outer pattern of solder balls is specified by the **Ball Perimeter Rows** and **Ball Perimeter Cols** properties. The space between solder balls, both horizontally and vertically, is specified by the **Ball Pitch** property.

The **Ball Chan Width**, **Ball Pitch** and **Ball Diameter** property values determine the number of rows and columns generated in the Perimeter layout. Specifically, the number of perimeter rows and columns is equal to:

(Ball Chan Width - Ball Diameter) / Ball Pitch

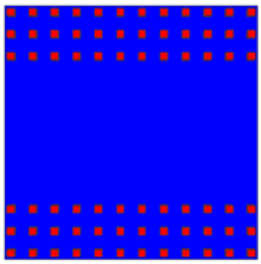
For example, if the Ball Chan Width is 10 mm, Ball Diameter is 1 mm and Ball Pitch is 3 mm, then the number of rows and columns generated is (10 - 1) / 3 = 3.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.7.2.3. Channel Ball Pattern

The following properties are used to specify the **Channel** solder pattern:

Table 8.25: Channel Ball Pattern Properties

• Ball Count • Ball Perimeter Rows • Ball Chan Width • Ball Pitch • Ball Perimeter Cols	

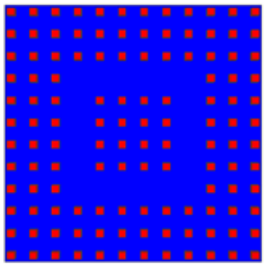
The Channel pattern is similar to the Perimeter pattern, except that the channels are generated only on two sides of the part. The number of rows in each channel is determined by the **Ball Chan Width**, **Ball Pitch** and **Ball Diameter** property values just like the Perimeter pattern.

8.8.7.2.4. Island Ball Pattern

The table below shows the properties used to specify the **Island** solder pattern. The Island pattern is similar to the Perimeter pattern, except that there is a grid of solder balls located in the center of the part. As with the Perimeter layout, the number of solder ball rows and columns on the perimeter of the part are determined by the **Ball Chan Width**, **Ball Pitch** and **Ball Diameter** property values.

The grid of solder balls located in the center of the part is determined by the **Ball Island Rows** and **Ball Island Cols** properties.

Table 8.26: Island Solder Pattern Properties

• Ball Count • Ball Perimeter Rows • Ball Island Rows • Ball Chan Width • Ball Pitch • Ball Perimeter Cols • Ball Island Cols	

8.8.7.2.5. Corner Solder Balls

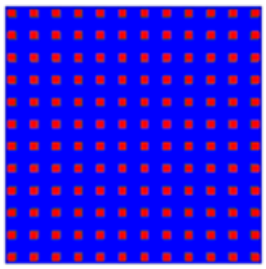
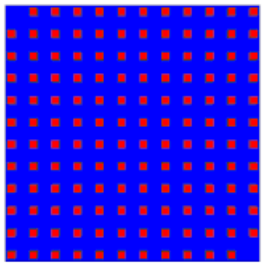
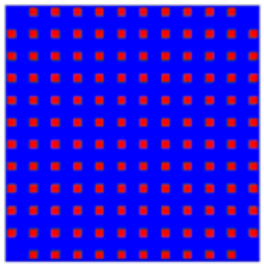
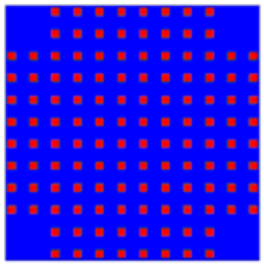
In some BGA solder patterns, one or more corner balls may be missing, resulting in some unique patterns. Although Sherlock is not capable of modeling such arbitrary solder ball patterns, users may use the **Ball Count** property to have Sherlock automatically remove one or more corner balls. For example, consider the pattern specified by the following properties:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ball Pattern = Full
- Ball Perimeter Rows = 12
- Ball Perimeter Cols = 12

Depending on how the Ball Count property is defined, Sherlock will generate the following patterns:

Table 8.27: Corner Ball Count Patterns

Ball Count = 144 No corners missing	
Ball Count = 142 2 corners missing	
Ball Count = 140 4 corners missing	
Ball Count = 128 16 corners missing	

Automatic corner ball removal is performed for Full, Perimeter and Island patterns whenever the Ball Count is less than the number of balls specified by the layout row and column properties. Corner balls are removed equally from all four corners of the pattern until the required number of removals is attained. Outer most solder balls are always removed first in each corner.

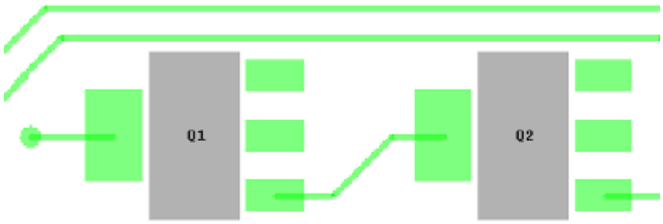
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.7.2.6. Automatic Consistency Checks

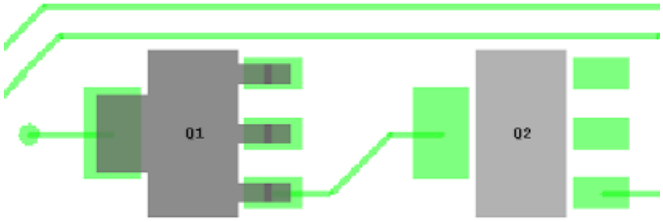
Sherlock performs a set of consistency checks whenever a BGA part or package model is created for visualization or analysis purposes. If any inconsistencies are found, a warning is printed to the Sherlock console and default values are used to eliminate the inconsistency. As such, it is important for all Sherlock users to use the **View 3D Model** button available in the Part Editor and Package Editor to ensure that the properties specified for a given part or package actually generate the desired BGA solder ball pattern.

8.8.8. 2D Lead Viewing

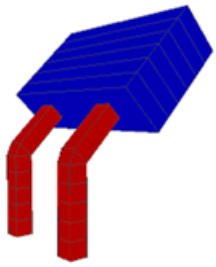
Starting in Version 5.0, the Layer Viewer optionally allows users to display a 2D representation of component leads using the **Leads** filter. For example, the following image shows a region of the tutorial board with the **Leads** filter disabled.



In this case, the Q1 and Q2 parts are displayed using simple rectangular outlines because the **Leads** filter is disabled. If the **Leads** filter is enabled, the following rendering would be generated:



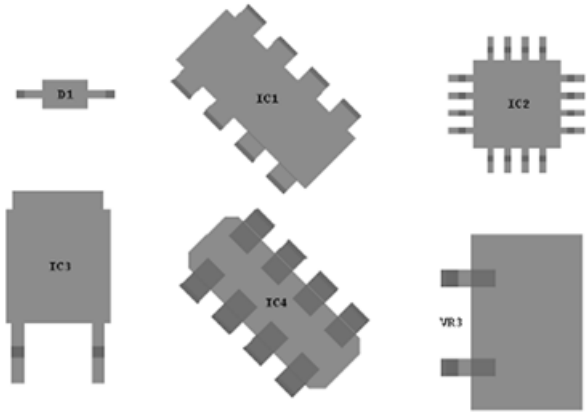
In this case, the Q1 is displayed by a set of rectangular outlines showing the location of each gullwing lead and the heatsink tab as viewed from above. The Q1 component is defined as a SOT-223 package and has the **Lead Modeling** part property set to **Enabled**. The Q2 part is still rendered by a simple rectangle because the **Lead Modeling** part property is **Disabled** for Q2.



All types of leads (for example, J-lead, Gullwing, Balls, etc) are supported, including through-hole leads. In most cases, the main body of the part will be seen in the same size and orientation regardless of the **Leads** filter setting because SMT leads and non-bent through-hole leads have no effect on the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

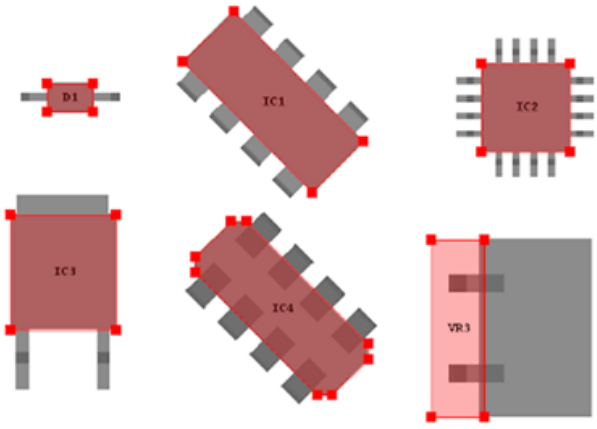
location of the part body in the XY plane. However, when displaying through-hole parts with bent leads, the exact size and orientation of the part and lead profile will vary depending on the height, bend and shoulder length of the lead.



For example, consider the through-hole part shown above with leads that have a 45 degree bend in them. When viewed from above in the Layer Viewer, the main body of the part will appear to one side of the center location, instead of directly above it, as can be seen in the image from the Layer Viewer shown here.

This image shows how most SMT and through-hole parts are still centered over the part label (which denotes the center location assigned to the part) when leads are rendered, but not the VR3 part which has 45-degree bent leads. In such cases, the 3D representation of the part and leads is projected onto the XY plane to be displayed in the Layer Viewer.

It is important to remember that the label associated with each part is always displayed at the location assigned to the part in the Parts List, regardless of the part/lead outline. Furthermore, when selecting a part for editing in the Component Editor, the simplified top-view outline of the part will be highlighted, not the part/lead outline, as can be seen here:



Highlighted regions always show the size and orientation of the part when viewed from above **without any consideration for lead orientation.** This approach is taken so that users can easily resize, rotate, and move the part using the selected region.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.9. FEA- Heat Sink Modeling

Ansys Sherlock has the capability to model various types of heat sinks in addition to the PCB, components, and mount points for Finite Element Analysis (FEA) purposes. Heat sinks are modeled with a set of properties defined in the layer viewer. Once added to a circuit card assembly (CCA), these heat sinks are used during analysis. Heat sinks may optionally be attached to mount pads which are attached to the PCB.

In this section, the following topics are covered:

8.9.1. Heat Sink Modeling
8.9.2. Heat Sink Modeling Properties
8.9.3. Creating Heat Sinks
8.9.4. Updating Heat Sinks
8.9.5. Heat Sink Attachment Points
8.9.6. Viewing Heat Sink 3D Model
8.9.7. Heat Sink Analysis

8.9.1. Heat Sink Modeling

The Sherlock analysis modules make use of the following properties for heat sink modeling.

Table 8.28: Heat Sink Modeling Properties

Heat Sink Layout	Heat Sink Material	Heat Sink Units
Heat Sink Length	Heat Sink Width	Heat Seat Thickness
Heat Sink Fin Height	Heat Sink Fin Width	Heat Seat Fin Thickness
Heat Sink Length Fin Pitch	Heat Sink Width Fin Pitch	Heat Sink Fin Bend
Heat Sink Fin Shoulder		

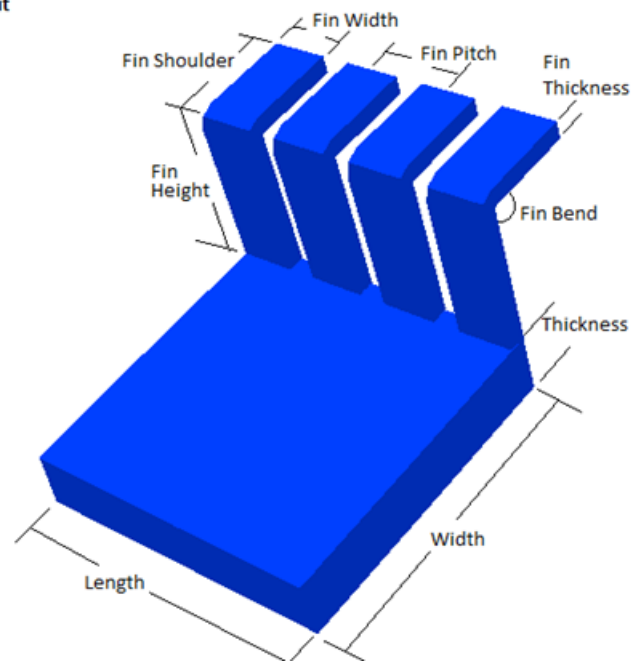
8.9.1.1. Heat Sink Layouts

The **Heat Sink Layout** property defines the basic shape of a heat sink. Supported values are:

Single Fin – heat sink with a single row of fins along the length of one edge of the heat sink base. Fins may be vertical or have a bend up to 90 degrees. The number of fins is governed by the length of the heat sink base, the fin length pitch, and the fin width.

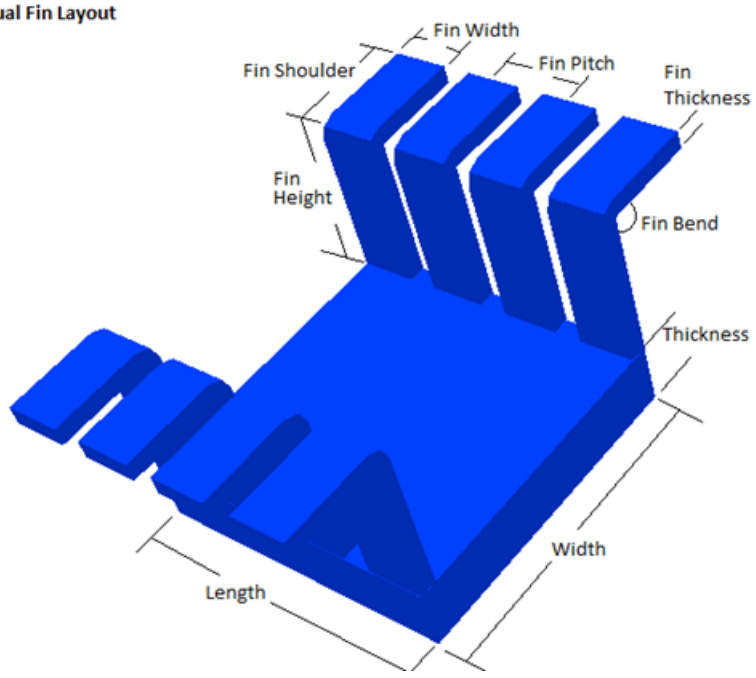
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Single Fin Layout



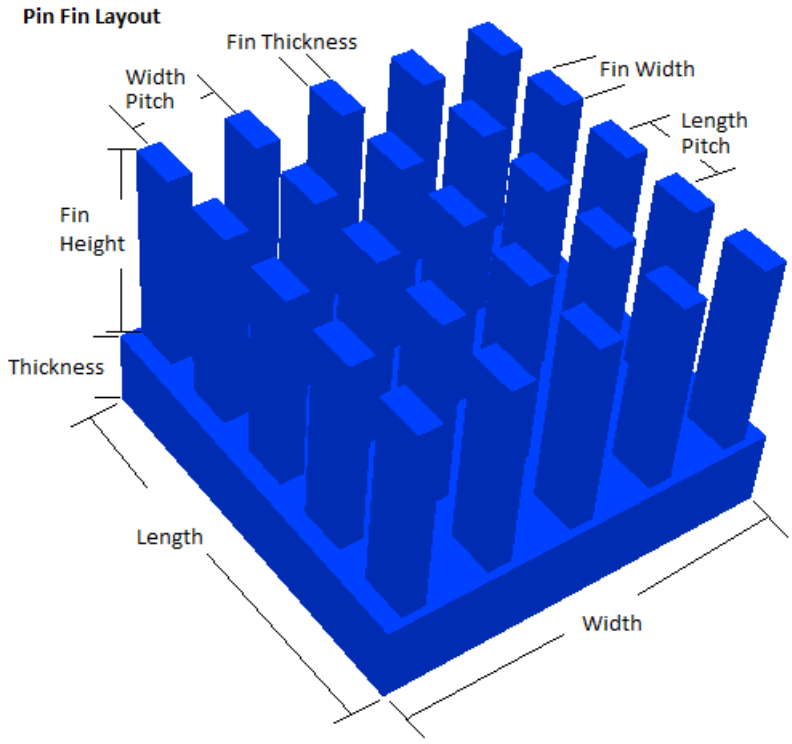
Dual Fin – heat sink with two rows of fins, each along the length of one edge of the heat sink base. Fins may be vertical or have a bend up to 90 degrees. The number of fins is governed by the length of the heat sink base, the fin length pitch, and the fin width.

Dual Fin Layout



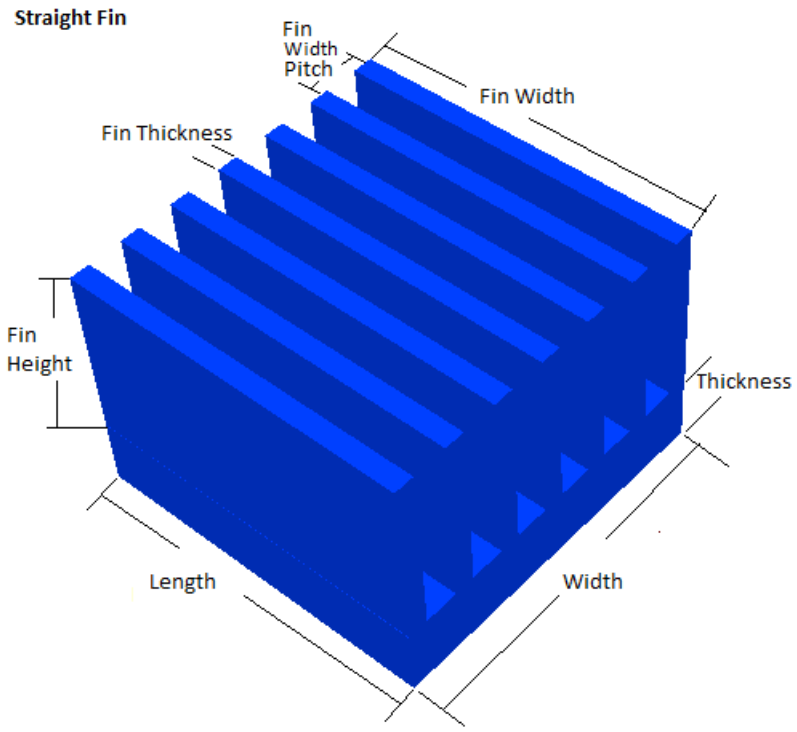
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pin Fin – heat sink with one or more rows and columns of fins. The number of fins along the length of the heat sink is governed by the length of the heat sink, the fin length pitch, and the fin width. The number of fins along the width of the heat sink is governed by the width of the heat sink, the fin width pitch, and the fin thickness.

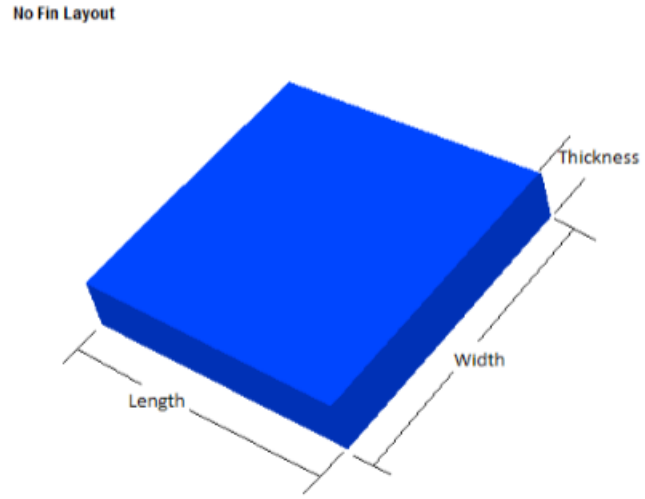


Straight Fin – heat sink with one or more rows of fin blades. The length of the fin blades is governed by the fin width. The number of fins is governed by the width of the heat sink, the fin width pitch, and the fin thickness.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



No Fin – heat sink with no fins or fin blades.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.9.2. Heat Sink Modeling Properties

Heat sinks are modeled based on a set of properties. The following properties define how a given heat sink layout is modeled.

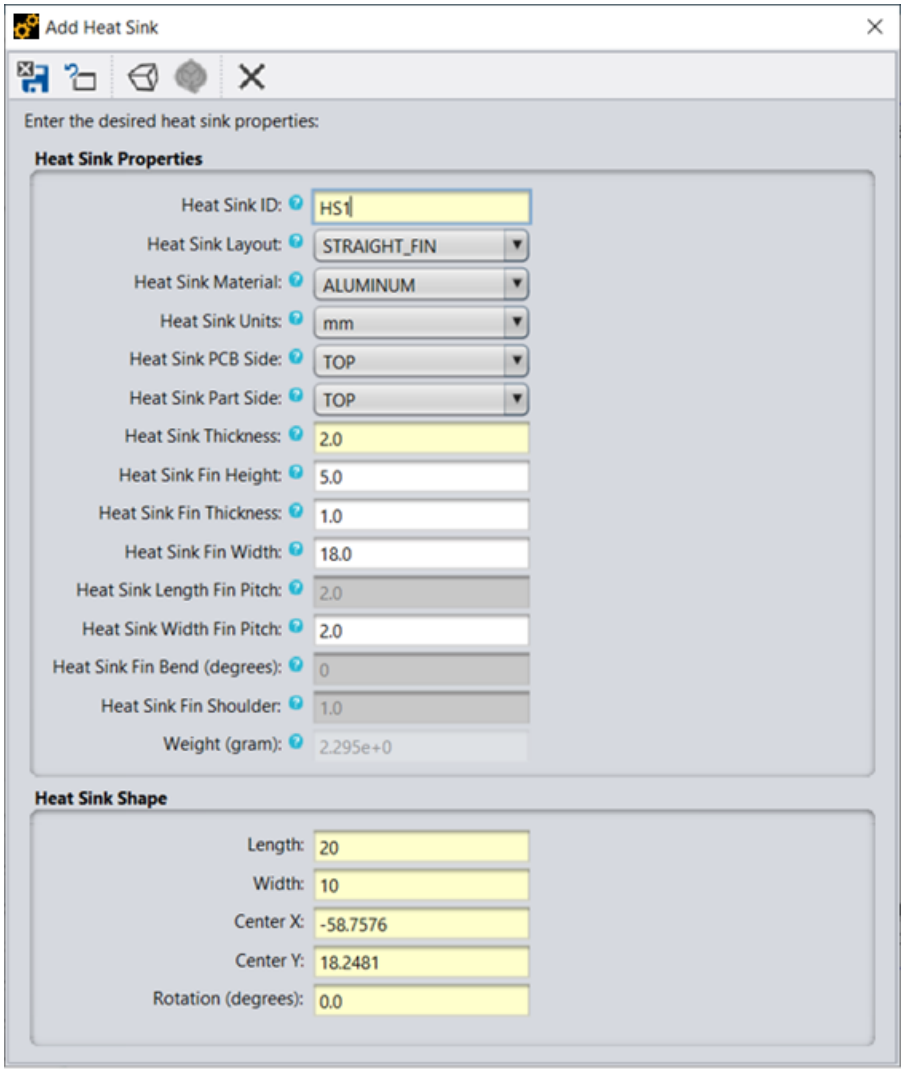
- **Heat Sink Length**: Length of the heat sink (usually maximum dimension).
- **Heat Sink Width**: Width of the heat sink (usually minimum dimension).
- **Heat Sink Thickness**: Thickness (height) of the heat sink base.
- **Heat Sink Fin Bend**: Indicates how far the heat sink fin is bent from vertical in degrees. When set to zero (0), the fin will be straight and vertical. When greater than zero, the fin will be modeled by a lower vertical segment and an upper shoulder segment that is bent from vertical by the specified amount. The length of the vertical segment is specified by the **Heat Sink Fin Height** property, while the length of the shoulder segment is specified by the **Heat Sink Fin Shoulder** property.

  For **Single-Fin** and **Dual-Fin** layouts, the angle may be between 0 and 90 degrees (inclusive). For all other layouts, the angle must be 0 degrees.
- **Heat Sink Fin Height**: Height of the heat sink fin for straight fins and the length of the vertical segment of bent fins. For bent fins, the height specifies the location where the center of the shoulder segment intersects the vertical segment.
- **Heat Sink Fin Width**: Width of the heat sink fin (usually the maximum dimension).
- **Heat Sink Fin Thickness**: Thickness of the heat sink fin (usually the minimum dimension).
- **Heat Sink Length Fin Pitch**: The center-to-center distance between adjacent fins along the length of the heat sink.
- **Heat Sink Width Fin Pitch**: The center-to-center distance between adjacent fins along the width of the heat sink.
- **Heat Sink Fin Shoulder**: The length of the non-vertical fin segment for bent fins.
- **Heat Sink Material**: Primary material of which the heat sink is made. Material names are defined in the material data file provided by Sherlock and/or a user-defined data file.
- **Heat Sink Units**: Unit of length for all sink measurement properties.
- **Weight**: The estimated heat sink weight is displayed when viewing heat sink properties. This is a read-only field only intended for reference.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.9.3. Creating Heat Sinks

Heat sinks are added in the **Layer Viewer**. Using the ODB++ Tutorial (p. 23)example, select **Edit Heat Sinks** from the **Edit** menu of the **Layer Viewer**. The **Heat Sink Editor** will now be indicated along the bottom of the **Layer Viewer**.



To add a heat sink, right-click one an area on one of the components displayed and select **Add Heat Sink**. This will display the **Add Heat Sink** dialog.

This dialog allows for the definition of all the properties that define the location of the heat sink and the properties of the heat sink. When adding a heat sink to a component, the default length and width values of the heat sink will be the same as the part.

Heat sinks may be placed on top of or underneath of parts. When placing underneath of parts, typically select either the single-fin or dual-fin heat sink layout as these two layouts line the fins along the outside of the edge of the heat sink.

To view a definition of each of the heat sink properties along with a diagram of how each is defined for every heat sink layout, click the question mark next to the desired property.

To add this heat sink, click the **Save the changes and exit the editor** button in the dialog.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

 Save the changes and exit the editor

 Reset the form to the currently saved values

 View the 3D model of the heat sink

 Refresh the 3D model

 Close the dialog

Links to subsections:

8.9.4. Updating Heat Sinks

In the layer viewer, verify the **Heat Sinks** folder is opened in the **Layers** menu and the desired side of the PCB is selected to see all heat defined heat sinks. Similar to other layer viewer editors, right-click the heat sink to be modified, then select **Edit Properties** from the context menu. The same dialog is presented for updated properties as is created for adding heat sinks.

From the context menu, you may also select to move the heat sink which will allow you to reposition the heat sink by using the mouse to move the heat sink to the desired location. You may rotate the heat sink one of the predefined offsets. Adjustments to the rotation other than 45 and 90 degree increments may be made from the heat sink properties dialog. You may copy the heat sink, which will result in an identical heat sink created and located next to the existing heat sink. You may delete the heat sink. When you are complete with the Heat Sink editor you must **Save** or **Apply** any changes for them to take affect.

Multiple heat sinks may also be selected and the properties of those heat sinks updated at the same time. When the properties for more than one heat sink are edited at the same time, any heat sink property which varies between the selected heat sinks will display the value <*VARIOUS*>. Any field which is left as the value <*VARIOUS*> when saving the form will leave the original heat sink value specified for that given field. Any other field values will be assigned to all selected heat sinks.

8.9.5. Heat Sink Attachment Points

Heat sinks may be located over top of mount points defined as mount pads or standoffs. When a heat sink is located over top of these mount points heat sink attachment points may be created between the heat sink and the mount point. To add an attachment point to a heat sink, place the heat sink over top of a mount point using the **Heat Sink Editor** in the **Layer Viewer.** Now make sure no heat sinks are selected, then right-click a mount point that should become an attachment point for the heat sink. If the mount point meets all the properties to become an attachment point, then a context menu should appear to add the selected mount point as a heat sink point. Repeat this process for each of the desired mount points. To remove a mount point as a heat sink attachment point, right-click the mount point and select **Remove Heat Sink Point**.

Attachment points for heat sinks must adhere to the following conditions:

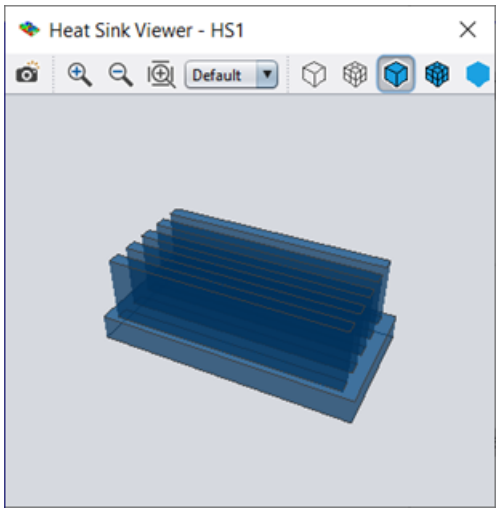
- Attachment points attach the heat sink to the PCB

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Attachment points must be fully contained by the heat sink outline
- Attachment points may not intersect or be contained by any part outline
- Attachment points may not be defined when a heat sink is under a part
- All attachment points assigned to a heat sink must have the same height
- Attachment points on heat sinks covering a part must be at least as tall as the height of the part

8.9.6. Viewing Heat Sink 3D Model

From the heat sink editor dialog, click the **View the 3D Model of the Heatsink** icon to launch a 3D representation of the heat sink as defined in the dialog. You may also right-click the heatsink in the **Layer Viewer** and select **View 3D Model**. To quickly see how changes to the heat sink properties in the **Heat Sink Editor** affect the model, click the **Refresh 3D Model** icon. Be aware, however, that you cannot view the 3D model when editing multiple heat sinks simultaneously, as their properties will vary.



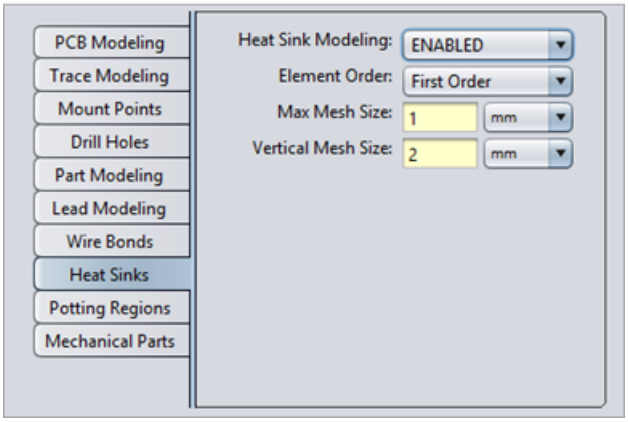
8.9.7. Heat Sink Analysis

Heat sinks can be selectively enabled/disabled in any FEA generated using any of the following menu options:

- Generate 3D Model
- Export FEA Model
- Edit Analysis Properties

In all cases, the **Heat Sinks** property tab is used to specify the following heat sink analysis properties:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Element Order** property specifies the type of FEA elements to use when modeling the heat sink. First order elements reduce analysis complexity and run time, but may not result in the most accurate results, depending on the type of material and structure of the heat sink. Second order elements use more nodes to define the heat sink, allowing it to model additional mechanical behaviors, like twisting, but also requires additional analysis time. Shell elements reduce analysis time even further by approximating the heat sink elements with special 2D elements.

The **Max Mesh Size** property specifies the desired size of each element generated by the Ansys Meshing algorithm (Bonded mesh). The setting also appllies to the legacy Sherlock meshing (Merged mesh) if you have enabled it in Mesh Settings (p. 85) and to any native FEA meshing algorithm (for Geometric models).

The **Vertical Mesh Size** property specifies the maximum vertical distance that can be spanned by a given element used to model the heat sink. If a 0 value is specified, then no limit is imposed and each element used in the heat sink model will be as tall as necessary.

Note:

The **Heat Sinks Modeling** tab is only enabled when **Part Modeling** is enabled.

8.9.7.1. Heat Sink Results

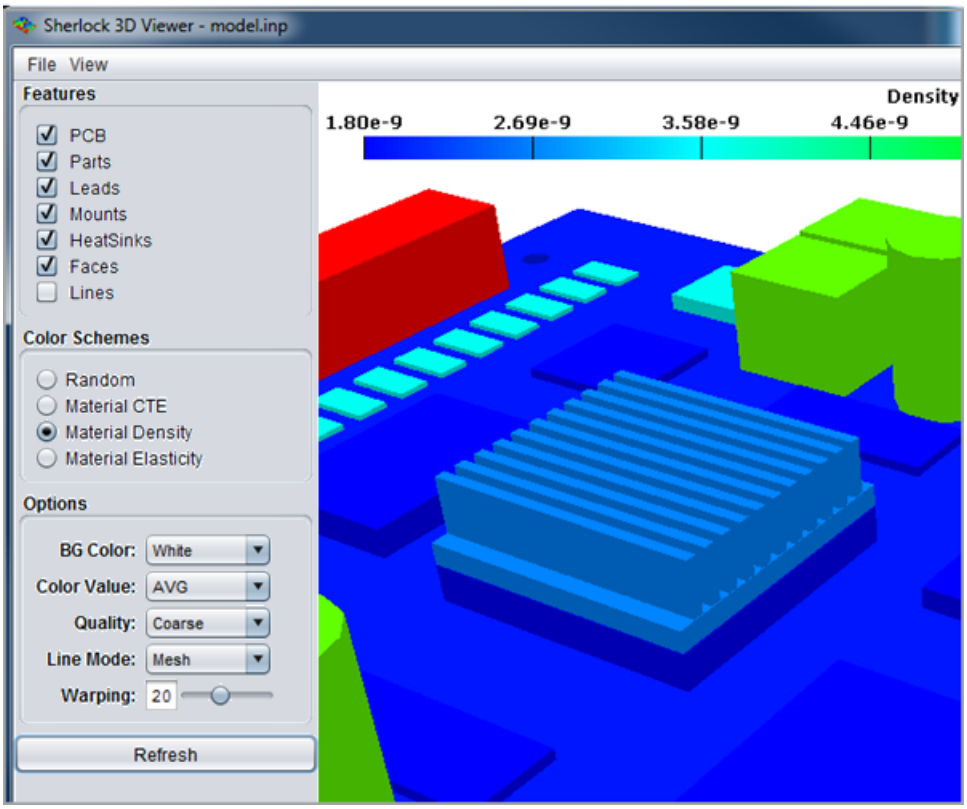
There are no specific analysis results generated by Sherlock for heat sinks. Much like mount points and mechanical parts, they are used to make the FEA model itself more accurate, thereby affecting all PCB and component results. Nonetheless, heat sink can be selectively viewed in the Sherlock **3D Viewer** as part of the FEA model and/or analysis results. For example, the following image shows a heat sink mounted on top of a large component:

Heat sinks may be selectively hidden and displayed in the **3D Viewer** by enabling or disabling **Heat Sinks** in the **Features** panel.

Note:

In the **3D Viewer**, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at 20°C only.

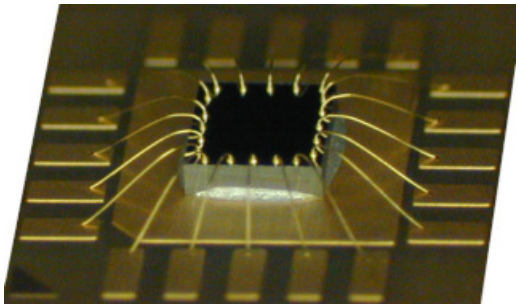
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.10. FEA- Wire Bonds

Wire bonding is a common method used to connect an integrated circuit to the pins or pads of the package in which it is contained. It is also used in some situations to make connections between a pair of ICs or between an IC and the PCB. In either case, the wires used are usually made of aluminum, copper, silver or gold. Wire diameters can be as small as 15 microns or as large as several hundred microns for high-powered applications.

Ansys Sherlock allows users to interactively define one or more wire bonds that attach to either a component or the PCB at designated locations. Users can also import/export batches of wire bond definitions via various file formats.



Note:

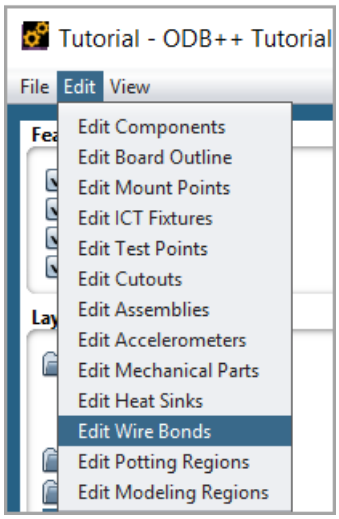
When wire bonds are enabled for FE analysis, the corresponding analysis results for mechanical shock, vibration, and ICT will each include a results table for each wire bond.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.10.1. Adding Wire Bonds
8.10.2. 3D Model Viewer
8.10.3. Editing Wire Bonds
8.10.4. Wire Bond Export
8.10.5. Wire Bond Import
8.10.6. 3D FEA Modeling
8.10.7. Wire Bond Results

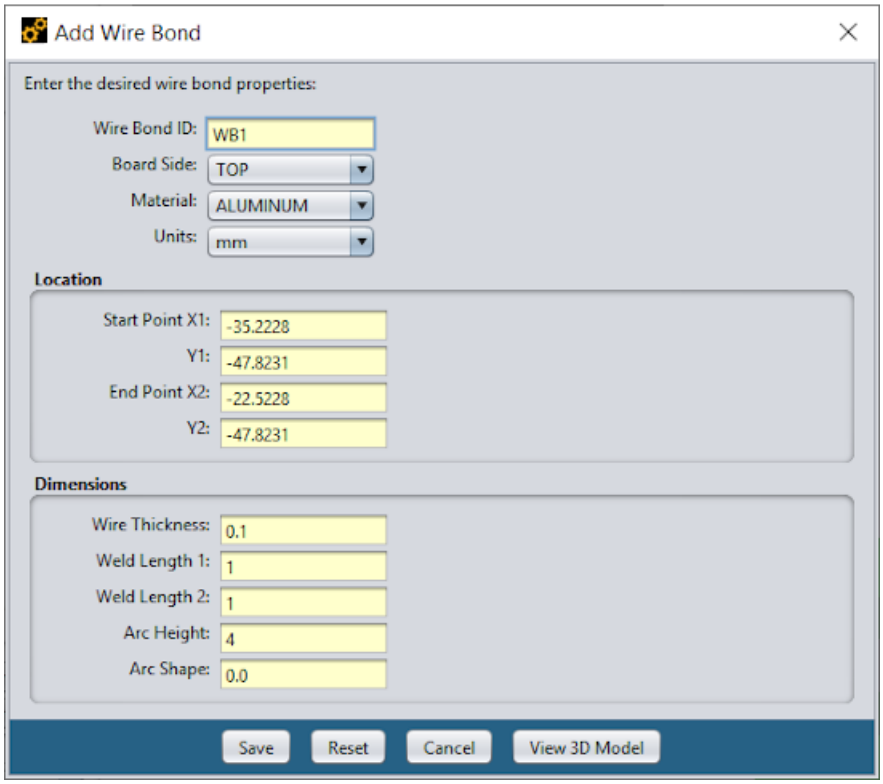
8.10.1. Adding Wire Bonds

Wire bonds are added using the **Edit > Edit Wire Bonds** menu option found in the 2D Layer Viewer main menu, accessible by double clicking **Layers** under **Inputs** in the Project Tree. When selected, the Wire Bond Editor control buttons will be displayed at the bottom of the 2D Layer Viewer panel and the **Mechanical > Wire Bonds** layer category will be enabled so that all existing wire bonds will be displayed. Wire bonds are displayed as thin dark yellow lines by default. You can change the display color assigned to wire bonds using the **Settings > General Settings > Color** menu option available in the Sherlock main menu.



To add a wire bond, right-click anywhere on the circuit card and select the **Add Wire Bond** option from the context menu. At that point, the **Add Wire Bond** dialog will be displayed, allowing you to specify the desired properties for the new wire bond. Sherlock automatically assigns a unique ID to each wire bond, but you are free to change the ID to whatever is required, provided it is still unique.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

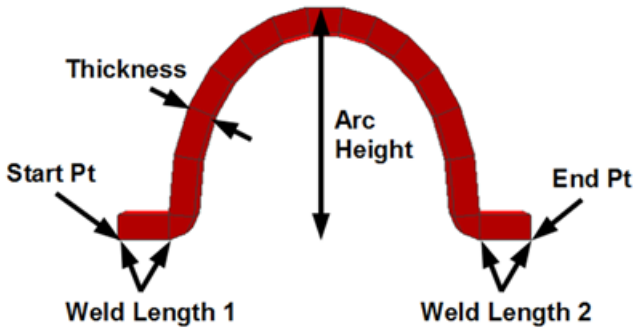


The **Board Side** property specifies the side of the board on which the wire bond is located. Sherlock will automatically determine which component(s), if any, are located under the wire bond whenever a 3D model is generated. This allows users to freely position wire bonds and components without having to maintain correlations between them.

The **Material** property specifies the type of material used for the wire bond, which is passed directly into the resulting 3D FEA model.

The **Location** panel defines the start and end points of the wire bond. Such locations are typically edited graphically in the layer viewer (more on this shortly), but you are free to enter exact locations in the dialog as well.

The **Dimensions** panel defines the relevant dimensions (illustrated below) of the wire bond. The **Wire Thickness** property defines the square cross section of the wire.



The **Weld Length** properties define the length of the foot on each end of the wire where it is connected to the underlying component or PCB.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Arc Height** property defines the height of the arc, measured from the HIGHEST foot position. For example, if the start point of the wire bond is sitting on top of a component and the end point is sitting on the PCB, then the Arc Height will be measured from the top of the component, not the PCB surface.

The **Arc Shape** property controls the shape of the Bezier curve generated by Sherlock to represent the wire as it bends from the start point to the end point (which may have different vertical coordinates), through the specified height. Bezier curves are well-suited for 3D modeling, because they are smooth enough to prevent any distortions, yet they can be controlled by a single property.

The **Arc Shape** property is specified as a numeric value between 0 and 1 that defines how pointy the arc will be, as depicted in the following images.

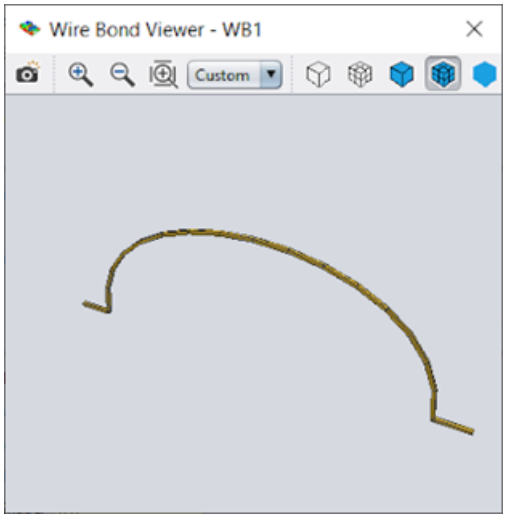
Table 8.29: Arc Shapes

		
Arc Shape = 0.0	**Arc Shape = 0.5**	**Arc Shape = 1.0**

An Arc Shape value of 0.0 results in a very full arc, that comes up almost vertically from each foot and has a wide, rounded form. An Arc Shape of 1.0 results in a narrow, almost triangular arc.

8.10.2. 3D Model Viewer

The **View 3D Model** button at the bottom of the wire bond property dialog can be used to visualize the impact of the various dimensional properties on the wire bond arc. Once pressed, the Wire Bond Viewer will be displayed, showing the 3D model for the current set of properties, assuming that each foot of the wire bond is sitting at the same vertical level.



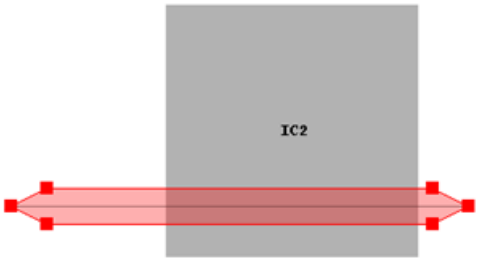
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As with other 3D viewers in Sherlock, you can pan, zoom and rotate the model as needed to examine the shape. You can also generate a snapshot image for documentation or reporting purposes.

If you change any of the property values in the property dialog, simply press the **View 3D Model** button again to update the 3D display using the new properties.

8.10.3. Editing Wire Bonds

Existing wire bonds can be edited graphically in the **2D Layer Viewer** or using the **Edit Wire Bond** dialog. In both cases, the editing process begins by left-clicking a wire bond in the **2D Layer Viewer** while holding down the SHIFT key to select the wire bond. Once selected, the wire bond will be highlighted by a red polygon with 6 control nodes, as shown in this image.



To change the location of either end point, simply left-click any of the control nodes and drag them to the desired location. The normal pan and zoom features provided by the **2D Layer Viewer** can be used at any time to more precisely position the wire bond end points as needed. To change the location of the wire bond without changing its overall length, left-click anywhere inside of the highlighted region and drag the entire wire bond to the desired location.

You can right-click anywhere in the selected wire bond region to display the following additional editing options:

- Edit Properties
- Copy Wire Bond
- Delete Wire Bond

The **Edit Properties** menu option displays the Edit Wire Bond dialog that is identical to the Add Wire Bond dialog previously discussed. The Edit Wire Bond dialog allows you to modify any of the properties associated with the wire bond.

The **Copy Wire Bond** menu option immediately copies the selected wire bond to a nearby location and selects the new wire bond for editing. Typically, the wire bond is then dragged to the proper location and the process is repeated for all the wire bonds needed by a given component. In this way, users can quickly define a set of identical wire bonds that vary only in their location.

The **Delete Wire Bond** menu option is used to delete an existing wire bond.

As with most other Sherlock 2D Layer editors, multiple wire bonds can be selected by SHIFT-left-clicking on the circuit card and dragging a rectangle around the desired wire bonds. Individual wire bonds can then be added to or removed from the selected set using SHIFT-left-click. With multiple

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wire bonds selected, the **Copy Wire Bonds** and **Delete Wire Bonds** context menu options can be used to copy or delete all of the wire bonds in the selected set.

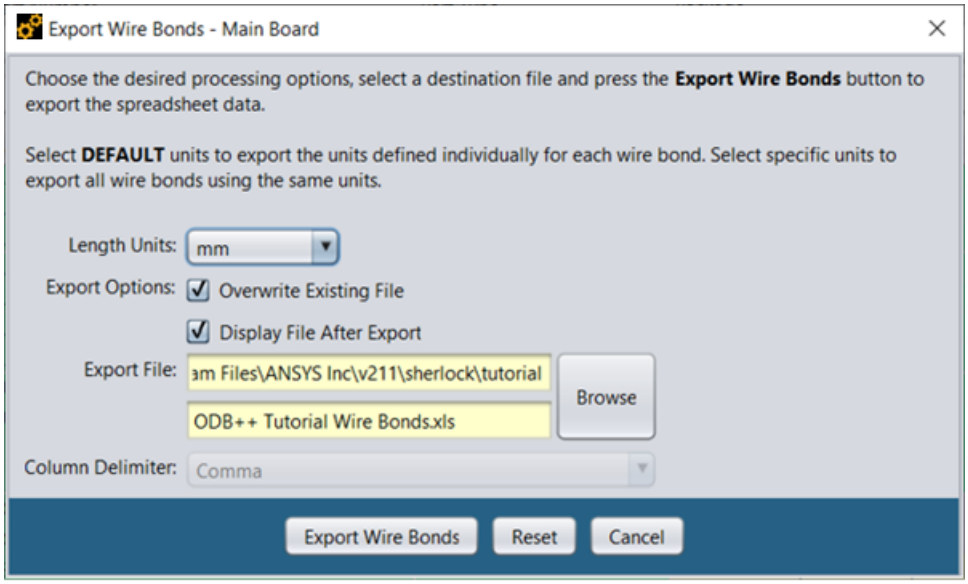
8.10.4. Wire Bond Export

In addition to the manual / graphical process discussed above, a batch of wire bond definitions can be imported from or exported to a spreadsheet or CSV file. This can significantly reduce the time required for data input in situations where the wire bond locations are known. The combination of the export and import capabilities can also be used to facilitate bulk editing needs, such as changing the material property for all wire bonds or just a subset.

All of the currently defined wire bonds can be exported to a spreadsheet or CSV file by selecting the **CCA > Export Wire Bonds** option from the Sherlock main menu. At that point, the **Export Wire Bonds** dialog (belowf) will be displayed, allowing you to specify the name, type and location of the file to be exported.

The following file formats are supported:

- Excel Spreadsheet (*.xls)
- Excel Spreadsheet XML (*.xlsx)
- Comma Separated Value (*.csv)



The **Length Units** property allows you to specify the units to be used for all wire bond properties, regardless of the units used in Sherlock to define those properties.

The **Overwrite Existing File** option indicates that any existing file with the same name should be overridden without further prompting.

The **Display File After Export** option specifies that the Windows application associated with the selected file format should be launched right after the file has been exported.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wire bond definitions are imported by adding a spreadsheet or CSV data file to the CCA and assigning one of the following file types to that file:

- Wire Bond (Excel)
- Wire Bond (CSV)

8.10.5. Wire Bond Import

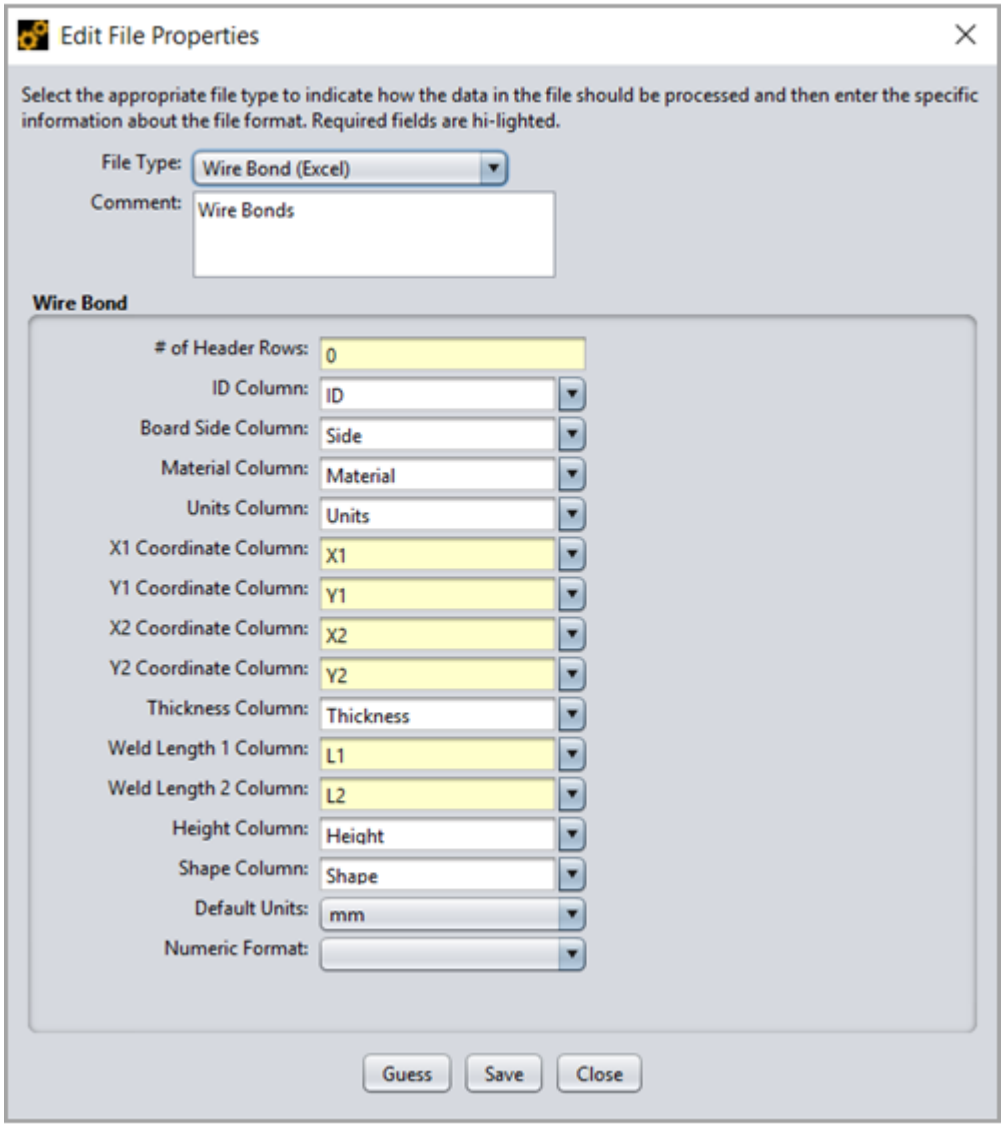
To add a file to the CCA, either select **CCA > Add Files** from the main Sherlock menu, or right click **Files** and select **Add File(s)** from the **Project Tree**. To assign a file type, open the **File Properties** editor (below) by right clicking the file, selecting **Edit Properties**, and then toggling the **File Type** drop down. With either file type assigned, a set of file parsing properties can then be used to designate the columns containing the desired data. Each column must begin with a text value containing the column name. Based on the column names, Sherlock will automatically attempt to match the appropriate column in the spreadsheet or CSV file with each wire bond property. Nonetheless, you are free to enter or select the proper column name for each wire bond property.

Once all of the required column names have been specified, press the **Save** button to parse the wire bond definitions and import them into Sherlock.

Note:

All wire bond definitions found in the file will be added to any existing set of wire bonds. If you want to overwrite the existing wire bonds, you should delete all the existing wire bonds in the **2D Layer Viewer** before importing from the file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.10.6. 3D FEA Modeling

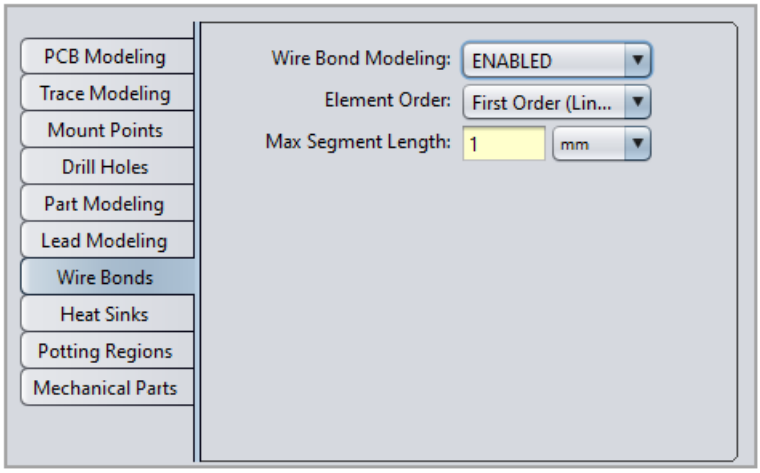
Wire bonds can be selectively enabled or disabled in the 3D FEA models created by each of the following operations:

- Generate 3D Model
- FE analysis
- Export FEA Model

In each case, the **Wire Bonds** tab (below) is used to specify the desired wire bond modeling properties.

When enabled, 3D models are created for each wire bond. Each wire bond foot is bonded to the underlying component or PCB elements, depending on the location of the wire bond and **Part Modeling** properties.

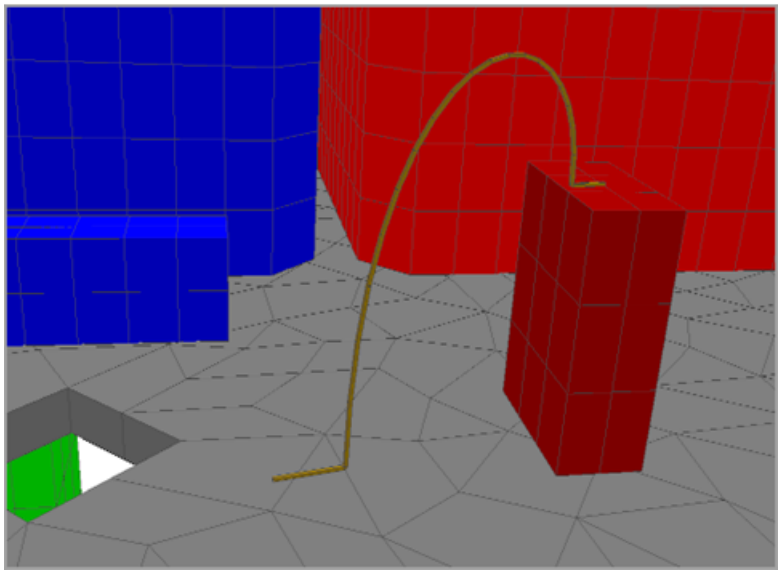
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Element Order** property specifies the type of element to be used in the FEA tool for the wire bond. First Order elements are bricks specified by 8 nodes, while Second Order elements are bricks specified by 20 nodes.

The **Max Segment Length** property specifies the maximum linear distance covered by each arc segment. A shorter length results in a smoother arc, but also increases the number of elements created for each arc, which increases analysis time.

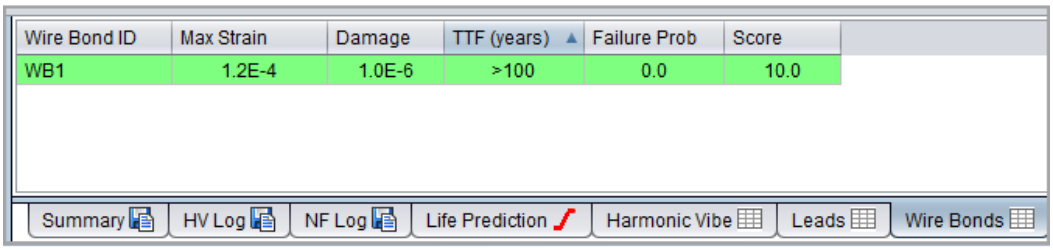
Sherlock automatically determines the proper Bezier arc for each wire bond based on the wire bond properties and the underlying component or PCB surfaces. As shown below, the arc will smoothly transition from one end point, through the maximum height, to the other end point. Proper specification of the foot length, thickness, arc height and arc shape properties allow for a wide range of different wire bonds to be effectively modeled.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.10.7. Wire Bond Results

When wire bonds are enabled for FE analysis, the corresponding analysis results for mechanical shock, vibration, ICT, and thermal mechanical will each include a results table for each wire bond. For vibration analysis, users may select one or more wire bonds and right-click to view the life prediction chart for the selected wire bonds.



8.11. FEA- Potting & Staking

Potting and staking is added to a circuit card to protect and/or support the circuit card and/or components. The resins used for such purposes can change the mechanical properties of the circuit card significantly and, as such, should be considered when running any FE analysis. To that end, Ansys Sherlock allows users to define one or more **Potting Regions** that represent a 3D volume on the board that is filled with a specified material. Sherlock will automatically fill the volume around all components that are either contained by or intersected by the potting region. Once defined, potting regions can be selectively included in any 3D model or FEA test generated by Sherlock to determine their effect on the mechanical properties of the board.

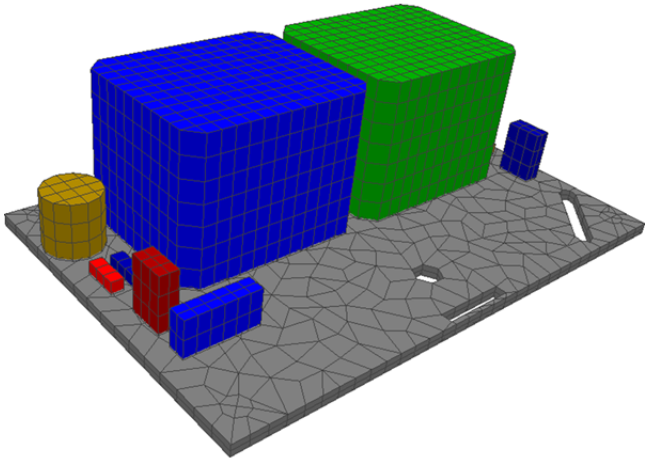
In this section, the following topics are covered:

8.11.1. Tutorial Project
8.11.2. 2D Modeling
8.11.3. PCB Shape
8.11.4. Rectangular Shape
8.11.5. Circular Shape
8.11.6. Slot Shape
8.11.7. Polygonal Shape
8.11.8. 3D Modeling
8.11.9. Potting and Part Elements
8.11.10. Potting and Lead Elements
8.11.11. Potting & Staking Examples

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

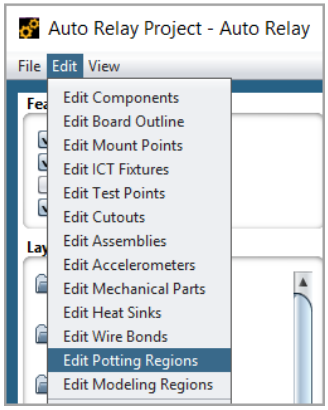
8.11.1. Tutorial Project

Throughout this section, we will use a relatively simple, but real-world, PCB example to show how potting regions and staking are used. All the design files for the PCB can be found in the **Auto Relay Project.zip** file located in the **tutorial** folder found in the **Sherlock Installation Directory**. That ZIP file can be imported as a Sherlock Project for training purposes. As shown here, the simple circuit card consists of two layers with a handful of through-hole parts, including some large relays, a couple LEDs (represented by the circular drums), some resistors and diodes mounted horizontally and 3 pin headers (represented by the tall rectangular shapes).



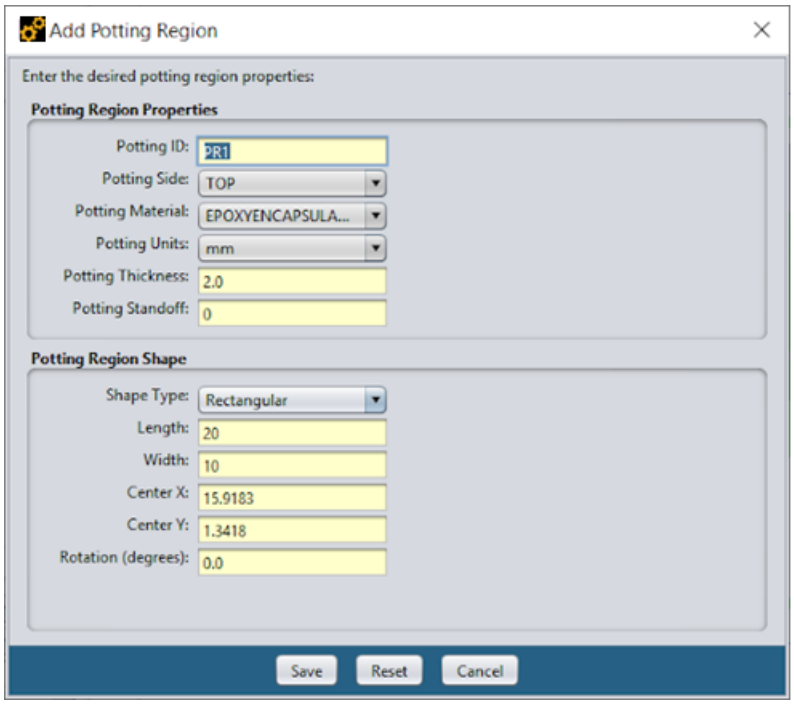
8.11.2. 2D Modeling

Potting regions are defined in the 2D Layer Viewer by selecting the **Edit > Edit Potting Regions** option from the main menu. At that point the **Potting Editor** control buttons will be displayed at the bottom of the Layer Viewer and you can add, modify or delete potting region definitions. When the Potting Editor is activated, it automatically enables the display of all top and bottom potting regions, as indicated in the **Potting Regions** section of the **Layers** panel. You can selectively enable/disable the top and bottom potting groups as needed while editing.



To add a potting region, right-click anywhere on the circuit card and select **Add Potting Region** from the context menu. At that point, the **Add Potting Region** dialog will be displayed, allowing you to specify the desired properties for the potting region.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Potting regions are assigned unique **IDs** to help identify them when exporting the 3D model to FEA tools for custom analysis. Sherlock will automatically assign a unique identifier, but you are free to use whatever ID you like.

The **Potting Side** property specifies the circuit card side on which the potting region is located.

The **Potting Material** property specifies the type of material used to fill the potting region. Only materials marked **POTTING** in the **Material Manager** are available for selection. See the Material Management (p. 153)User Guide section for more details.

The **Potting Standoff** and **Potting Thickness** properties define the vertical base and top of the potting region, measured from the circuit card surface. If **Potting Standoff** is non-zero, the potting region would normally be called staking between two or more parts.

The **Potting Region Shape** panel allows you to specify the shape of the potting region outline, which includes the following choices:

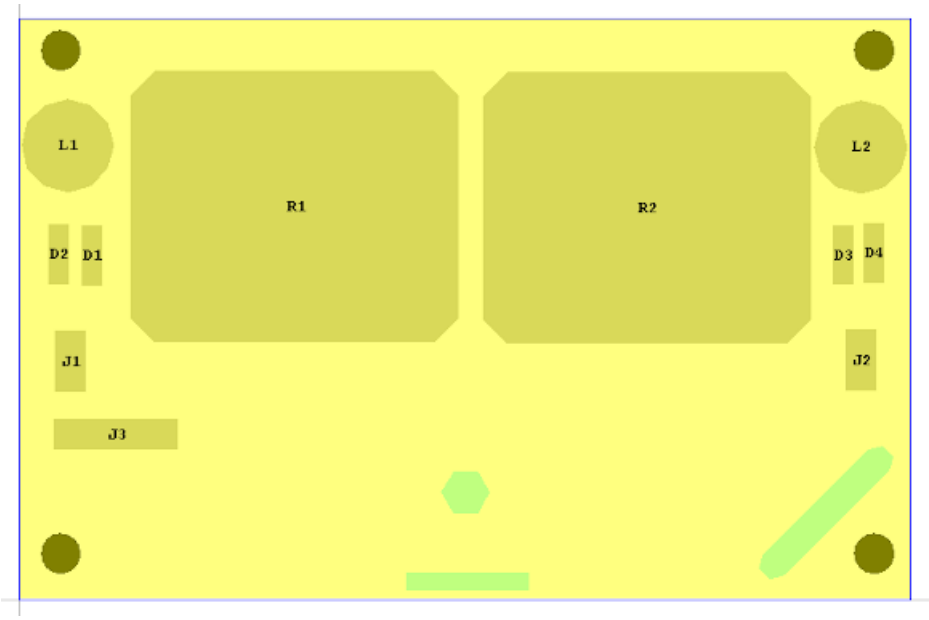
- Polygonal
- Rectangular
- Slot
- Circular
- PCB

These shape selections allow you to easily define a wide variety of potting region outlines. You can freely switch between the shape types, although some details will naturally be lost when converting from certain shape types to other shape types.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

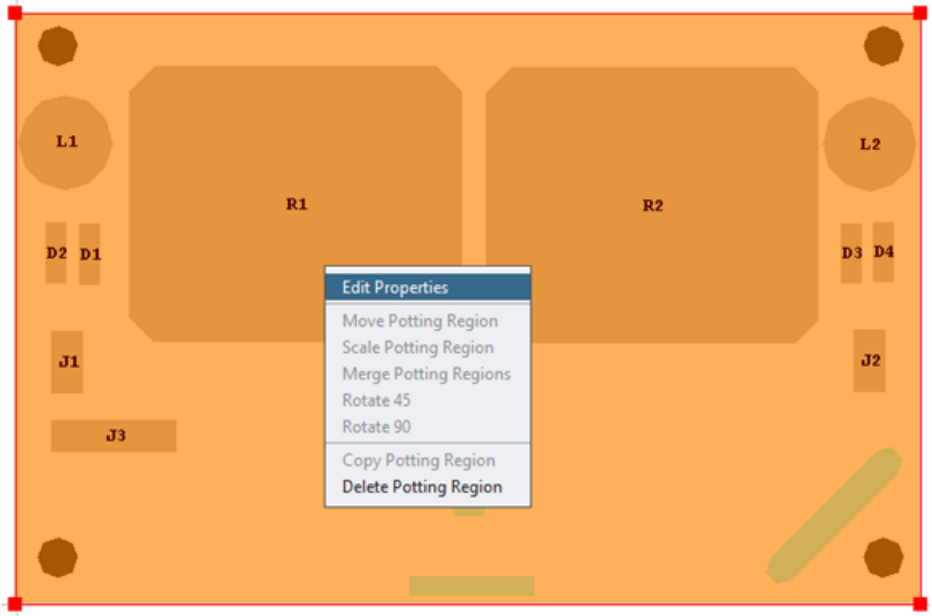
8.11.3. PCB Shape

In the example shown previously, (p. 617) the PCB shape has been chosen, which requires no other shape properties to be defined. The PCB shape always matches the exact shape of the currently defined PCB outline, as can be seen here:



Potting regions are displayed as semi-transparent yellow areas in the **Sherlock 2D Layer Viewer**. In this case, the potting region extends across the entire PCB, covering or enveloping all parts on the top of the circuit card.

We can change the shape and/or size of the potting region at any time in the Potting Editor by right-clicking the mouse inside of the potting region and selecting the **Edit Properties** menu item. At that point, the potting region will be highlighted as a semi-transparent red area, with red squares displayed at each node along the outline.

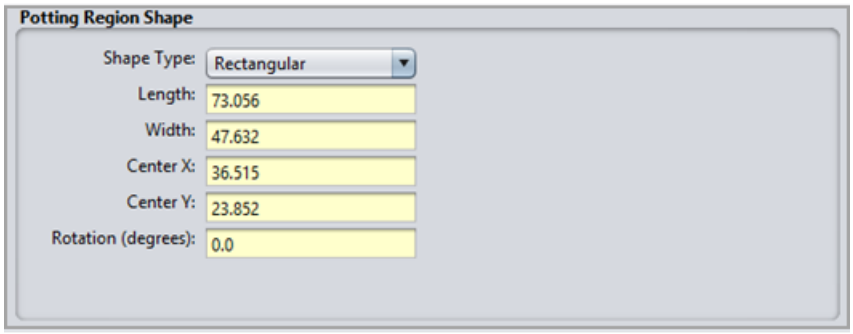


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

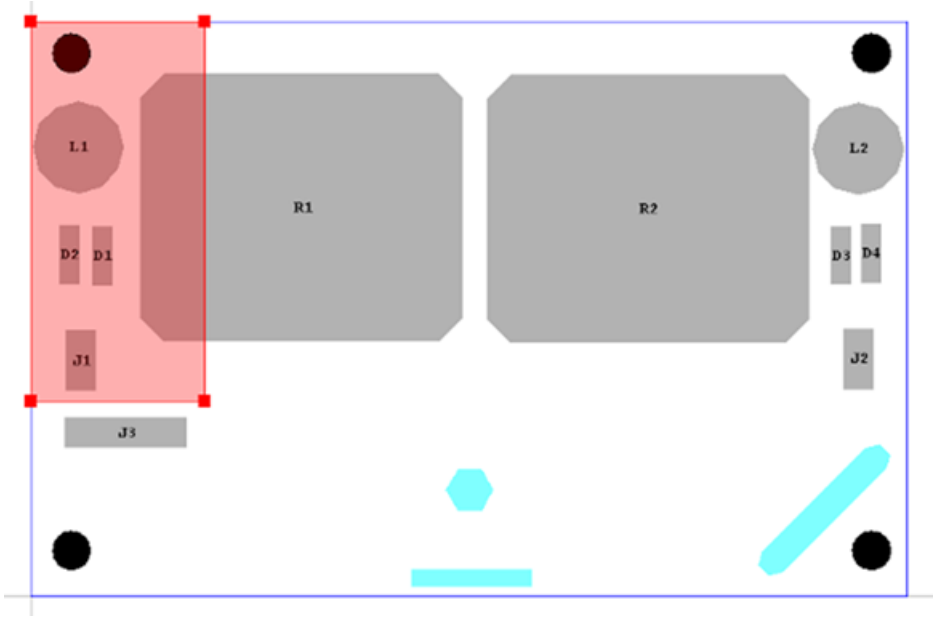
Notice in this case that the menu options used to change the size, location or orientation of the potting region are disabled. Similarly, the square control nodes displayed along the outline, which are normally used to drag or scale the outline nodes are also disabled. This is because the PCB shape always takes on the shape of the PCB outline itself and therefore cannot be altered.

8.11.4. Rectangular Shape

To change the shape of the potting region, we can choose **Rectangular** in the **Edit Potting Region** dialog and then define the appropriate shape properties. When converting from one shape to another, Sherlock automatically determines the most appropriate dimensions for the new shape based on the dimensions of the old shape.



When the changes are saved, the outline of the potting region can now be changed inside of the **Potting Editor** by dragging the square red nodes using the mouse because the shape is not PCB anymore. Since a **Rectangular** shape was selected, dragging any of the four corner nodes will result in the rectangular being stretched or contracted in the given direction, keeping the opposite corner fixed in place.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

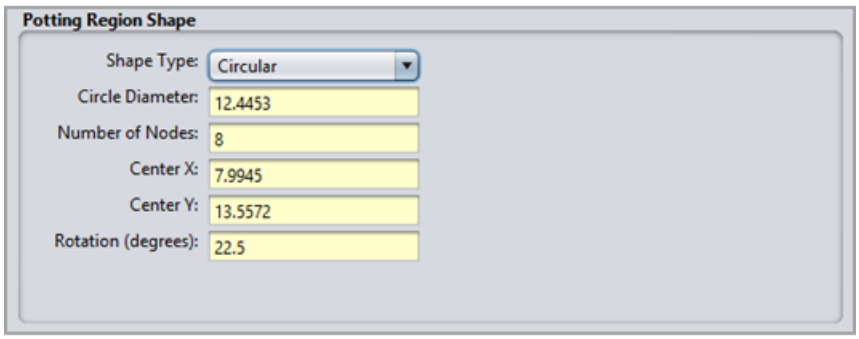
Through a combination of dragging and scaling operations using the mouse, we can easily re-define the potting region to cover only a portion of the circuit card.

Note:

Rectangular shapes can be rotated by an arbitrary angle to model a desired orientation. When the square red outline nodes are dragged with the mouse, the rectangle will retain the proper orientation.

8.11.5. Circular Shape

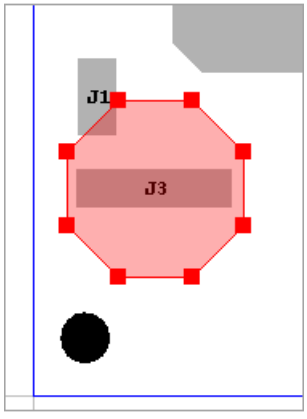
The **Circular** shape can be used to easily model circular potting regions, where you control the number of segments used to represent the circle. The **Circle Diameter** defines the diameter using the units specified by the **Potting Units** property in the top dialog panel.



The **Number of Nodes** specifies the desired number of line segments and nodes to be used to represent the circle, allowing you to define any regular N-gon.

The **Center X** and **Center Y** properties define the location of the circular region, while the **Rotation** property defines the orientation of the circular region in degrees.

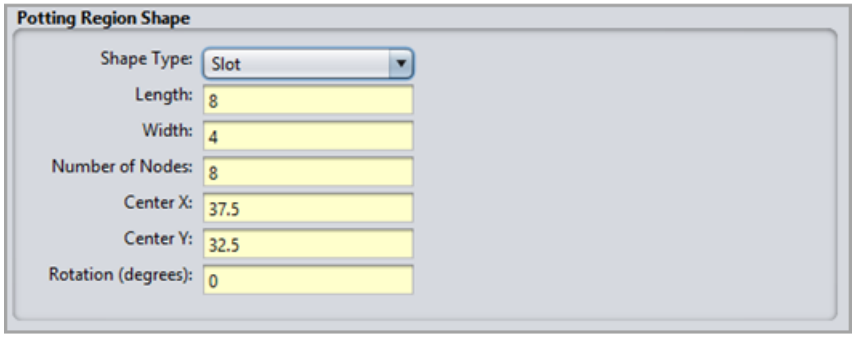
The **Rotation** property allows you to precisely position the nodes used to define the polygon. For example, the octagon shown below has been rotated by 22.5 degrees so that the sides of the octagon are parallel to the X and Y axes.



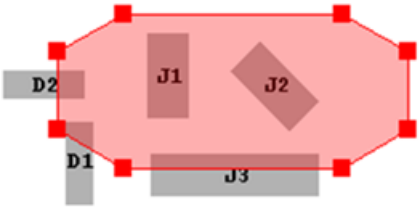
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.11.6. Slot Shape

The **Slot** shape is a combination of the rectangular and circular shapes. The shape is formed by two parallel lines along it's length, with rounded end caps.



The **Length** and **Width** properties define the overall size of the shape. The **Number of Nodes** defines the total number of nodes used to draw both end caps and must be an even number. The **Center X**, **Center Y** and **Rotation** properties define the location and orientation of the shape.

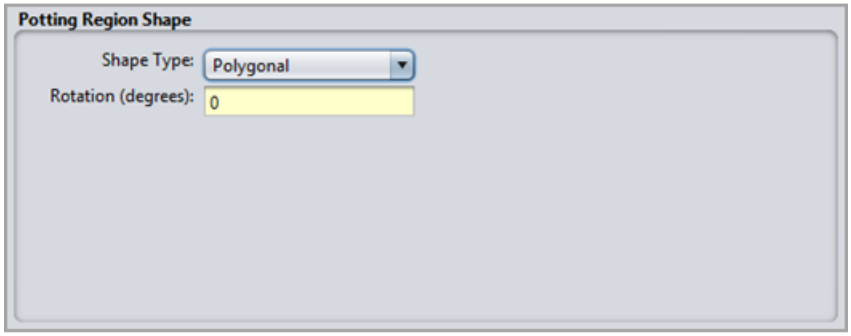


When editing a slot shape, left-click in the center of the shape to drag the shape to a new location, maintaining the current rotation. You can also left-click and drag any of the red squares to change the size of the slot shape. If you drag any of the four nodes closest to the center of the slot, the shape will be modified like a rectangle with the opposite corner fixed in place and the adjacent sides scaled accordingly. If you drag any of the other nodes, the slot shape will be modified like a circle, with the center fixed in place and all sides scaled to match the mouse motion. With a little practice you'll be able to quickly scale slot shapes to model many different potting situations.

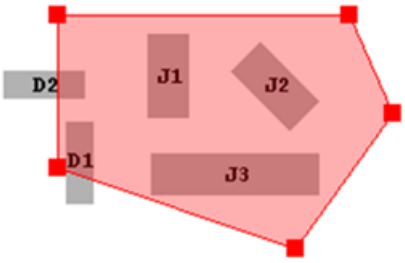
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.11.7. Polygonal Shape

The **Polygonal** shape allows you to specify arbitrary potting regions defined by a set of 3 or more nodes. The only property used to modify a polygonal shape is the **Rotation** property, which allows you to easily change the orientation.



The graphical editing of polygonal shapes differs from the other geometric shapes discussed above in that when you drag a selected node, only that node location is modified, all other nodes in the polygon remain fixed in place.



Nodes can be added or removed from the polygon using the following interactions:

- Left-click any node in the polygon while holding the SHIFT key to insert a new node halfway between the selected node and the next node in the polygon. The newly inserted node can then be dragged to wherever it is needed to define the proper shape.
- Left-click any node in the polygon while hold the CONTROL key to delete the selected node. Any attempt to delete the last 3 nodes in the polygon will be silently ignored.

Note:

Ansys Sherlock automatically detects if you attempt to drag a node to a location that would create a self-intersecting polygon. In such cases, the drag operation is silently ignored.

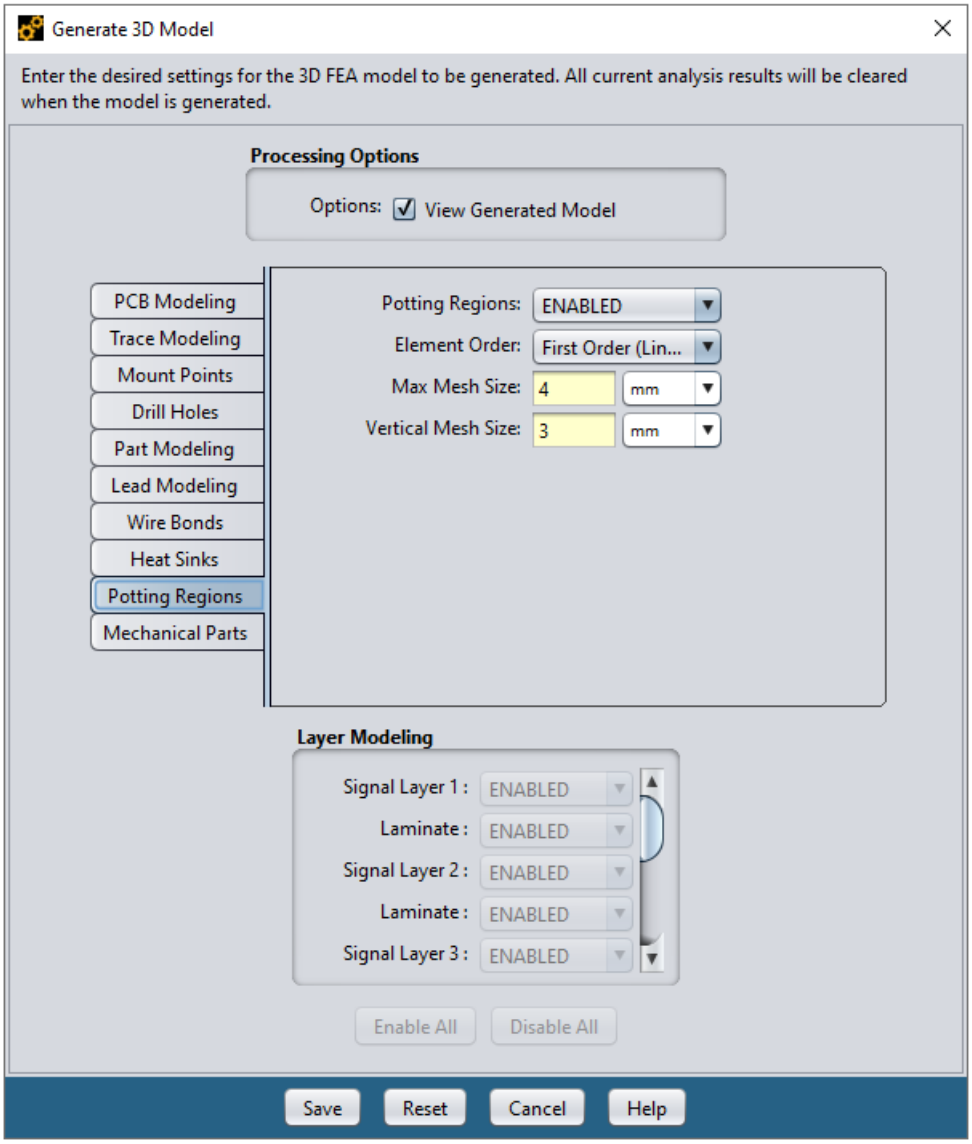
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.11.8. 3D Modeling

With one or more potting regions defined in the **2D Layer Viewer** you can selectively enable or disable the inclusion of all potting regions when performing any of the following operations:

- Generating a 3D model (by right-clicking any FE analysis in the **Project Tree**)
- Running any FE analysis task
- Exporting an FEA model

In all such cases, the **Potting Regions** tab is used to enable/disable potting region modeling and to specify the type and size of elements to be generated (when appropriate).



Note:

The **Potting Regions** tab is only enabled if the Mesh Type property in the **PCB Modeling** tab is set to **Bonded**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Element Order** property specifies the type of element to be used by the FEA tool when modeling the potting region.

The **Max Mesh Size** and **Vertical Mesh Size** properties specify the desired horizontal and vertical mesh dimensions for all elements used to model the potting region. The exact interpretation of these properties depends on the FEA tool being used.

8.11.9. Potting and Part Elements

When generating 3D elements for potting regions, Sherlock automatically determines all components overlapped by any potting region and tailors the potting region elements accordingly. Specifically, Sherlock automatically deals with all of the following situations:

- If a component is completely contained by or overlaps the potting region, then potting elements will be created around all appropriate sides of the component.
- If the top of a component is below the top of the potting region, then potting elements will be created above the component up to the top of the potting region.
- If the bottom of a component is above the bottom of the potting region, then potting elements will be created below the bottom of the component down to the bottom of the potting region.

To reduce the number of TIE constraints required to bond potting region elements and potted part elements, Sherlock generates a single horizontal mesh for all potting regions and potted parts that touch each other, allowing potting and part elements to share nodes in the horizontal plane. The mesh is then stratified in the vertical dimension, based on the vertical base and top of each potting region and part. This vertical stratification allows the potting and part elements to share nodes in the vertical direction as well, bonding all potting regions and parts to each other without any TIE constraints. This usually reduces the time needed for analysis. The only TIE constraints generated for the potting or part elements are those needed to bond them to the PCB.

Note:

Part Modeling Restrictions in a Potting Region: There are a couple restrictions regarding how the Corner Face property is used when modeling parts in a potting region. If the Corner Face property is *not* TOP_BOTTOM and leads are *not* being modeled, then the Corner Face and Corner Shape properties are ignored, and the part is modeled as a SQUARE shape. This is necessary because the potting region and parts are meshed at the same time in the XY plane and extruded in the Z dimension. See Corner Face Restrictions (p. 560).

8.11.10. Potting and Lead Elements

If lead modeling is enabled for one or more potted parts, the lead elements will be included in the 3D model, but they may coexist with potting elements in the same 3D space. This approach is taken because:

a. Lead elements tend to be relatively small, which increases the complexity of the mesh generated for potting regions and potted parts.

b. The rounded corners of various leads, like DIP and Gullwing, would cause very small slabs to be created in the vertical direction across the entire potting region.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

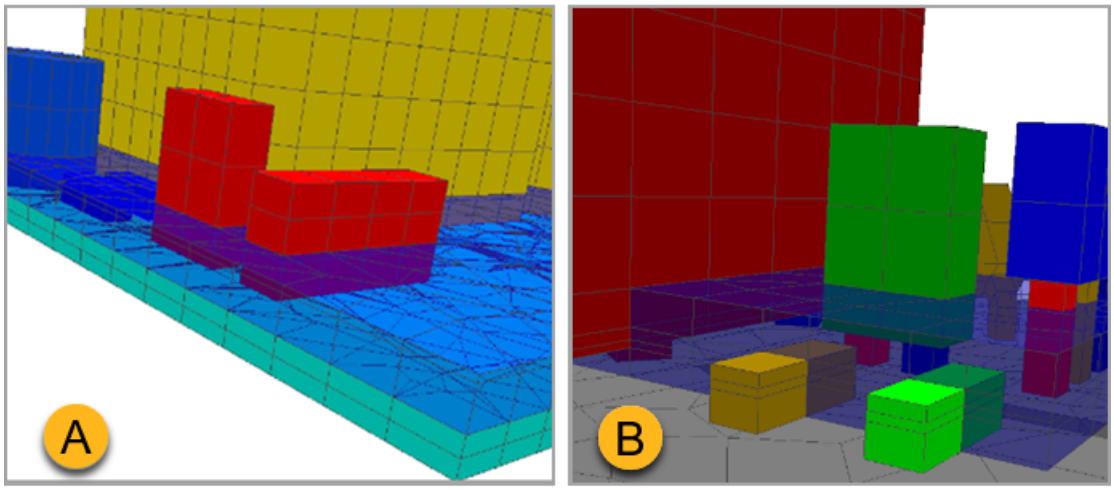
c. The bonds between the potting, part and PCB elements are far more significant in determining the overall mechanical properties of the circuit card than any bonds between the potting region and lead elements.

In the section above Potting and Part Elements (p. 625), see the note **Part Modeling Restrictions in a Potting Region** which explains how the choice *not* to model leads affects part modeling.

8.11.11. Potting & Staking Examples

Despite their name, potting regions in Sherlock can be used to model several different potting and staking situations commonly found on circuit cards. In this section, we present just a few of the most common situations and how they are modeled in Sherlock.

One of the most common examples is when a resin is poured over the entire board to a given thickness. In Sherlock, that is easily modeled as a single potting region with a PCB shape, a desired thickness and a standoff value of 0. As shown here (A), Sherlock creates potting elements that flow around, under and over all parts.



When viewed using the **Sherlock 3D Viewer**, potting elements are displayed using a semi-transparent color so that other features are still visible. In this specific example, the colors used are based on the plasticity of the various elements.

Note:

You can selectively enable/disable the display of potting and/or part elements in the **Sherlock 3D Viewer** to better view them.

Another common form of potting adds resin to only a portion of the PCB surface, which may completely or partially contain one or more parts. In this example, a rectangular potting region partially covers a couple surface mounted parts, completely contains a few leaded parts and flows under a larger leaded part.

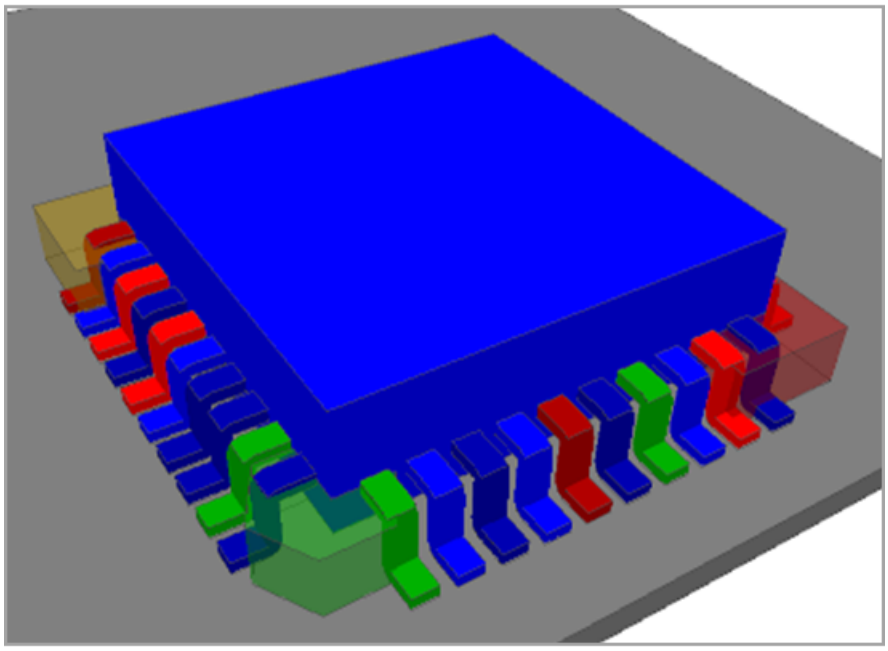
The image above (B) shows the vertical stratification used for the potting region and all potted parts. The vertical stratification is created using the bottom and top of all the potted parts, taking lead standoff into account.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The use of one or more potting regions to secure a large part to the PCB is typically called staking. Such situations are modeled in Sherlock by defining one or more potting regions, using whatever properties are appropriate for each staking area.

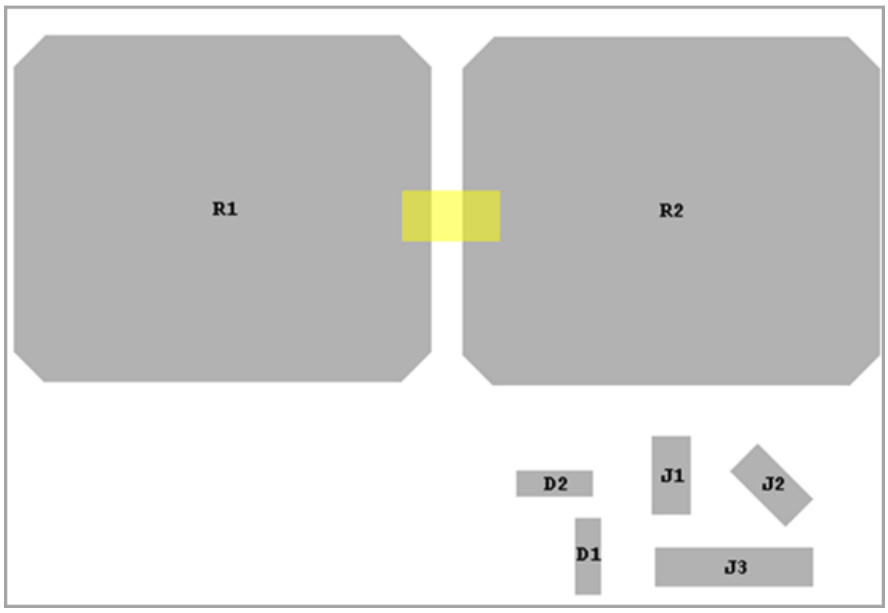
In the example below, the large leaded part is secured to the PCB using two rectangular and two hexagonal potting regions in each of the IC corners. The potting elements flow under the IC and co-exist with nearby lead elements.

The image was generated directly by the **Sherlock 3D Viewer**, using a random color scheme for all features.

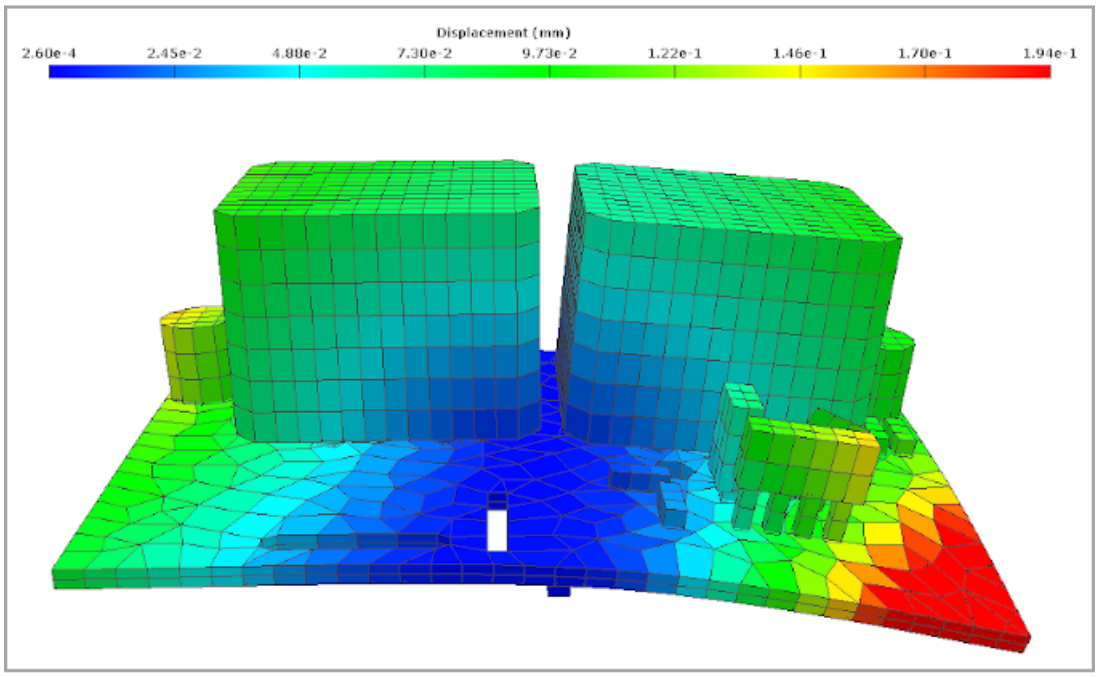


All the examples shown so far used potting regions with a standoff value of zero. In other words, the potting region sits directly on the PCB and flows under any leaded part. If the standoff value is greater than zero, the potting region will float above the PCB, bonded only to adjacent parts. For example, consider the following potting region defined across two large relays. In this case, the relays are 20 mm tall and with a lead standoff of 1 mm. The potting region has a standoff value of 14 mm and a thickness of 8 mm. In other words, the staking material sits 1 mm above the top of the relays and extends down between the relays another 7 mm.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

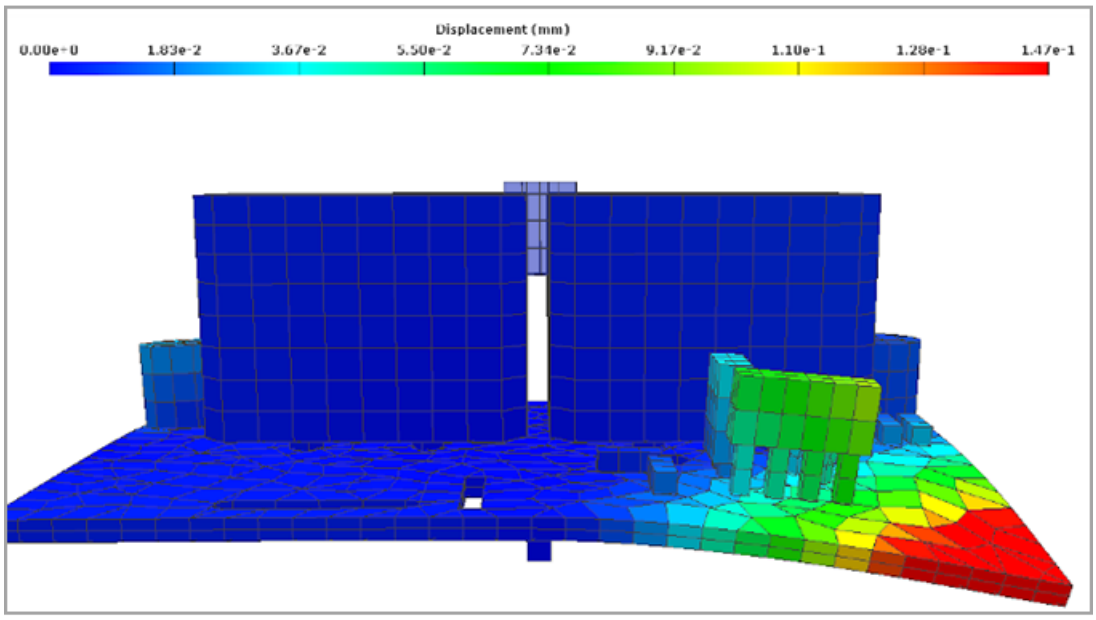


The following images show ICT results for a static load, with and without potting modeling enabled. Without staking, the static force on the right side of the board causes the gap to widen between the two relays.



With staking between the relays, the gap between the relays stays basically the same (depends on the mechanical properties of the potting region material). Note how the staking elements sit above and between the parts. With a smaller standoff value for the potting region, the staking elements would only be between the two relays.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.12. FEA- Mechanical Parts

Ansys Sherlock provides the ability to create mechanical parts that are placed on a circuit card and used during FE analysis. Mechanical parts are defined by a 2D shape definition and extruded to 3D with the definition of a given height for a part. Mechanical parts may be circular, rectangular, or any polygonal shape. Mechanical part definitions also include a material definition and may be used when generating the FEA model.

In this section, the following topics are covered:

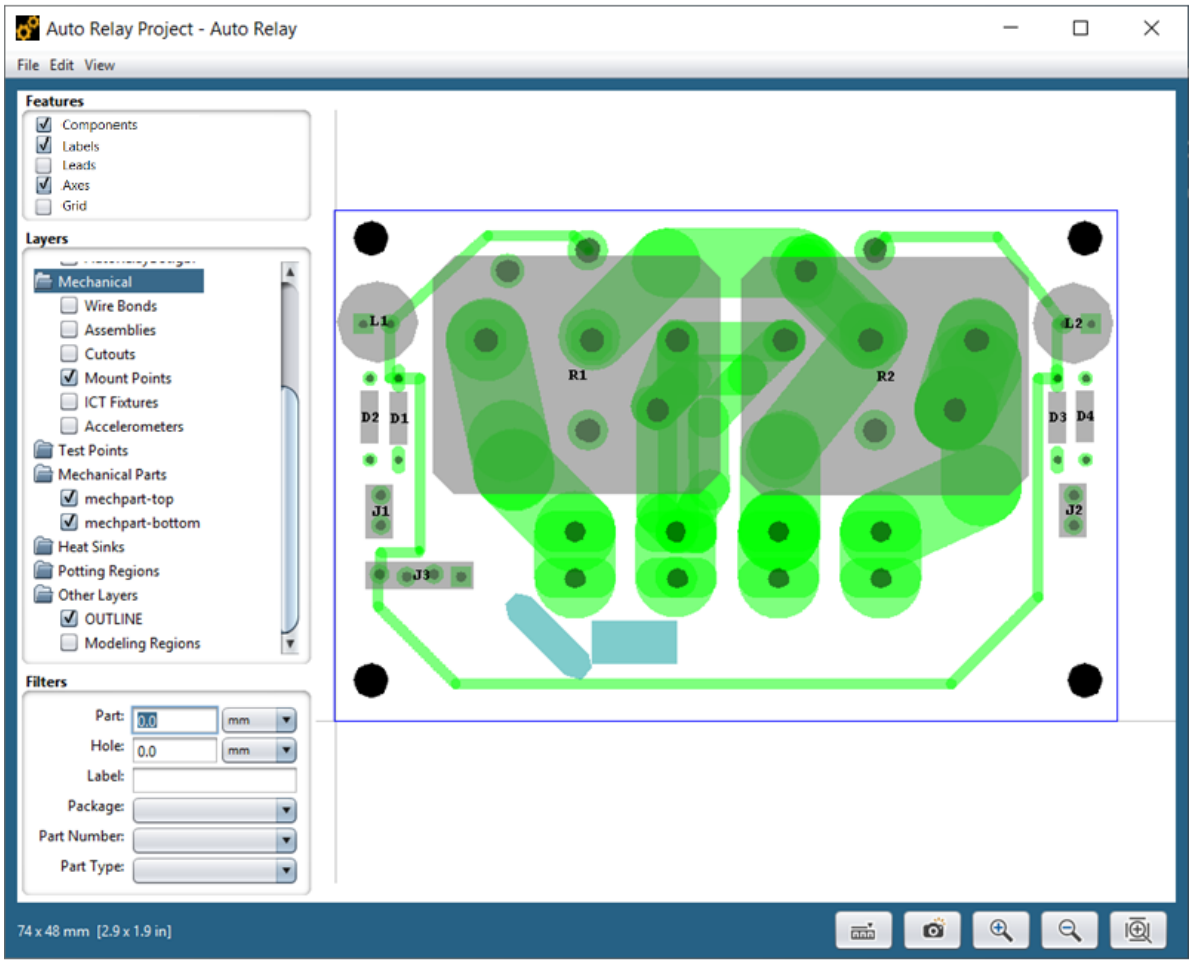
8.12.1. Mechanical Part Viewing
8.12.2. Mechanical Part Editing
8.12.3. Mechanical Part Properties
8.12.4. Mechanical Part Multi-Edit
8.12.5. Mechanical Part Attachment Points
8.12.6. FEA Model

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.12.1. Mechanical Part Viewing

Mechanical parts can be viewed in the Layer Viewer by opening the **Mechanical Parts** folder in the **Layers** control panel and selecting the desired mechanical part layer (top or bottom) to show. As shown in the example below, mechanical parts are displayed using the medium cyan color and they can take on a variety of shapes and orientations.

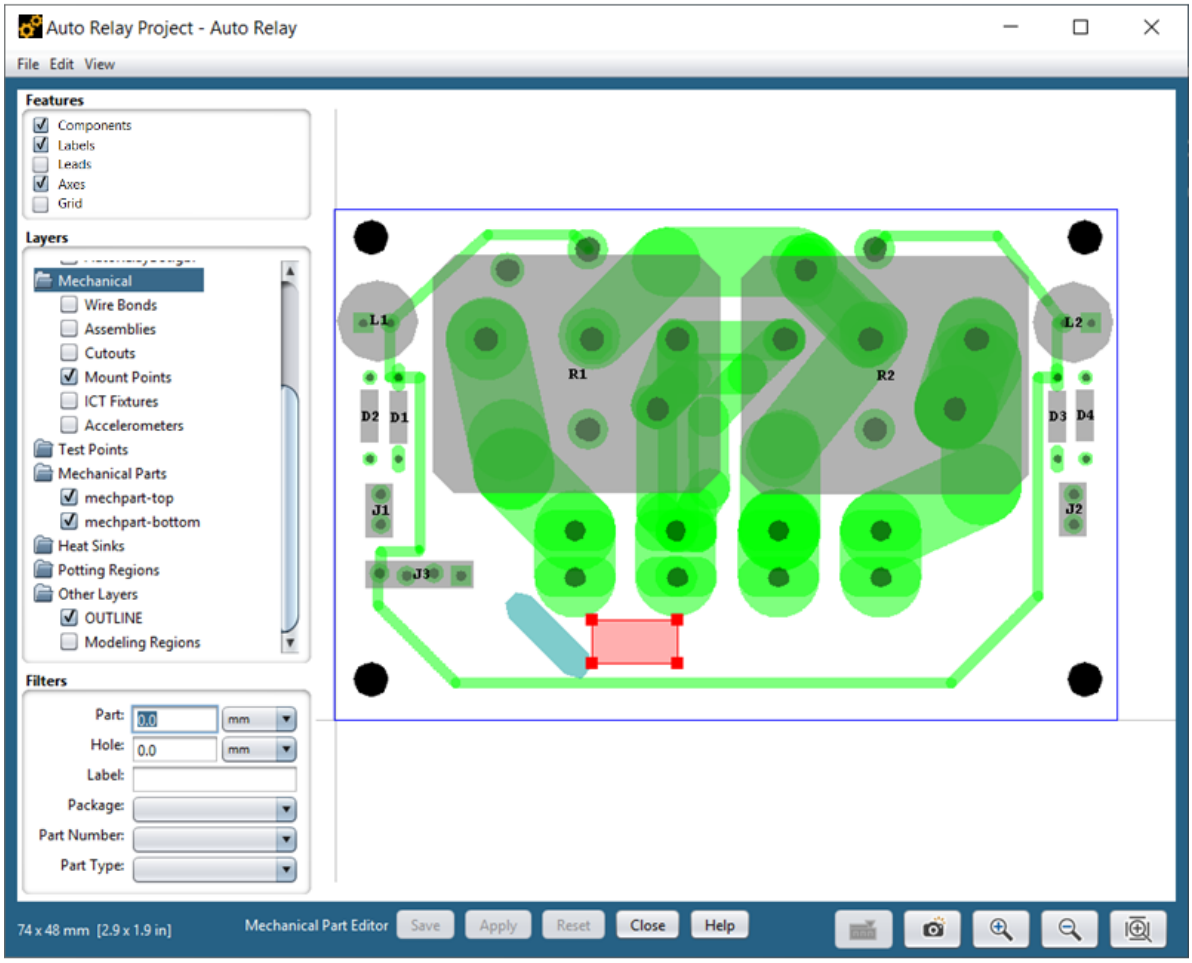
In this example, there is a rectangular part at bottom-center, and a slot shape part rotated 45 degrees just in the lower-right section of the PCB.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.12.2. Mechanical Part Editing

Mechanical parts can be added, deleted and edited using the **Edit > Edit Mechanical Parts** menu option available in the **Layer Viewer** to enter the **Mechanical Part Editor** mode.



When the **Mechanical Part Editor** mode is enabled, the following mouse controls can be used to select or modify mechanical parts:

- Shift-Left-click any mechanical part to select only that mechanical part
- Shift-Left-click and drag to select one or more mechanical parts within a rectangular region
- Control-Left-click to add/remove a mechanical part from the selection set

Selected parts will be displayed with a red border and control points. Once selected, mechanical parts can be moved to a new location by left-clicking inside any of the selected parts and dragging them to the desired location. When more than one mechanical part is selected, all parts will be translated by the same amount, thereby staying in the same relative positions.

The red control points can be used to graphically scale an individual part. The exact transformation performed depends on the shape of the part being scaled. For example, with a rectangular part, the point opposite that selected will remain fixed, while the selected control point and the two adjacent control points are scaled to match the current mouse location. On the other hand, with a circular

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

part, when any control point is dragged, the center of the part will remain fixed and the diameter of the part will be scaled to match the current mouse location. Such an approach makes it easier to translate and scale mechanical parts as needed to graphically place them in a desired location and orientation relative to other PCB features.

In addition to the mouse actions used to select mechanical parts, the following mouse actions can be used to add new mechanical parts or edit properties for existing mechanical parts:

- Right-click anywhere in the PCB to add a new mechanical part
- Right-click a mechanical part to display the context menu

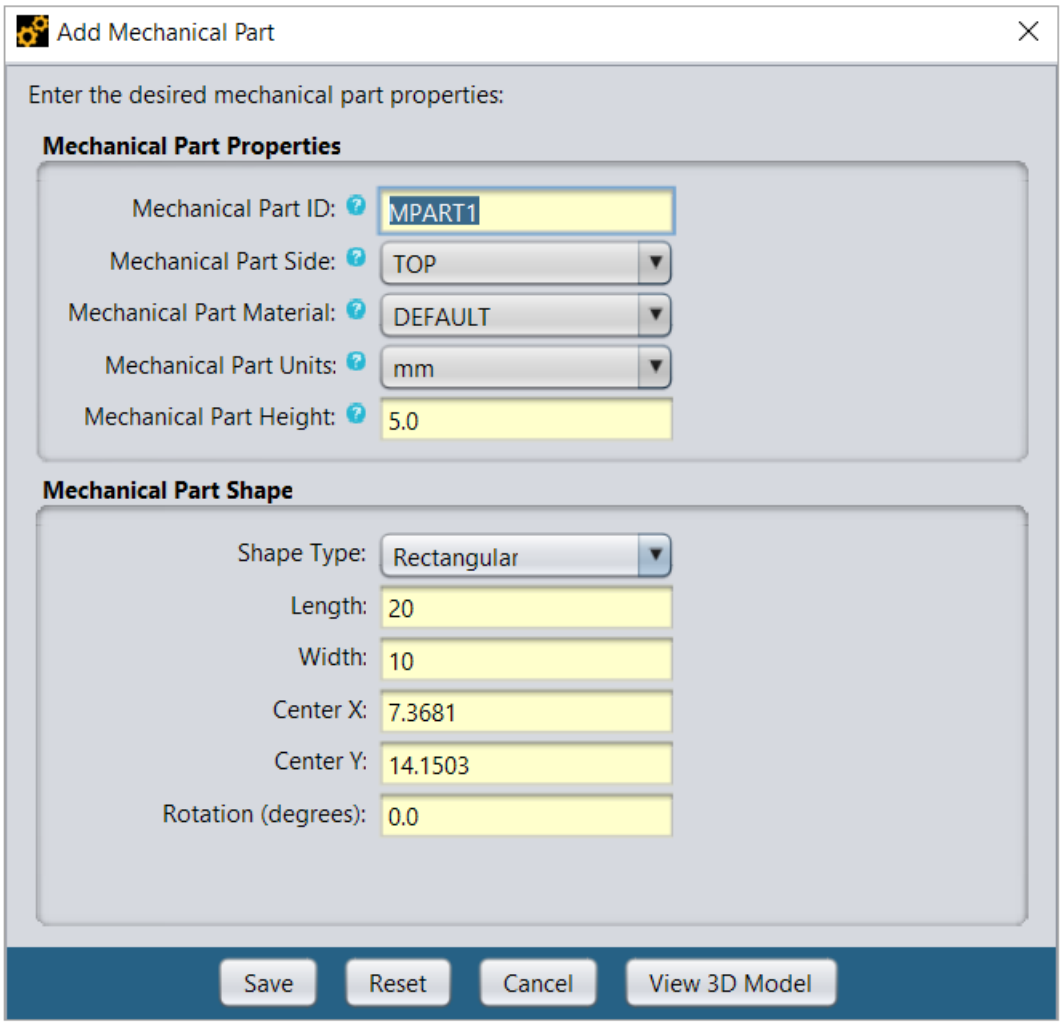
The mechanical part context menu provides options for:

- Editing mechanical part properties
- Moving, scaling and rotating parts
- Merging mechanical parts
- Adding, copying and deleting parts

8.12.3. Mechanical Part Properties

The Mechanical Part Properties dialog is displayed whenever the **Add Mechanical Part** or **Edit Properties** options are selected in the mechanical context menu, allowing users to specify exact shape and location properties as needed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Shape Type** property defines the geometric shape of the part, which controls how it can be edited. The following shape types are supported:

Table 8.30: Mechanical Part Shape Types

Rectangular	Defined by length and width dimensions, a center location and an angle of rotation.
Circular	Defined by a diameter, number of nodes, a center location and an angle of rotation.
Slot	Defined by length and width dimensions, number of nodes, a center location and an angle of rotation.
Polygonal	Defined by a set of node locations and an angle of rotation.
PCB	Defined by the board outline of the PCB.

The Circular and Slot shapes allow you to specify the number of nodes to use when approximating arcs used to form those shapes. For circular shapes, the nodes are evenly spaced along the circumference of the circle. For slot shapes, 4 nodes are used to define the rectangular part of the slot and the remaining nodes are evenly spaced along the semi-circles on both ends of the slot.

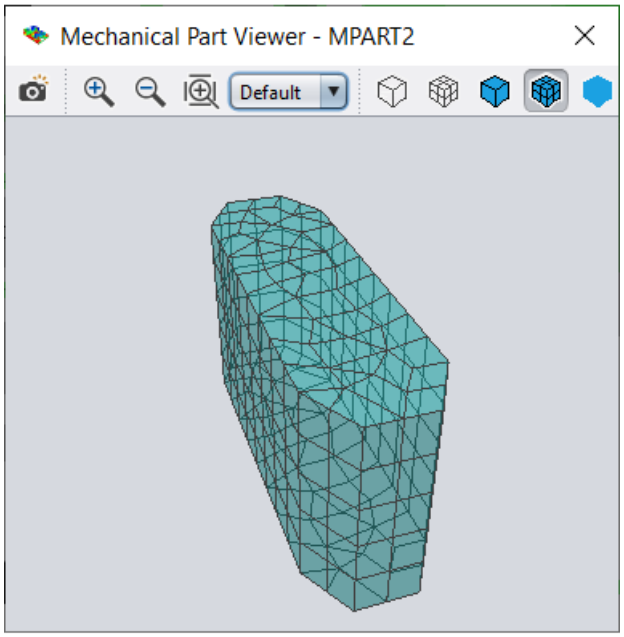
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For all shapes, an angle of rotation can be specified to rotate the shape in the XY plane. When scaling shapes graphically using the control points, the angle of rotation is preserved, and the shape dimensions are automatically updated to reflect the scale operation.

In addition to the shape properties, properties to select the side of the PCB to contain the part, the height of the mechanical part, and the material the part consists of are all defined for each mechanical part.

The estimated mechanical part weight is displayed when viewing mechanical part properties. This is a read-only property only intended for reference.

Since mechanical parts are three-dimensional objects, the **View 3D Model** button may be used to see how changing various mechanical part properties affect the shape of the part while editing the part.



Multiple mechanical parts may be merged into a single part if the defined parts overlap and have similar height and material properties. By selecting all the parts to merge into a single part, right-click to display the context menu and select **Merge Mechanical Parts**. Following this operation, any overlapping parts which can be merged will be merged into a single part.

8.12.4. Mechanical Part Multi-Edit

Multiple mechanical parts may also be selected, and the properties of those mechanical parts updated at the same time. When the properties for more than one mechanical part are edited at the same time, any mechanical part property which varies between the selected mechanical parts will display the value <*VARIOUS*>. Any field which is left as the value <*VARIOUS*> when saving the form will leave the original mechanical part value specified for that given field. Any other field values will be assigned to all selected mechanical parts.

Multiple mechanical parts may only be edited at the same time if the mechanical parts are of the same shape type.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.12.5. Mechanical Part Attachment Points

Mechanical parts may be located over top of mount points defined as mount pads or standoffs. When a mechanical part is located over top of these mount points mechanical part attachment points may be created between the mechanical part and the mount point. To add an attachment point to a mechanical part, place the mechanical part over top of a mount point using the **Mechanical Part Editor** in the **Layer Viewer.** Now make sure no mechanical parts are selected, then right-click a mount point that should become an attachment point for the mechanical part. If the mount point meets all the properties to become an attachment point, then a context menu should appear to add the selected mount point as a mechanical part point. Repeat this process for each of the desired mount points.
To remove a mount point as a mechanical part attachment point, right-click the mount point and select **Remove Mechanical Part Point.**

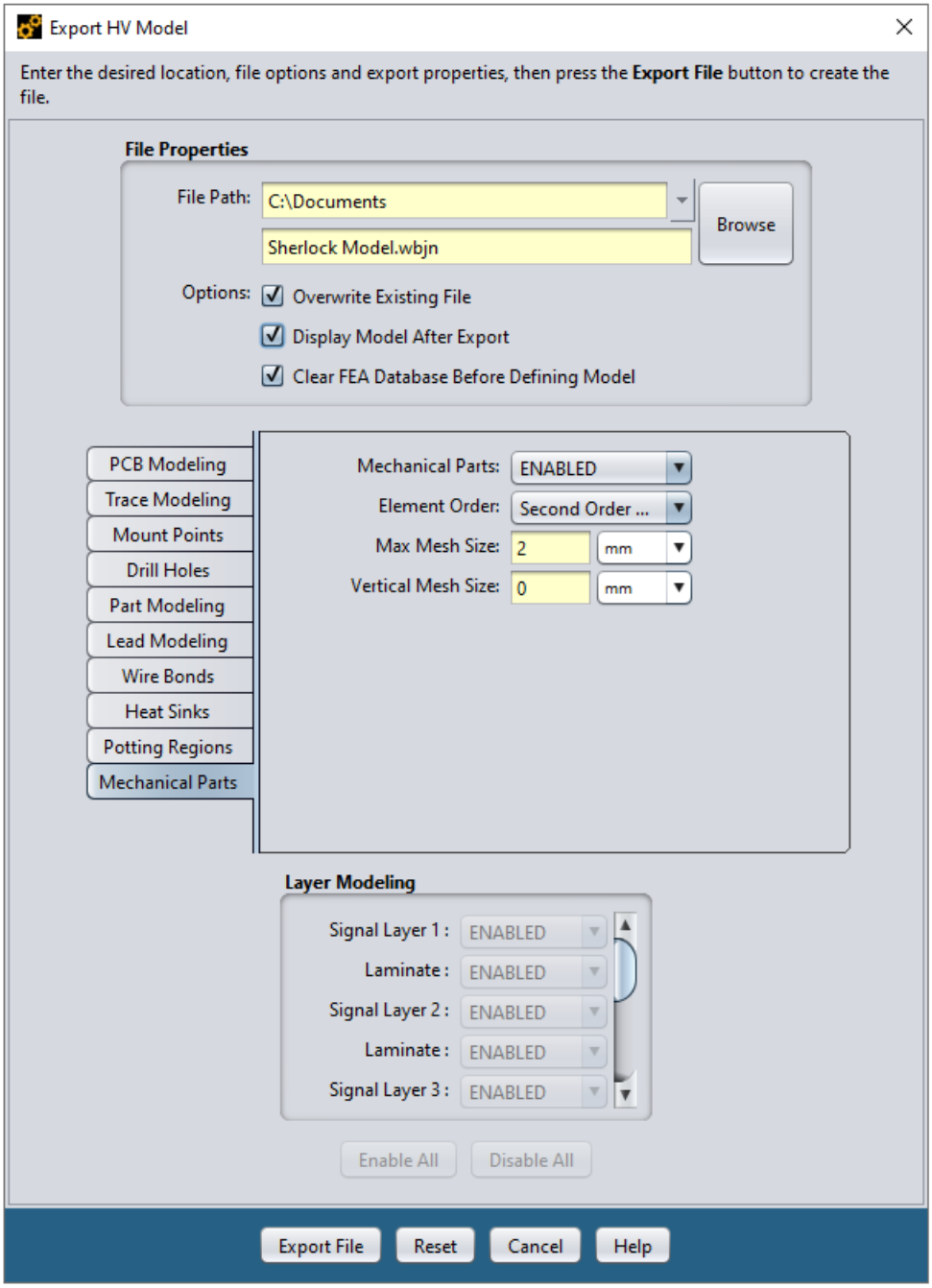
Attachment points for mechanical parts must adhere to the following conditions:

- Attachment points attach the mechanical part to the PCB
- Attachment points must be fully contained by the mechanical part outline
- All attachment points assigned to a mechanical part must have the same height
- Attachment points on mechanical parts covering a component must be at least as tall as the height of the component

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.12.6. FEA Model

Mechanical parts may also be included when generating a 3D FEA model for export or analysis. The analysis settings and export model dialogs all include an option for **Mechanical Parts**. Select this tab to **enable** or **disable** mechanical parts in the FEA model.



8.13. FEA- Sub-Assembly Analysis

The **Finite Element Analysis (FEA)** used by Ansys Sherlock as the basis for **In-Circuit Testing (ICT) Analysis** and **Shock**, **Vibration**, and **Thermal Mechanical Analysis** is a powerful modeling tool for circuit card assemblies consisting of printed circuit boards, components and mount points. Sherlock

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

extends such analysis capabilities to include one or more sub-assembly CCAs that are attached to a main CCA as either mezzanine or edge-connected cards. This document shows how the sub-assembly analysis feature is used in Sherlock.

Note:

The examples shown in this document are based on the **Assembly Tutorial.zip** project archive provided as part of the Sherlock Tutorial package. Use the **Project > Import Project** menu item to import the project from the tutorial folder in the Sherlock Installation Directory.

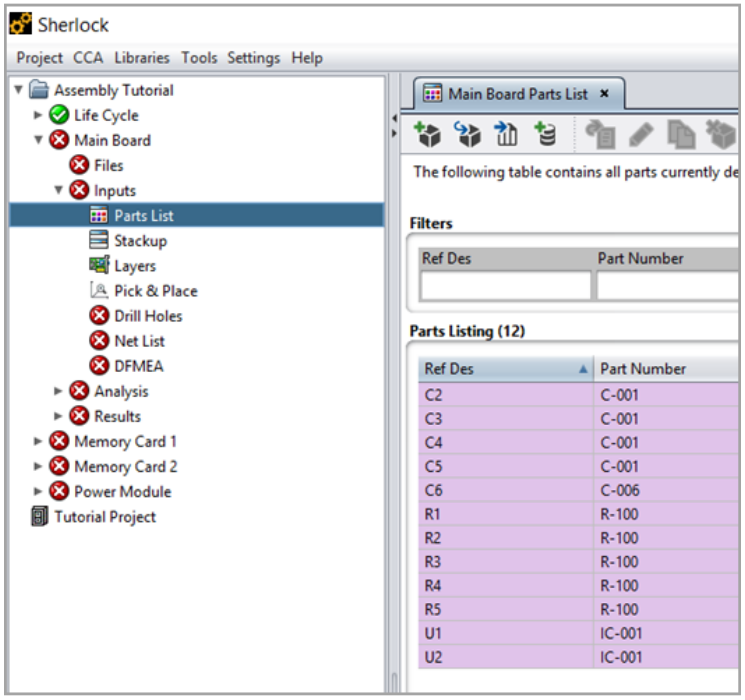
In this section, the following topics are covered:

8.13.1. Tutorial Review
8.13.2. Baseline Analysis
8.13.3. Adding an Assembly
8.13.4. Defining Assembly Points
8.13.5. Multiple Assemblies
8.13.6. Assembly Results

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

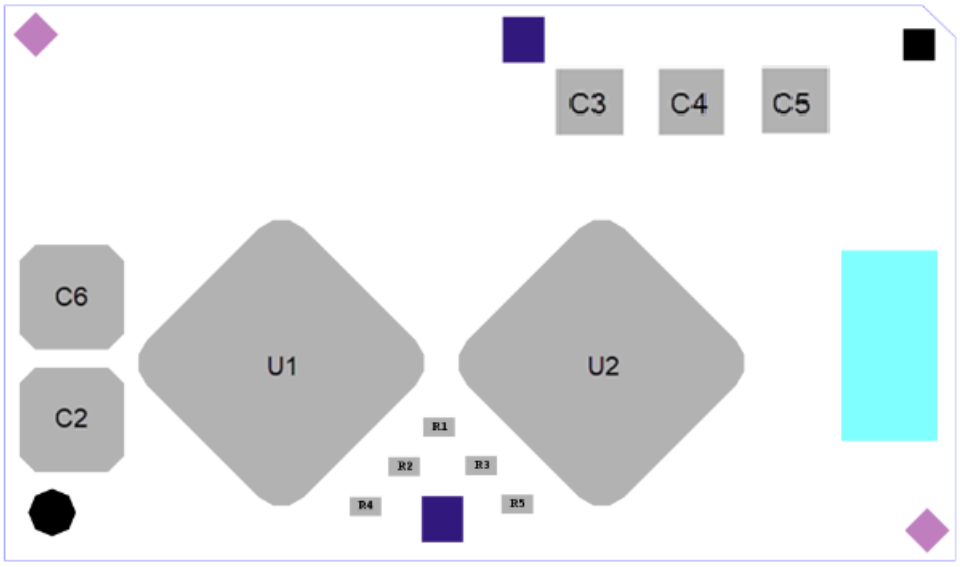
8.13.1. Tutorial Review

The following snapshot shows the overall composition of the Assembly Tutorial project:



The project consists of four CCAs called Main Board, Memory Card 1, Memory Card 2, and Power Module. The CCAs are defined just like any other Sherlock CCA, with various components and mount points located on the top and bottom of each board. We've also defined two ICT fixtures (the blue squares) and two ICT test points (the purple diamonds) to facilitate ICT analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



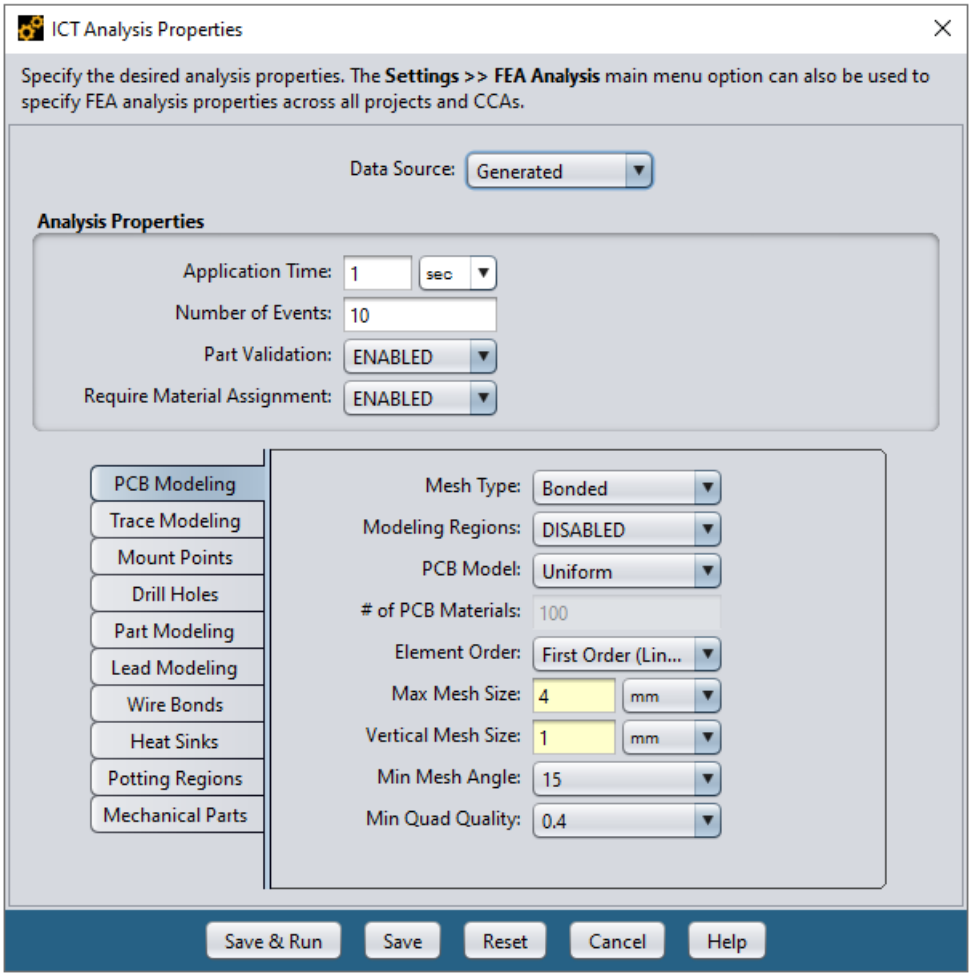
Note:

For demonstration purposes, each CCA only has a handful of parts, but there are no restrictions on the types of CCA's that can be used as sub-assemblies or the main board.

8.13.2. Baseline Analysis

Before we start adding assemblies to the main board, let's generate some baseline results for just the main board itself. Start by right-clicking the **Analysis > ICT Analysis** node in the **Project Tree** for the **Main Board** and select the **Edit Properties** in the context menu. Enter the settings shown in the image below and press the **Save & Run** button to perform an ICT analysis on the Main Board.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



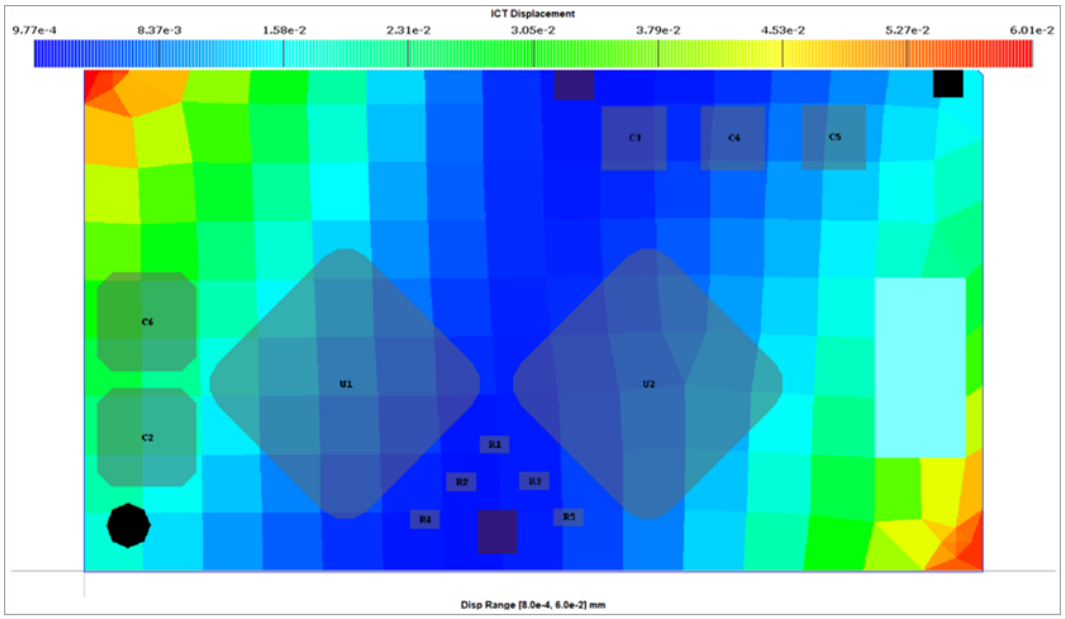
Note:

The icon next to **ICT Analysis** may indicate a warning from the analysis run. If so, double-click the **ICT Analysis** item. Then click the **Issues** tab to see the warning. With this tutorial, a warning regarding Part validation will most likely be displayed.

After the ICT analysis task is completed, the ICT Displacement layer clearly shows the expected results as seen from the **Layer Viewer**, shown below.

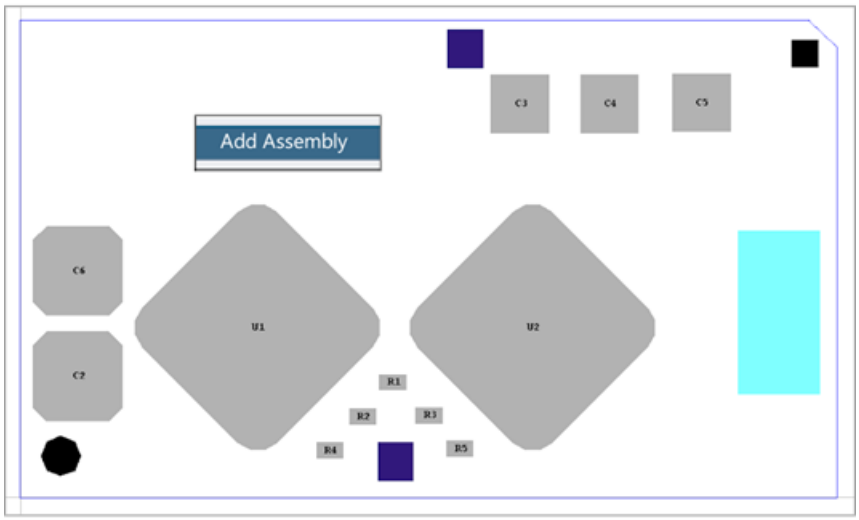
With two fixtures located in the middle of the board and two test points pushing down on opposite corners, the board flexes the most under both test points, with relatively no flex in the middle of the board.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.13.3. Adding an Assembly

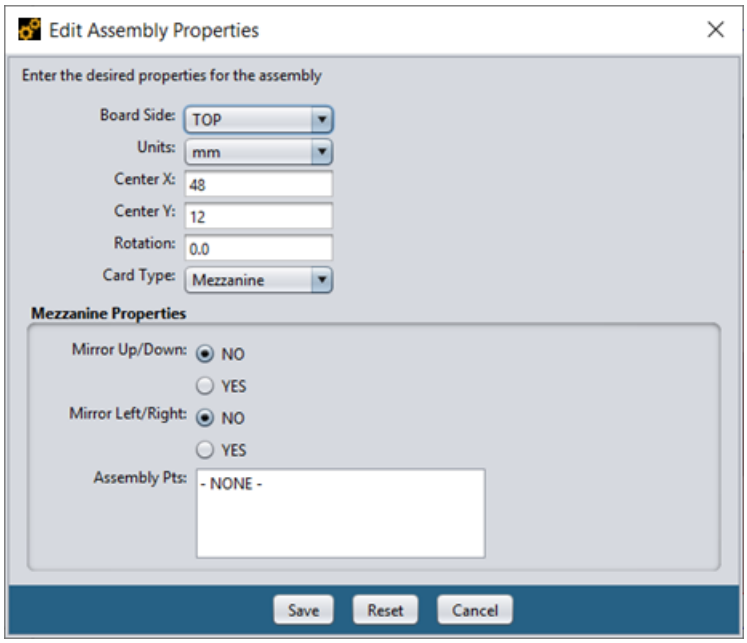
All assembly management is done graphically using the **Layer Viewer**, so select **Edit > Edit Assemblies** from the **Layer Viewer Main Menu** to start assembly edit mode.



To add an assembly to the Main Board, right-click anywhere in the **Layer Viewer** content panel to display and select the **Add Assembly** menu item. Where you click becomes the default center of the circuit board you are adding, but you can modify the coordinates later.

When **Add Assembly** is selected, the **Add Assembly** dialog will appear, allowing you to select the desired CCA to be used as the assembly and to define how that assembly is to be oriented.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

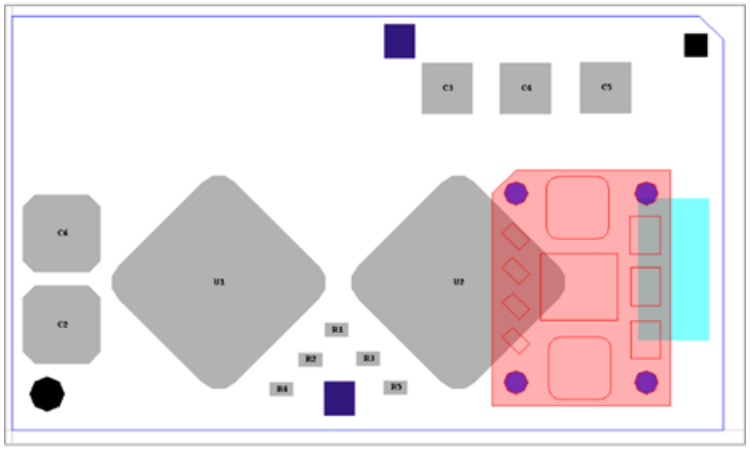


The coordinates are based on the location of your **Add Assembly** mouse-click earlier, but you can easily move the assembly around after it is created. Of course, if you know the exact coordinates from the design files, you can enter them now.

At this point, the most important properties are the **Assembly**, **Board Side**, and **Card Type**. For this example, the Power Module assembly is placed on top of the Main Board, oriented as a Mezzanine card. As you will see, Mezzanine cards are parallel to the main card and are attached with one or more assembly points.

Sherlock also allows you to rotate and flip the assembly in the up/down and/or left/right directions to orient it properly relative to the main card. In this case, no rotation or mirroring is required so we can simply press **Save** to add the assembly.

After saving, an outline of the assembly will appear in the **Layer Viewer** for the Main Board, centered over the mouse location used to launch the dialog.

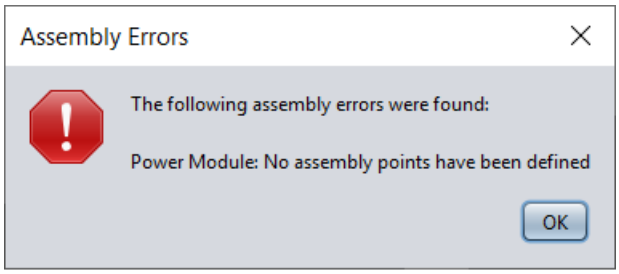


Select the newly added assembly by holding the SHIFT key and left-clicking anywhere inside of the assembly PCB or component outline. The selected assembly will be highlighted by a red transparent color.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Left-click and drag the selected assembly to place it in the desired location on the Main Board.

Press the **Apply** button to save your changes and stay in the **Assembly Editor**. At that point, the following error message will appear:



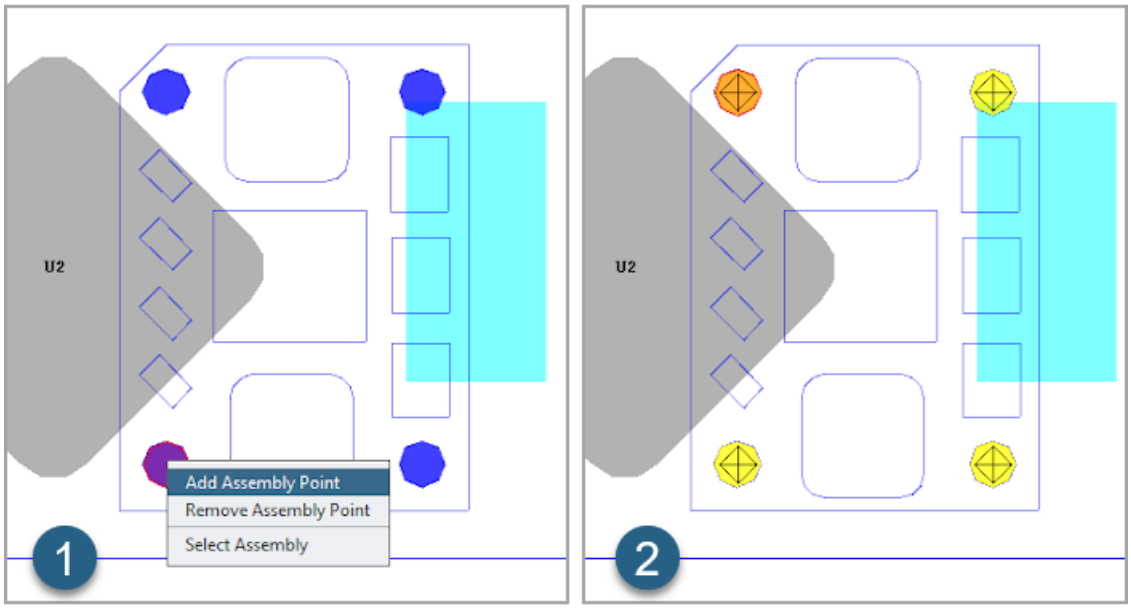
If you recall, Mezzanine cards must be attached to the Main Board using one or more assembly points. Since we haven't made any such attachments yet, Sherlock is reminding us that the assembly definition is not yet complete.

8.13.4. Defining Assembly Points

Assembly points are defined by associating one or more existing standoffs or mount pads with the assembly. The standoffs or mount pads may be initially defined in either the main CCA or the assembly CCA. However, in order to model reality properly the assembly points used to attach Mezzanine cards:

- Must be located between the main CCA and the assembly CCA
- May not be located inside of a component on either CCA
- And must all be of the same height.

In our example (1, below) four standoffs have already been defined in the Power Module CCA. All four of the standoffs are on the bottom side of the Power Module CCA and they are all the same height. The standoffs are denoted in the assembly outline by the solid blue circles, providing an easy way to identify them.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define the standoffs as assembly points, right-click anywhere OUTSIDE of the assembly outline to deselect the assembly. Then, right-click INSIDE of any one of the standoff circles to display the assembly point menu and select the **Add Assembly Point** menu item.

At that point, the solid blue circle will be replaced by a yellow circle (2) with an inscribed diamond pattern to denote the assembly point.

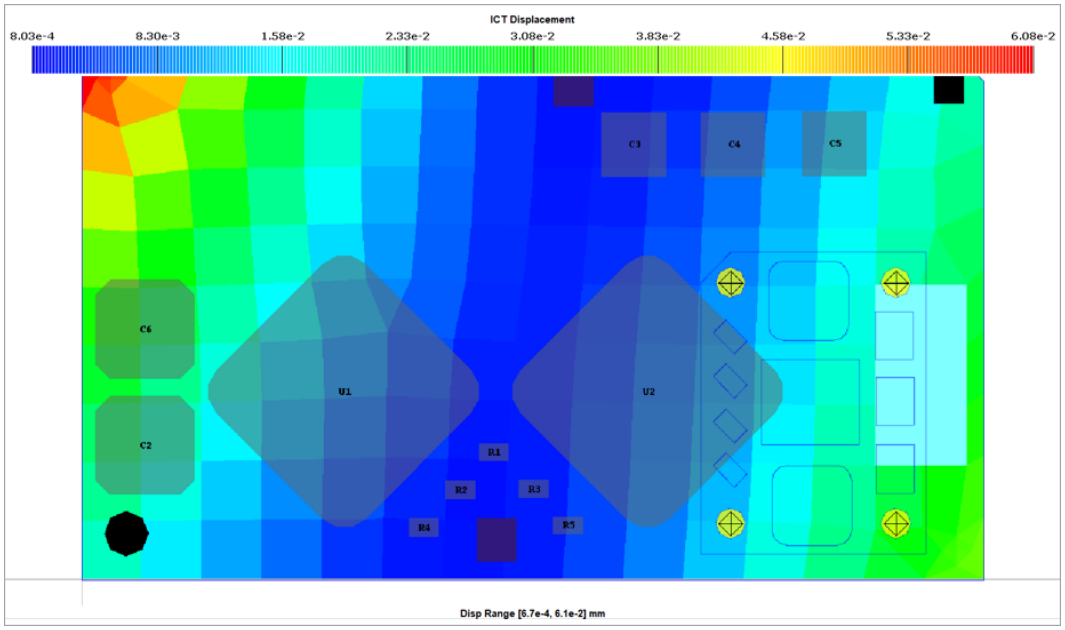
Repeat the process for all four standoffs defined in the Power Module to create four assembly points.

With assembly points defined, you can now **Save** the assembly editing changes and re-run **ICT Analysis** to see how the assembly alters the results.

Note:

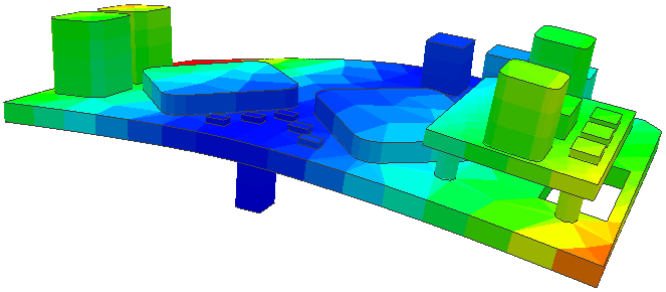
Ansys Sherlock saves different sets of modeling properties for each FE analysis task. As such, be sure to modify properties for sub-assemblies under the same FE analysis task. For example, when exporting a model from the Natural Frequency analysis task, you should modify the Natural Freqency modeling properties of each sub-assembly as needed.

By comparing the new **ICT Displacement** results to the baseline results, we can see that the lower right corner of the Main Board is no longer displaced by the same amount on the opposite side of the board. As expected, the addition of the Power Module sub-assembly has stiffened the Main Board on the right-hand side, causing it to flex less than the left-hand side.



A 3D rendering of the ICT model clearly shows the Mezzanine card in the desired location, as well as the relative displacement of each part of the model under the ICT loads defined.

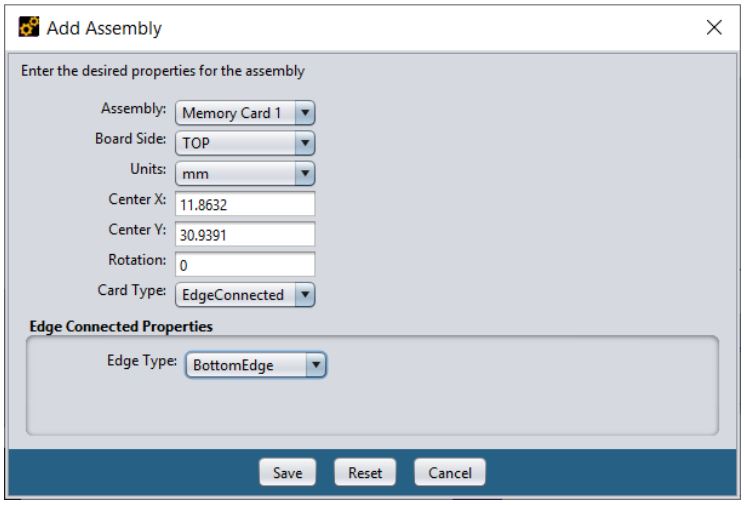
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See Viewing and Managing Analysis Results (p. 785) for more details about viewing 3D results.

8.13.5. Multiple Assemblies

To support real-world assemblies, Sherlock allows you to attach multiple assembly CCAs to a given CCA. Moreover, it allows you to attach an assembly CCA to an assembly CCA that is attached to a main CCA. To prevent infinite loops and other nasty modeling situations, Sherlock ensures that a given CCA can be used at most once as an assembly in a project and that all assemblies are defined in the same project.



To demonstrate how multiple assemblies are used, let's add Memory Card 1 and Memory Card 2 to the Main Board as edge-connected cards. To that end, enter the Assembly Editor mode in the Layer Viewer, right-click inside of the Main Board and select **Add Assembly** from the context menu.

Select **Memory Card 1** as the Assembly choice and **EdgeConnected** as the Card Type.

At that point, the bottom set of properties will change to allow you to specify the **Edge Type** for the assembly. In this case, select the **BottomEdge** to indicate that the bottom edge of Memory Card 1 should be attached to the Top side of the Main Board.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As before, the default center location will be automatically set to the mouse location when the context menu was selected, but you can easily drag the new assembly to the proper location.

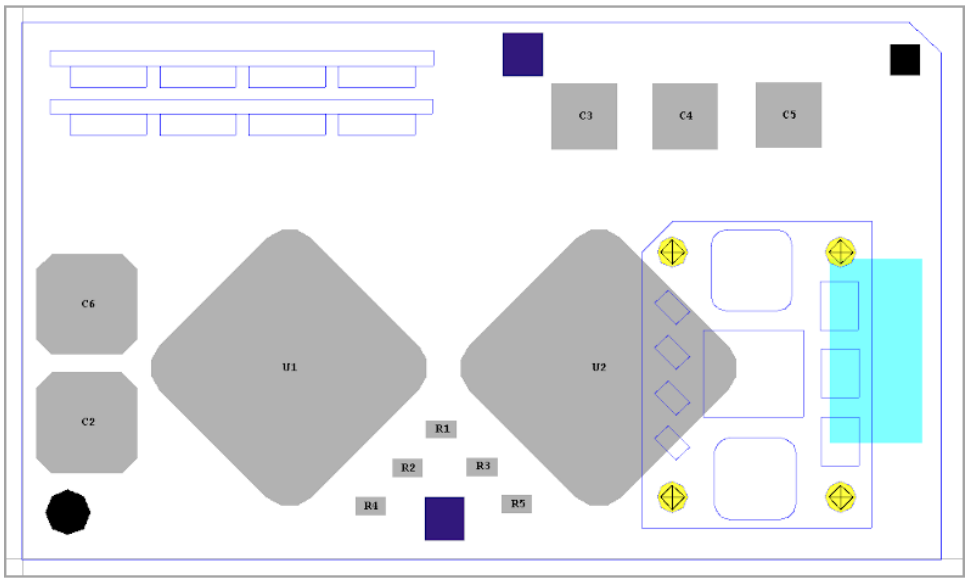
Note:

Unlike Mezzanine cards, there are no mirror properties for Edge-Connected cards. The Board Side, Rotation and Edge Type properties are enough to orient the assembly card as needed relative to the main card.

Note:

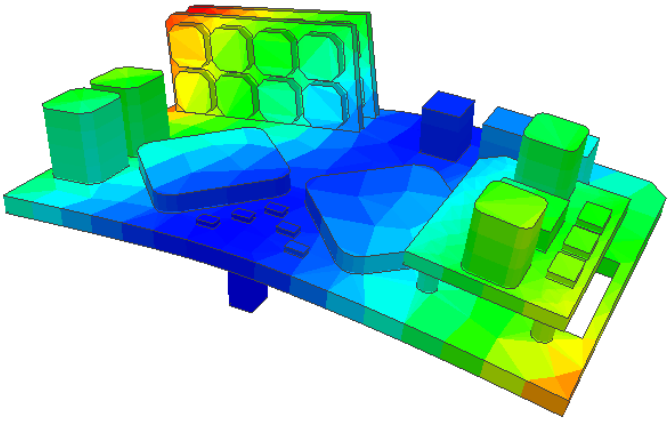
Similarly, no assembly points are defined for Edge-Connected assemblies because Sherlock assumes that the assembly card edge itself is attached to the main card.

Repeat the edit steps above to add Memory Card 2 to the Main Board in addition to Memory Card 1 as shown here:



The assembly outlines shown in the **2D Layer Viewer** provide a convenient mechanism for moving assemblies around and reviewing analysis results. However, you must always keep in mind the side of the main board where each assembly is attached. To ensure you've got everything in the right place, you can either export a 3D STEP model of the board and view it using an appropriate viewer, or you can run a crude ICT or Natural Frequency analysis of the board and view the results using the **Sherlock 3D Viewer**, as shown here:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8.13.6. Assembly Results

When one or more assemblies are defined for a given CCA and an analysis is performed on that CCA, Sherlock automatically generates tabular and graphical results for each sub-assembly CCA as well as those for the main CCA. By the nature of the models generated, such sub-assembly results are different than would otherwise be determined if the assembly CCA was analyzed by itself. For example, the natural frequency displacements for each individual CCA are much different than those for the combined assembly.

When sub-assembly results are generated, they are stored as the analysis results for the individual CCA and can be accessed just like any analysis results. For example, here are the ICT analysis results for the Power Module assembly after analyzing the Main Board:

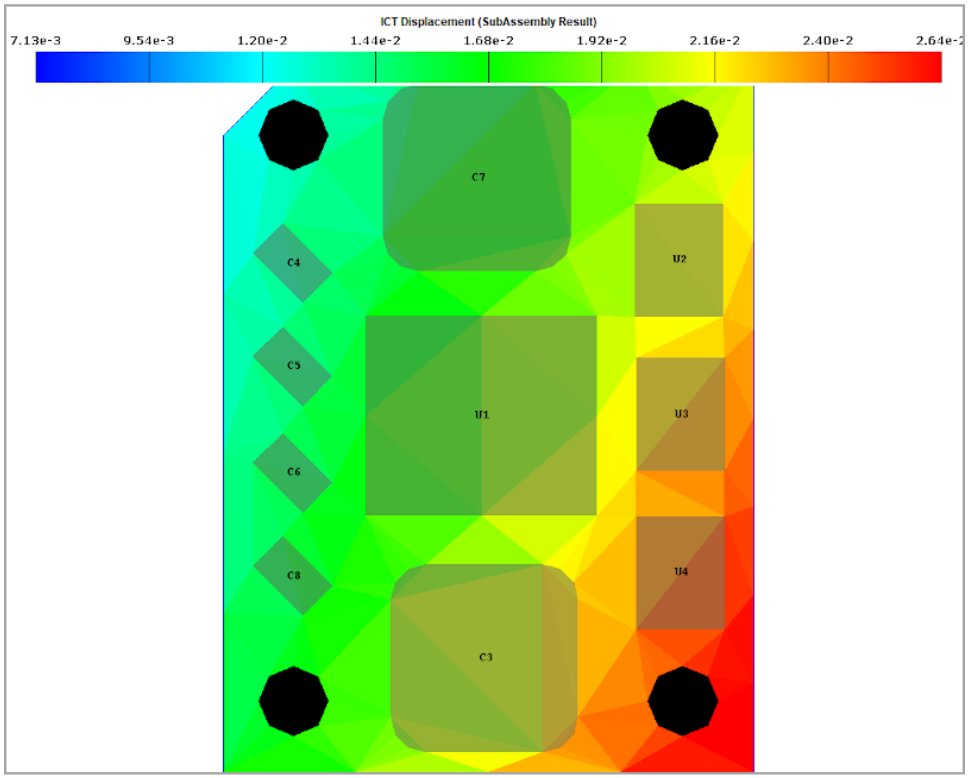
RefDes	Package	Part Type	Side	Material	Weight	Max Disp	Max Strain	TTF (years)	Failure Prob	Score	Overstress...
C3	CAP-ALUM-C	CAPACITOR	TOP	EPOXYENCAPSULANT	3.29E-1	2.2E-2	2.2E-5	>100	0.0	10.0	10.0
C4	0805	CAPACITOR	BOTTOM	EPOXYENCAPSULANT	2.59E-3	9.9E-3	3.8E-5	>100	0.0	10.0	10.0
C5	0805	CAPACITOR	BOTTOM	EPOXYENCAPSULANT	2.59E-3	1.1E-2	8.8E-6	>100	0.0	10.0	10.0
C6	0805	CAPACITOR	BOTTOM	EPOXYENCAPSULANT	2.59E-3	1.2E-2	1.0E-5	>100	0.0	10.0	10.0
C7	CAP-ALUM-C	CAPACITOR	TOP	EPOXYENCAPSULANT	3.29E-1	1.6E-2	1.7E-5	>100	0.0	10.0	10.0
C8	0805	CAPACITOR	BOTTOM	EPOXYENCAPSULANT	2.59E-3	1.3E-2	4.5E-5	>100	0.0	10.0	10.0
U1	TO-252	IC	TOP	EPOXYENCAPSULANT	1.53E-1	2.0E-2	1.8E-5	>100	0.0	10.0	10.0
U2	FLATPAK4	IC	TOP	EPOXYENCAPSULANT	1.08E-2	2.1E-2	2.0E-5	>100	0.0	10.0	10.0
U3	FLATPAK4	IC	TOP	EPOXYENCAPSULANT	1.08E-2	2.3E-2	9.8E-6	>100	0.0	10.0	10.0
U4	FLATPAK4	IC	TOP	EPOXYENCAPSULANT	1.08E-2	2.6E-2	1.5E-5	>100	0.0	10.0	10.0

Note:

When performing an analysis on a project with sub-assemblies, Sherlock uses the analysis settings for each sub-assembly to determine what analysis to perform. If the sub-assembly results are not what you expect, verify the **Analysis Properties** for each circuit card. For example: If you run an analysis where **Lead Modeling** is set to **ENABLED** but the sub-assembly results do not show lead results, edit the **Analysis Properties** for the sub-assembly: Set **Lead Modeling** to **ENABLED**, **Save** the changes, and run the analysis again.

Graphical results are also stored for the assembly CCA and are accessible via the Layer Viewer for that CCA. For example, here are the ICT Displacement results for the Power Module assembly.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You should note that a SubAssembly Result notation has been added to the result title to indicate that the analysis was done as part of the analysis of the Main Board.

You should also note that the spectrum used to color the layer is the same one used for the Main Board itself (which is why there are no deep blues or bright reds seen in the layer). This approach allows you to properly compare 2D result images for the Main Board and all sub-assembly CCA's.

8.14. Advanced BGA Modeling

Section Links:

8.14.1. Creating the Detailed Component

8.14.2. Properties of Detailed BGA Component

8.14.3. Exporting the Detailed Component to Ansys Mechanical

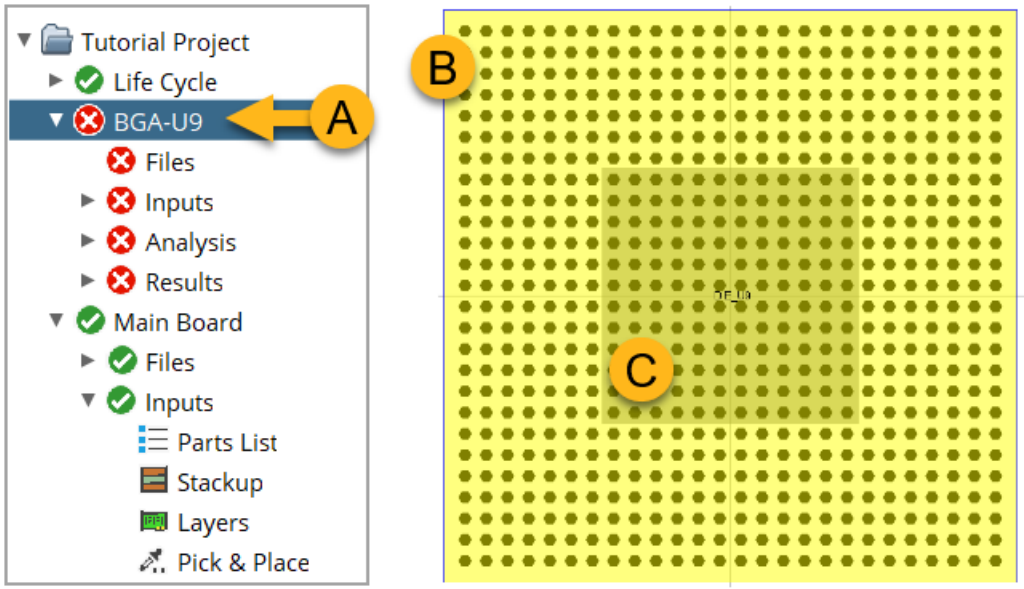
Starting with release 2024 R2, Sherlock automates the process of taking a BGA component from the Parts List and modeling it in greater detail for export to Ansys Mechanical. In short, Sherlock models the BGA as a small, rectangular circuit board, converting the BGA elements to CCA elements. Previously, this kind of modeling had to be done manually. Now Sherlock does much of the work for you. Once the transformation is complete, the new Detailed Component has the following characteristics:

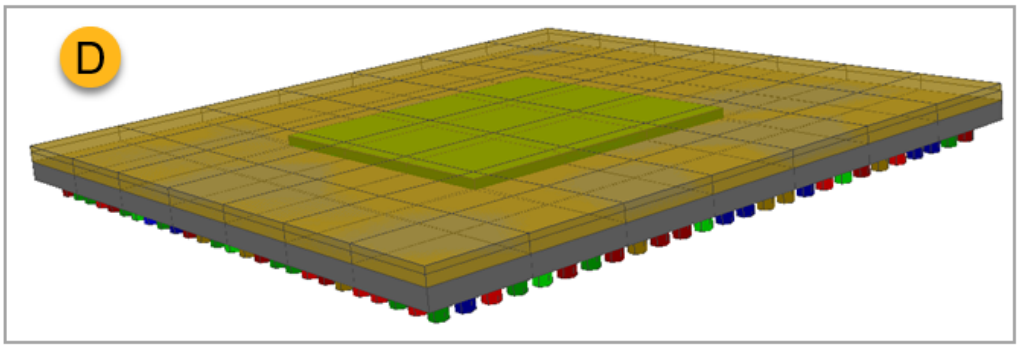
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The new Detailed Component appears in the **Project Tree** as a CCA (A, in the image below). For details on the CCA's properties, see CCA Properties (p. 651) below. Image B shows how the Detailed Component appears in Sherlock's Layer Viewer. In image D, it is shown in the 3D Viewer.

> **Note:**
>
> For the purpose of explanation, the instructions in this section use BGA component U9 from the Sherlock tutorial as an example. See Tutorial: Project Overview (p. 8).

- To model the die, Sherlock adds a single leadless part (C, below) centered on the CCA. For details on the CCA's properties, see Die Properties (p. 653) below.
- To model the overmolding, Sherlock creates a potting region. The yellow color coding in image B represents the potting. For details on the potting region properties, see Overmold-Potting Region Properties (p. 652) below.
- Sherlock creates mount points to model the balls. For details on the mount point properties, see Mount Point - Ball Properties (p. 654) below.





Tip:

Remember, the Detailed Component model is intended for export to Ansys Mechanical. Since it is modeled as a PCB, you can export the Detailed Component exactly as you would any other PCB. See FEA- Ansys Workbench™ Integration (p. 747) .

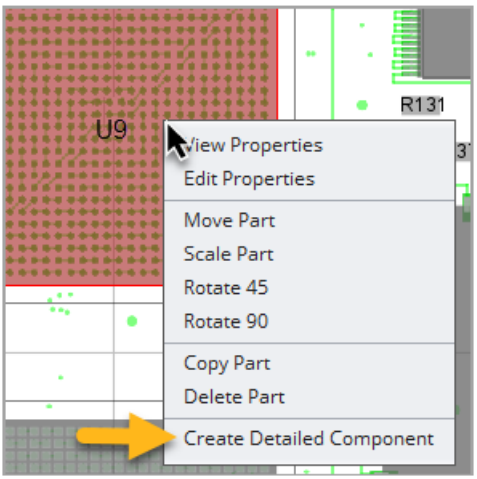
8.14.1. Creating the Detailed Component

Tip:

Before creating the Detailed Component, verify the BGA part properites are accurate and complete.

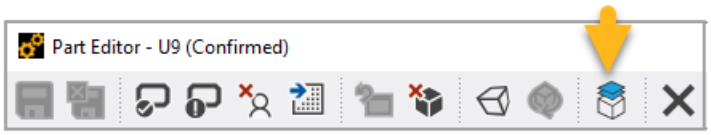
There are two ways you can launch the BGA transformation to a Detailed Component.

- From the Layer Viewer, select **Edit Components** from the **Edit** menu. Right-click the BGA component of interest and in the context menu select **Created Detailed Component**. Sherlock will ask for confirmation before creating the detailed component. In the image below, BGA component U9 has been selected.



- Or, from the Part Editor, click the **Create Detailed Component** icon shown below. Sherlock will ask for confirmation before creating the detailed component. The icon is active only for BGA components.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Before the CCA creation is initiated, Sherlock may prompt you to provide information if additional part properties are required. In the example of component U9 from the tutorial circuit board, you must specify the following:

- Set the **Overmold Material** to **OVERMOLD-BGA** (**Package** tab in the Part Properties Editor).
- Enable **Potting Usage** for the overmold material using the Material Editor.
- Specify **Ball Count** and related properties in the **Ball** tab. **Ball Count** = 100. **Ball Perimeter Rows** = 10. **Ball Perimeter Cols** = 10.

8.14.2. Properties of Detailed BGA Component

The sections below explain how Sherlock determines the properties for the Detailed Component which, remember, is modeled as a small CCA. In Sherlock, you can edit the properties of the Detailed Component just as you would any CCA.

Links:

8.14.2.1. CCA Properties

8.14.2.2. Overmold-Potting Region Properties

8.14.2.3. Die Properties

8.14.2.4. Mount Point - Ball Properties

8.14.2.1. CCA Properties

The new detailed BGA component, which appears as a CCA in the **Project Tree**, will have the following properties:

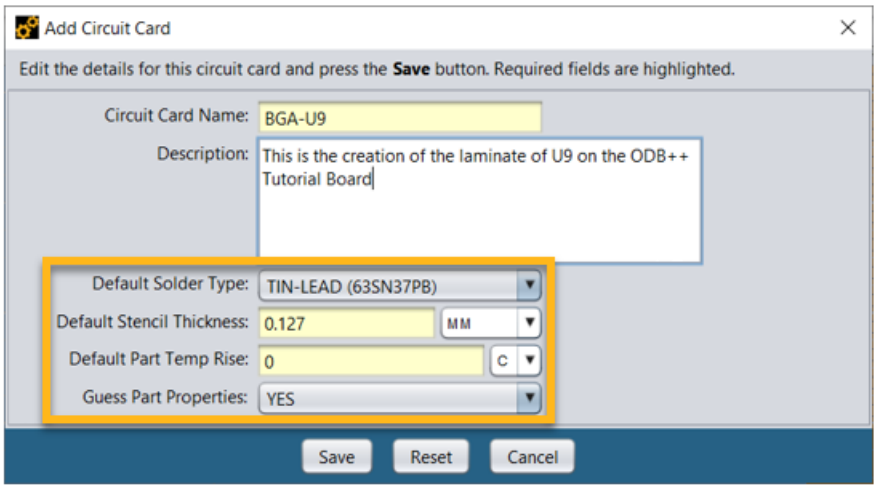
- The name of the circuit card will be **BGA-<RefDes>** (**BGA-U9**, for example).
- The circuit card will include three layers, each 100% Substrate Material.

Stackup Layers

Layer	Type	Material	Construction	Thickness	Density	CTExy	CTEz	Exy	Ez
1	SIGNAL	LAMINATE-BGA (100.0%) / RM_L1		0.5 oz	1.9000	15.000	15.000	15,168	15,168
2	Laminate	Generic FR-4 / LAMINATE-BGA (100.0%)	DEFAULT (2116)	28.13 mil	1.9000	15.000	15.000	15,168	15,168
3	SIGNAL	LAMINATE-BGA (100.0%) / RM_L3		0.5 oz	1.9000	15.000	15.000	15,168	15,168

- The **Default Solder Type**, **Default Stencil Thickness**, **Default Part Temp Rise**, and **Guess Part Properties** will be the default values from Sherlock, as shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Other CCA Properties**: The shape of the new circuit board will be rectangular with the following properties:

CCA Property of the New Detailed Component	Based on the Property of the Original BGA
Board Units	Package Units
Board Length	Package Length
Board Width	Package Width
Board Thickness	Laminate Thickness
Conductor Material	Substrate Material
Conductor Percentage	100
All other parameters	Default Sherlock Parameters

8.14.2.2. Overmold-Potting Region Properties

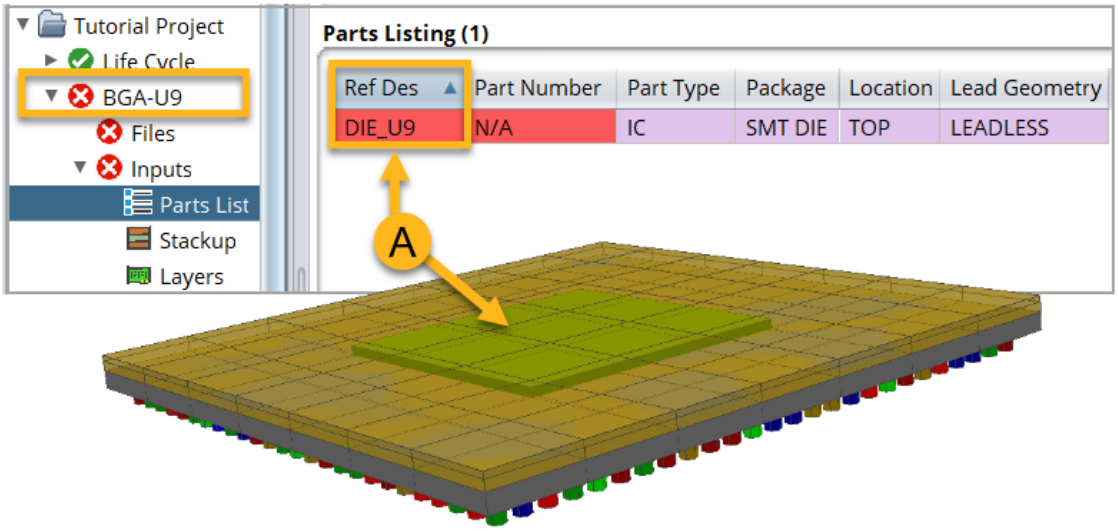
Sherlock creates the overmold as a potting region with the following properties:

Potting Region Property on the New Detailed Component	Default Value
Potting Side	TOP
Potting Material	Overmold Material of the orginal BGA component
Potting Units	Package Units of the orginal BGA component
Potting Thickness	Overmold Thickness of the orginal BGA component
Potting Standoff	0
Potting Region Shape Type	PCB

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.14.2.3. Die Properties

To represent the die of the new detailed model, Sherlock creates a part centered on the new CCA (A, below). The name of this part will be **Die_<RefDes>**. As an example, for BGA U9, the name of the part will be **DIE_U9**. The **Part Type** will be **IC**.



Package Properties of Component Representing the Die	
Package Name	DIE
Package Type	DIE
Package Mount	SMT
Package Units	Die Units from Original BGA
Package Length	Die Length from Original BGA
Package Width	Die Width from Original BGA
Model Part	ENABLED
Corner Shape	SQUARE
Corner Face	TOP_BOTTOM
Material	SILICON

Location Properties of Component Representing the Die	
Board Side	TOP
Location Units	Package Units from original BGA part properties
X Coordinate	0
Y Coordinate	0
Rotation	0
Mirrored	N

Solder Properties of Component Representing the Die	
Solder Model	DIE MODEL

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.14.2.4. Mount Point - Ball Properties

In the detailed component model, Sherlock creates mount points to represent the balls. The mount points have the following properties:

Properties of Mount Points (MP) that Represent the Balls	
Mount Point ID	• Netlists: Mount Point IDs correspond to NetList IDs. • Ball Pattern: Corresponds to the assigned Ball number based on the ball pattern.
MP Type	Mount Pad
MP Units	Based on Ball Units property of original BGA
MP Side	Bottom
MP Height	Uses Ball Height property of original BGA
MP Material	Uses Ball Material property of original BGA
MP State	ENABLED
Mount Point Shape Properties	
Shape Type	Circular
Clircle Diameter	The average of the Ball Diameter, Ball Package Diameter, and Ball Pad Diameter.
# Nodes	6
Center X	Netlist or Ball Pattern X Coordinate
Center Y	Netlist or Ball Pattern Y Coordinate
Rotation	0

8.14.3. Exporting the Detailed Component to Ansys Mechanical

The Detailed Component model is intended for export to Ansys Mechanical. Since it is modeled as a PCB, you can export it exactly as you would any other PCB. See FEA- Ansys Workbench™ Integration (p. 747) for instructions on exporting to Mechanical.

Tip:

For tutorials on running an analysis on the Detailed Component in Ansys Mechanical, visit the Ansys Learning Hub.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 9: FEA Analysis Types

In this section, the following topics are covered:

9.1. FEA- Natural Frequency

9.2. FEA- In-Circuit Testing (ICT)

9.3. FEA- Mechanical Shock

9.4. FEA- Harmonic Vibration

9.5. FEA- Random Vibration

9.6. FEA- Thermal Mech (Thermo-Mechanical)

9.7. FEA- Virtual Accelerometer

9.8. FEA- Ansys Workbench™ Integration

9.9. FEA- Ansys Icepak Integration

9.10. Strain Mapping in Sherlock

9.1. FEA- Natural Frequency

Natural frequency analysis (or Modal Response analysis) is the basis for most FEA processing procedures. The purpose of the analysis is to determine the resonant frequencies of the circuit card and component assembly, based on the structure defined and material properties assigned to each 3D element. The natural frequency predictions can then be used to determine how the board will respond to different types of shock and vibration loads.

9.1.1. Natural Frequency Input Data

9.1.2. Natural Frequency Analysis Properties

9.1.3. Natural Frequency Analysis Results

9.1.1. Natural Frequency Input Data

The Natural Frequency analysis module makes use of the following input data for its calculations:

- Parts List

> **Tip:**
>
> The following section explains which part properties Sherlock uses in the Natural Frequency analysis: Tables: Required Part Properties per Analysis Type (p. 326).

- Size and location of all parts, plated through-holes, and cutouts
- Size and location of all mount points

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Circuit card mechanical properties (stackup data)
- Circuit card outline
- Mesh properties

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

> **Warning:**
>
> Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate results, see the section Compatibility with Earlier Versions of Sherlock (p. 5).

9.1.2. Natural Frequency Analysis Properties

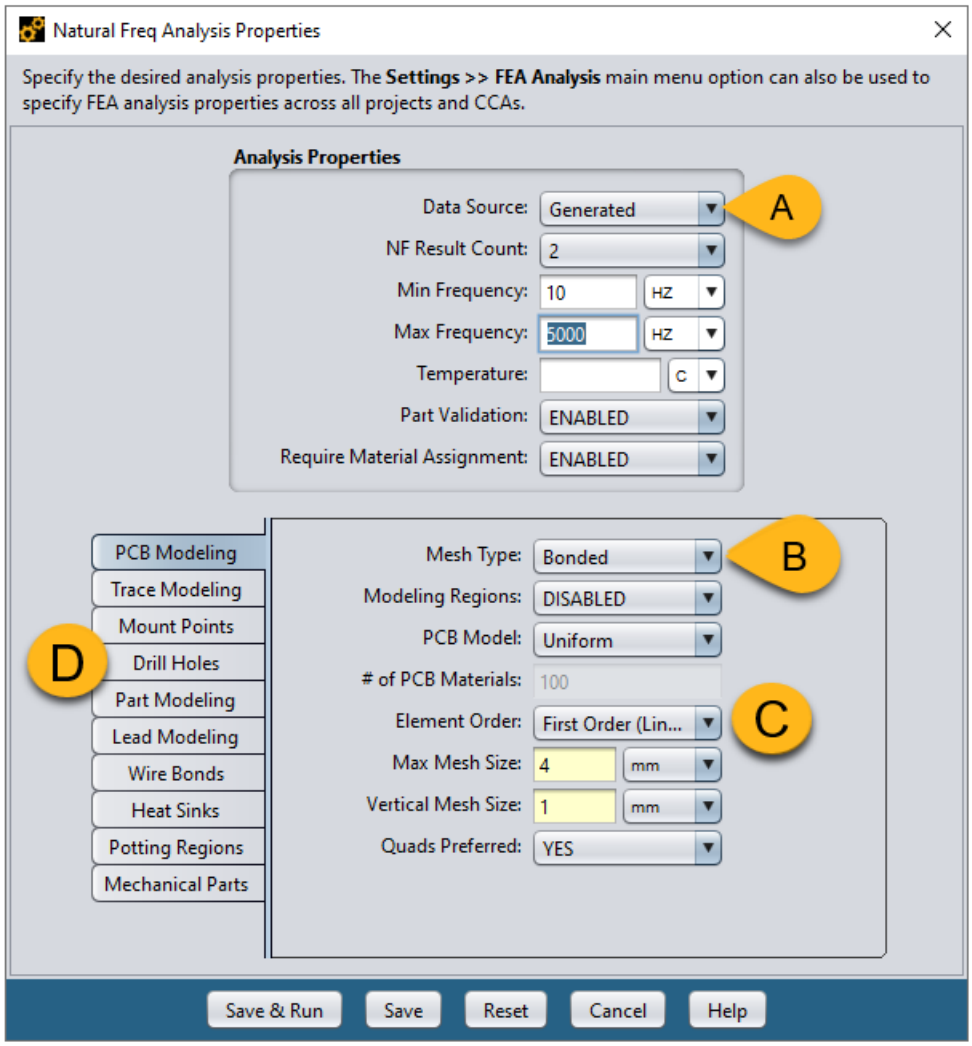
To specify all input properties and perform natural frequency analysis, right-click the **Natural Freq** node in the Project Tree and select the **Edit Properties** option from the context menu. As a convenience, you can also run the analysis immediately, using the last properties entered, by selecting the **Run Analysis Task** option from the context menu.

9.1.2.1. Generated Data Source

When using the **Generated** data source (A, below), you may specify the type of model to be generated. **Bonded** model, which is generated by the default Ansys Meshing algorithm, is typically the only option available in the **Mesh Type** dropdown menu (B). If you have enabled **Legacy Meshing** (see Mesh Settings (p. 85)), **Merged** will also appear in the **Mesh Type** menu. You may select other mesh properties using the **Mesh Properties** options (C). Each mesh property form is selectable by the indicated tabs (D). See the FEA Overview (p. 443) for more details on these options.

The **Analysis Properties** form allows users to indicate the frequency range to be covered by the analysis and number of graphical result layers to be generated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Analysis Properties

- **Data Source**: Choose **Generated** or **Imported**. When you select **Generated**, you are tasking Sherlock to run the FE model in batch mode based on the FE solver selected. If **Imported** is selected, Sherlock allows you to import a Natural Frequency results file.
- **NF Result Count**: This specifies how many 2D results layers Sherlock generates. Each layer shows the modal shape of the PCB for the respective modal frequencies.
- **Min Frequency**: Minimum frequency of the frequency range used for the Natural Frequency Analysis.
- **Max Frequency**: Maximum frequency of the frequency range used for the Natural Frequency Analysis.
- **Temperature**: Specify the temperature Sherlock uses when calculating temperature-dependent properties. If the field is left blank, Sherlock uses the default value of 20°C.
- **Part Validation**: If ENABLED, a Part Validation (p. 263) will be run prior to the FEA analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

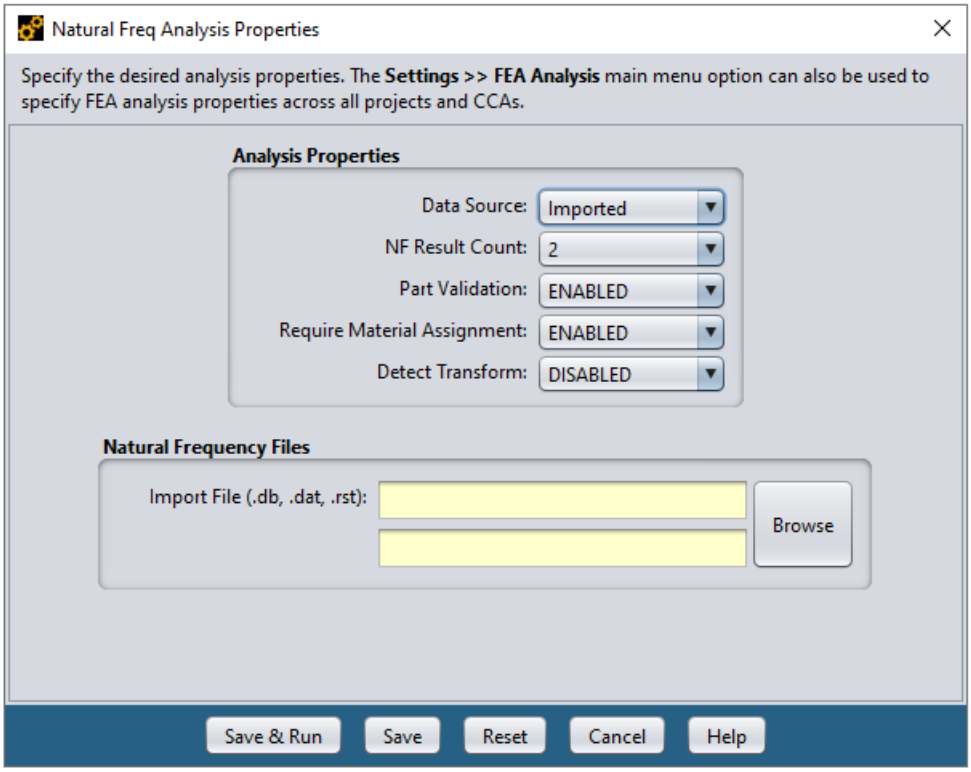
- **Require Material Assignment**: When ENABLED, Sherlock generates an error and halts the FEA analysis if a material is set to DEFAULT.

9.1.2.2. Imported Data Source

Natural frequency results can be imported into Sherlock by selecting the **Imported** data source as shown below. When selected, simply enter the name and location of the Model File and Result File.

Tip:

For detailed, step-by-step instructions for importing results from Ansys Workbench, see the following section: FEA- Ansys Workbench™ Integration (p. 747).



The supported file types are `*.db`, `*.dat`, and `*.rst`.

When the **Detect Transform** is ENABLED, Sherlock will attempt to detect any Transforms applied to the imported model and make adjustments if required. This ensures the imported model matches Sherlock's original model. Select MANUAL to enter Transform properties manually. If a Transform exists and is not detected, Sherlock will not be able to provide component results.

When running the **Detect Transform** feature, Sherlock uses multiple threads by default. This optimizes detection performance. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

The other settings uner **Analysis Properties** are explained in the previous section Meshing Settings (p. 85).

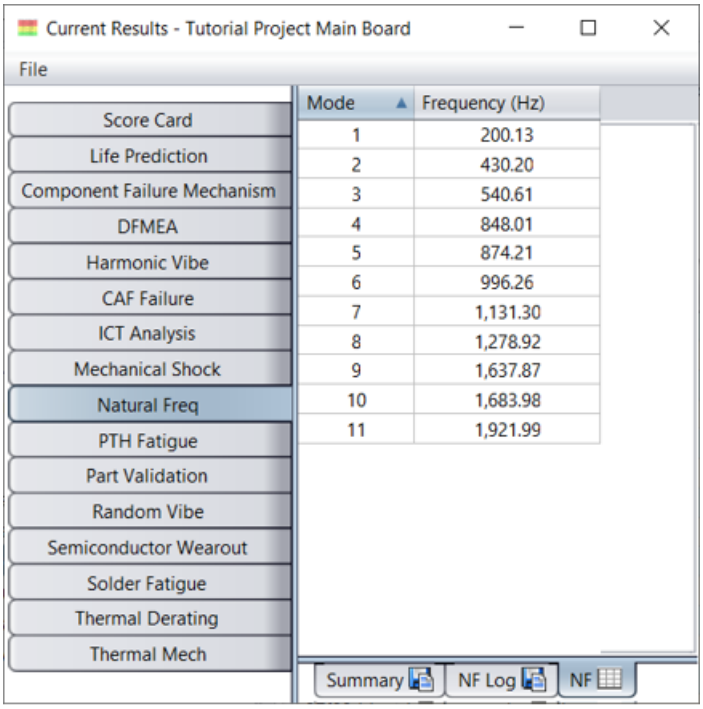
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.1.3. Natural Frequency Analysis Results

The results generated by the Natural Frequency analysis module include a **Summary Panel** showing all the input settings used for the analysis, as well as the predicted natural frequencies. The Natural Frequency analysis module also generates one or more 2D layers showing the predicted deflection of the PCB for each modal frequency.

9.1.3.1. Summary Panel

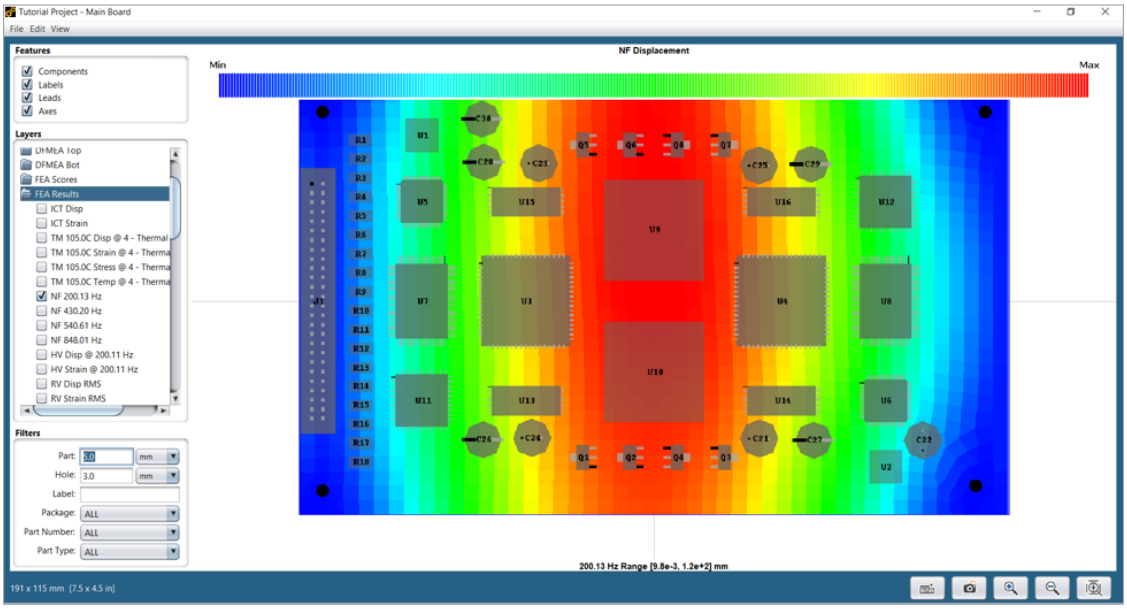
The **NF Table** tab at the bottom of the **Natural Frequency** results show the modes and frequencies, in Hz.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.1.3.2. Graphical Results Layers

The **Natural Frequency Analysis Module** generates one or more 2D result layers depending on the value specified in the **NF Result Count** input property. Each result layer shows the maximum displacement of each PCB element at a given natural frequency, as shown here:



In this example, the first natural frequency, at 200.13 Hz, is depicted. The displacement values are colored based on their magnitude, with the minimum displacement values colored blue and the maximum values colored red. Since the displacement results are abstract values, no exact values are displayed on the color legend.

9.1.3.3. 3D Model / Results Viewing

You can view the 3D FEA model and/or results by right-clicking the **Natural Freq** node in the Project Tree and selecting either the **View 3D Model** or **View 3D Results** option from the context menu. In both cases, the **Sherlock 3D Viewer** will be launched allowing you to interactively view the 3D data. See Viewing and Managing Analysis Results (p. 785) for more details.

Note:

In the **3D Viewer**, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at 20°C only.

9.1.3.4. Log Files

If the **Show FEA Logs** option is enabled in the **Settings > FEA Settings** dialog, then a **Log Panel** will be displayed along with the other analysis results containing the log generated by the FEA engine during the analysis process. Advanced users can review such log data to determine if everything was processed as expected and/or to trouble-shoot FEA processing issues.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.2. FEA- In-Circuit Testing (ICT)

In-Circuit Testing (ICT) uses a collection of test probes and test fixtures on one or both sides of a circuit card to test electrical connections during the manufacturing process. Each test probe exerts a mechanical force on a specific circuit card location, called the test point, as determined by the design. The combined effect of all these test point forces displaces the circuit card during the test, causing mechanical stresses to be experienced by each solder joint. If the stress values are high enough one or more solder joints could fail. The ICT Analysis module provided by Ansys Sherlock models the mechanical stresses exerted by the test points and fixtures and scores each circuit card part based on the predicted stresses for that part.

In this section, the following topics are covered:

9.2.1. ICT Input Data

9.2.2. ICT Analysis Properties

9.2.3. ICT Analysis Results

9.2.1. ICT Input Data

The **ICT Analysis Module** makes use of the following input data for the analysis calculations:

- Parts List

  Tip:

  The following section explains which part properties Sherlock uses in the ICT analysis: Tables: Required Part Properties per Analysis Type (p. 326).

- Size and location of all parts, plated through-holes, and cutouts
- Size and location of all test points and test fixtures
- Circuit card mechanical properties (stackup data)
- Circuit card outline
- Mesh Properties.

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

Warning:

Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate results, see the section Compatibility with Earlier Versions of Sherlock (p. 5).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.2.1.1. ICT Test Fixtures

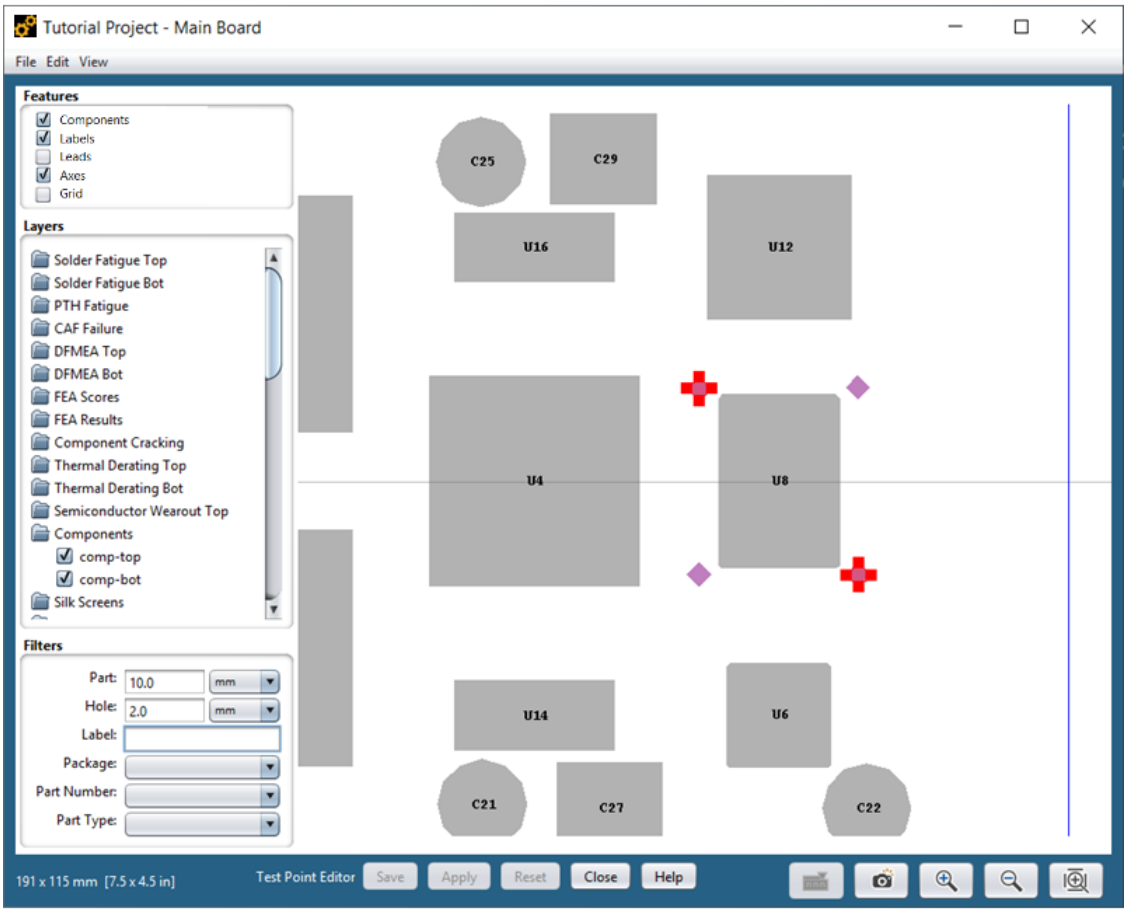
ICT analysis depends on the current placement and type of ICT Fixtures associated with the circuit card being analyzed. See the section on Mount Points & Fixtures (p. 479) for details concerning the creation and management of ICT Fixtures.

9.2.1.2. ICT Test Points

Much like ICT Test Fixtures, the test points used by the **ICT Analysis Module** can be added, modified and deleted using the **Test Point Editor** provided by the **CCA Layer Viewer**. To access the **Test Point Editor**:

- Double-click the **Inputs > Layers** entry for the circuit card being analyzed to display the **Layer Viewer**
- Select **Edit > Edit Test Points** from the main menu of the **CCA Layer Viewer**

At that point, the **Test Point Editor** buttons will be displayed at the bottom of the **Layer Viewer**, as shown in the following screen shot:



In this case, four test points are shown as purple diamonds located near the corners of part U8. Two of the test points have been selected for editing, as denoted by the red outlines and corner

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

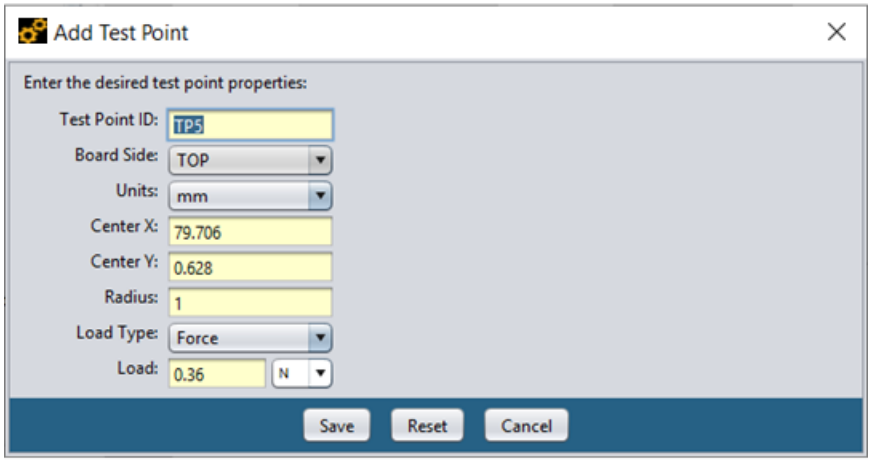
nodes. Single or multiple test points can be selected using the same keyboard and mouse iterations described for test fixtures.

Note:

By default, test points are displayed as purple diamonds in the **Layer Viewer**. You can customize the color using the **Settings > Color Settings** option in the **Main Menu**.

9.2.1.2.1. Adding a Test Point

To add a test point, simply right-click on the circuit card at the approximate location for the test point and select **Add Test Point** from the context menu. At that point, the **Add Test Point** dialog will appear, indicating the default properties to be assigned to the new test point, as shown below.



You may specify the exact location using the **Center X** and **Center Y** fields or you may simply move the fixture to the proper location after it has been created. Other than the **Board Side** and **Force** values, all other test point properties can be edited using either the dialog form or graphical controls (discussed in the next sub-section).

Note:

The **Test Point ID** must be a unique value for each test point. Sherlock automatically selects an available ID, but you are free to use whatever ID you want.

9.2.1.2.2. Modifying Test Point Properties

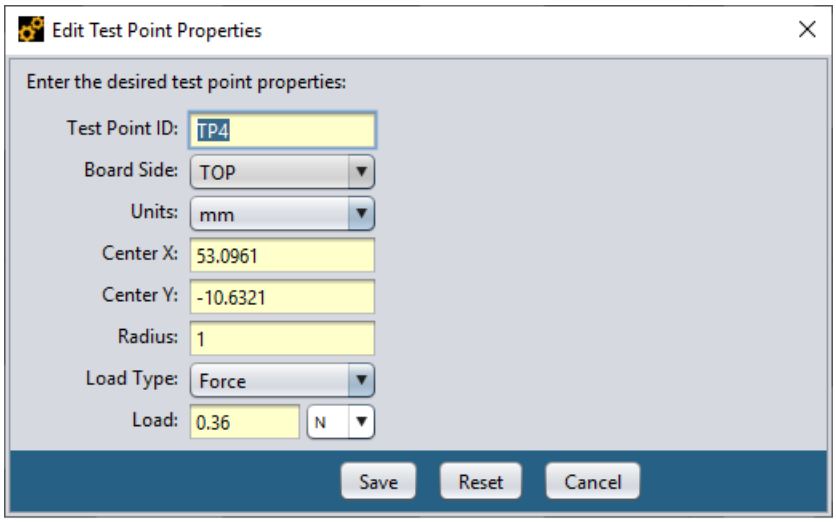
To move one or more selected test points, left-click inside of any selected test point and drag the test point(s) to the new location. To resize one or more test points, left-click any of the highlighted corners and drag that corner to change the size of all selected test points. Finally, to access the context menu for one or more selected fixtures, right-click inside of any selected fixture to display the following menu items:

- Edit Properties
- Move Test Point(s)
- Scale Test Point(s)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Delete Test Point(s)

Other than **Edit Properties**, the menu items are self-explanatory. When the **Edit Properties** menu item is selected for a single test point, the **Edit Test Point Properties** dialog will be displayed (see below) and will be identical to the **Add Test Point** dialog shown above. Simply modify the desired test point properties and press **Save** to update the test point.



When multiple test points are selected, only a subset of the test point properties will be editable, as shown here. For each property shown, if the value is the same for each selected test point, then that value will be displayed. Otherwise, <VARIOUS> will be shown in the property field to indicate that the selected test points have different values for that property.

To modify one or more property values for all selected test points, simply update the property field(s) and press the **Save** button. At that point, all selected test points will be updated. If <VARIOUS> is specified for a given property value, then no changes will be made to that property in any of the selected test points.

9.2.1.2.3. Saving Test Point Changes

After adding, modifying or deleting test points, you should press the **Save** or **Apply** buttons at the bottom of the **Layer Viewer** to save your changes. The **Apply** button will save the changes but keep the **Test Point Editor** open, allowing you to make additional changes. The **Save** button saves the changes and automatically closes the **Test Point Editor**.

9.2.1.2.4. Importing Test Point Locations

ICT Test Points can be imported from the following types of design files:

- Test Point (CSV) - comma separated file containing test point data
- Pick & Place (CSV) - comma separated file containing pick & place data
- Pick & Place (DELIMITED) - delimited file containing pick & place data
- Pick & Place (FIXED) - Fixed column file containing pick & place data
- Pick & Place (ODB++) - ODB++ file containing pick & place data

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the first (most common) case, a spreadsheet file is used to provide the test point data, while in all other cases the test point data is extracted from the Pick & Place data by looking for reference designators starting with a specific Test Point Prefix.

For example, consider the following CSV file:

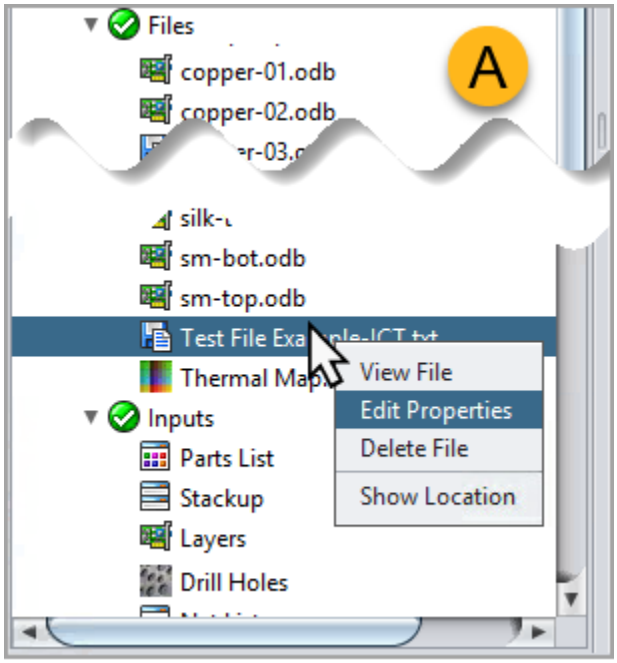
"Test Point Example"

"X","Y","Board Side","Force"

1,1,"TOP",5.2

-1,-1,"BOTTOM",0.36

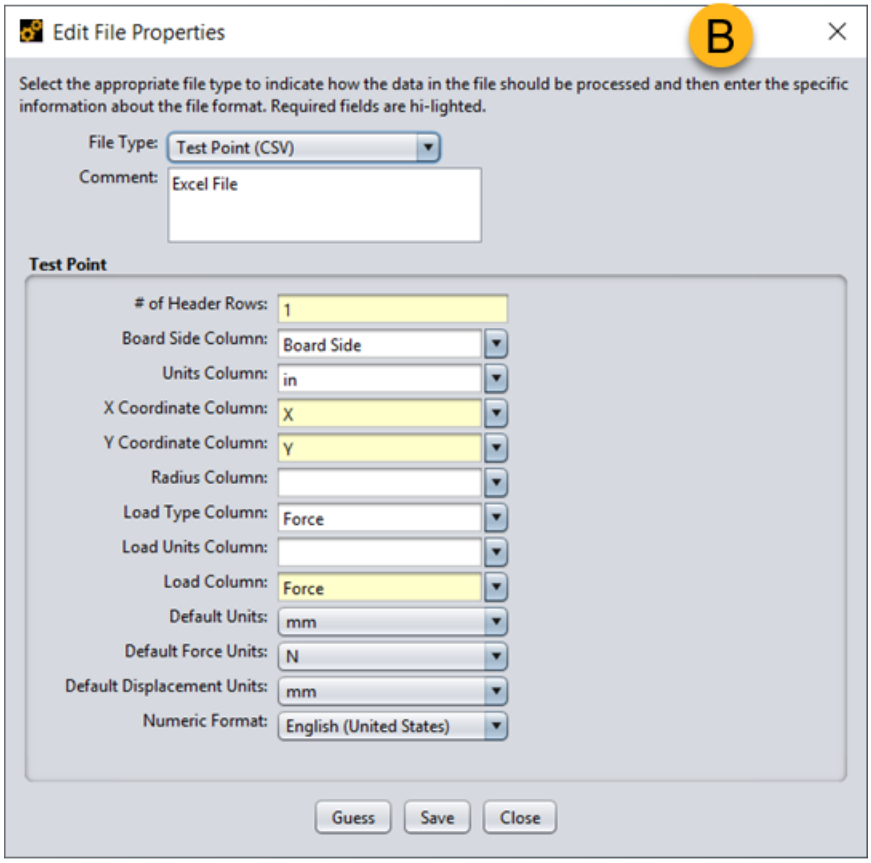
To add a file to the CCA, either select **CCA > Add Files** from the main Sherlock menu, or right-click **Files** and select **Add File(s)** from the **Project Tree** as shown below (A). To assign a file type, open the **Edit File Properties** dialog (B, below) by right-clicking the file, selecting **Edit Properties**, and then toggling the **File Type** drop-down menu.



The two test points are defined, providing their X, Y coordinates (required), board side (optional) and force (required). The test point data from this file can be imported into Sherlock by selecting the **Test Point (CSV)** file type in the **Edit File Properties** dialog shown below. At that point, the list of properties will be displayed, allowing you to specify the columns to be used for specific data. In many cases, Sherlock will automatically guess the appropriate columns, but you may have to specify one or more columns depending on the column headers defined in the file.

When the **Save** button is pressed, Sherlock will parse the file and add all the test points found. For identification purposes, Sherlock assigns a string ID starting with TP to all test points defined in the file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.2.1.2.5. Importing Test Points From Pick & Place Files

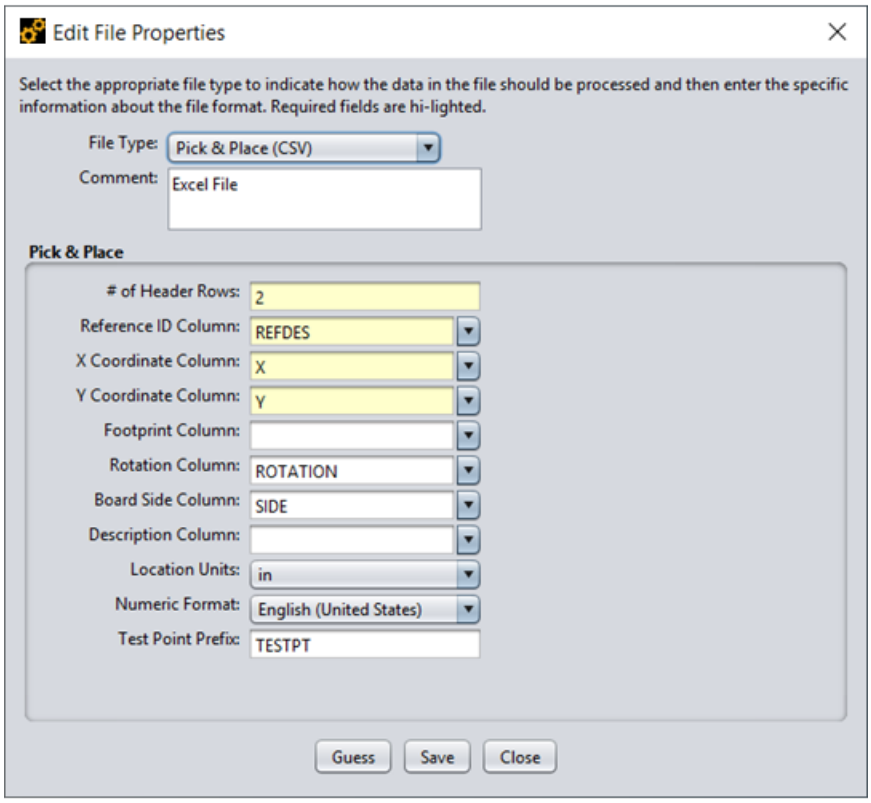
In some cases, the Pick & Place design files associated with a given circuit card will contain entries for some or all of the test points defined for that circuit card. In such cases, Sherlock can be used to automatically import test point definitions from the Pick & Place data and create corresponding test points with default properties. For example, consider the following Pick & Place spreadsheet file:

Component Pick & Place Data				
Total Components: 221				
REFDES	X	Y	ROTATION	SIDE
TESTPT1	1	1	0	TOP
TESTPT2	-1	-1	0	BOTTOM
C1	-2.7	-1.65	0	TOP
C2	-2.4	-1.65	0	TOP
C3	-2.4	-1.9	180	TOP

In this case, two test points are defined using the TESTPT prefix in their reference designators, providing their X,Y coordinates and the side of the board on which they are located.

After examining the file contents, you need only specify the prefix used by the test point definitions as one of the file properties in the Pick & Place (CSV) edit dialog, as shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When Sherlock imports the Pick & Place data it will automatically treat all reference designators starting with that prefix as a test point and will create a default test point accordingly. Then, you need only select groups of test points and set their forces to complete the input process.

9.2.1.3. ICT Mesh

The **ICT Analysis Module** creates a finite element model by creating a mesh that includes all selected parts, holes, cutouts, test fixtures and test points. You can display the current ICT mesh by selecting the **ICT Mesh** layer (if available) in the **Other Layers** folder of the **CCA Layer Viewer**. You may also re-create the ICT mesh at any time by selecting **Edit > Update ICT Mesh** from the **Edit** menu in the **CCA Layer Viewer**.

Note:

When examining the ICT mesh, you might notice that fixtures are always modeled by a collection of triangles just like parts, but no such triangles exist for test points. Test points are modeled by a single mesh node located approximately at the specified test point coordinates. A single node is used because the **ICT Analysis Module** assumes the force is applied at a single point.

9.2.2. ICT Analysis Properties

The ICT Analysis task can be executed whenever the following requirements are met:

- At least one test fixture is defined

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- At least one test point is defined

To edit the analysis properties, right-click the **ICT Analysis** entry in the **Project Tree** and select the **Edit Properties** menu item to display the ICT Analysis Properties dialog. As a convenience, you can also run the analysis immediately, using the last properties entered, by selecting the **Run Analysis Task** option from the context menu.

Note:

Sherlock reports ICT analysis results in terms of microstrain. You can enable or disable this feature using the **Engine Properties** dialog. (On the main menu, go to **Settings > FEA Analysis**.) See Analysis Properties (p. 448) in this chapter for more information.

9.2.2.1. Generated Data Source

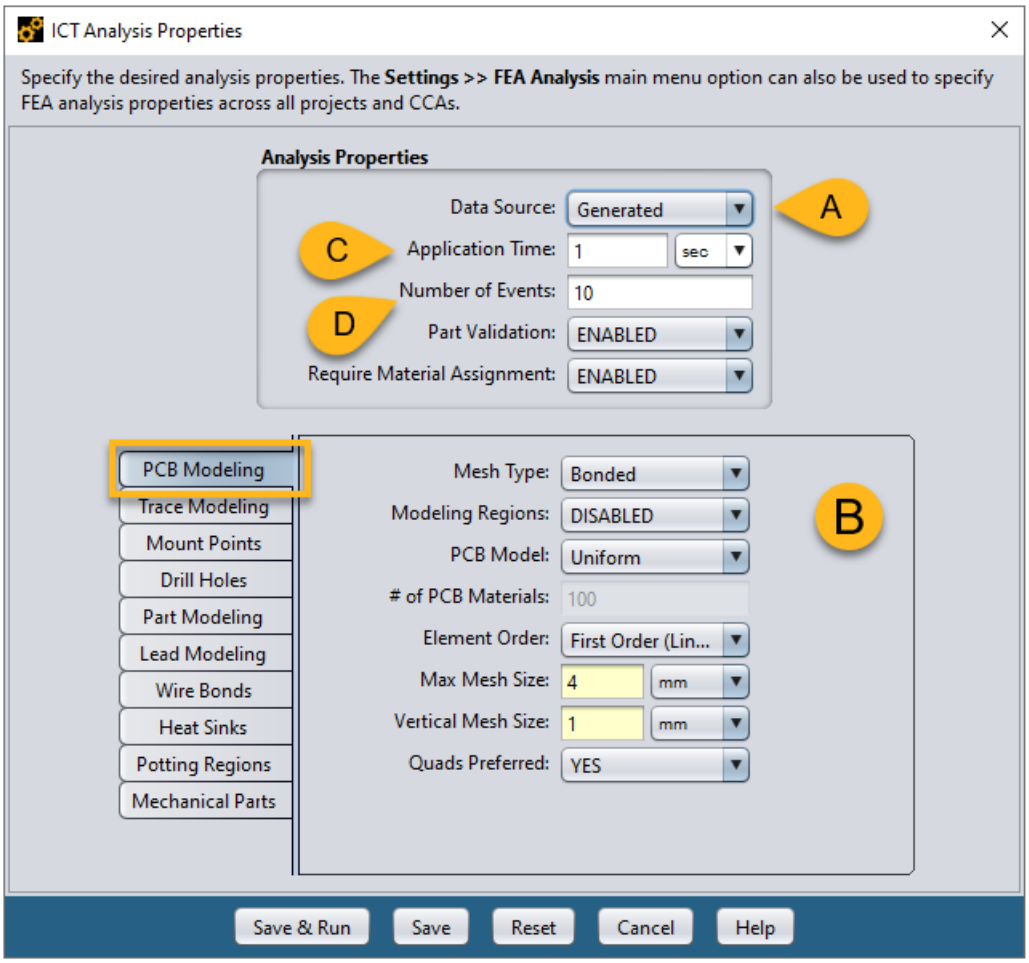
When using the **Generated** data source (A, below), you may specify mesh properties using the various settings in the **PCB Modeling** form (B). See the FEA Overview (p. 443) for more details.

Unlike other FEA analysis tasks, ICT analysis does not use any Life Cycle Events. To provide Time-to-Failure Calculations, the **Application Time** property combined with the maximum component strain are used.

Application Time (C) is the time it takes to apply the load– that is, the time between no load to full load.

Number of Events (D) specifies the number of times the load is applied.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.2.2.2. Imported Data Source

ICT analysis results can be imported into Sherlock by selecting **Imported** for the data source (A, below). When selected, you simply enter the locations and names of the FEA model and results files.

The supported file types are `*.db`, `*.dat`, and `*.rst`.

Tip:

For detailed, step-by-step instructions for importing results from Ansys Workbench, see the following section: FEA- Ansys Workbench™ Integration (p. 747).

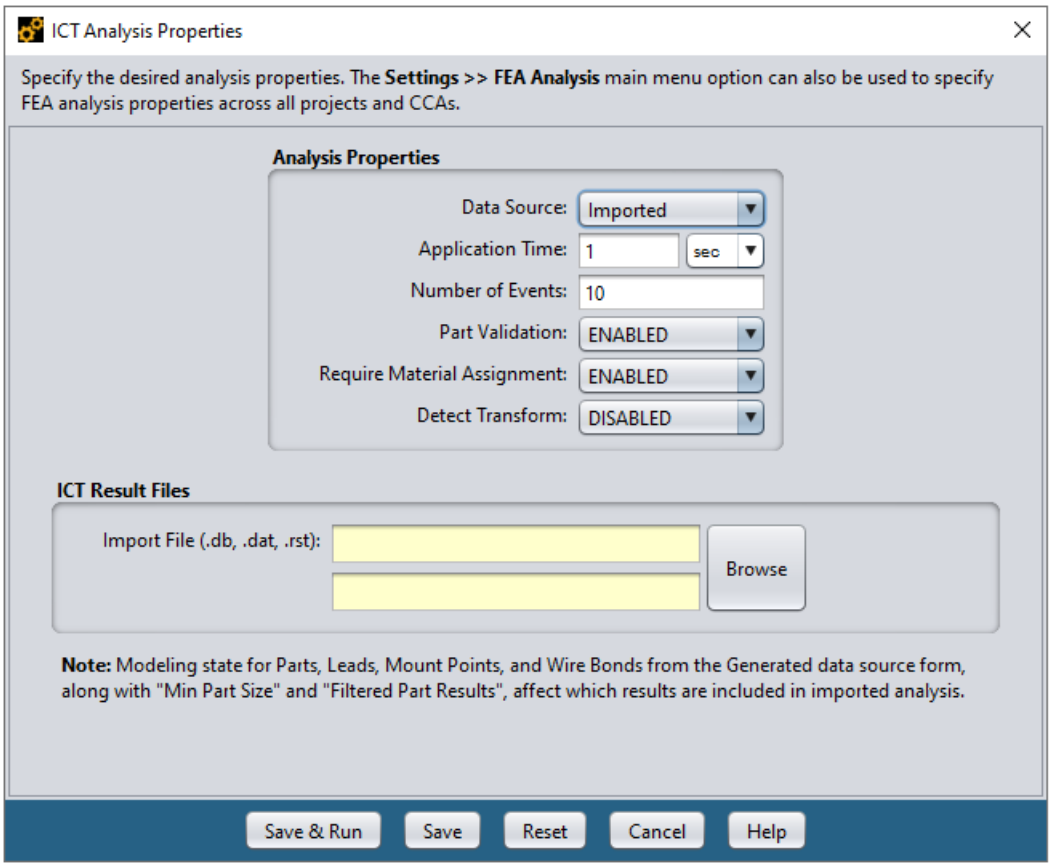
The **Detect Transform** property (B) is located under the **ICT Analysis Properties**. When the **Detect Transform** is **ENABLED**, Sherlock will attempt to detect any Transforms applied to the imported model and make adjustments if required. This ensures the imported model matches Sherlock's original model. Select **MANUAL** to enter Transform properties manually. If a Transform exists and is not detected, Sherlock will not be able to provide component results. When running the **Detect Transform** feature, Sherlock uses multiple threads by default. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **Save & Run** button is pressed, Sherlock will import both the model and the results to determine the reliability results.

Note:

If lead results are not generated after importing results which contain leads, do the following: Enable **Lead Modeling** on the **Generated Data Source** (p. 668) form (shown above) and perform the imported analysis again.



9.2.3. ICT Analysis Results

The results generated by the **ICT Analysis Module** show the maximum displacement and strain experienced by each component during the analysis, as well as the location and force or displacement of each test point. The analysis results are used to assign a score to each part and a combined score for the circuit card itself. Sherlock displays a summary panel showing the overall scores and range of bending forces experienced by the board, as well as a table showing individual results for all parts associated with the circuit card. We now describe each analysis result in more detail.

9.2.3.1. Summary Panel

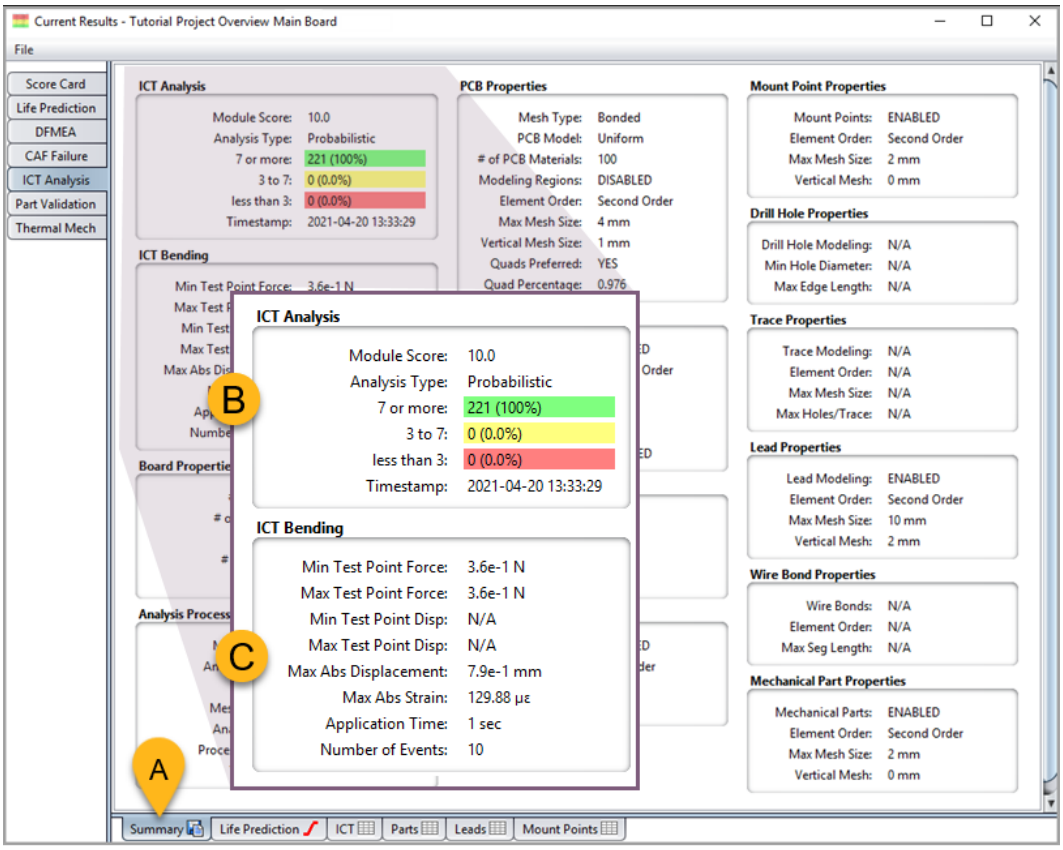
When the ICT Analysis process is finished, a GREEN check mark will appear next to the **ICT Analysis** entry in the **Project Tree**. Double-click that entry to display the **ICT Analysis Results** panel and select the **Summary** sub-tab (A, below).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Summary Panel** (B) shows the overall distribution of scores assigned to each part analyzed, along with the overall score assigned to the circuit card itself.

The **ICT Bending** (C) panel shows the minimum and maximum test point forces and test point displacement applied to the board, and the maximum displacement and strain predicted by the ICT analysis model. It also includes the **Application Time** and **Strain per Event** used as analysis inputs.

The **PCB Properties**, **Analysis Process**, and various **Mesh Properties** panels show the statistics for the number of inputs and properties used for the analysis.

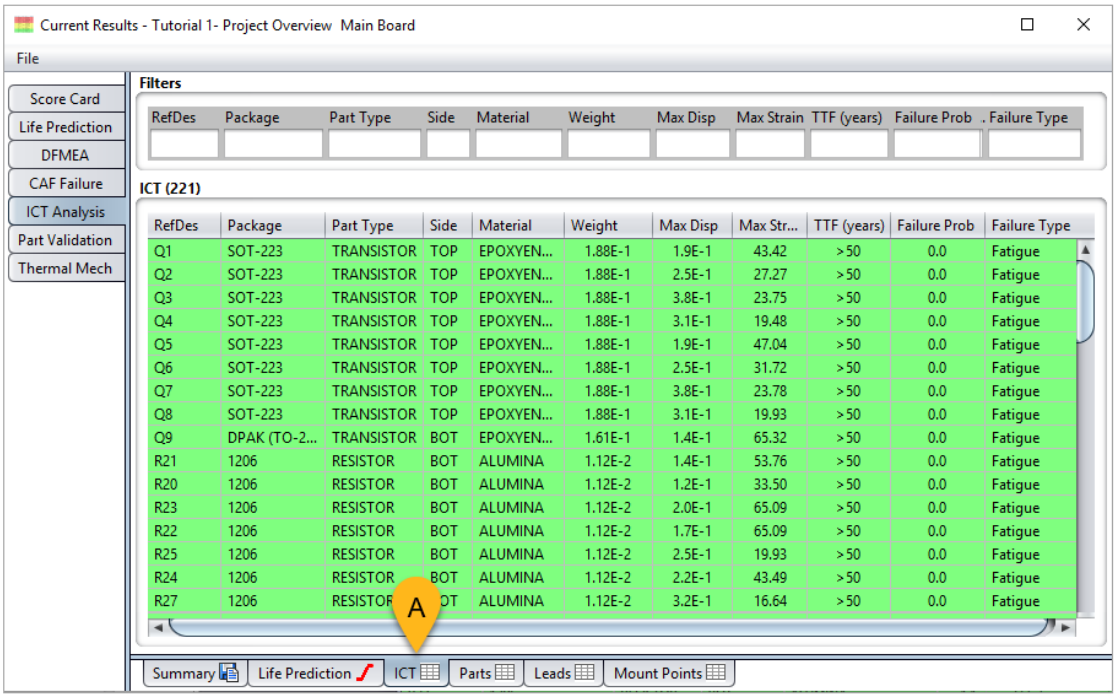


9.2.3.2. ICT Table

Select the **ICT Table** sub-tab (A, below) to examine the detailed results generated for each part analyzed.

The table rows are color-coded based on the score assigned to each part, which is based primarily on the maximum strain experienced by that part according to the ICT model. As with all other Sherlock tabular results, you can double-click any row to view the properties associated with a given part and/or select one or more rows to be exported to a CSV file. See the section on Data Export (p. 799) for more details.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



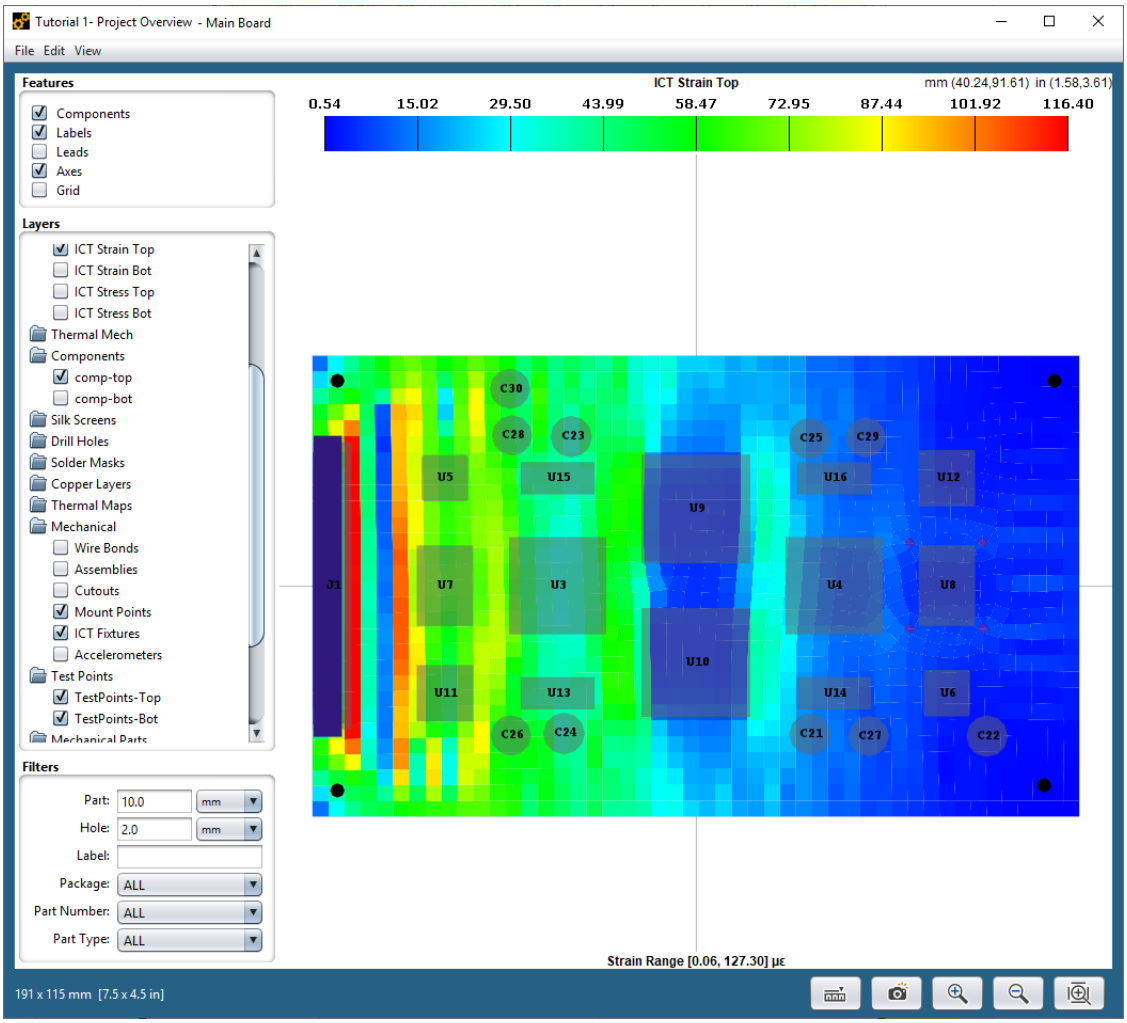
9.2.3.3. Graphical Results Layers

The analytic results and scores generated by the **ICT Analysis Module** can be viewed graphically using the **CCA Layer Viewer** shown below. The layer shows the combined strain (in both the X and Y directions) predicted over the board surface for the given test point forces:

In addition to the strain results, we've selected the **ICT Fixture**, **Test Point** and **Component** layers so that we can fully understand the results. With a single test fixture located along the left side of the board and four test points located near the right side of the board, the maximum strain values are seen near the text fixture, as expected. The red areas also show relatively high strain values in between the large components.

The raw results shown by the displacement and strain layers is useful for validating the analysis model, but they don't clearly show how each component is affected by the predicted strains. To see clearly how each part is affected, we can examine the score layers generated by the ICT Analysis Module. In this case, the score assigned to each top component is shown in a color-coded region.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



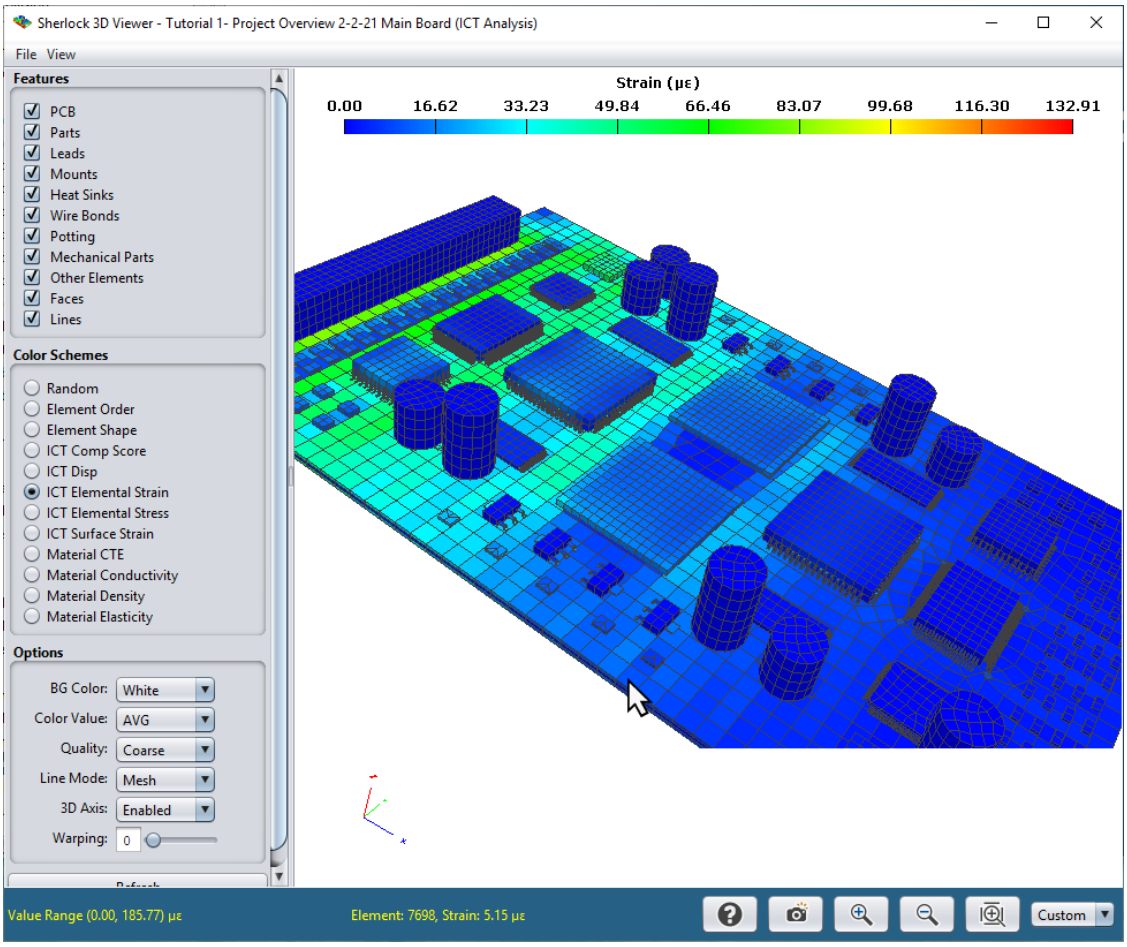
9.2.3.4. 3D Model / Results Viewing

You can view the 3D FEA model and/or results by right-clicking the **ICT Analysis** node in the **Project Tree** and selecting either the **View 3D Model** or **View 3D Results** option from the context menu. In both cases, the **Sherlock 3D Viewer** will be launched, as shown below, allowing you to interactively view the 3D data. See Viewing and Managing Analysis Results (p. 785) for more details.

Note:

In the **3D Viewer**, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at 20°C only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.2.3.5. Log Files

If the **Show FEA Logs** option is enabled in the **Settings > FEA ANALYSIS** dialog, then a **Log Panel** will be displayed along with the other analysis results containing the log generated by the FEA engine during the analysis process. Advanced users can review such log data to determine if everything was processed as expected and/or to trouble-shoot FEA processing issues.

9.3. FEA- Mechanical Shock

Mechanical shock is the sudden application of single or multiple, but non-periodic, physical loads due to acceleration or deceleration that results in significant displacement or deformation. Common events that induce mechanical shock include drops, crashes, firing, impact, or explosions. The performance of a solder joint when subjected to mechanical shock is primarily dictated by the ductility of the solder and the fragility of the interconnect structure (intermetallic region, interface between the bond bad, and the printed board, etc.). The strengths of these regions and the amount of stress transmitted to them during the Shock Event will determine whether failure occurs. This assessment is based on calculating the board strain (or curvature) for the shock pulse using a transient dynamic finite element ana-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lysis (FEA) and equations developed by Steinberg. This strain is compared to the maximum allowable strain, and the probability of failure of the component is predicted.

Tip:

Sherlock offers another option for running a Mechanical Shock analysis: It's a fast, non-FE analysis which generates predictions based on strain maps you provide—from an image file or a CSV file. See Strain Mapping in Sherlock (p. 769).

In this section, the following topics are covered:

9.3.1. Input Data

9.3.2. Mechanical Shock Analysis Properties

9.3.3. Mechanical Shock Analysis Results

9.3.1. Input Data

The **Mechanical Shock Analysis Module** makes use of the following input data for the analysis calculations:

- Life Cycle Reliability Goals and Mechanical Shock Events
- Parts List

Tip:

The following section explains which part properties Sherlock uses in the Mechanical Shock analysis: Tables: Required Part Properties per Analysis Type (p. 326).

- Size and location of all parts, through-holes, mount points, and cutouts
- Circuit card mechanical properties (stackup data)
- Circuit card outline
- Mesh properties
- Solder properties

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for this analysis module.

Warning:

Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate results, see the section Compatibility with Earlier Versions of Sherlock (p. 5).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.1.1. Mechanical Shock Events

The external loads used during Mechanical Shock analysis are based by one or more **Mechanical Shock Events** defined in the project **Life Cycle**. See Life Cycle Management (p. 343) for details.

9.3.1.2. Mount Points

Mechanical Shock analysis depends on the current placement and type of mount points associated with the circuit card being analyzed. See the section on FEA Mount Points & Fixtures (p. 479) for details concerning the creation and management of mount points.

9.3.2. Mechanical Shock Analysis Properties

To specify all input properties and perform mechanical shock analysis, right-click the **Mechanical Shock** node in the Project Tree and select the **Edit Properties** option from the context menu. As a convenience, you can also run the analysis immediately, using the last properties entered, by selecting the **Run Analysis Task** option from the context menu.

Note:

Sherlock reports Mechanical Shock analysis results in terms of microstrain. You can enable or disable this feature using the **Engine Properties** dialog. (On the main menu, go to **Settings > FEA Analysis**.) See Analysis Properties (p. 448) in this chapter for more information.

Topics in this section:

9.3.2.1. Generated Data Source

9.3.2.2. Imported Data Source

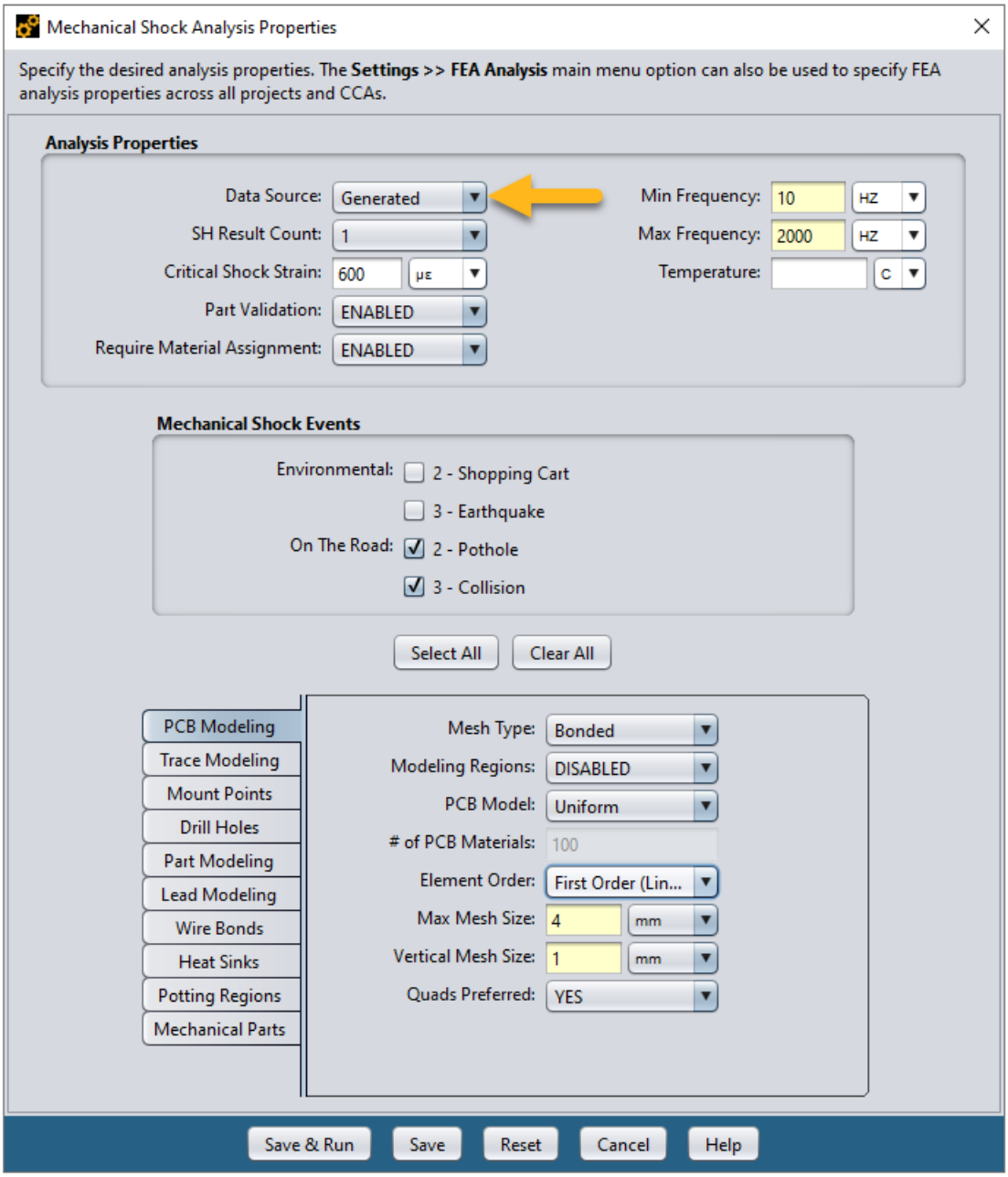
9.3.2.3. Strain Map Data Source

9.3.2.4. LS-DYNA Imported Data Source

9.3.2.1. Generated Data Source

When using the **Generated** data source, you may specify the type of model to be generated using the **Mesh Type** dropdown menu (Merged or Bonded) and other mesh properties using the **Mesh Properties** form. Each mesh property form is selectable by the indicated tabs. See the FEA Overview (p. 443) for more details.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The **Analysis Properties** form allows users to specify the range of frequencies used for a natural frequency scan, which is required for all FEA processing and is automatically performed by Sherlock.
- The **SH Result Count** property specifies the number of shock displacement result layers to be generated. Results are chosen based on maximum nodal displacement in the result step.
- **Critical Shock Strain** is the maximum strain a component can experience without being over-stressed. By default, the field is blank. When blank, Sherlock calculates critical shock strain for each component based on its dimensions.

  If the FE analysis determines that the PCB surface strain under the component's footprint is greater than the calculated critical shock strain *and* if the cycles-to-failure for the PCB surface

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

strain of the component's footprint is lower than 1000 cycles, then Sherlock classifies the failure as overstressed.

In cases where the calculated critical shock strain is greater than the PCB surface strain *and* the calculated cycles-to-failure is greater than 1000, then this condition is *not* considered to be an overstress failure. Generally, a component surviving 1000 cycles is not considered prone to overstress.

However, when you explicitly provide a critical shock strain value and the PCB surface strain under the component's footprint is higher than the critical shock strain, then Sherlock will *not* overwrite your definition of overstress and will treat the component as having failed from overstress.

- The **Mechanical Shock Events** form displays a list of all Shock Events (p. 347) you have created. (See Shock Event in the *Sherlock User's Guide* (p. 347) for a review on creating Life Cycle Events.) You can select which Events you want included in the analysis. Sherlock will automatically perform a separate analysis for each Mechanical Shock Event selected in the list.

Tip:

The **Numeral Damping** setting (**Damping Modifier** in earlier versions of Sherlock) is located in the **Sherlock Settings** dialog (**Main Menu > Settings > FEA Analysis** tab **> Analysis Properties** panel). See FEA Analysis Settings (p. 446) for additional information.

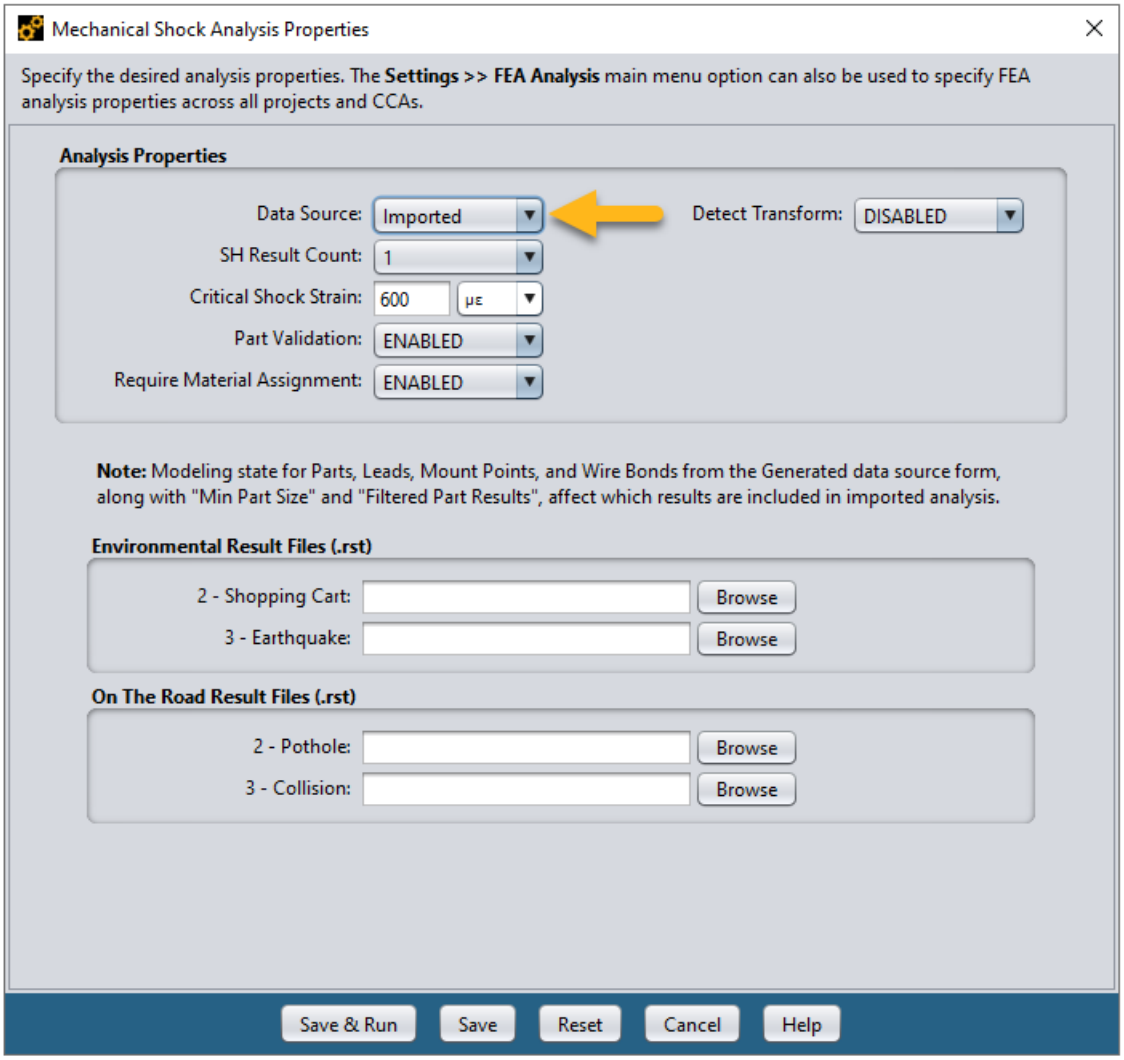
9.3.2.2. Imported Data Source

Note:

If you wish to import LS-DYNA analysis results, see the section below, LS-DYNA Imported Data Source (p. 680).

Mechanical Shock analysis results can be imported into Sherlock by selecting the **Imported** data source. When selected, as shown here, you must enter the locations and names of the model and results files (`.rst`) used for the Mechanical Shock Event analysis runs.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tip:

For detailed, step-by-step instructions for importing results from Ansys Workbench, see the following section: FEA- Ansys Workbench™ Integration (p. 747).

When the **Detect Transform** is ENABLED, Sherlock will attempt to detect any Transforms applied to the imported model and make adjustments if required. This ensures the imported model matches Sherlock's original model. Select MANUAL to enter Transform properties manually. If a Transform exists and is not detected, Sherlock will not be able to provide component results. When running the **Detect Transform** feature, Sherlock uses multiple threads by default. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

If any of the Event result file entries are blank, then Sherlock will assume that no analysis was performed for those Events.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **Save & Run** button is pressed, Sherlock will import both the model and the results to determine the reliability results.

Note:

When importing data, it is your responsibility to assign the proper results file to each Event. Sherlock has no other way of determining the association.

Note:

If lead results are not generated after importing results which contain leads, enable **Lead Modeling** on the **Generated Data Source** (p. 676) form and perform the imported analysis again

---

9.3.2.3. Strain Map Data Source

Select the **Strain Map Data Source** to generate reliability predictions based on data from strain map images and CSV files. See Strain Mapping in Sherlock (p. 769) below.

9.3.2.4. LS-DYNA Imported Data Source

To import LS-DYNA analysis results, select the **LS-DYNA Data Source** as shown below. This selection is available only for Mechanical Shock Analysis, and you must have selected Ansys as the **Analysis Engine** (see the section above, Selecting the FEA Engine (p. 445)).

Using the interface shown below, enter the directory that contains all the `d3plot` and `input.k` files generated from the LS-DYNA analysis.

Note:

Before importing LS-DYNA results, decide what kind of results you want to see. Sherlock gives you two options: **Max Over Time** or **Max Displacement Time**.

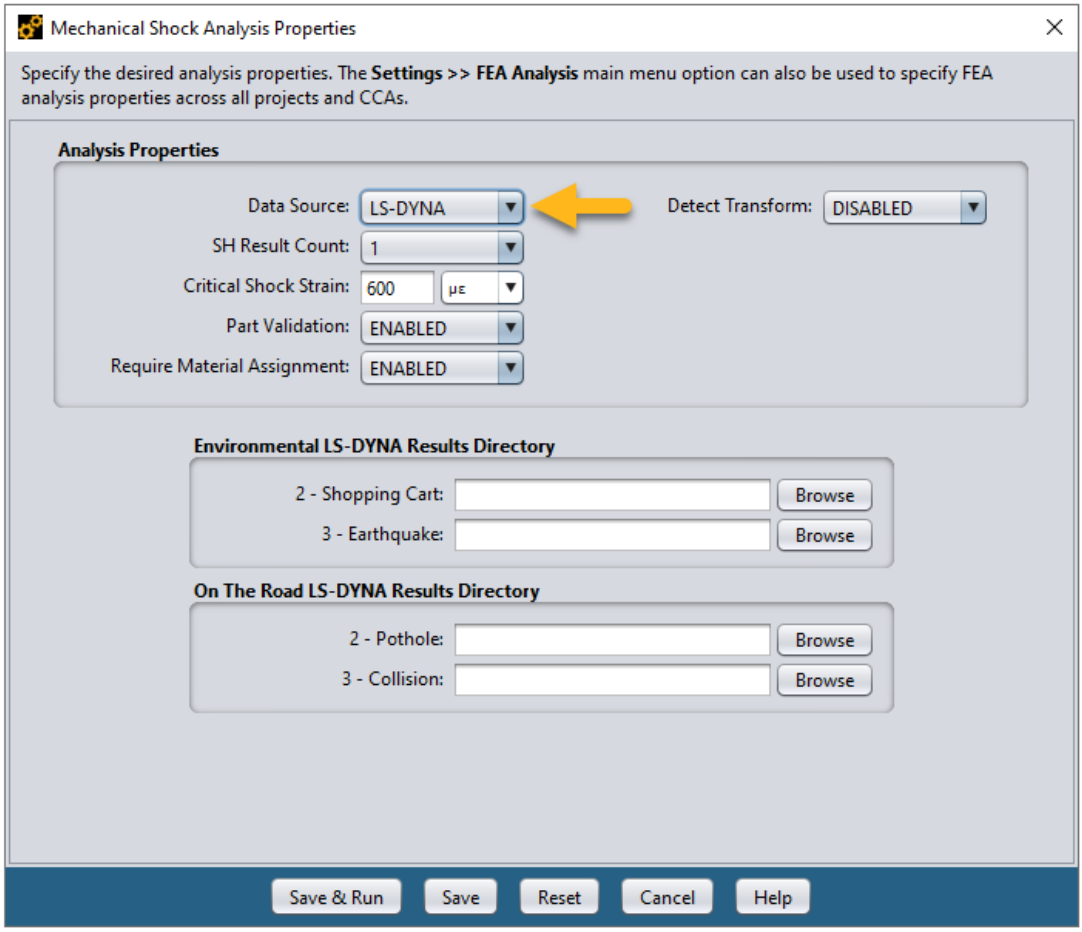
- **Max Over Time**: When reporting strain for an element, Sherlock considers all time steps and reports the maximum stress the element saw. This means the strain result for one element may represent a different time step than that of another element.
- **Max Displacement Time**: The reported strain results for all elements will come from the same time step—the time when maximum displacement occurred.

The **LS-DYNA Strain Settings** option is found in Sherlock's **Settings** menu (p. 77). From the Main Menu: **Settings** > **FEA** > **Analysis Properties** > **LS-DYNA Strain Import**.

Note:

When LS-DYNA is selected as the **Data Source**, Sherlock will not calculate strain results for orthotropic materials.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When the **Detect Transform** is ENABLED, Sherlock will attempt to detect any Transforms applied to the imported model and make adjustments if required. This ensures the imported model matches Sherlock's original model. Select MANUAL to enter Transform properties manually. If a Transform exists and is not detected, Sherlock will not be able to provide component results. When running the **Detect Transform** feature, Sherlock uses multiple threads by default. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

If any of the Event result file entries are blank, then Sherlock will assume that no analysis was performed for those Events.

When the **Save & Run** button is pressed, Sherlock will import both the model and the results to determine the reliability results.

Note:

Sherlock currently supports only the standard solid elements with one point integration. For additional information on this, refer to the following sections in the LS-DYNA Keyword User's Manual:

- Volume 1, section 1-357 for a brief explanation of solids
- Volume 1, section 37-68 on SECTION_SOLID

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.3. Mechanical Shock Analysis Results

The **Mechanical Shock Analysis Module** generates the following types of results:

- Overall summary of the scores assigned to each part analyzed,
- Maximum displacement and strain values across all parts and Shock Events analyzed,
- Table of maximum displacement and strain values for each part analyzed, and
- Graphical layers showing color-coded part scores (top and bottom sides), and
- Graphical layers showing maximum displacement and strain during Shock Events.

Note:

Remember, when you view the analysis results from an imported LS-DYNA file, how the results are displayed depends on which option you chose: **Max Over Time** or **Max Displacement Time**. These settings are explained above in LS-DYNA Imported Data Source (p. 680).

We'll now look at each of these types of results.

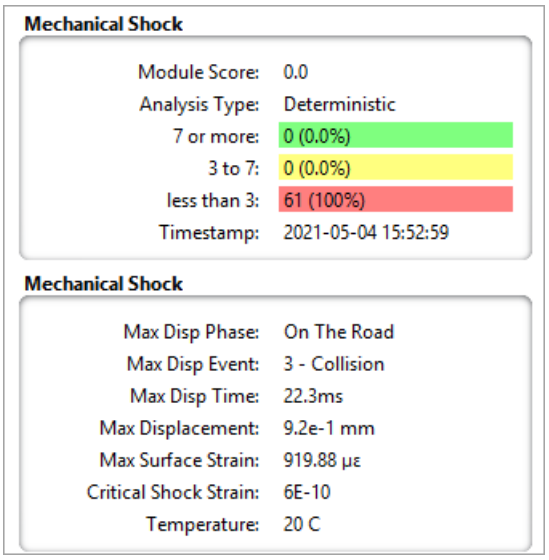
9.3.3.1. Summary Panel
9.3.3.2. Tabular Mechanical Shock Results
9.3.3.3. Life Prediction Curve
9.3.3.4. Graphical FEA Scores
9.3.3.5. Graphical FEA Results
9.3.3.6. 3D Model / Results Viewing
9.3.3.7. Log Files

9.3.3.1. Summary Panel

The **Summary Panels**, shown below, for the **Mechanical Shock Analysis Module** provide an easy-to-read overview of the results for all parts analyzed. A break-down of the scores assigned to each part analyzed is provided, clearly showing the relative number of problems found.

The second summary panel also shows the **Life Cycle Phase** and **Life Cycle Shock Event** that caused the maximum displacement of the circuit card. The time of maximum displacement (relative to the start of the shock pulse defined for that Event), as well as the predicted displacement and resulting strain, is listed. Although each of the Shock Events contributes to the predicted lifetime for the parts, you can use these maximum values as a sanity check of the results based on the **Life Cycle** definitions.

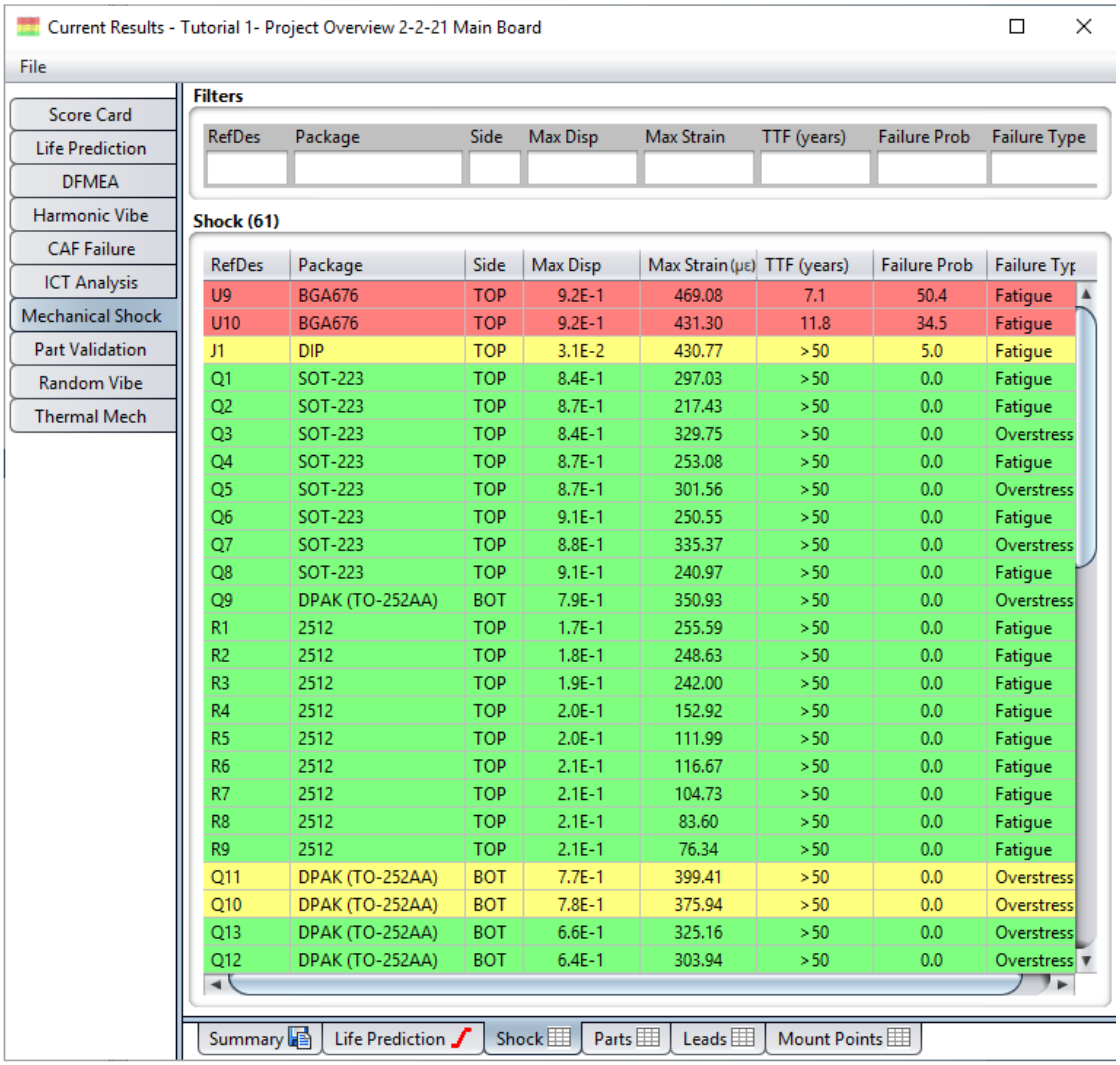
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.3.2. Tabular Mechanical Shock Results

Select the Shock sub-tab at the bottom of the results panel to view the detailed results for each part analyzed. The color-coded tabular results produced by the **Mechanical Shock Analysis Module** provide a convenient way to focus on the specific parts that may fail prematurely because of Mechanical Shock Events.



As with other color-coded displays in Sherlock, RED means that the predicted score is less than 3 (less than a 3X safety factor), YELLOW means the score is between 3 and 7, and GREEN means that the score is 7 or more, indicating there is a very small probability that the part will fail. In this case, we see that two parts are predicted to have significant problems, falling well short of the desired **Life Cycle Reliability Goals**, with many others in the Warning range.

Note:

The **Settings > Score Settings** editor can be used to change the score-color mapping to suit your specific needs.

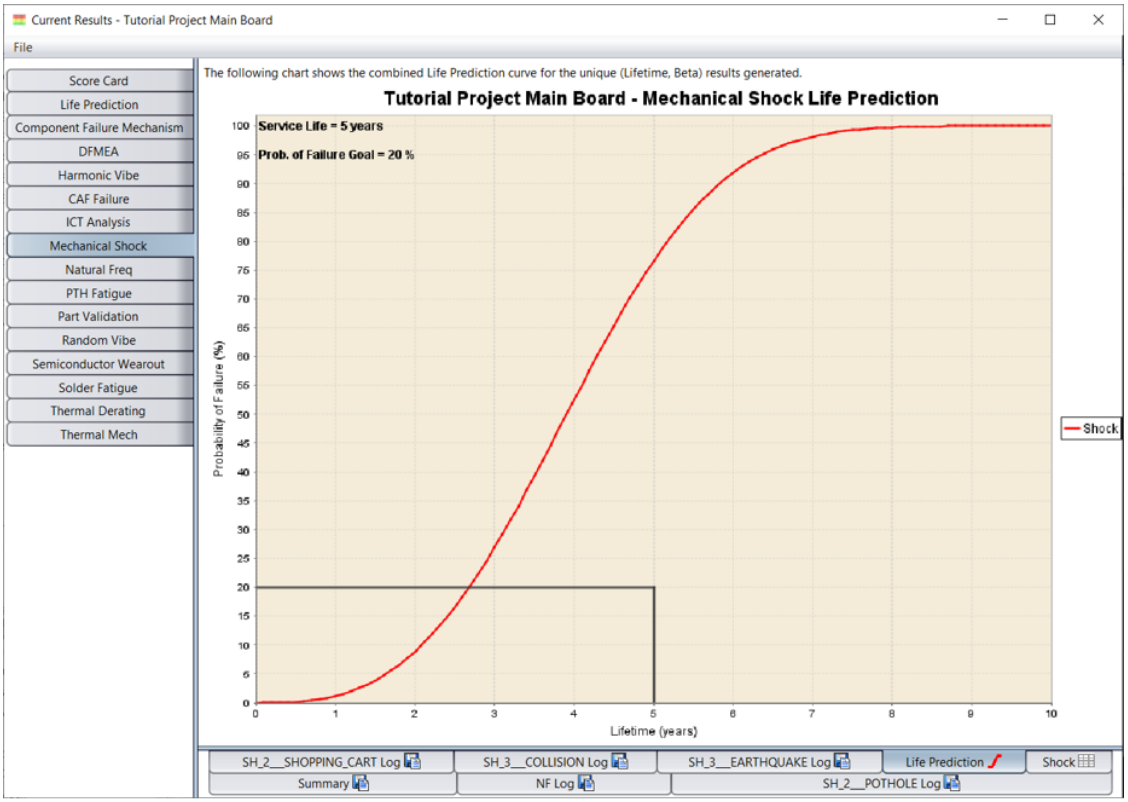
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can double-click any row in the table to view all the properties associated with a selected part. When viewing current results, the properties will be from the current **Parts List.** When viewing archived results, the properties will be from a snapshot of the **Parts List** taken at the time the archive was made.

You can export one or more rows from the results table by right-clicking a row and selecting the Export Table option from the context menu. See the section on Reporting and Data Export (p. 799) more details. You may also view the life prediction chart for any selected part by selecting the View Life Prediction option from the context menu on the table.

9.3.3.3. Life Prediction Curve

A Life Prediction curve can be generated for the circuit card itself, based on the **Life Prediction** curves for each of the parts analyzed, as shown here:



Note:

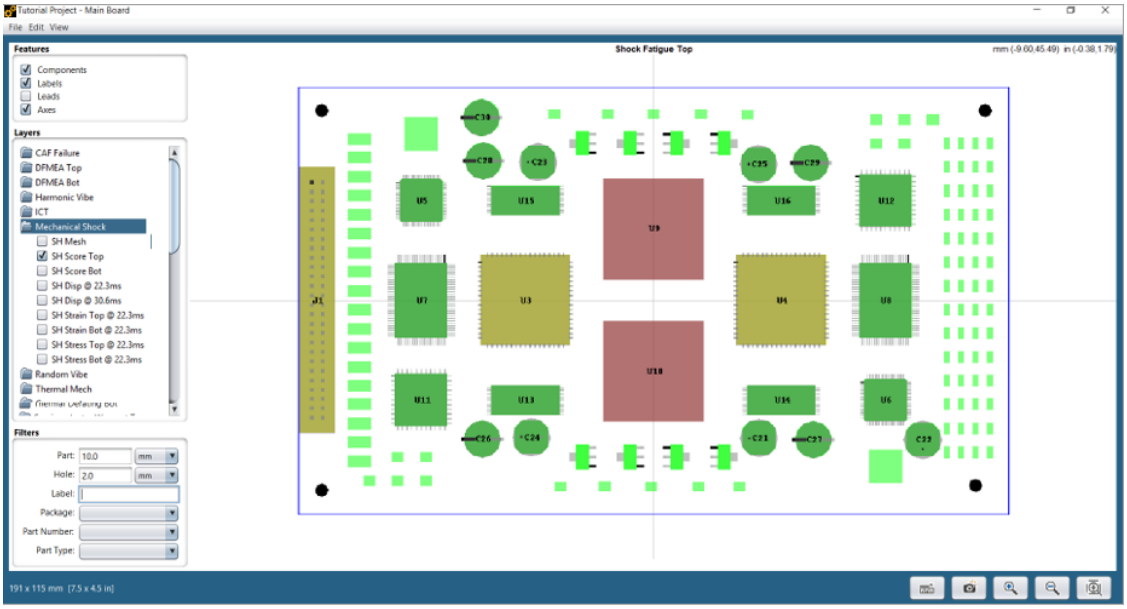
Time-to-failure calculations for leads are not available in the Mechanical Shock Analysis. It is planned as a future enhancement.

The **Life Prediction** chart shows all the critical information about circuit card reliability over a long period of time. The rectangular region in the lower left-hand corner of the chart graphically shows the desired **Reliability Goals** for this project, bounded on top by the desired **Probability of Failure** and bounded on the right by the desired **Service Life**. If the curve stays below the top line up to the **Service Life**, then the circuit card is predicted to meet its goals. In this case, the circuit card fails to meet its reliability goals when subjected to the Mechanical Shock Events defined in the **Life Cycle**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.3.4. Graphical FEA Scores

The **Mechanical Shock Analysis Module** generates graphical layers depicting component scores for the top and bottom sides of the PCB. Such displays can be used to quickly correlate potential problems with geographic and/or dimensional properties, as shown here:

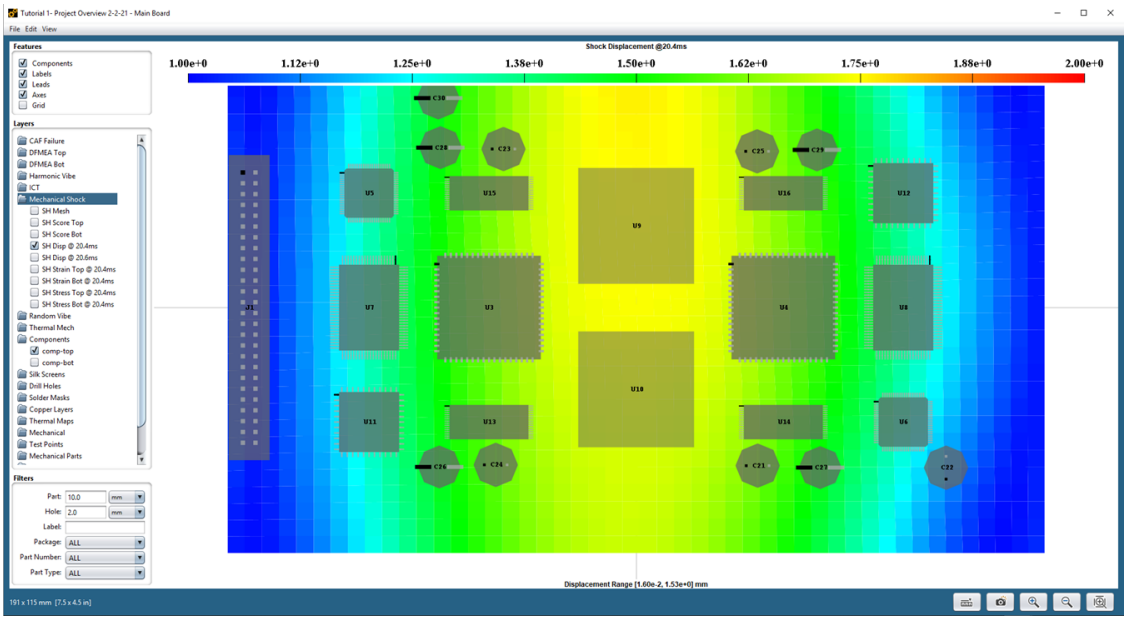


In this case, the color-coded FEA Scores – SH Top layer clearly shows the problem parts. It also shows the relative scores of all parts analyzed. Notice, the colored-region for each part analyzed are always displayed in the score layer, even though we are filtering parts smaller than 5mm in the **Components – comp-top** layer. We see that the troubled parts are located symmetrically around the center of board, almost exactly half-way between the center of the board and the 4 corner mount points. The regions in which the parts are located must be experiencing the highest strain during the shock pulse.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.3.5. Graphical FEA Results

The **Mechanical Shock Analysis Module** also generates graphical result layers showing the maximum displacement and strain values experienced during the worst Shock Event analyzed. For example, the following image shows the maximum displacement values, which occurred 20.4 ms after the start of the Shock Wvent (the **Summary Panel** indicates which Shock Event caused by maximum displacement):



The color legend indicates the range of displacement values being displayed (in this example, from 1.00 mm to 2.00 mm). You can customize the color range using the **Settings > Color** dialog. See the FEA Overview (p. 443) for more details.

9.3.3.6. 3D Model / Results Viewing

You can view the 3D FEA model and/or results by right-clicking the **Mechanical Shock** node in the Project Tree and selecting either the **View 3D Model** or **View 3D Results** option from the context menu. In both cases, the Sherlock 3D Viewer will be launched allowing you to interactively view the 3D data. See Viewing and Managing Analysis Results (p. 785) for more details.

Note:

In the **3D Viewer**, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at 20°C only.

9.3.3.7. Log Files

If the **Show FEA Logs** option is enabled in the **Settings > FEA Analysis** dialog, then a **Log Panel** will be displayed along with the other analysis results containing the log generated by the FEA engine during the analysis process. Advanced users can review such log data to determine if everything was processed as expected and/or to trouble-shoot FEA processing issues.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4. FEA- Harmonic Vibration

Solder joints provide electrical, thermal, and mechanical connections between electronic components and a printed board. When the printed board is subjected to vibration, it will experience global and local changes to the board shape and curvature. The degree of bending will be different for specific components and the area of the printed board to which they are attached. This behavior will introduce strain into the second-level solder joint. With repeated exposure, damage will accumulate, leading to crack propagation and eventual failure of the solder joint. Vibration-induced solder joint fatigue is influenced by the type of vibration, the shape of the vibration spectrum, the size and shape of the printed board, printed board in-plane material properties, support conditions, component design, component material properties, location of the component, solder joint geometry, and solder joint material. Ansys Sherlock calculates time to failure using a modified Steinberg that takes board-level strain into account.

This chapter covers Harmonic Vibration analysis.

Random Vibration analysis is treated in a separate chapter: See FEA- Random Vibration (p. 698).

In this section, the following topics are covered:

9.4.1. Harmonic Vibe Input Data

9.4.2. Harmonic Vibration Analysis Properties

9.4.3. Harmonic Vibration Analysis Results

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.1. Harmonic Vibe Input Data

The Harmonic Analysis module make use of the following input data for the analysis calculations:

- Life Cycle Reliability Goals and Harmonic/Random Vibration Events
- Parts List

> **Tip:**
>
> The following section explains which part properties Sherlock uses in the Harmonic Vibe analysis: Tables: Required Part Properties per Analysis Type (p. 326).

- Size and location of all parts, plated through-holes, mount points and cutouts
- Circuit card mechanical properties (stackup data)
- Circuit card outline
- Mesh properties
- Solder properties

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for the analysis module.

> **Warning:**
>
> Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate results, see the section Compatibility with Earlier Versions of Sherlock (p. 5).

9.4.1.1. Harmonic Events

The external loads used during Vibration analysis are based on one or more **Harmonic Events** (p. 354) defined in the project **Life Cycle**. See the Life Cycle Management (p. 343) section for details.

9.4.1.2. Reuse Modal Results Setting

The **Reuse Modal Results** feature, when enabled, significantly reduces solve time when running a Harmonic or Random Vibration analysis that has multiple Events. By default, Sherlock runs a modal (Natural Frequency) analysis prior to each Vibration Event. But when **Reuse Modal Results** is activated, Sherlock runs only one modal analysis and reuses the results for all Vibration Events.

The **Reuse Modal Results** option is located in the **FEA Settings/Engine Properties** panel (p. 446) of the **Sherlock Settings** window. To use this feature, the **FEA Engine** must be set to **Ansys**, also in the **Engine Properties** panel.

If you intend to import results with the **Reuse Modal Results** setting enabled, note the following:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If **Save Modal Results** is not enabled in Ansys Mechanical, then the `.mode` file must be in the same import directory.
- For the Harmonic Vibration analysis, you can use the Modal Reuse option with the **Use Lead Nodal Forces** option.

9.4.1.3. Mount Points

Harmonic Vibration Analysis depends on the current placement and type of mount points associated with the circuit card being analyzed. See the section on Mount Points and Fixtures (p. 479) for details concerning the creation and management of mount points.

9.4.2. Harmonic Vibration Analysis Properties

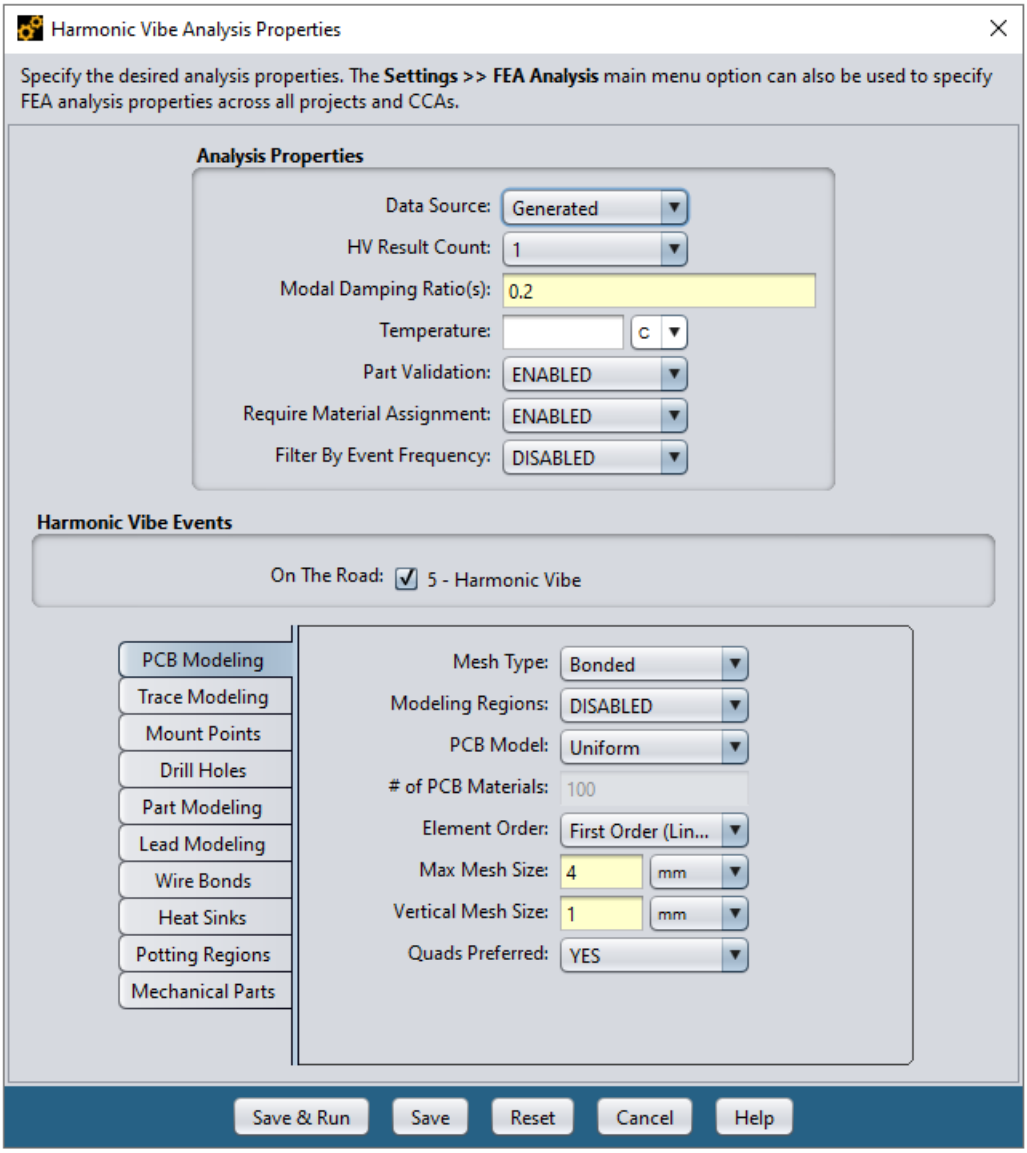
Vibration analysis properties can be specified by right-clicking the **Harmonic Vibe** node in the **Project Tree** and selecting the **Edit Properties** option from the context menu. You may also run the analysis immediately, using the last properties entered, by selecting the **Run Analysis Task** option from the context menu.

9.4.2.1. Generated Data Source

When using the **Generated** data source, users may specify mesh properties using the mesh properties forms. Each form is selectable by the indicated tabs. The **PCB Modeling** mesh properties can be seen in the image below. See the FEA Overview (p. 443) for more details.

The **Analysis Properties** form allows users to specify the range of frequencies to be scanned to determine the natural frequency of the circuit card. A natural frequency scan is required for harmonic vibration analysis and is automatically performed by Sherlock. The range defined must be larger than the range for any Harmonic Events being used for the analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **HV Result Count** property indicates the number of modal peaks per Harmonic Vibe Event Sherlock uses when computing damage. It also determines the number of result layers (which show maximum displacement) Sherlock will generate from the Harmonic Vibe Event. When choosing which results to display, Sherlock selects those frequencies which resulted in maximum global and local deflections.

The **Damping Modifier** specifies the damping of the circuit card assembly. A damping modifier of 0.1 represents very little damping and is akin to rigid mounting. A damping modifier of 1.0 is more representative of chassis mounted circuit card assemblies and will lower the transmissibility.

Temperature: Specify the temperature to be used when generating temperature-dependent properties. The default is 20°C.

Part Validation: When ENABLED, Sherlock runs a Part Validation analysis prior to the FEA analysis.

Require Material Assignment: When enabled, Sherlock generates an error and will halt the FEA analysis if a material is set to DEFAULT.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filter by Event Frequency: When ENABLED, any frequencies found in the Imported Results are ignored if they do not fall within the frequency range defined by the selected Harmonic Profile.

The **Harmonic Vibe Events** form provides a list of all Harmonic Events (p. 354) currently defined in the project **Life Cycle**. Sherlock will automatically perform a separate vibration analysis for each Vibration Event selected in the list. Reliability results will then be based on those results. This allows users to ignore specific Events during an analysis run or determine the effects of just one or two Events on the reliability results.

When running Harmonic Vibration, make sure the frequency range (min frequency and max frequency), at the very least, covers the expected natural frequency range. If you don't know the natural frequency range, you can run a Natural Frequency analysis before proceeding with Harmonic Vibration

---

Note:

If any mode of the natural frequency occurs outside the specified frequency range, the analysis may be incorrect. Make sure the frequency range encompasses all essential modes.

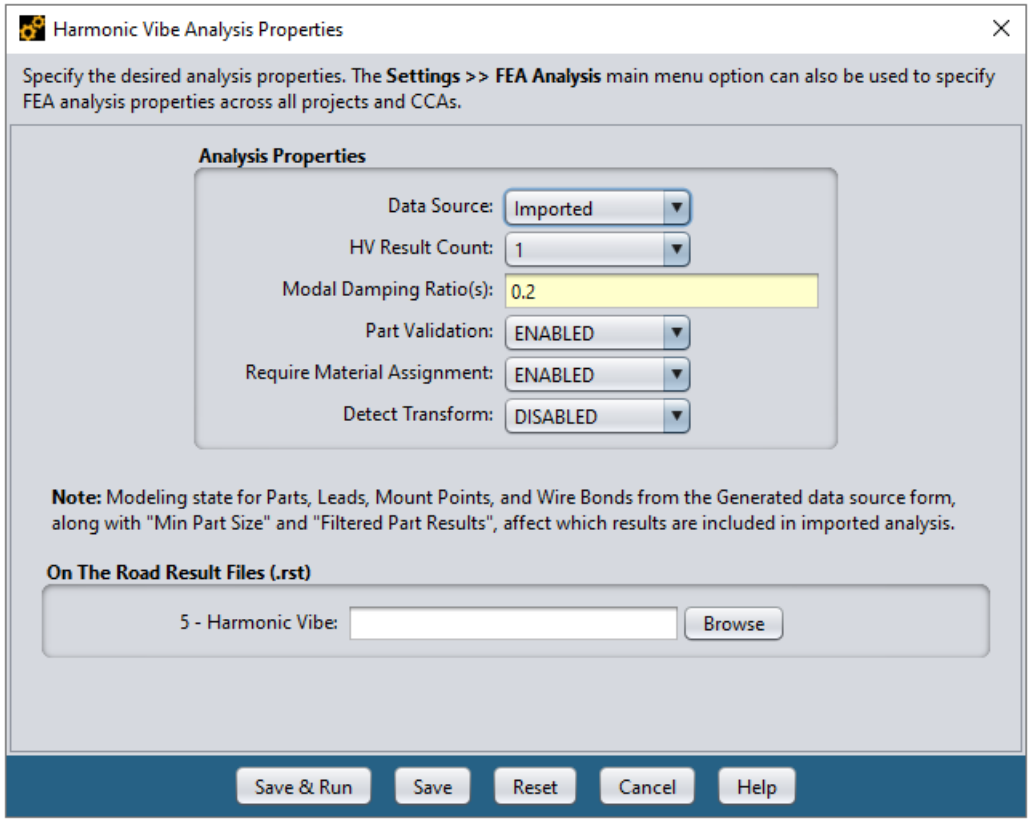
9.4.2.2. Imported Data Source

Vibration analysis results (`.rst`) can be imported into Sherlock by selecting the **Imported** data source. When selected, as shown below, enter the results files used for both the Natural Frequency analysis and all the Vibration Event analysis runs.

Tip:

For detailed, step-by-step instructions for importing results from Ansys Workbench, see the following section: FEA- Ansys Workbench™ Integration (p. 747).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When the **Detect Transform** is ENABLED, Sherlock will attempt to detect any transforms applied to the imported model and make adjustments if required. This ensures the imported model matches Sherlock's original model. Select MANUAL to enter Transform properties manually. If a Transform exists and is not detected, Sherlock will not be able to provide component results. When running the **Detect Transform** feature, Sherlock uses multiple threads by default. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

If any of the file entries are blank, then Sherlock will assume that no analysis was performed for those Events.

When the **Save & Run** button is pressed, Sherlock will import both the model and the results to determine the reliability results.

Note:

When importing data, it is the user's responsibility to assign the proper results file to each Event. Sherlock has no other way of determining the association.

Note:

If lead results are not generated after importing results which contain leads, enable **Lead Modeling** on the **Generated Data Source** (p. 690) form and perform the imported analysis again. This applies to both Harmonic and Random Vibration results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.3. Harmonic Vibration Analysis Results

The Harmonic Vibration module generate the following types of results:

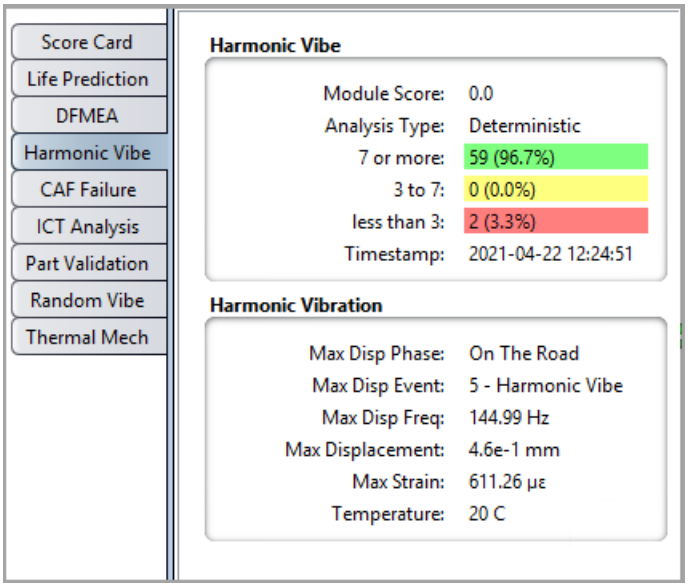
- Overall summary of the scores assigned to each part analyzed
- Maximum displacement and strain values across all parts, leads, and Vibration Events analyzed
- Table of maximum displacement and strain values for each part and lead analyzed
- Life prediction curve for the circuit card
- Graphical layers showing color-coded part scores (top and bottom sides)
- Graphical layers showing maximum displacement and strain during Vibration Events

We'll now take a look at each of these types of results.

9.4.3.1. Summary Panel
9.4.3.2. Harmonic Vibe Tabular Results
9.4.3.3. Harmonic Vibe Life Prediction Curve
9.4.3.4. Harmonic Vibe Graphical Results Layers
9.4.3.5. Harmonic Vibe 3D Model / Results Viewing
9.4.3.6. Log Files

9.4.3.1. Summary Panel

The **Summary Panel** shows the overall distribution of scores assigned to each part analyzed, along with the overall score assigned to the circuit card itself.

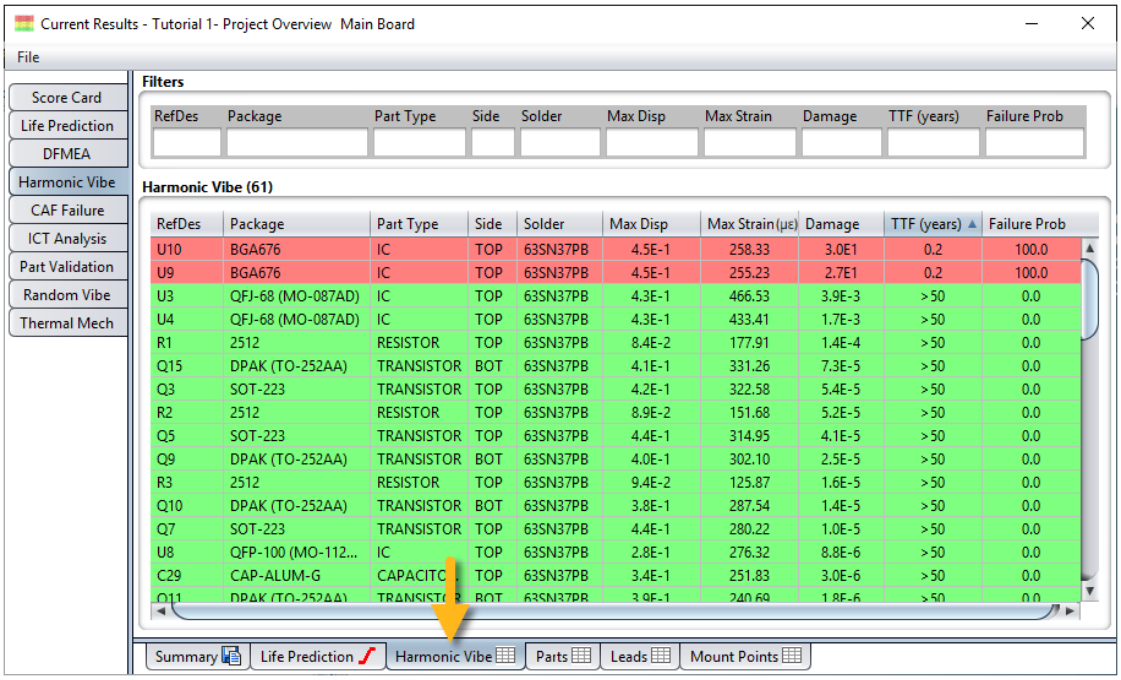


The **Harmonic Vibration** panel shows the name of the Phase and Harmonic Vibe Event that caused the maximum displacement of the circuit card, along with the maximum displacement and strain values. The **Max Disp Freq** value indicates the frequency at which the maximum results occurred during the harmonic sweep.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.3.2. Harmonic Vibe Tabular Results

Select the **Harmonic Vibe** sub-tab at the bottom of the results panel to view the detailed results for each part analyzed. The color-coded tabular results provide a convenient way to focus on the specific parts that may fail prematurely because of Vibration Events. When lead modeling is enabled, select the **Leads** sub-tab to view detailed results for each lead analyzed. When lead modeling is enabled, the strain, damage, time to failure, and score in the **Harmonic Vibe** will be for the lead on that component with the largest strain value.



As with other color-coded displays in Sherlock, RED means that the predicted score is less than 3 (less than a 3X safety factor), YELLOW means the score is between 3 and 7, and GREEN means that the score is 7 or more, indicating there is a very small probability that the part will fail.

Note:

In **Score Settings** (select from the main menu **Settings > Score**) you can change the score-color mapping to suit your specific needs.

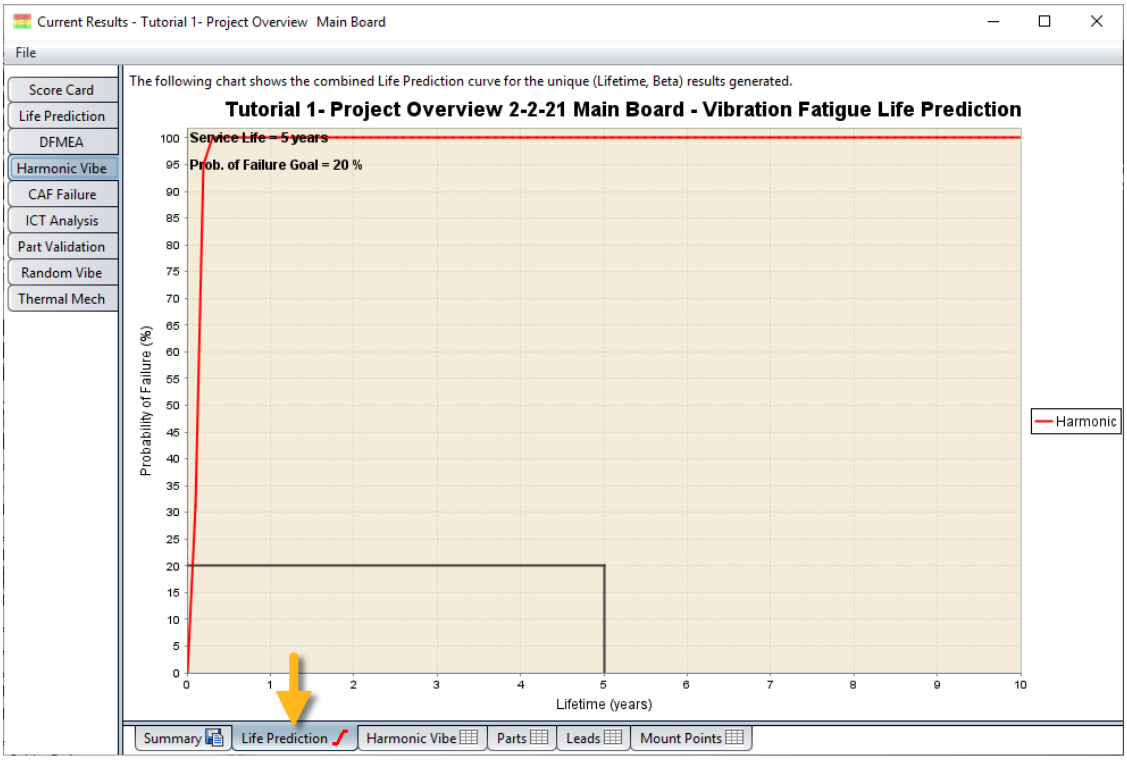
You can double-click any row in the table to view all the properties associated with a selected part. When viewing current results, the properties will be from the current **Parts List**. When viewing archived results, the properties will be from a snapshot of the **Parts List** taken at the time the archive was made.

You can export one or more rows from the results table by right-clicking a row and selecting the **Export Table** option from the context menu. See Reporting and Data Export (p. 799) for more details. You may also view the life prediction chart for any selected part or lead by selecting the **View Life Prediction** option from the context menu on the table.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.3.3. Harmonic Vibe Life Prediction Curve

Vibration analysis modules generate a **Deterministic** result. As such, a **Life Prediction Curve** can be generated for the circuit card itself, based on the **Life Prediction** curves for each of the parts analyzed, as shown here:

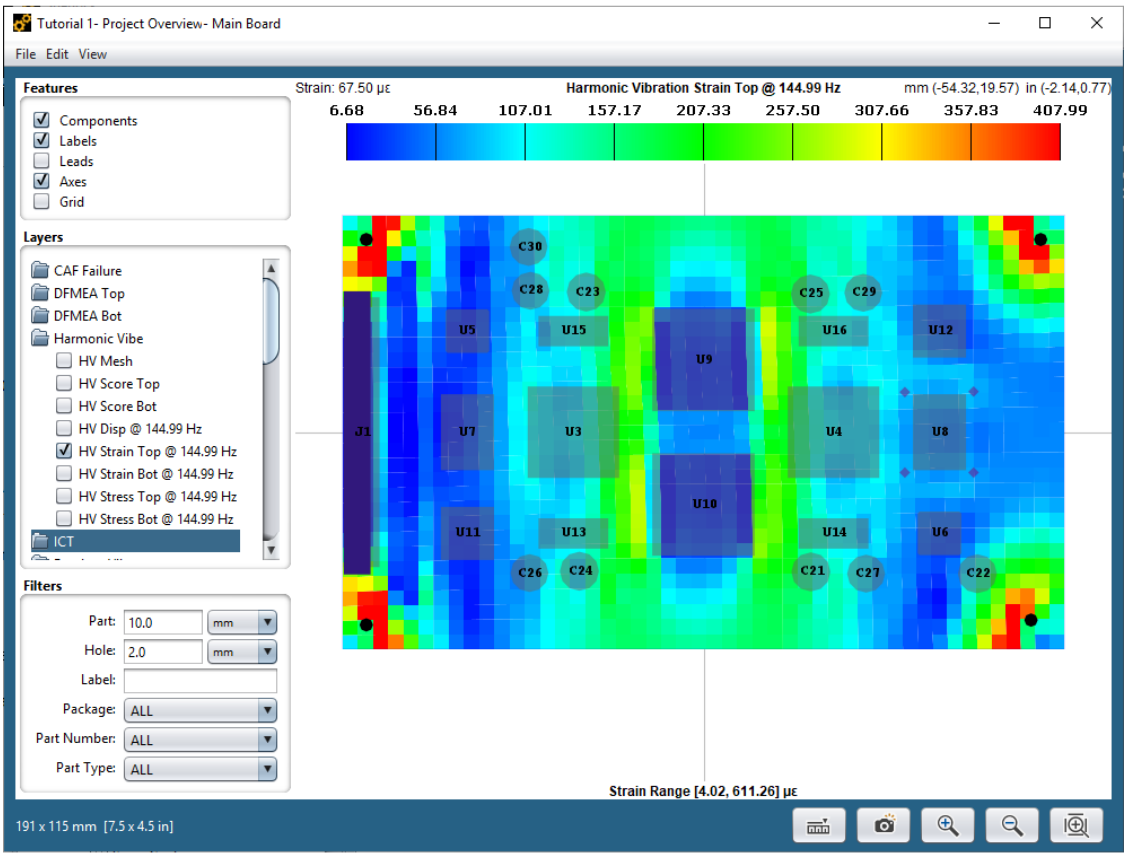


The **Life Prediction** chart shows all the critical information about circuit card reliability over a long period of time. The rectangular region in the lower left-hand corner of the chart graphically shows the desired **Reliability Goals** for this project, bounded on top by the desired **Probability of Failure** and bounded on the right by the desired **Service Life**. If the curve stays below the top line up to the **Service Life**, then the circuit card is predicted to meet its goals. In this case, the circuit card fails to meet its reliability goals when subjected to the Vibration Events defined in the **Life Cycle**.

9.4.3.4. Harmonic Vibe Graphical Results Layers

The Harmonic Vibe module generates graphical result layers showing the displacement ,stress, and strain values experienced during the worst Vibration Event analyzed. For example, the following image shows the surface Equivalent Strain on the top surface of the board during a harmonic vibration analysis:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The color legend indicates the range of displacement values being displayed, from 6.68 με to 2.9e-1 με.

You can customize the color range using the **Settings > FEA Settings** dialog. See the FEA Overview (p. 443) for more details.

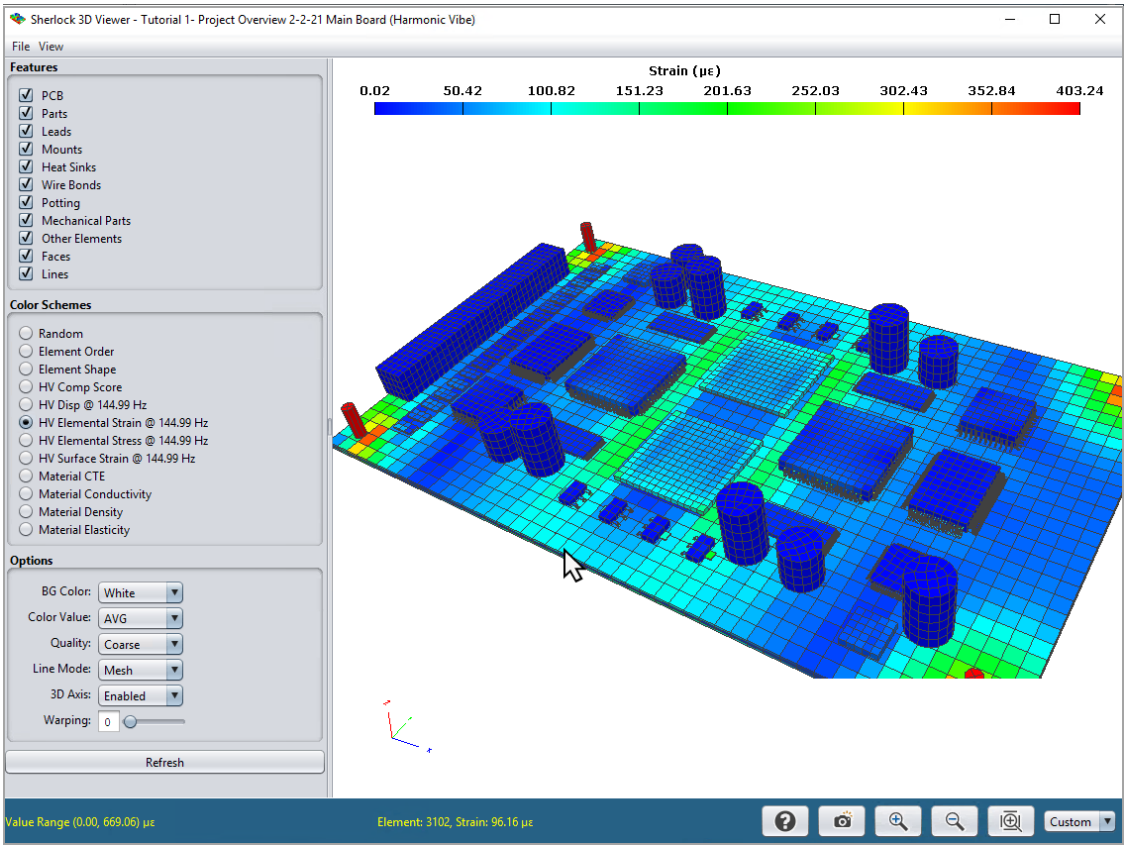
9.4.3.5. Harmonic Vibe 3D Model / Results Viewing

You can view the 3D FEA model and/or results by right-clicking the **Harmonic Vibe** node in the Project Tree and selecting either the **View 3D Model** or **View 3D Results** option from the context menu. In both cases, the **Sherlock 3D Viewer** will be launched allowing you to interactively view the 3D data. See Viewing and Managing Analysis Results (p. 785) for more details.

Note:

In the **3D Viewer**, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at 20°C only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.4.3.6. Log Files

If the **Show FEA Logs** option is enabled in the **Settings > FEA Settings** dialog, then a **Log Panel** will be displayed along with the other analysis results containing the log generated by the FEA engine during the analysis process. Advanced users can review such log data to determine if everything was processed as expected and/or to trouble-shoot FEA processing issues.

9.5. FEA- Random Vibration

Solder joints provide electrical, thermal, and mechanical connections between electronic components and a printed board. When the printed board is subjected to vibration, it will experience global and local changes to the board shape and curvature. The degree of bending will be different for specific components and the area of the printed board to which they are attached. This behavior will introduce strain into the second-level solder joint. With repeated exposure, damage will accumulate, leading to crack propagation and eventual failure of the solder joint. Vibration-induced solder joint fatigue is influenced by the type of vibration, the shape of the vibration spectrum, the size and shape of the printed board, printed board in-plane material properties, support conditions, component design, component material properties, location of the component, solder joint geometry, and solder joint material. Ansys Sherlock calculates time to failure using a modified Steinberg that takes board-level strain into account.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This chapter covers Random Vibration analysis. Harmonic Vibration Analysis is treated in a separate chapter: See Harmonic Vibration Analysis (p. 688).

Tip:

Sherlock offers another option for running a Random Vibe analysis: It's a fast, non-FE analysis which generates predictions based on strain maps you provide—from an image file or a CSV file. See Strain Mapping in Sherlock (p. 769).

In this section, the following topics are covered:

9.5.1. Random Vibe Input Data

9.5.2. Random Vibration Analysis Properties

9.5.3. Random Vibration Analysis Results

9.5.1. Random Vibe Input Data

The Random Vibration analysis module makes use of the following input data for the analysis calculations:

- Life Cycle Reliability Goals and Random Vibration Events
- Parts List

Tip:

The following section explains which part properties Sherlock uses in the Random Vibration analysis: Tables: Required Part Properties per Analysis Type (p. 326).

- Size and location of all parts, plated through-holes, mount points and cutouts
- Circuit card mechanical properties (stackup data)
- Circuit card outline
- Mesh properties
- Solder properties

If any of the input data listed above is changed, Sherlock will automatically clear the analysis results for the analysis module.

When you select Ansys or Abaqus as the solver, Sherlock will perform a random vibration analysis. (In previous versions of Sherlock, Abaqus would perform a 1 G harmonic sweep.) When NX Nastran is chosen (Windows version only), Sherlock performs a 1 G harmonic sweep and post processes those

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

results to provide life predictions. (Abaqus and NX Nastran solvers are available only in the Windows version of Sherlock.)

> **Warning:**
>
> Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate results, see the section Compatibility with Earlier Versions of Sherlock (p. 5).

9.5.1.1. Random Events

9.5.1.2. Single Modal Analysis Setting

9.5.1.3. Mount Points

9.5.1.1. Random Events

The external loads used during Vibration analysis are based on one or more Random Events (p. 350) defined in the project **Life Cycle**. See the Life Cycle Management (p. 343) section for details.

9.5.1.2. Single Modal Analysis Setting

The **Single Modal Analysis** feature, when enabled, significantly reduces solve time when running a Harmonic or Random Vibration analysis that has multiple Events. By default, Sherlock runs a modal (Natural Frequency) analysis prior to each Vibration Event. But when **Single Modal Analysis** is activated, Sherlock runs only one modal analysis and reuses the results for all Vibration Events.

The **Single Modal Analysis** option is located in the **FEA Settings/Engine Properties** panel (p. 446) of the **Sherlock Settings** window. To use this feature, the **FEA Engine** must be set to **Ansys**, also found in the **Engine Properties** panel.

If you intend to import results with the **Single Modal Analysis** setting enabled, note the following:

- If **Save Modal Results** is not enabled in Ansys Mechanical, then the `.mode` file must be in the same import directory.
- **Single Modal Analysis** *cannot* be used with the **Use Lead Nodal Forces** option in Random Vibration when the **Data Source** is set to **Imported**.

9.5.1.3. Mount Points

Random Vibration analysis depends on the current placement and type of mount points associated with the circuit card being analyzed. See the section on Mount Points and Fixtures (p. 479) for details concerning the creation and management of mount points.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

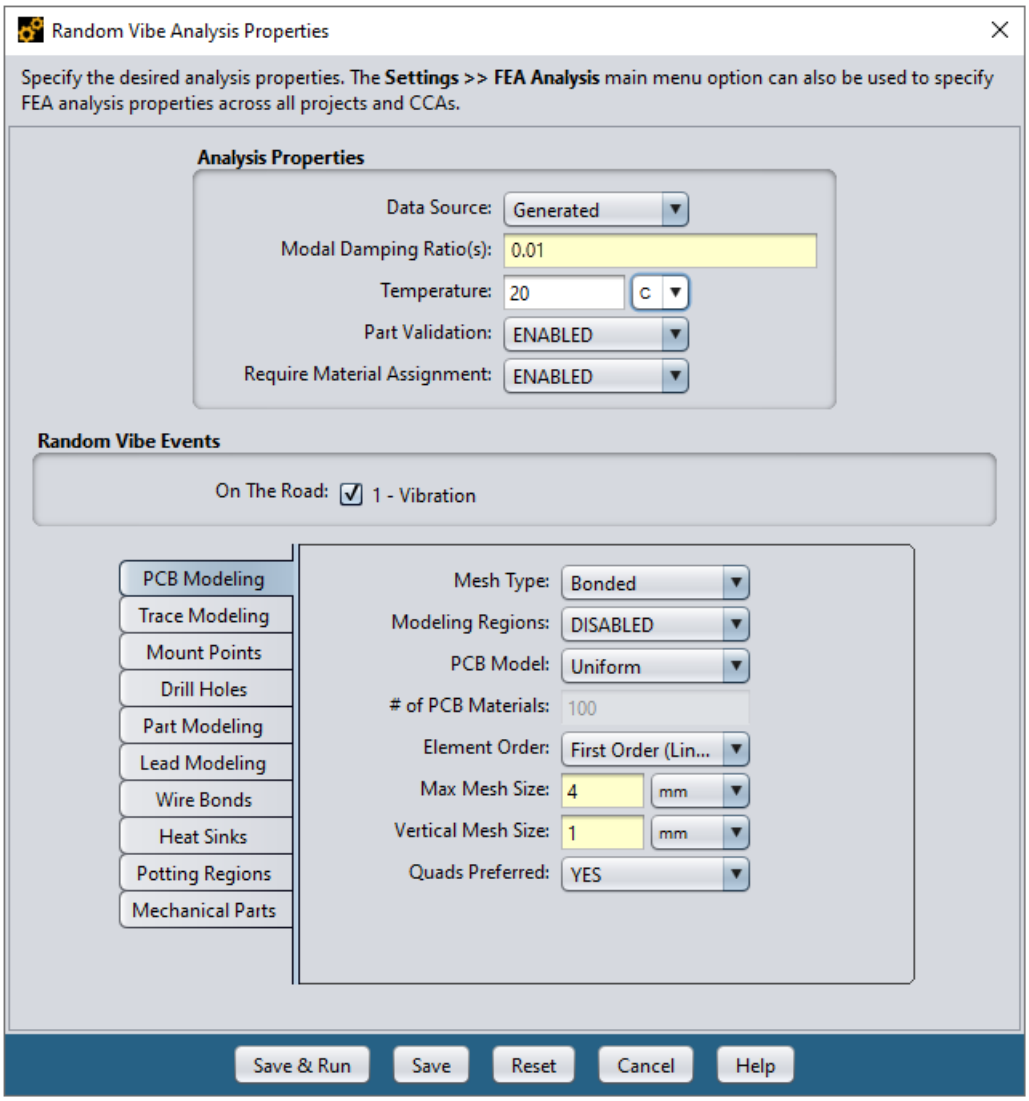
9.5.2. Random Vibration Analysis Properties

Vibration analysis properties can be specified by right-clicking the **Random Vibe** node in the **Project Tree** and selecting the **Edit Properties** option from the context menu. You may also run the analysis immediately, using the last properties entered, by selecting the **Run Analysis Task** option from the context menu.

9.5.2.1. Generated Data Source

When using the **Generated** data source, users may specify mesh properties using the mesh properties forms. Each form is selectable by the indicated tabs. The **PCB Modeling** mesh properties can be seen in the image below. See the FEA Overview (p. 443) for more details.

The **Analysis Properties** form allows users to specify the range of frequencies to be scanned to determine the natural frequency of the circuit card. A natural frequency scan is required for both Harmonic and Random vibration analysis and is automatically performed by Sherlock. The range defined must be larger than the range for any Harmonic Events being used for the analysis.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Modal Damping Ratio(s)** specifies the damping of the circuit card assembly. A default of 0.01 is used.

The **Random Vibe Events** form provides a list of all Random Vibration Events (p. 350) currently defined in the project **Life Cycle**. Sherlock will automatically perform a separate vibration analysis for each Vibration Event selected in the list if the option **Single Modal Analysis** is set to **DISABLED** in FEA Settings (p. 446). If the option is **ENABLED**, Sherlock will run a single modal analysis and reuse the results for multiple Vibration Events. Reliability results will then be based on those results. This allows users to ignore specific Events during an analysis run or determine the effects of just one or two Events on the reliability results.

Note:

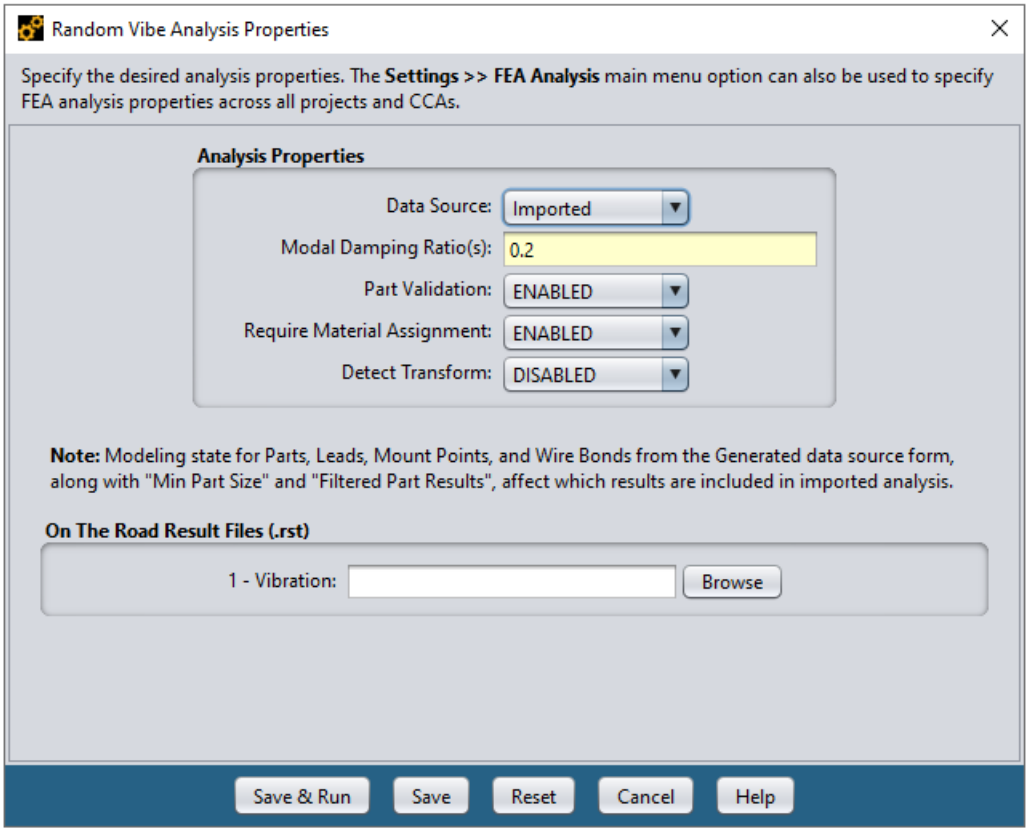
Accelerometer results are currently not supported by Random Vibration Analysis.

9.5.2.2. Imported Data Source

Vibration analysis results (`.rst`) can be imported into Sherlock by selecting the **Imported** data source. When selected, as shown below, enter the results files used for both the Natural Frequency analysis and all the Vibration Event analysis runs.

Tip:

For detailed, step-by-step instructions for importing results from Ansys Workbench, see the following section: FEA- Ansys Workbench™ Integration (p. 747).



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **Detect Transform** is ENABLED, Sherlock will attempt to detect any Transforms applied to the imported model and make adjustments if required. This ensures the imported model matches Sherlock's original model. Select MANUAL to enter Transform properties manually. If a Transform exists and is not detected, Sherlock will not be able to provide component results. When running the **Detect Transform** feature, Sherlock uses multiple threads by default. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

If any of the file entries are blank, then Sherlock will assume that no analysis was performed for those Events.

When the **Save & Run** button is pressed, Sherlock will import both the model and the results to determine the reliability results.

Note:

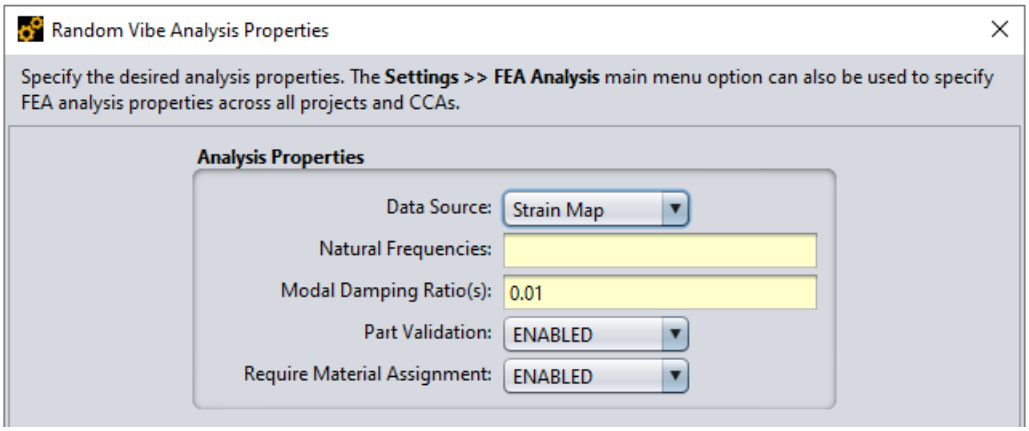
When importing data, it is the user's responsibility to assign the proper results file to each Event. Sherlock has no other way of determining the association.

Note:

If lead results are not generated after importing results which contain leads, enable **Lead Modeling** on the **Generated Data Source** (p. 690) form and perform the imported analysis again. This applies to both Harmonic and Random Vibration results.

9.5.2.3. Strain Map Data Source

This option is explained in Strain Mapping in Sherlock (p. 769).



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.5.3. Random Vibration Analysis Results

The Random Vibration analysis module generates the following results:

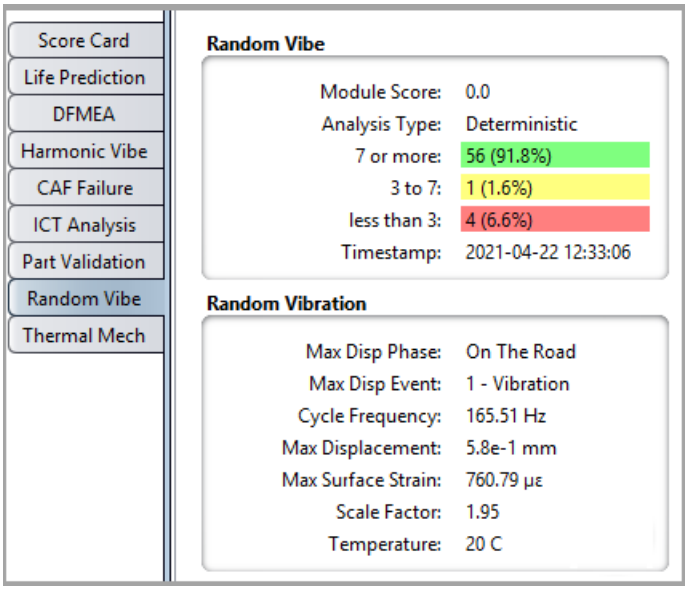
- Overall summary of the scores assigned to each part analyzed
- Maximum displacement and strain values across all parts, leads, and Vibration Events analyzed
- Table of maximum displacement and strain values for each part and lead analyzed
- Life prediction curve for the circuit card
- Graphical layers showing color-coded part scores (top and bottom sides)
- Graphical layers showing maximum displacement and strain during Vibration Events

We'll now take a look at each of these types of results.

9.5.3.1. Summary Panel
9.5.3.2. Random Vibration Tabular Results
9.5.3.3. Random Vibe Life Prediction Curve
9.5.3.4. Random Vibe Graphical Results Layers
9.5.3.5. Random Vibe 3D Model / Results Viewing
9.5.3.6. Log Files

9.5.3.1. Summary Panel

The **Summary Panel** shows the overall distribution of scores assigned to each part analyzed, along with the overall score assigned to the circuit card itself.

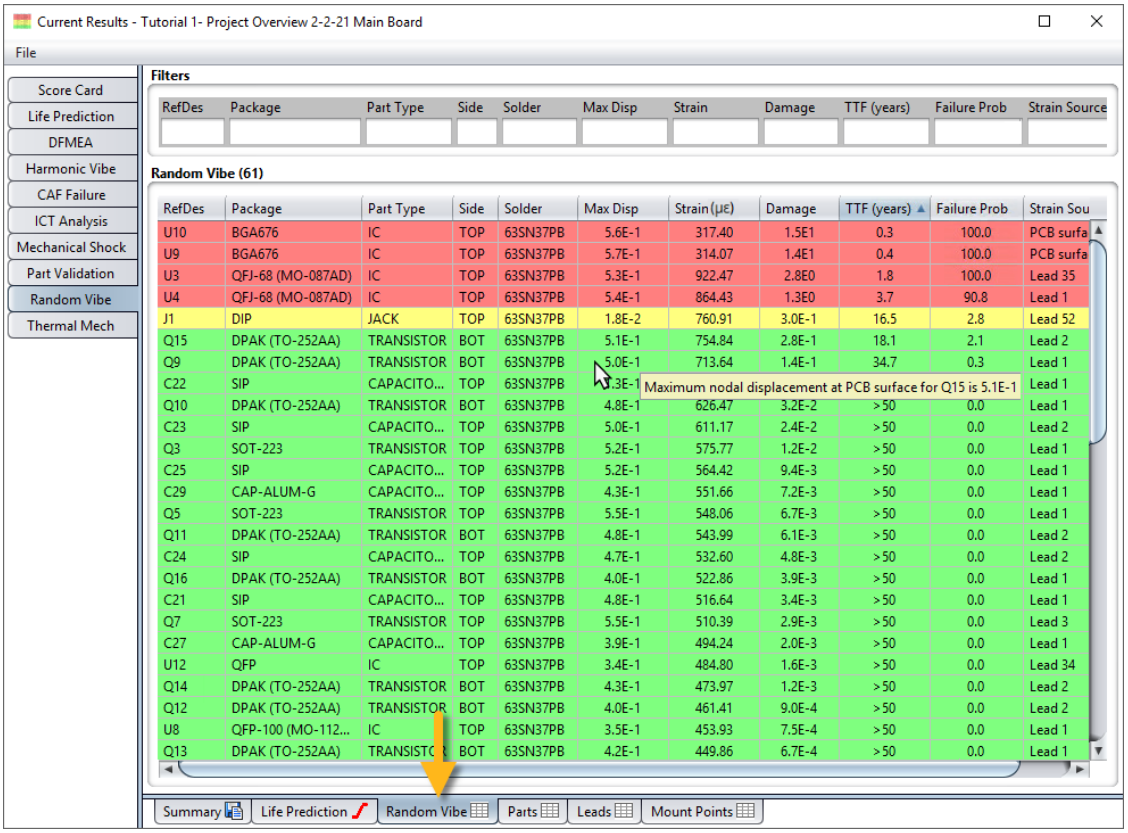


The **Random Vibration** panel shows the name of the Phase and Random Vibe Event that caused the maximum displacement of the circuit card, along with the maximum displacement and strain values. The **Cycle Frequency** value represents the number of positive zero-crossings per second.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.5.3.2. Random Vibration Tabular Results

Select the **Random Vibe** sub-tab at the bottom of the results panel to view the detailed results for each part analyzed. The color-coded tabular results provide a convenient way to focus on the specific parts that may fail prematurely because of Vibration Events. When lead modeling is enabled, select the **Leads** sub-tab to view detailed results for each lead analyzed. When lead modeling is enabled, the strain, damage, time to failure, and score in the **Random Vibe Results Table** will be for the lead on that component with the largest strain value.



As with other color-coded displays in Sherlock, RED means that the predicted score is less than 3 (less than a 3X safety factor), YELLOW means the score is between 3 and 7, and GREEN means that the score is 7 or more, indicating there is a very small probability that the part will fail.

Note:

The **Settings > Score Settings** editor can be used to change the score-color mapping to suit your specific needs.

You can double-click any row in the table to view all the properties associated with a selected part. When viewing current results, the properties will be from the current **Parts List**. When viewing archived results, the properties will be from a snapshot of the **Parts List** taken at the time the archive was made.

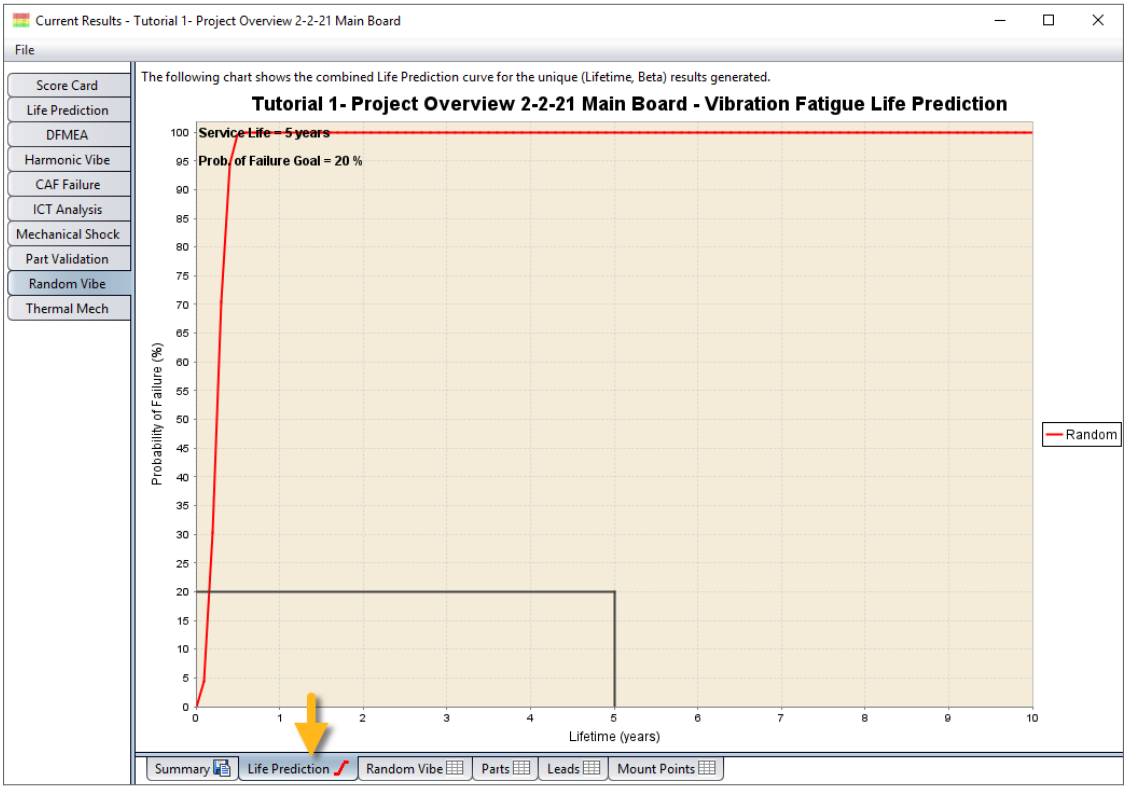
You can export one or more rows from the results table by right-clicking a row and selecting the **Export Table** option from the context menu. See Reporting and Data Export (p. 799) for more details.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may also view the life prediction chart for any selected part or lead by selecting the **View Life Prediction** option from the context menu on the table.

9.5.3.3. Random Vibe Life Prediction Curve

Vibration analysis modules generate a **Deterministic** result. As such, a **Life Prediction Curve** can be generated for the circuit card itself, based on the **Life Prediction** curves for each of the parts analyzed, as shown here:

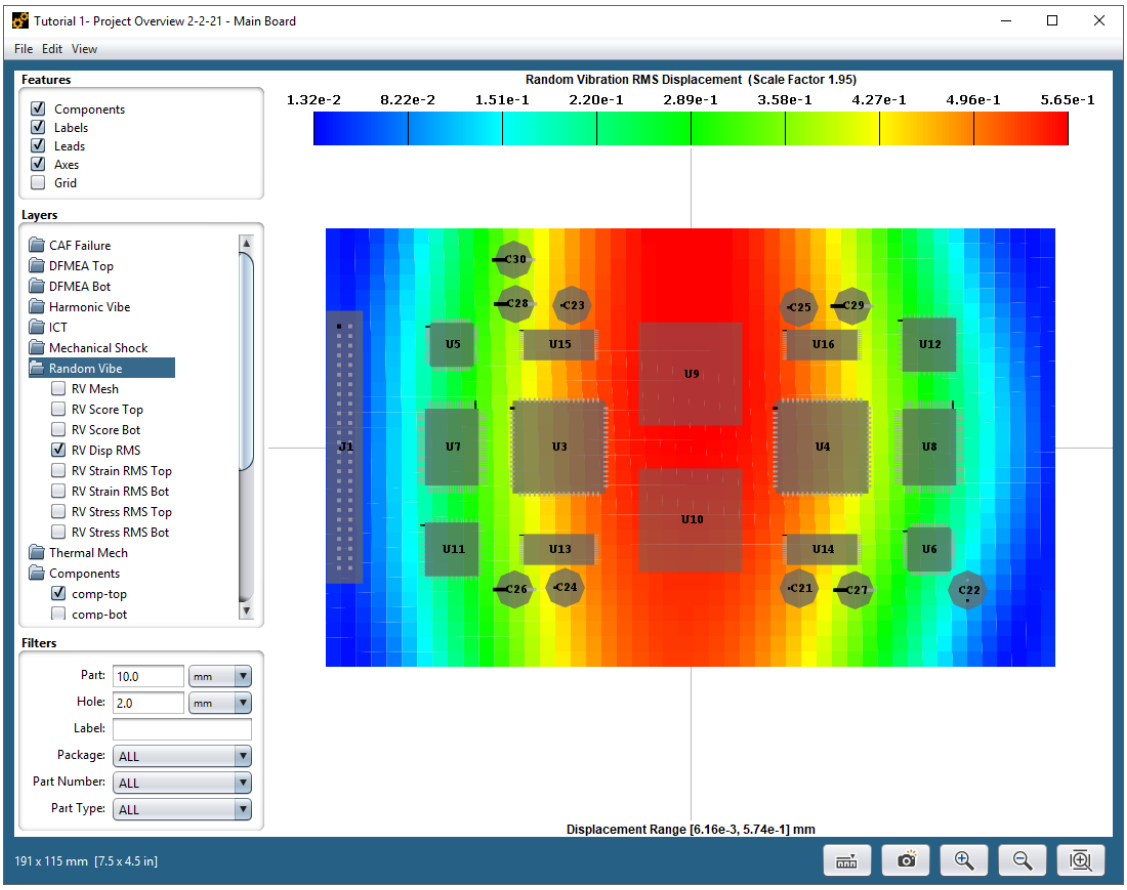


The **Life Prediction** chart shows all the critical information about circuit card reliability over a long period of time. The rectangular region in the lower left-hand corner of the chart graphically shows the desired **Reliability Goals** for this project, bounded on top by the desired **Probability of Failure** and bounded on the right by the desired **Service Life**. If the curve stays below the top line up to the **Service Life**, then the circuit card is predicted to meet its goals. In this case, the circuit card fails to meet its reliability goals when subjected to the Vibration Events defined in the **Life Cycle**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.5.3.4. Random Vibe Graphical Results Layers

Graphical and Tabular results for Random Vibe are displayed as RMS values due to the nature of the analysis. By default, the Displacement, Strain, and Stress values plotted under Random Vibration analysis are for 1.95 sigma. The sigma value can be adjusted by changing the **Random Vibe Scale Factor** (see **Random Vibe Scale Factor** under Analysis Properties (p. 448)). The Scale Factor must be set before running the analysis in Sherlock or importing the `.rst` file into Sherlock. The following image shows the 1.95 sigma RMS displacements experienced by the PCB during a Random Vibration simulation.



The color legend indicates the range of displacement values being displayed, from 1.32e-2 mm to 5.65e-1 mm.

You can customize the color range using the **Settings > FEA Settings** dialog. See the FEA Overview (p. 443) for more details.

9.5.3.5. Random Vibe 3D Model / Results Viewing

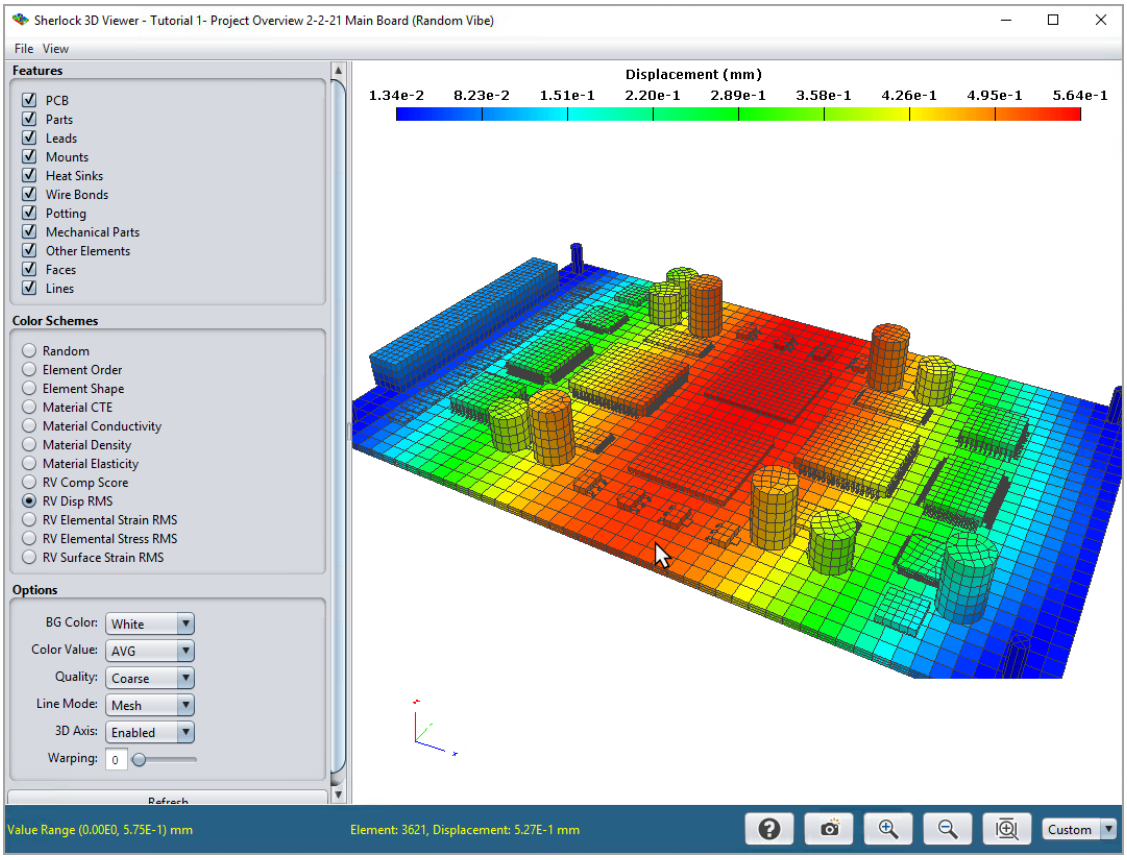
You can view the 3D FEA model and/or results by right-clicking the **Random Vibe** node in the Project Tree and selecting either the **View 3D Model** or **View 3D Results** option from the context

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu. In both cases, the **Sherlock 3D Viewer** will be launched allowing you to interactively view the 3D data. See Viewing and Managing Analysis Results (p. 785) for more details.

Note:

In the **3D Viewer**, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at 20°C only.



9.5.3.6. Log Files

If the **Show FEA Logs** option is enabled in the **Settings > FEA Settings** dialog, then a **Log Panel** will be displayed along with the other analysis results containing the log generated by the FEA engine during the analysis process. Advanced users can review such log data to determine if everything was processed as expected and/or to trouble-shoot FEA processing issues.

9.6. FEA- Thermal Mech (Thermo-Mechanical)

Sherlock's Thermal Mech feature analyzes Circuit Card Assemblies (CCA's) to assess potential solder failures caused by thermal cycling. These failures are driven by differences in the coefficients of thermal expansion (CTE) among the various materials which comprise the assembly. The different expansion rates of the parts and ensuing deformations account for stress in the assembly. The different expansion rates and the ensuing deformations during thermal cycling causes stress in the solder joints. Repeated exposure to such conditions leads to fatigue, eventually resulting in joint failure.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you run a Thermal Mech Analysis, Sherlock captures the strain that drives solder fatigue failure. When calculating the strain range (Δstrain) for ball grid arrays (BGA's), Sherlock uses elemental equivalent strain from the minimum and maximum temperature contained in the Thermal Profile. For other leaded devices, Sherlock also provides the strain range for the leads. For components that are not leaded (or when you choose not to include leads in the analysis), Sherlock provides board strain ranges. Sherlock does not make any predictions with the strain range as of Sherlock 2023 R1. However, the strain range could be easily exported to make failure predictions outside Sherlock.

Running a Thermal Mech Analysis involves setting up a Finite Element (FE) model of the board and analyzing the model using the Ansys Mechanical solver. The Sherlock interface provides a streamlined method for setting up the FE model with the appropriate boundary conditions to simulate various thermal loading scenarios. You define thermal load(s) using Sherlock's **Thermal Event Editor** (p. 343) under **Life Cycle** in the **Project Tree**.

Once the model is solved, Sherlock determines the strain range (Δstrain) for each part and tabulates the results. Sherlock also displays results graphically: Board displacement, stress, and strain are displayed in the **Layer Viewer** (2D) and the **3D Results Viewer**. More on the results are provided in the section below Thermal Mech Analysis Results (p. 730).

> **Warning:**
>
> Exercise caution when using multiple versions of Sherlock on the same project. To avoid inaccurate results, see the section Compatibility with Earlier Versions of Sherlock (p. 5).

In this section, the following topics are covered:

9.6.1. Thermal Mech and Material Properties
9.6.2. Thermal Mech and Input Data
9.6.3. Overview of Thermal Mech Analyses
9.6.4. Thermal Mech Properties Dialog
9.6.5. Types of Thermal Mech Analyses
9.6.6. Summary of Analysis Combinations
9.6.7. Thermal Mech and Mount Points
9.6.8. Part Temp Rise and Thermal Map FAQ
9.6.9. Imported Data Source
9.6.10. Thermal Mech Analysis Results

9.6.1. Thermal Mech and Material Properties

The **Thermal Mech Analysis Module** makes use of the following material properties when building FEA models:

- Density
- Elastic Modulus
- Poisson Ratio
- Coefficient of Thermal Expansion

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Thermal Conductivity
- Secant Modulus (BGA's only)

In Sherlock, you can define part properties in the **Material Manager** (p. 153). To define solder and laminate material properties, use the **Solder Manager** (p. 145) and **Laminate Manager** (p. 133) respectively.

Sherlock runs a linear elastic, static structural analysis or a steady state thermal analysis followed by a static structural analysis, depending on the thermal boundary conditions. Creep properties for the solder are considered only for BGAs as a temperature dependent secant modulus. In general, thermo-mechanical simulation which targets solder failure in electronics uses creep models such as Garofalo or Anand's plastic model. However, running a board level analysis with creep models involving thousands of solder geometries is computationally expensive. On the other hand, simplifying the analysis with a linear elastic solder model will under-predict damage.

Sherlock strikes a balance by using the secant modulus for solders. You can run a simple pull test in Ansys Mechanical with creep models and identify the secant modulus at a strain of your choice and then use it in Sherlock for the solder material. Sherlock does not require you to use its default material properties. For example, Sherlock provides a set of secant properties for SAC305 and SnPb at 0.2% strain from a tensile pull simulation carried out at strain rate of 1e-4 strain/second. If this value does not suit your purposes, you can override this property with a custom secant modulus.

Note:

Missing Property Values: In the unlikely event thermal conductivity values are missing for a material, Sherlock logs an analysis issue (and console message) such as the one shown here:

```
TM: No thermal conductivity defined for FERRITE, using 1.0e-
4 W/m-K
```

As the message indicates, Sherlock automatically uses a default thermal conductivity value of 1E-4 W/m-K. To eliminate such warnings, use the Material Manager (p. 153) to add the proper value.

9.6.2. Thermal Mech and Input Data

The **Thermal Mech Analysis Module** makes use of the following input data:

- Thermal Events (p. 344) and Custom Uniform Temperatures
- Part Temperature Rise, Thermal Map (CSV), Thermal Map (Image)
- Parts List (p. 32)

Tip:

The following section explains which part properties Sherlock uses in the Thermal Mech analysis: Tables: Required Part Properties per Analysis Type (p. 326).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Size and location of all parts (p. 30), plated through-holes, and cutouts (p. 489)
- Circuit card mechanical properties (stackup data (p. 473))
- Mesh Properties (p. 473)
- Mount Points (p. 479)

Note:

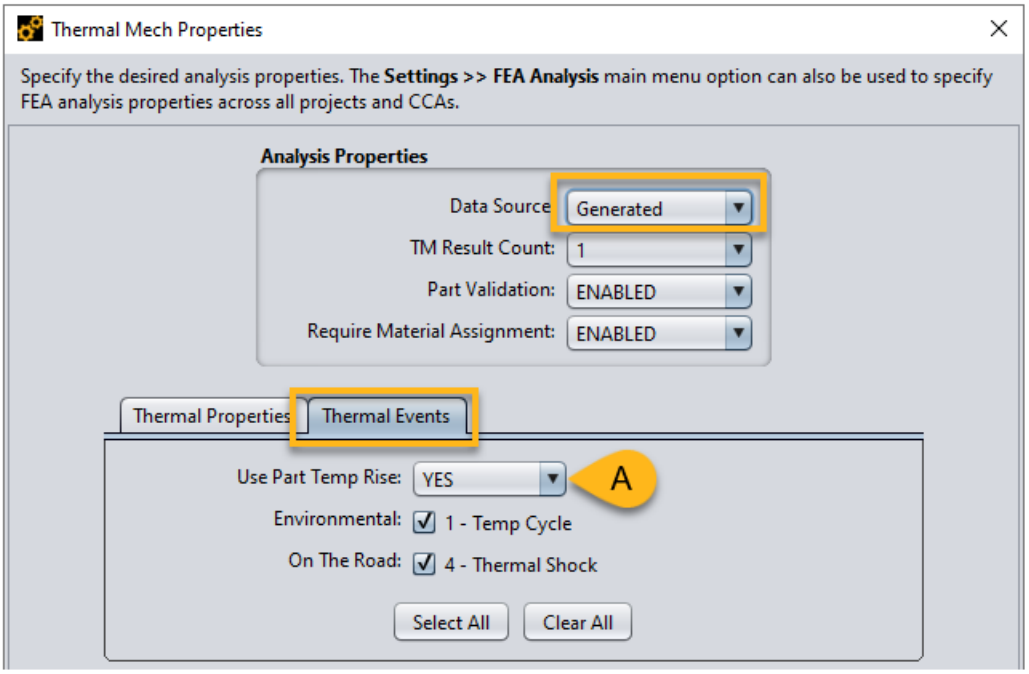
For BGAs, you may choose to import a Net list (p. 395). This allows you to change the location of the solder balls.

9.6.3. Overview of Thermal Mech Analyses

Sherlock allows you to perform several kinds of Thermal Mech Analyses. The following list is an overview and will be further explained in subsequent sections.

- **Using Thermal Events**: Thermal Events in Sherlock are temperature profiles you create which are applied to the electronic assembly during a Thermal Mechanical Analysis. When you create a Thermal Event, you define a temperature profile and specify the number of cycles it will be applied to the electronic assembly. (See Project Life Cycles (p. 343) for how to create Thermal Events.)

  For this kind of analysis, you select **Generated** as the data source in the **Thermal Mech Properties** window, as shown below. Use the **Thermal Events** tab to select which Thermal Events apply to the analysis.

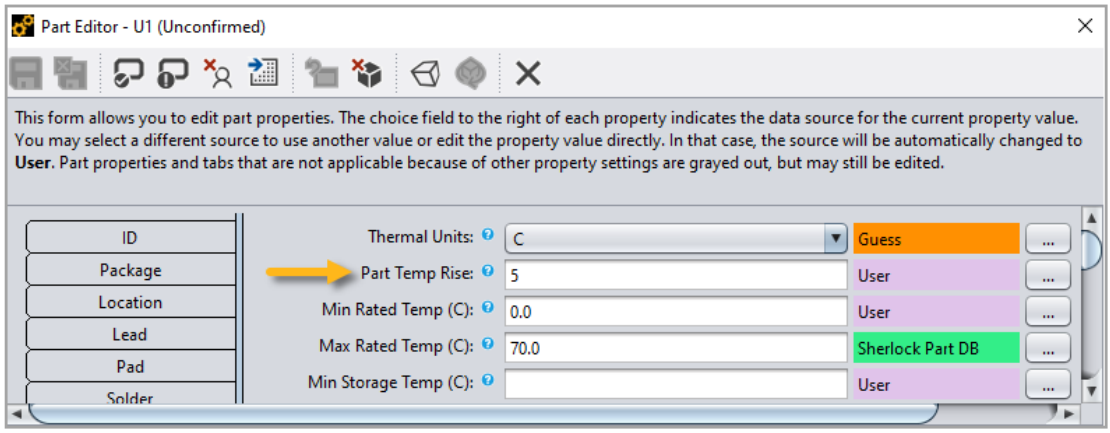


- **Thermal Events with Part Temperature Rise**: Select **YES** for **Part Temp Rise** (A, above) if you want Sherlock to incorporate the Part Temp Rise property into its analysis. **Part Temp Rise** is a part property that specifies a part's temperature rise that occurs during the device's operating conditions.

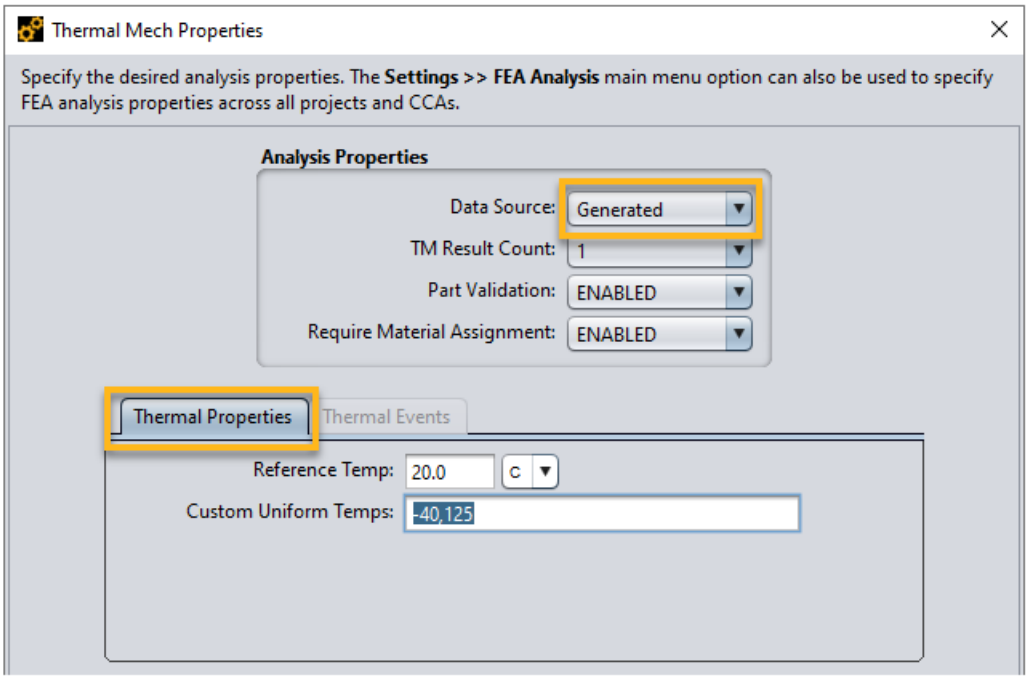
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a given Thermal Event, Sherlock will raise the temperature for those parts for which **Part Temp Rise** has been set and will solve for temperature distribution across the board for the new thermal load. The thermal distribution is then used in the structural analysis to solve for deformation and strain.

For this kind of analysis, you must first specify the **Part Temp Rise** property for the relevant parts in your Parts List, using the **Part Editor** (see below). As explained later, it is *not* necessary to set **Part Temp Rise** for *all* the parts in your Part List.



- **Isothermal Analysis:** In Sherlock, you can run an isothermal thermo-mechanical analysis using custom temperature points. This is an easy way to run a quick ramp-up or ramp-down analysis without defining Thermal Events. This feature is exclusively used for running isothermal analysis and does not support any type of heat transfer analysis. Sherlock uses the custom temperature points to apply the uniform temperature in Ansys Mechanical. To run an isothermal analysis, choose **Generated** as the **Data Source**, and enter the temperature information using the **Thermal Properties** tab as shown below.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Conduction Analysis**: Sherlock also supports thermal conduction analysis. If you know the temperature of the part at each step of the thermal profile or have the temperature distribution of the TOP and (or) BOTTOM face of the PCB, you can run thermal analysis using Part Temperature Rise and Thermal Mapping techniques in Sherlock.

  Part Temperature Rise allows you to define the temperature rise of components caused by heat generation during operating conditions. On the other hand, if you have information on the absolute temperature of the parts, you can use Thermal Map (CSV). More details on how to add Part Temperature Rise to the parts of a CCA, how to add thermal Image/CSV and, more importantly, how the different techniques are associated to the Thermal Events will be discussed later.

Note:

If you select Abaqus (available in the Windows version only) to run the Thermal Mech analysis (see Selecting the FEA Engine (p. 444)), the following features are not supported:

- Part Temperature Rise
- Thermal Map using a CSV
- Thermal Map using an Image

9.6.4. Thermal Mech Properties Dialog

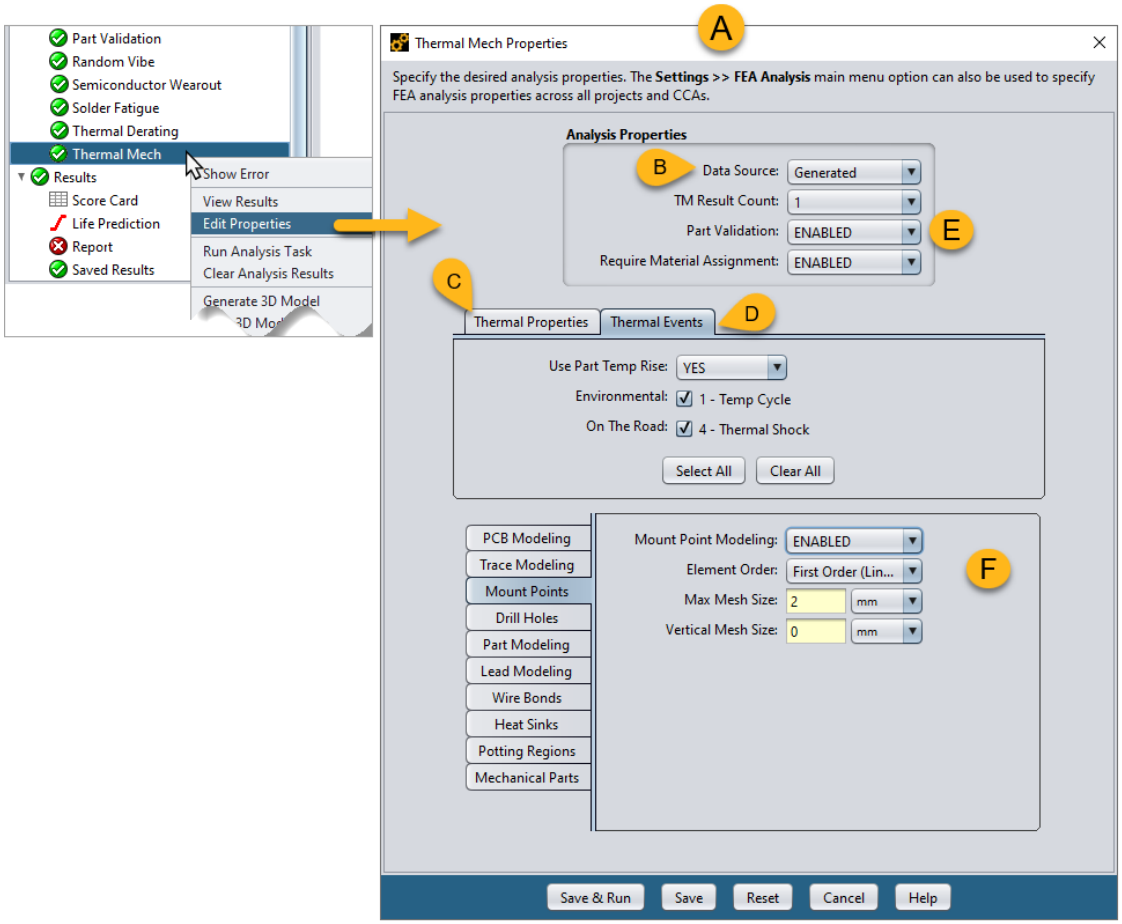
The **Thermal Mech Properties** dialog is the interface for setting up any Thermal Mech analysis. To open the dialog, right-click **Thermal Mech** in the **Project Tree** as shown below and select **Edit Properties** in the context menu.

The inputs required for the analysis (with **Data Source** (B) set to **Generated**) can be seen in the image below. They are categorized into: **Thermal Properties**(C), **Thermal Events** (D), **Analysis Properties** (E), and **Modeling Properties** (F).

Note:

If you change any of the input data, Sherlock will automatically clear previous Thermal Mech analysis results. If you want to keep your results, be sure to save them before proceeding.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Data Source (B): From this dropdown menu, specify how Sherlock will obtain the results used to calculate strain. Select one of two data sources: **Generated** or **Imported**.

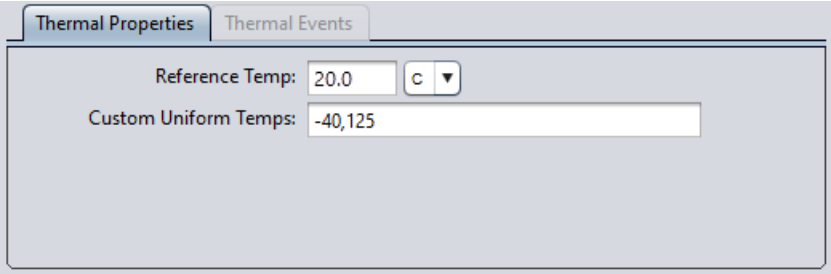
- **Generated**: When selected, you are tasking Sherlock to *generate* the results which are then used to calculate strain. Sherlock runs the FE model in batch mode based on the FE solver selected and post processes the results for strain.

- **Imported:** Select this option if you wish to import results files created by other applications such as Ansys Mechanical (.rst) or Abaqus (.odb). Sherlock performs the post-processing calculations and determines strain based on the result files provided.

Thermal Properties / **Thermal Events** tabs:

- **Thermal Properties** (C above): The **Thermal Properties** tab allows you to run a ramp-up or ramp-down analysis requiring minimal inputs. This feature is exclusively used for running isothermal analysis and does not support any type of heat transfer analysis. Sherlock uses the custom temperature points (**Custom Uniform Temps**) to apply the uniform temperature to the assembly.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The reference temperature for the ramp is set using the **Reference Temp** field. This defines the temperature at which the model exhibits no thermal expansion. Typically, room temperature (20°C) is chosen as shown in the example above. Use the dropdown menu to change the units of temperature if desired (C, F, or K).

Note:

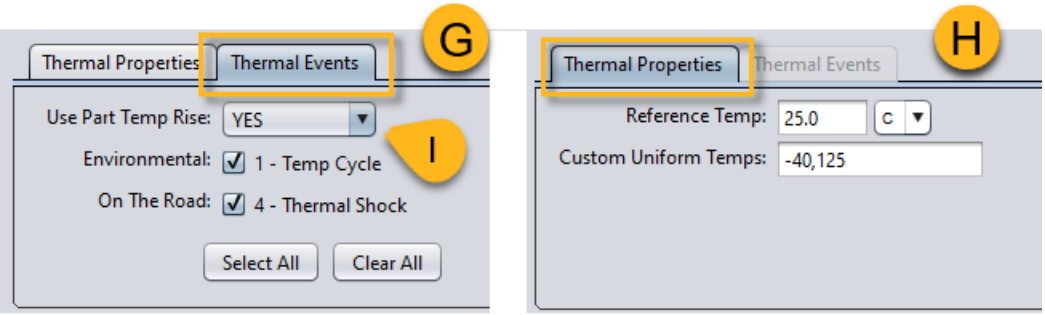
The temperature unit selected from the dropdown menu applies only to the **Reference Temperature**. Units for the **Custom Uniform Temps** field are handled differently as explained below.

As shown (H, below), the **Custom Uniform Temps** field can take multiple temperature values, separated by a comma. The default unit is Celsius. However, each temperature you enter can have its own unit. For example:0°C,32°F, 273.15 K all specify the same temperature using different units.

If a single temperature is provided, Sherlock will run a ramp-up (or ramp-down) analysis from the reference temperature. If you define multiple temperature data points, Sherlock will run a multi-step structural analysis with each step corresponding to one of the defined temperature points. For example, when Reference Temp is set to 25°C and Custom Uniform Temps of -40°C and 125°C are applied, Sherlock will run a two-step static structural analysis, where the first step is ramped from 25°C to -40°C and the second is ramped from 25°C to 125°C.

- **Thermal Events Tab** (G, below): Thermal Events in Sherlock are temperature profiles applied to the electronic assembly during a Thermal Mechanical analysis. When you create a Thermal Event, you define a temperature profile and specify the number of cycles it will be applied to the electronic assembly. The example shown below comes from the tutorial project: Two Thermal Events have been selected: **1 - Temp Cycle** and **2 - Thermal Shock**. How to create a Thermal Event is explained in the Life Cycle Management (p. 344) section.

  When running an analysis based on Thermal Events, you may select the option for Sherlock to **Use Part Temp Rise** (I, below). This is a part property which you can define using the **Part Editor**. These settings will be explained in more detail. See Thermal Events with Part Temperature Rise (p. 718) below.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When **Use Part Temp Rise** is set to NO, Sherlock will skip the thermal analysis (unless the Thermal Event is associated with Thermal Maps) and run an isothermal analysis by applying uniform temperature from the Thermal Profile throughout the circuit under static structural analysis.

It should be noted that **Part Temp Rise** value set under each part will be used only if **Use Part Temp Rise** is set to YES. In the **Part Editor**, you can set **Part Temp Rise** as a positive temperature value, or you can set it to DEFAULT. When set to DEFAULT, Sherlock does not apply any temperature boundary conditions to the part. Rather, Sherlock solves for its respective nodal temperature based on other boundary conditions.

Note:

If the **Thermal Events** tab is inactive, clear any data from the **Custom Uniform Temperature** field under the **Thermal Properties** tab. Any data in the field automatically deactivates the tab.

Tip:

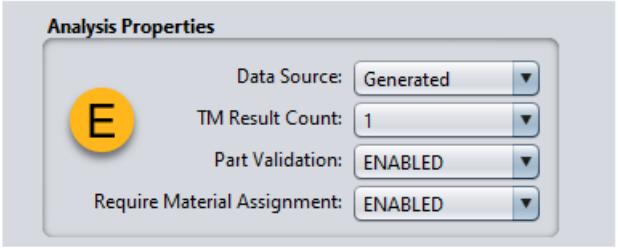
For a basic introduction on Project Life Cycles, refer to Project Life Cycle (p. 10) in the *Project Overview* tutorial.

Read Life Cycle Management (p. 343) for an overview of the following concepts: Life Cycle Management, Phases, and Events.

For instructions on creating Thermal Events and Thermal Profiles, read the section Thermal Event (p. 344).

Analysis Properties Panel (E, below)

- **TM Result Count**: This value specifies the number of results Sherlock will display in the **Layer Viewer** in the event of multiple Thermal Events. For example, if you set the **TM Result Count** to 2, if there are multiple Events, Sherlock will display results for the two Thermal Events that caused the highest strain values. Currently, Sherlock caps this value at 4. Setting it to 1 gives you the worst-case scenario.
- **Part Validation**: When this is ENABLED, Sherlock will run a parts property validation against the Part Library and/or the Approved Vendor List before building the thermal mech model. This prevents long-running analysis based on part properties that have not been validated.
- **Required Material Assignment**: If enabled, you must explicitly define material assignments. If set to NOT ENABLED- Sherlock uses default material properties.



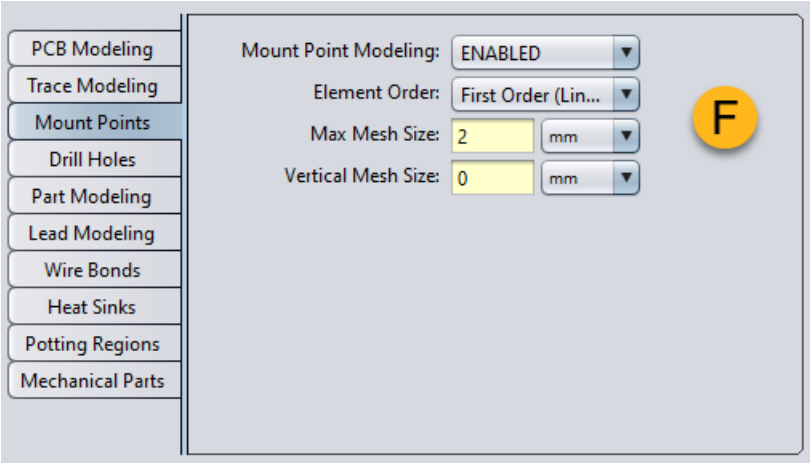
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeling Properties Panel:

The Modeling Properties panel in the **Thermal Mech Properties** dialog provides the necessary options to control mesh as well as the geometry of the FE model. The Finite Element Analysis (FEA) model created by Sherlock for Thermal Mech analysis is basically the same as those created by other analysis modules, such as Mechanical Shock, Harmonic Vibration and Random Vibration. The only difference is that temperature dependent material properties are included in the model for the solder geometry if defined. All other model properties, such as the inclusion of PCB traces, parts, leads, and heat sinks, for example, are used the same way as in the other analysis modules.

Tip:

FEA part modeling is discussed in detail in FEA- Part Modeling (p. 552).



Note:

If you wish to model component leads in your analysis, it is currently recommended that you export the project file to Ansys Workbench Products. Generate the FEA model, run the analysis in Workbench, and import the results back into Sherlock. See FEA-20 Ansys Workbench Integration for details.

For a full explanation of the Modeling Properties tabs and options, see FEA Modeling Properties (p. 552). Remember, you can apply specific FEA analysis options across all projects and CCAs by using the **FEA Analysis Settings** dialog found under Sherlock's main menu: **Settings > General Settings > FEA Analysis**. See FEA Analysis Settings (p. 444) for a guided explanation.

9.6.5. Types of Thermal Mech Analyses

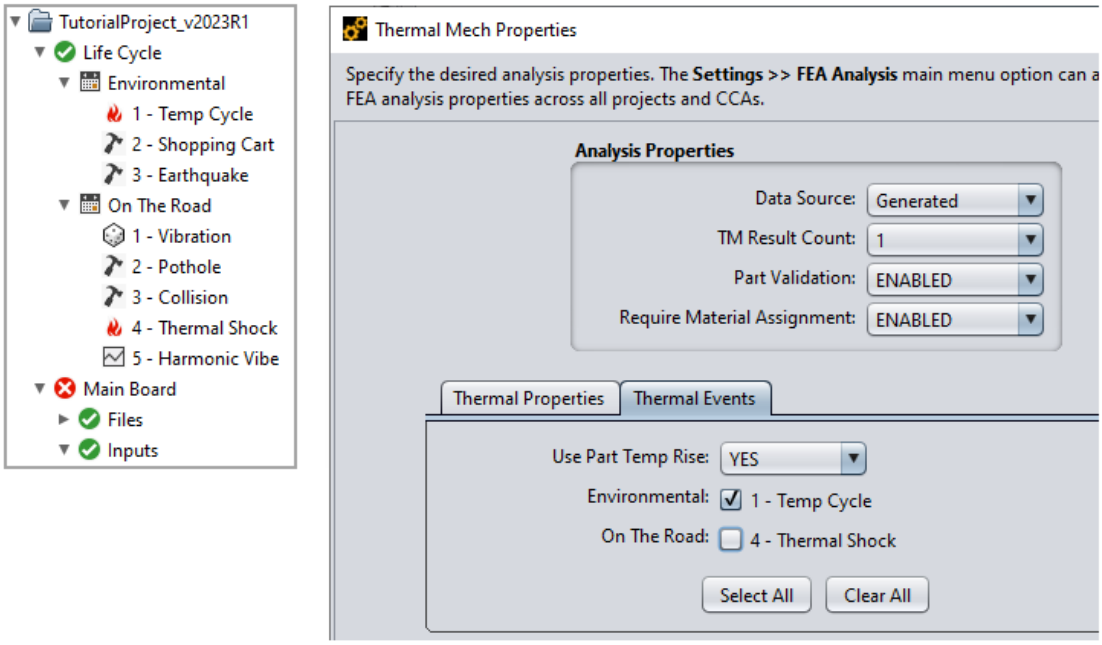
Sections Below:

9.6.5.1. Thermal Events with Part Temperature Rise
9.6.5.2. Thermal Events without Part Temperature Rise
9.6.5.3. Thermal Events with Thermal Map (CSV)
9.6.5.4. Thermal Events with Thermal Map (Image)

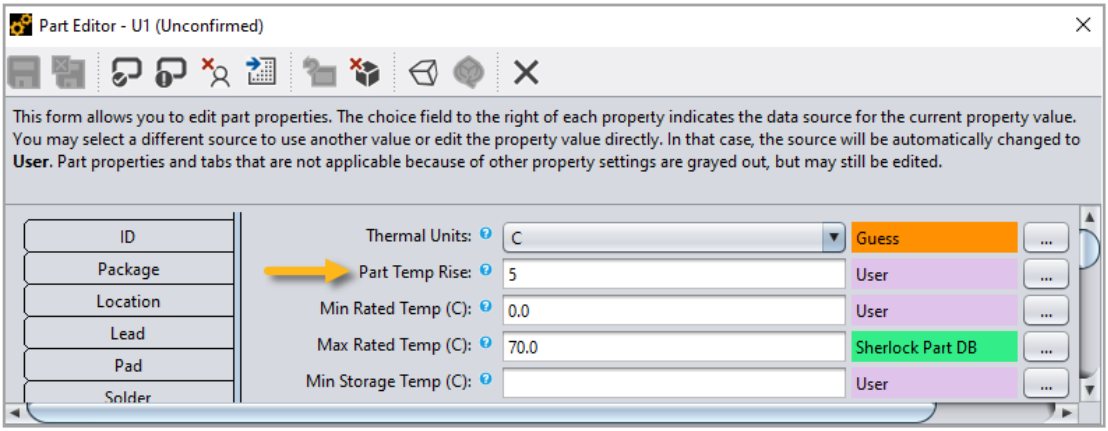
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.6.5.1. Thermal Events with Part Temperature Rise

Thermal Events (C) with Part Temperature Rise: Part Temperature Rise allows you to define the temperature rise of components due to heat generation during operating condition. You can run thermo-mechanical analysis with Part Temperature Rise by setting the **Use Part Temp Rise** to **YES**, as shown below. Additionally, you must specify the temperature rise for the parts from Parts List.

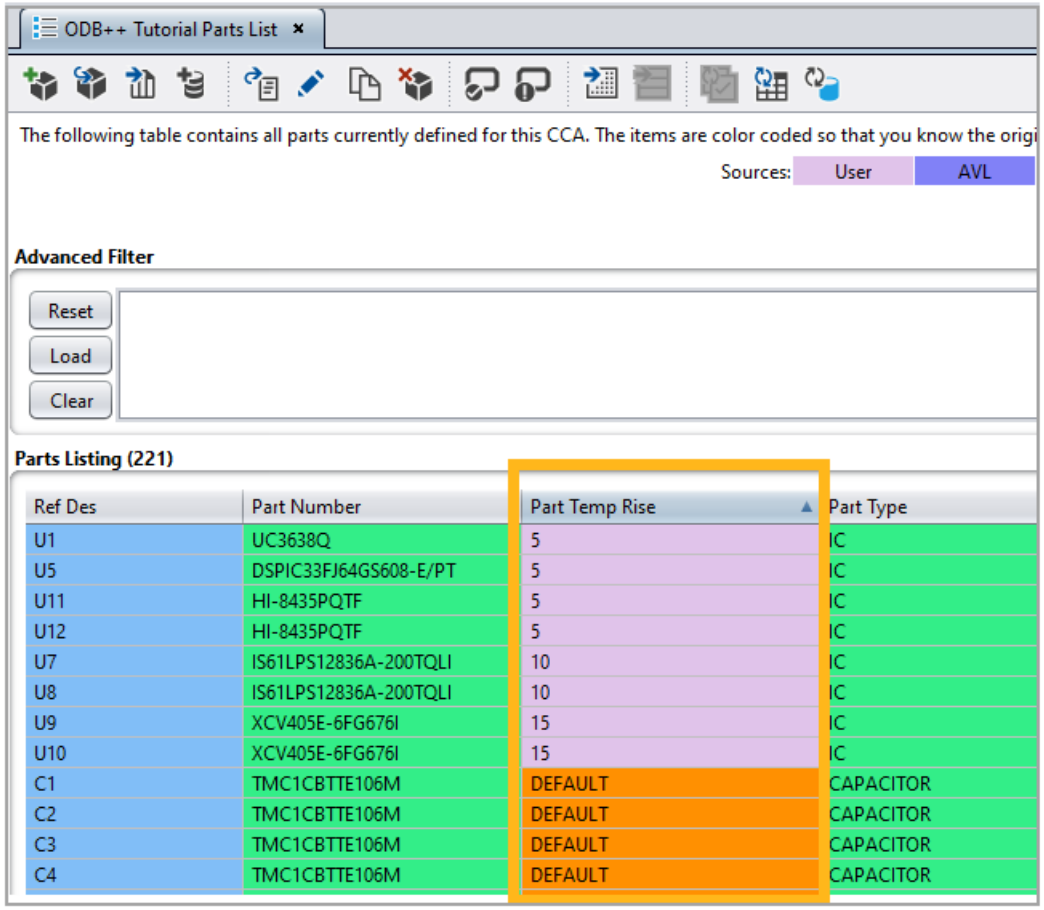


For a given Thermal Event, Sherlock will raise the temperature for those parts for which **Part Temp Rise** has been set and solves for temperature distribution across the board for the new thermal load. The thermal distribution is then used to solve for deformation and strain from structural analysis. You can set the temperature rise for each part in the circuit from **Part Editor** as shown below.



Part Temp Rise for all the parts in the circuit card can be viewed under the Parts List as shown below. You may need to customize the table to see the **Part Temp Rise** column as explained in Customizing the Parts List (p. 191).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



It should be noted that Part Temp Rise value set under each part will be used only if **Use Part Temp Rise** is set to YES. In the **Part Editor**, **Part Temp Rise** can either take a positive value for temperature or can be set to DEFAULT. When a part is set to DEFAULT, Sherlock does not apply any temperature boundary condition to the part but rather solves for its respective nodal temperature based on other boundary conditions.

With the Part Temp Rise set for the components, Sherlock runs the analysis for the selected Thermal Event(s). Once the analysis is complete, Sherlock will read the result file and determine the strain range for the Thermal Event.

9.6.5.2. Thermal Events without Part Temperature Rise

When **Use Part Temp Rise** is set to NO, Sherlock will skip thermal analysis (unless the Thermal Event is associated with Thermal Maps) and run an isothermal analysis by applying uniform temperature from Thermal Profile throughout the circuit under static structural analysis.

9.6.5.3. Thermal Events with Thermal Map (CSV)

Thermal Events (C) with Thermal Map (CSV): Sherlock also supports running thermal analysis by assigning temperature to Parts using CSV file. This feature is termed as Thermal Map (CSV). With this new feature, Users can assign temperature to the Parts using their RefDes in the CSV file. The CSV file format is as shown below (A).

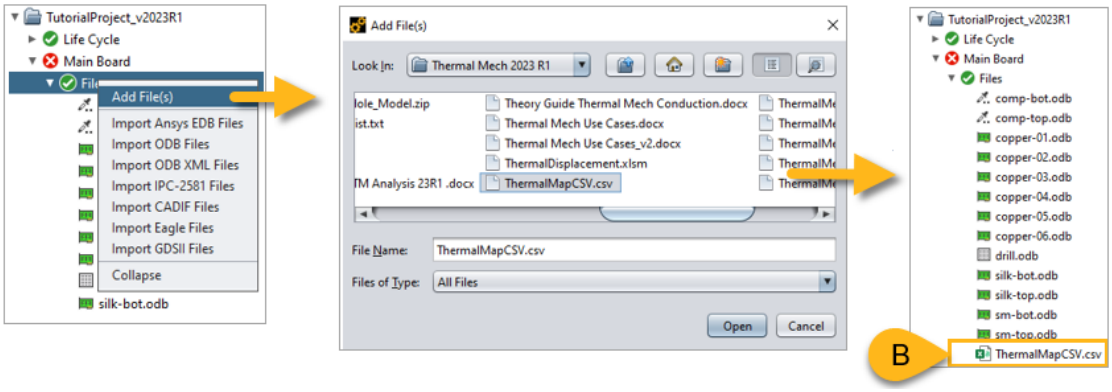
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	A	B
1	RefDes	Temp
2	U5	70
3	U7	75
4	U11	70
5	U9	80
6	U10	80
7	U12	70
8	U8	75

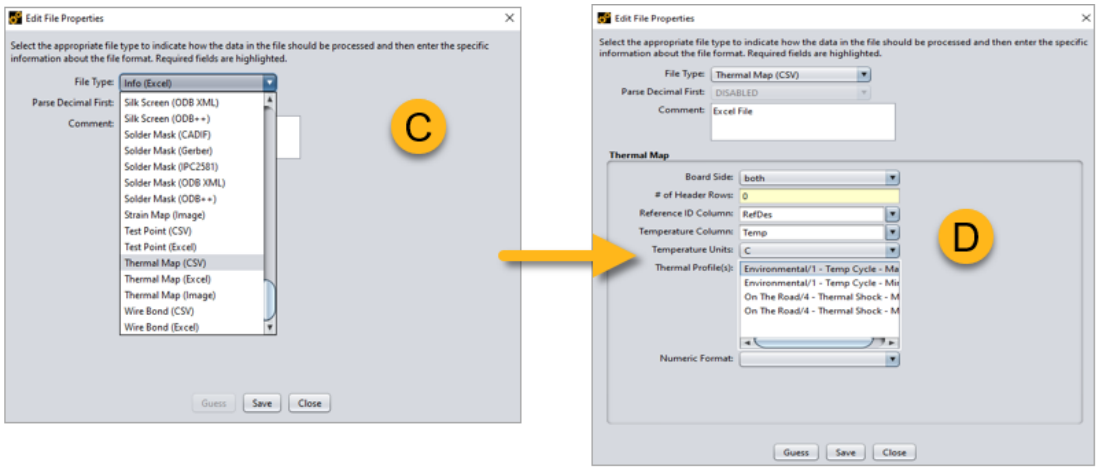
A

It should be noted that with **Part Temp Rise**, the temperature rise value in **Part Editor** is added to the part in addition to the temperature from Thermal Profile during the analysis. Unlike Part Temp Rise, temperature from CSV files is directly applied to the parts as is and is not an additional temperature value on top of the values from the Thermal Profile.

A CSV file formatted as shown above (A) can be added to appropriate Sherlock project under Files as shown below. Once the file has been added to the **Project Tree**, it should be edited for Thermal Map (CSV) application as explained in the example below.



Example: To edit the CSV file, right-click the **ThermalMapCSV.csv** (B, above) in the project tree. This opens the **Edit File Properties** window (C, below). In **Edit File Properties** window, change the **File Type** to **Thermal Map (CSV)**. This displays the **Thermal Map** panel (D).



Thermal Map (CSV) options in the **Edit File Properties** window:

- **Board Side**: The option is immaterial when using a Thermal Map (CSV).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

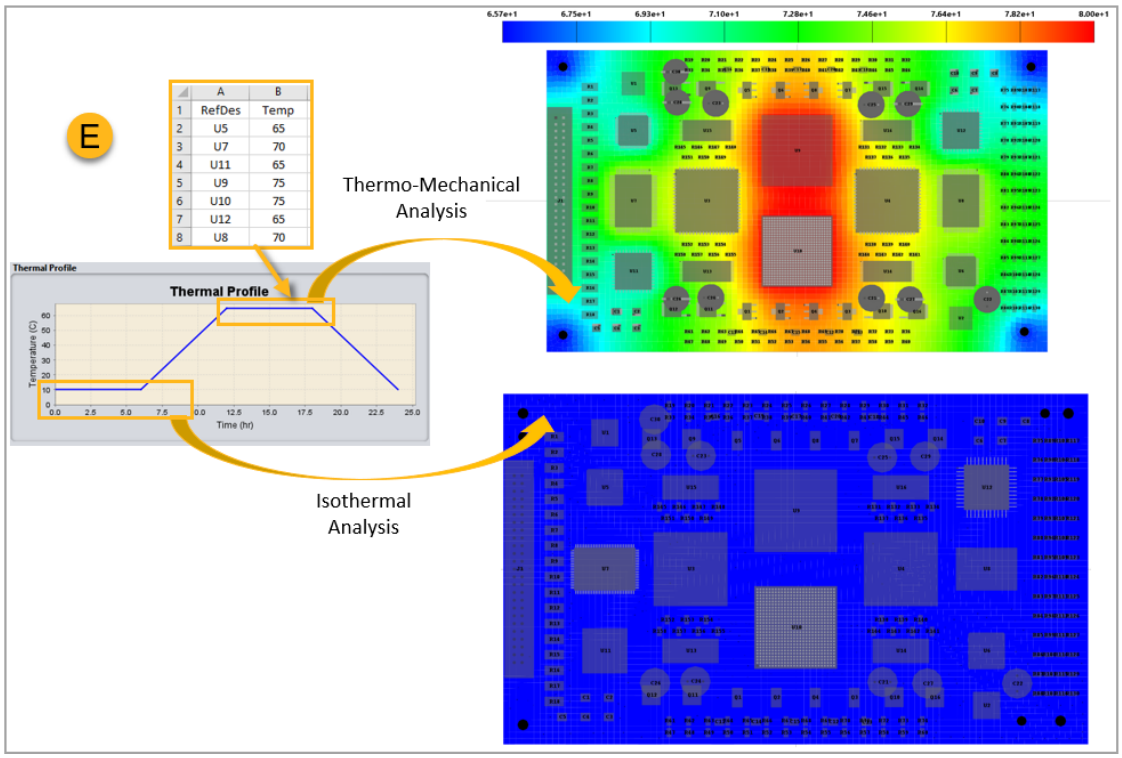
- **# of Header Row**: Specify the number of header lines in the CSV file.
- **Reference ID Column**: Specify which column contains the reference designators (RefDes).
- **Temperature Column**: Specify the column that lists the temperature data.
- **Temperature Units**: Select the units used in the CSV file.
- **Thermal Profile(s)**: The panel lists the minimum and maximum step of all the Thermal Events defined under the project Life Cycle. It is imperative that the CSV file is associated with one or more Thermal Events.

Example of a Thermal Mech Analysis with a Thermal Map (CSV)

Consider a Thermal Mech analysis where the CSV file shown above (A) is associated with the **Environment 1-Temp Cycle - Max**, as shown in image D. Sherlock runs a two-step thermal analysis followed by a two-step structural analysis using the temperatures from the RTH file obtained from the previous thermal analysis.

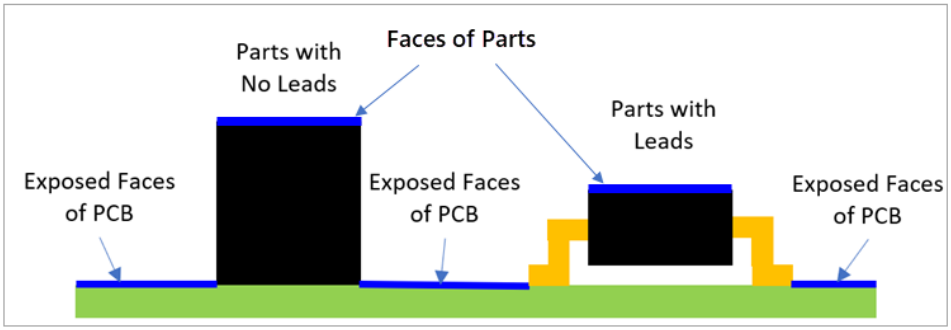
As illustrated below (E), the two steps in the analysis correspond to the minimum temperature and the maximum temperature specified in the Thermal Profile of the selected Thermal Event. When the CSV is associated to **Environment 1-Temp Cycle - Max**, the part temperatures in the field are applied to the maximum step of the analysis only.

If no file is associated to the minimum step of the Thermal Event (**Phase 1/Thermal Event - Min**), Sherlock does *not* run a thermal analysis but instead applies the minimum temperature from the event as a uniform temperature across the circuit card. The temperature distribution across the circuit boards used for the static structural analysis is shown below (E).

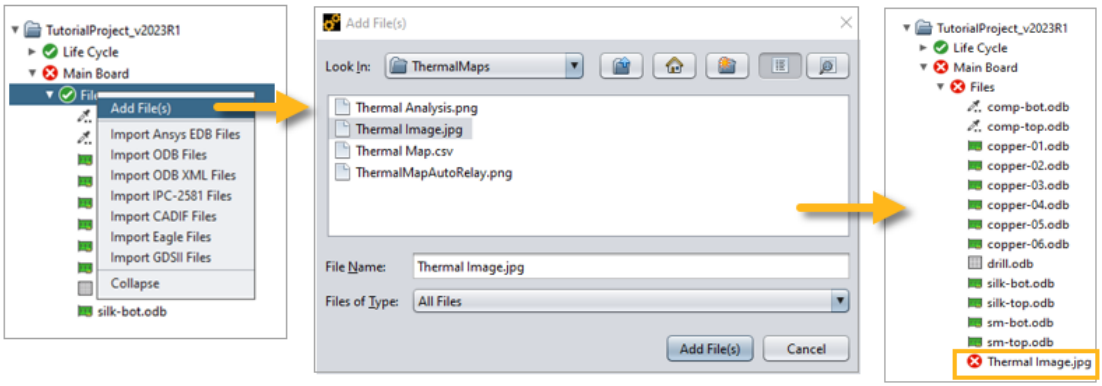


9.6.5.4. Thermal Events with Thermal Map (Image)

Thermal Events with Thermal Map (Image): You can provide temperature distribution data in image form, and Sherlock will run the thermal analysis using the image's temperature data and sequentially carry out a structural analysis to determine displacements and strains. When Thermal Map (Image) is used, boundary conditions are applied to all the exposed faces of the circuit card, including parts as well as the PCB, as shown below. This stands contrast to how a Thermal Map (CSV) is used, where Sherlock applies temperature as a boundary condition to parts only.



- The process of adding a Thermal Map (Image) file to Sherlock is similar to a Thermal Map (CSV), as shown below.



Note:

The file used in the following example is packaged with your Sherlock software in the following location (for a standard installation):

Windows:

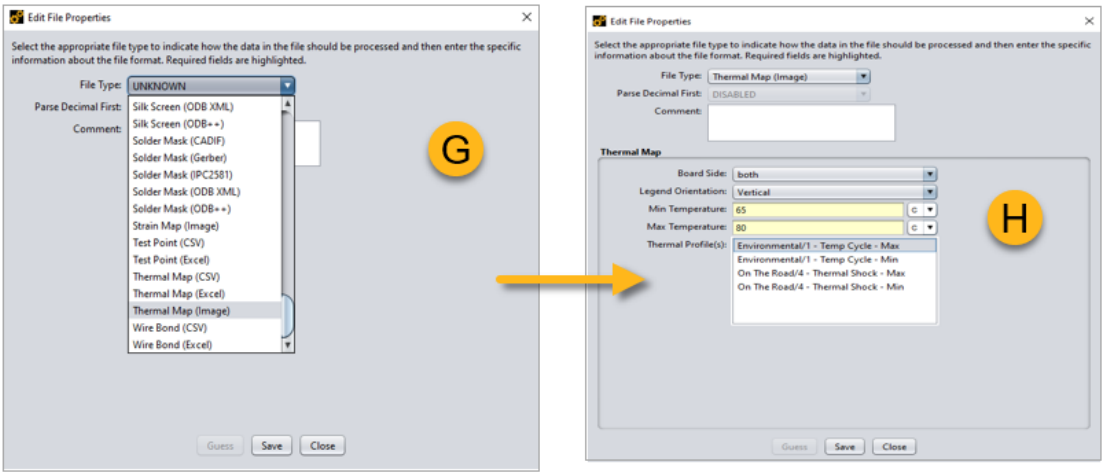
```
C:/Program Files/ANSYS Inc/v251/sherlock/tutorial/ThermalMaps
```

Linux:

```
/ansys_inc/v251/sherlock/tutorial/ThermalMaps
```

Once the file has been added to the **Project Tree**, it should be edited for a Thermal Map (Image) application: Right-click the image file in the **Project Tree** and select **Edit Properties** in the context menu. This opens the **Edit File Properties** window (G). You must change the **File Type** to **Thermal Map (Image)**. This displays the **Thermal Map** (Image) options panel (H).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Thermal Map (image) options in the **Edit File Properties** window:

- **Board Side**: Although this setting is not used for a Thermal Map (CSV), it is an important setting for using a Thermal Map (image). Use the dropdown menu to select whether the thermal map image applies to the top, bottom, or both sides of the board.

- **Legend Orientation**: Indicate whether the legend in the temperature map is horizontal or vertical. This helps Sherlock associate the minimum and maximum temperatures with the correct ends of the legend bar.

- **Min Temperature** and **Max Temperature**: Enter the minimum and maximum temperatures displayed in the thermal image so Sherlock knows the temperature range conveyed in the image. In the example shown above, the minimum temperature is set to 65°C, and the maximum temperature is set to 80°C.

Tip:

The mount points of an electronic board are generally mounted to the casing. During operation, represented by a Thermal Cycle in Sherlock, the mount points are generally exposed to the Thermal Cycle temperature. So, it is important to provide accurate **Minimum Temp** and **Maximum Temp** during Thermal Mapping (Image).

When using a Thermal Map (CSV), Sherlock maintains the Mount Points at appropriate temperature based on the Thermal Cycle. In contrast, when using a Thermal Map *(Image)*, Sherlock respects the **Minimum Temp** and **Maximum Temp** you entered. Sherlock does not force the Thermal Cycle temperature at Mount Points because this changes the contour you are providing in the thermal image.

However, when mount points are modelled as Mount Pads, Sherlock will maintain a temperature consistent with the Thermal Event. Therefore, it is important that you provide accurate **Min** and **Max Temperature** so they are consistent with that of the mount point. A mismatch in temperature near the mount point will cause high gradients across the mount pad.

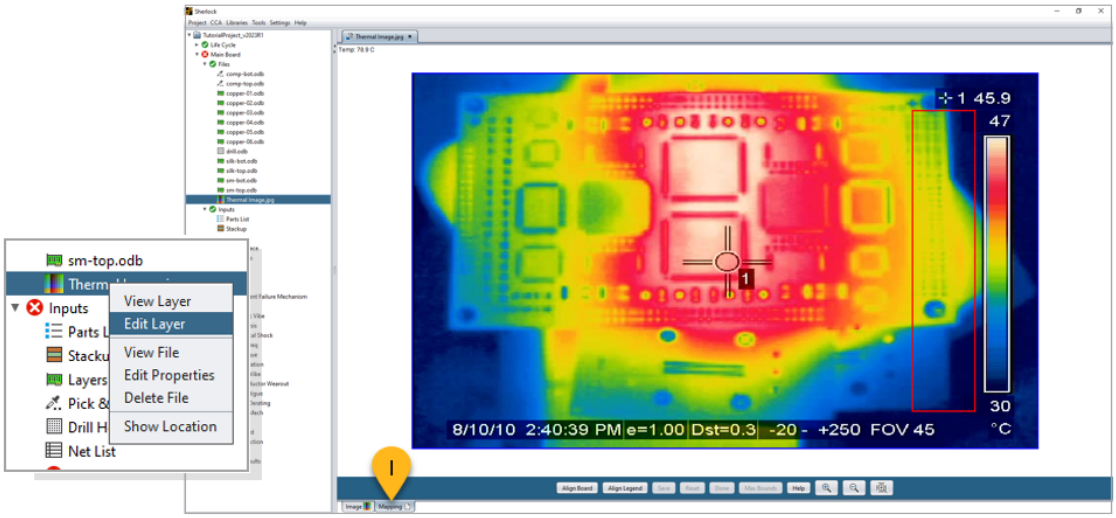
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Thermal Profiles**: As with the Thermal Map (CSV), the **Thermal Profile(s)** selection panel lists the minimum and maximum steps of all the Thermal Events defined under the project Life Cycle (see H above). While using an image file, it is imperative to associate one (or more) Thermal Events with the image file.

Example of a Thermal Mech Analysis with a Thermal Map (Image)

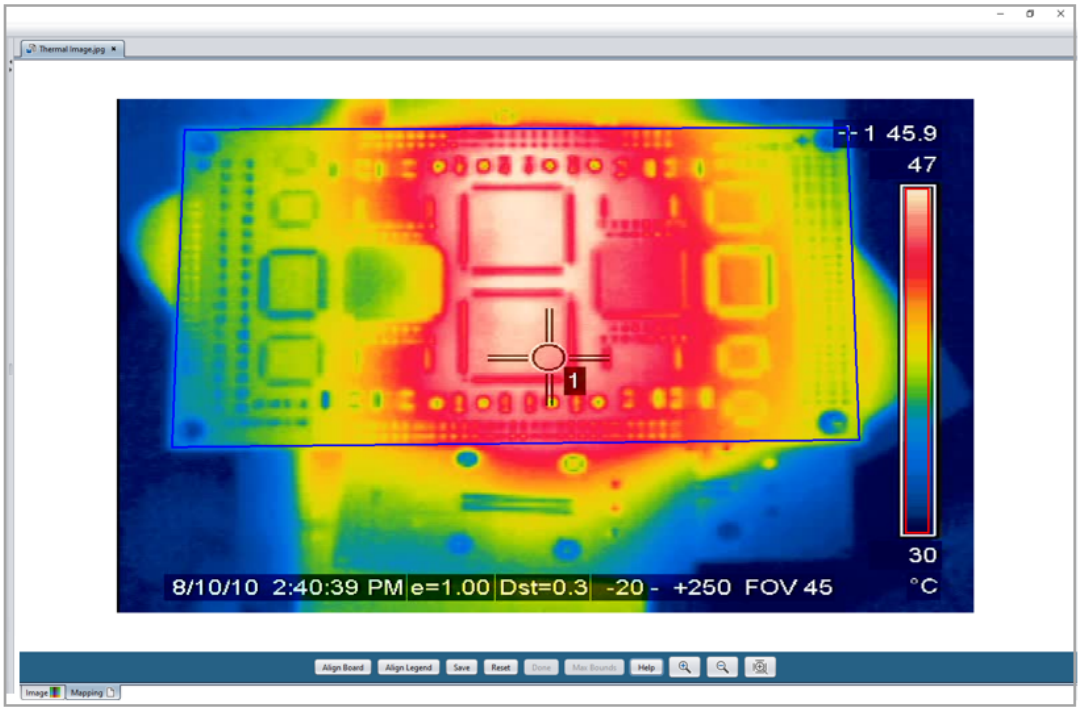
The continuing example illustrates how Sherlock performs the Thermal Mech analysis when the **ThermalImage.jpg** file is associate with the **Environment 1-Temp Cycle - Max** Thermal Profile. In the Thermal Map panel (H, above), the image is associated with **Environment 1-Temp Cycle-Max** (H, above). Click **Save** to save the association.

Once the image association has been saved, you must align the image to the board outline and align the legends correctly. To access the alignment options, right click the **Thermal Image** in the **Project Tree** and select **Edit Layer** in the context menu, as shown below.



The option will open the **Mapping** tab (I, above), where you can align the board and the legend using the **Align Board** button and the **Align Legend** buttons. Sherlock marks the board with a blue outline and the legend with a red outline.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

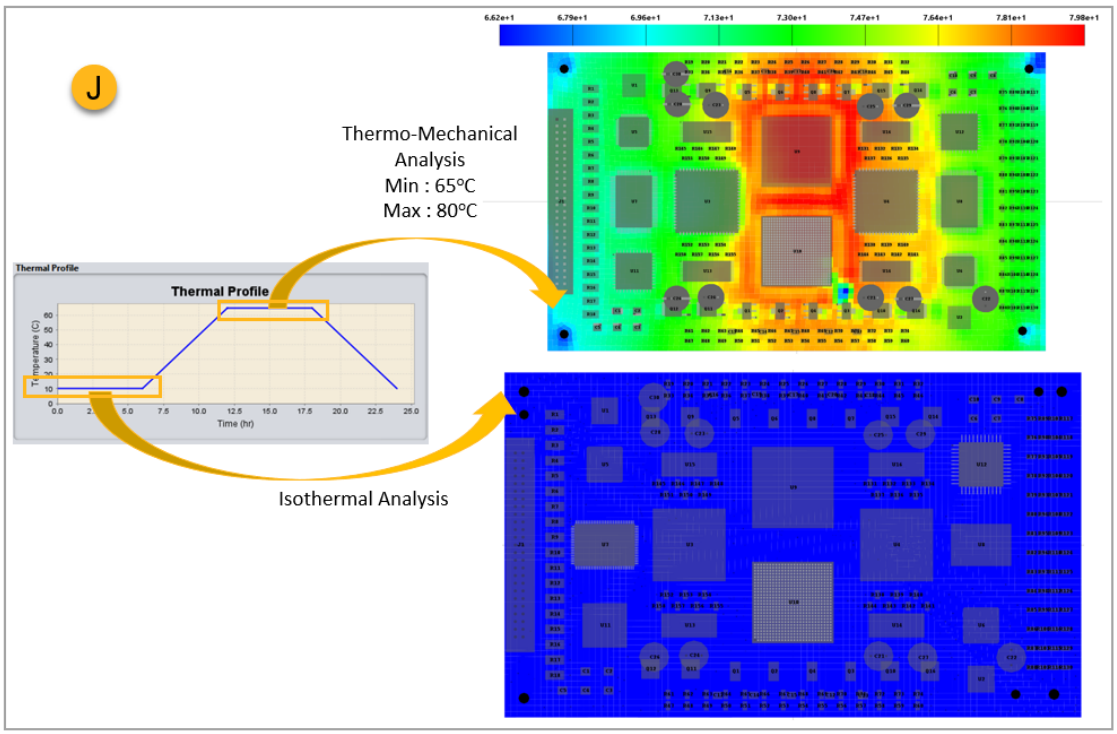


When the alignments are saved, Sherlock gives you the option to view an overlay of the thermal image and the board within the **Layer Viewer**, as shown above. The image shows us, at the maximum temperature of the cycle, the Mount Points are at 65°C as per the settings, and the maximum temperature in the contour is 80°C represented by the reddish white in the image.

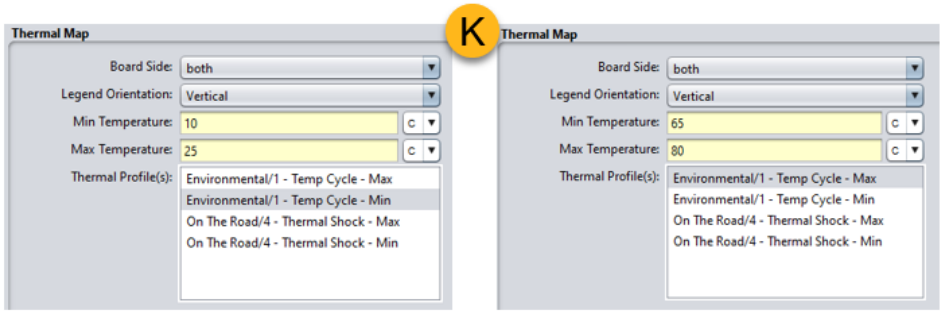
During the Thermal Mech analysis, at the maximum temperature step, Sherlock applies the temperature distribution (from the image) to the exposed faces of the PCB, to the parts, and to any potting regions. All the other nodes will be solved for during thermal analysis. To obtain strain results, Sherlock follows the thermal analysis with a static structural analysis.

In the example below (J), the Thermal Map (Image) is associated with the maximum step only of the Thermal Event (**Environment 1-Temp Cycle**). The minimum step is not associated with any image or CSV file, so Sherlock treats this step as an isothermal loading condition.

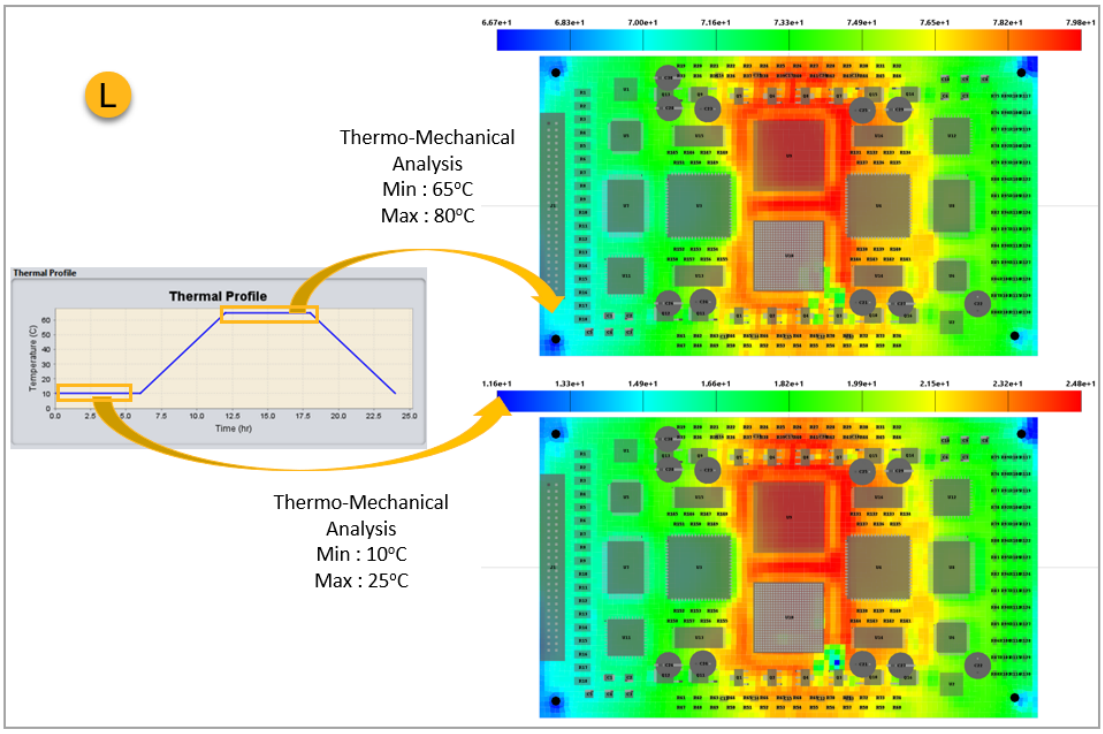
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Sherlock also allows you to assign separate images for the minimum and the maximum step of a Thermal Event as shown (K, below). The temperature distribution for the two steps based on the maps is also shown (L) for reference.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.6.6. Summary of Analysis Combinations

For any given Thermal Event, Sherlock runs a static structural analysis without conduction using the Ansys solver for all the analyses, assuming isothermal temperature rise. For an analysis that involves conduction, Sherlock runs a thermal analysis followed by a static structural analysis using results from the previous thermal run.

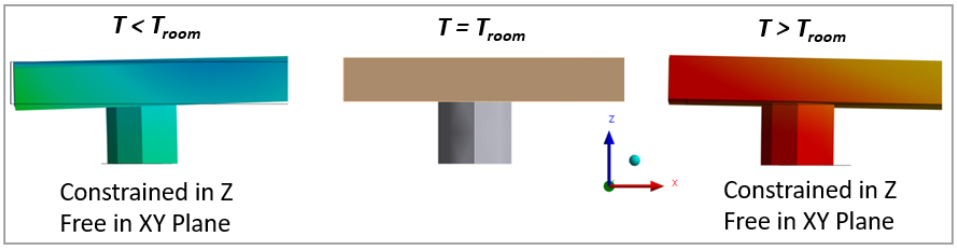
When one of the steps of a Thermal Event (the minimum or maximum temperature) is associated with a Thermal Map (CSV/Images) while the other step is not, Sherlock will carry out a static structural analysis (isothermal rise) for the step *without* the Thermal Map and carries out a thermo-mechanical analysis for the step *with* the Thermal Map. The table below shows the different combinations possible in Sherlock.

	Thermal Event	**Thermal Condition**	**Analysis**
Case 1	Both Steps	No Part Temp Rise No Thermal Map	Static Structural
Case 2	Both Steps	Part Temp Rise	Thermal + Structural
Case 3	Step 1 – Min Step 2 – Max	No Thermal Map Thermal Map	Static Structural Thermal + Structural
Case 4	Step 1 – Min Step 2 – Max	Thermal Map Thermal Map	Thermal + Structural Thermal + Structural

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.6.7. Thermal Mech and Mount Points

For all the analyses in Thermal Mech, Sherlock allows thermal displacements at the mount points based on the chassis material. Users should understand that the Mount Points are not fixed but are provided a displacement boundary condition during structural analysis based on the temperature and the CTE of the chassis material. If the mount points are modelled as Mount Holes, Sherlock uses the average CTE of chassis and the board material. This is to ensure accurate in-plane expansion of the chassis material on the board. The graphical representation of thermal displacements is shown below. At room temperature/reference temperature, the mount point remains undeformed. At different temperatures, the nodes of the mount points are displaced with respect to the center of the mount points based on the CTE of chassis.



9.6.8. Part Temp Rise and Thermal Map FAQ

1. **Can two Images/CSV be associated to a single Thermal Event ?**

   A step (minimum or maximum temperature) of a given Thermal Event cannot carry more than one thermal image. When Image-A is associated to a Thermal Event-Max step, if Image-B is associated to the same step, Image-B will override Image-A. The analysis will be run using Image-B.

2. **When does Part Temp Rise takes precedence?**

   Sherlock uses Part Temp Rise for running Thermal Mech, only when **Use Part Temp Rise** is set to YES. If you want no effect from Part Temp Rise during a Thermal Mech analysis, you must set it to NO.

3. **Can one step of the Thermal Event be associated to a Thermal Map (Image) and other step be associated to a Thermal Map (CSV)?**

   Yes.

4. **Does *potting* get temperature loads during Part Temp Rise or Thermal Map (CSV)?**

   No, potting geometry does *not* get temperature loads as boundary conditions whether from Part Temp Rise or Thermal Map (CSV). Temperature boundary conditions are applied only to the Parts. However, when running Thermal-Mech using Thermal Map (Images), Sherlock does apply temperature loading condition to the potting geometry.

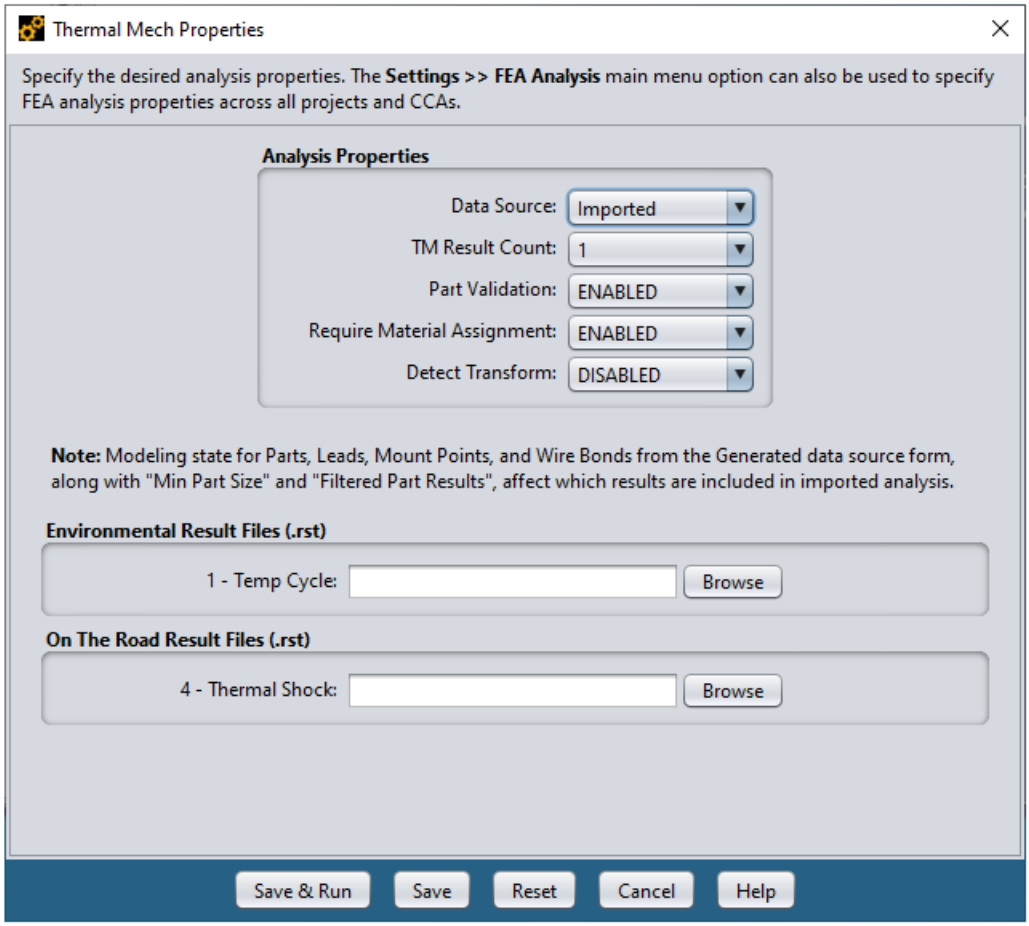
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.6.9. Imported Data Source

In the **Thermal Mech Properties** dialog, when you choose **Imported** as the **Data Source**, the interface changes as shown below. Use this interface to import thermal mech analysis results (`.rst`) from Ansys Workbench to Sherlock.

Tip:

For detailed, step-by-step instructions for importing results from Ansys Workbench, see the following section: FEA- Ansys Workbench™ Integration (p. 747).



When the **Detect Transform** is ENABLED, Sherlock will attempt to detect any Transforms applied to the imported model and make adjustments if required. This ensures the imported model matches Sherlock's original model. Select MANUAL to enter Transform properties manually. If a Transform exists and is not detected, Sherlock will not be able to provide component results. When running the **Detect Transform** feature, Sherlock uses multiple threads by default. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

If any of the file entries are blank, then Sherlock will assume that no analysis was performed for those Events.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **Save & Run** button is pressed, Sherlock will import both the model and the results to determine the reliability results.

Note:

When importing data, it is your responsibility to assign the proper results file to each Event. Sherlock has no other way of determining the association.

Note:

If lead results are not generated after importing results which contain leads, enable **Lead Modeling** on the **Generated Data Source** (p. ?) form and perform the imported analysis again

---

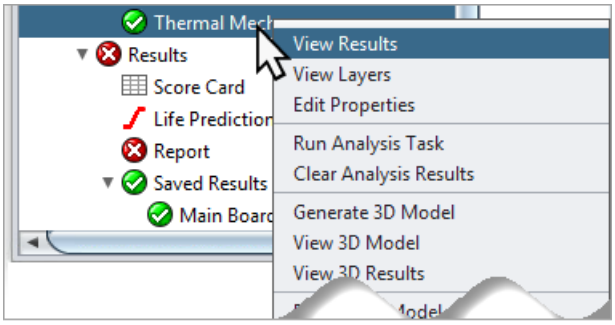
9.6.10. Thermal Mech Analysis Results

Once the analysis is complete, Sherlock offers several ways to view the results. While there is a complete guide to navigating analysis results in Viewing and Managing Analysis Results (p. 785), the section below highlights features relevant to Thermal Mech. Similar to other modules, Sherlock presents the results in tabular form as well as in graphical form which visualizes board and the elemental strains. Irrespective of the type of thermo-mechanical analysis, once the analysis is complete, you can access results from:

- Results Table (p. 730)
- Layer Viewer (p. 738)
- Sherlock 3D Viewer (p. 739)

9.6.10.1. The Results Tables

When a Thermal Mech analysis is performed, Sherlock generates tabular results for each part, modeled lead, mount point, and wire bond. To access the **Current Results Table**, right-click **Thermal Mech** in the **Project Tree** and select **View Results** in the context menu.



The **View Results** option opens the **Current Results** window with the **Thermal Mech** tab selected (A, in the image below). Use the bottom row of tabs (B) to access results on different aspects of

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the analysis: **Summary**, **TM** (Thermal Mech), **Parts**, and **Leads**. In certain conditions, other tabs (such as an **Error** or **Issues** tab) might appear.

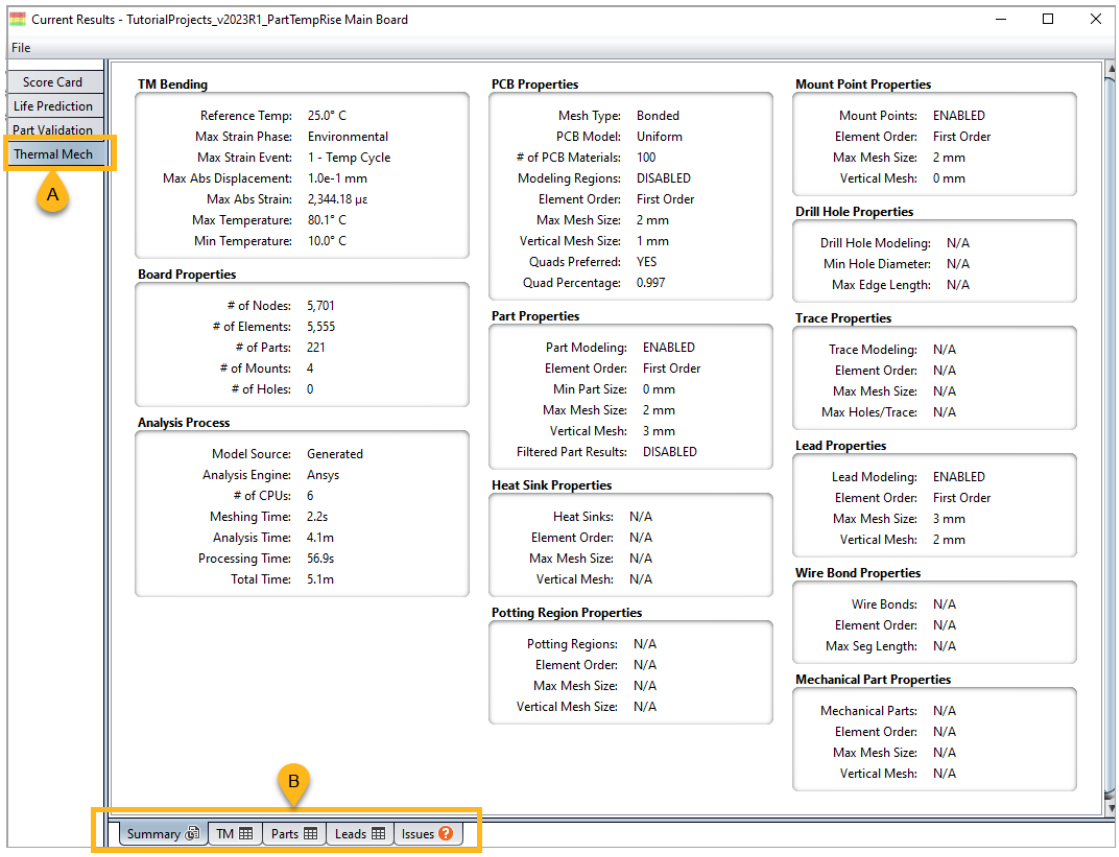
Tip:

Sherlock alerts you to errors or issues encountered during the analysis: An error icon or an issues icon will appear in the **Project Tree** next to **Thermal Mech**:

Thermal Mech - Issue: The analysis ran to completion but Sherlock identified issues you should investigate.

Thermal Mech - Error: Analysis did not run to completion due to errors.

To get more information, click the **Issues** or **Errors** tab in the **Current Results** window. The example below shows the results of an analysis where Sherlock identified issues that should be investigated (note the **Issues** tab at the bottom).



Current Results Tabs for Thermal Mech

- **Summary**: The **Summary** tab provides essential information about the analysis such as the reference temperature, maximum board displacement and strain, the Phase and Event at which the maximum delta strain took place along with the maximum and minimum temperatures the board experienced. In a sense, the information provides displacement and strain from the worst-case Event out of all the Events used during the analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

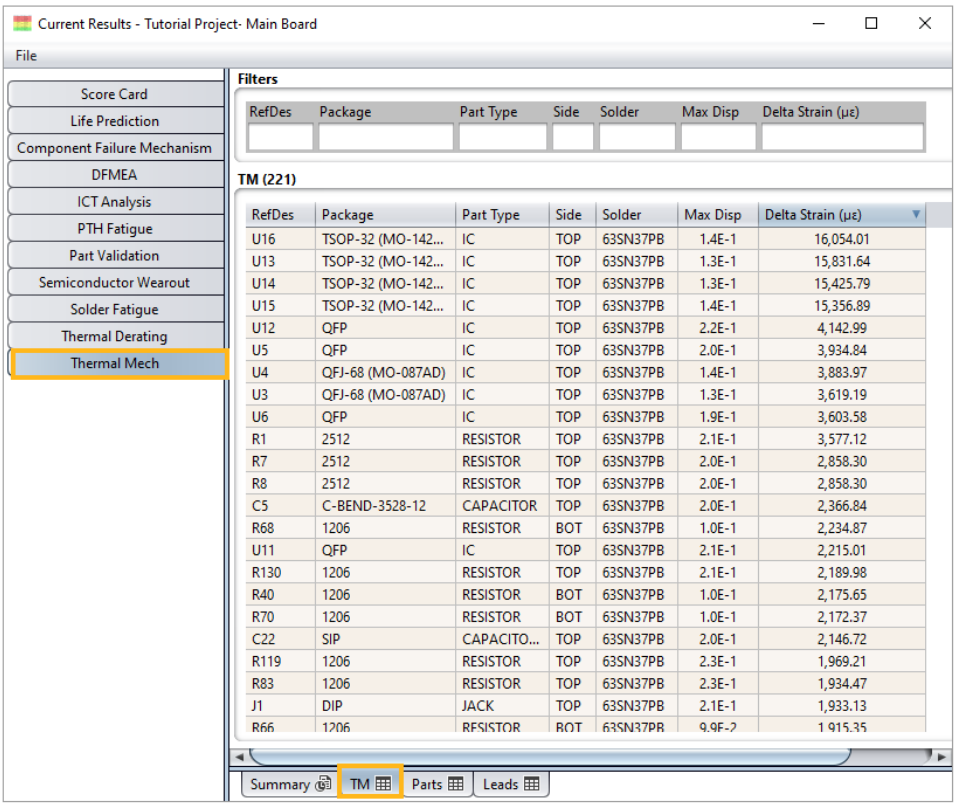
If the analysis used Part Temp Rise or Thermal Mapping, the Max and Min Temperature will *not* be the maximum and minimum temperatures from the Event—but rather, the maximum and minimum temperatures the board experienced during the analysis.

Sherlock also provides pre-processing information for the analysis under **Board Properties**, **PCB Properties**, **Part Properties**, and so on. Finally, information regarding solve time is also listed under **Analysis Process**.

- **TM (Thermal Mech)**: The **TM** tab, as shown below, contains the main table which displays the delta strain for each part analyzed during the analysis. Strain values are in units of microstrain (με).

Note:

For more information on how Sherlock calculates delta strain, licensed customers may refer to the Sherlock Theory Reference.



Tip:

Results tables in Sherlock are customizable. To add, remove, or reorder columns, right-click any cell and select **Customize**. The customization interface is identical to the one for customizing Parts tables (p. 239).

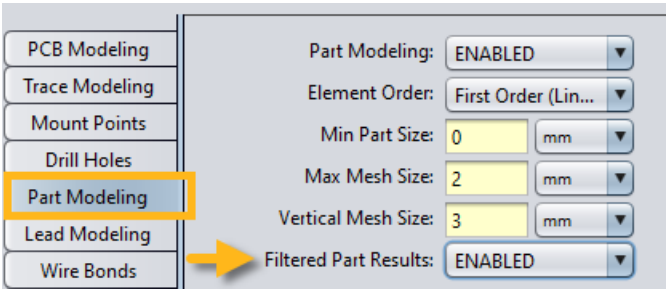
The **TM** table provides the following information:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **RefDes**: The reference designator of the part.

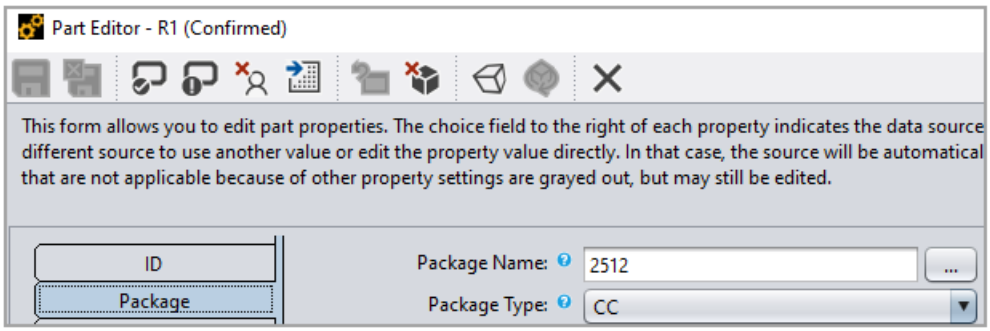
Tip:

Why aren't all the parts appearing on the Results Table? It's important to remember, all parts modeled as finite element models will appear in the results. Non-modeled parts appear only if you request it when configuring the analysis. As explained in FEA Model Properties (p. 562), if you wish non-modelled parts to appear in the results, set **Filtered Part Results** to ENABLED as shown below. This setting is found in the **Thermal Mech Properties** dialog. (p. 713)



– **Package**: The column provides the package name of the part as specified by the **Package Name** property in the **Part Editor**.

– **Part Type**: The column provides the package/part type as specified by the **Package Name** and **Package Type** properties in the **Part Editor**, as shown below.

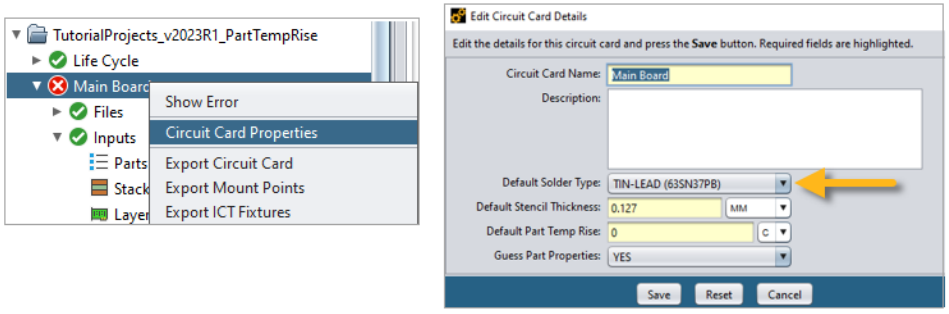


– **Side**: This column indicates which side of the board the part is located. Any part that is not leaded uses board strain for delta strain calculations. If the part is located on the top side (**TOP**) of the board, Sherlock uses board strain from the top side of the PCB. If the part is located on the bottom (**BOT**), delta strain is based on surface strain from the bottom surface of the board.

– **Solder**: The column shows the solder material specified for the part. By default, all parts have their **Solder Material** property set to DEFAULT. This means Sherlock will use the circuit card's default solder type. You can change the **Solder Material** for an individual part by using the **Solder** tab in **Part Editor**.

To determine or to set the **Default Solder Type** for the circuit card, right-click the circuit card in the **Project Tree** and select **Circuit Card Properties** as shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



– **Max Disp**: The column gives the maximum board displacement (in mm) at the footprint of the part. It is important to understand that the maximum displacement happened during the Event specified under the **Delta Strain Event** column. If the part is leaded, the displacement provided in the column is for the footprint of the part which includes the leads.

Note:

For a figure illustrating the footprints as described above, see PCB Strain in the *Sherlock Theory Reference*.

– **Delta Strain (με)**: This column provides maximum delta strain for the part among the all the Events included in the analysis. If the component is leaded, the column gives the delta strain for the leads. If the component is not leaded, the delta strain is calculated from the board surface strain.

Note:

For details on delta strain calculation, see Thermal Mech Theory in the Sherlock Theory Reference.

– **Strain Source:** This column gives the source of delta strain calculation. If the component is leaded, the column will read **Lead Strain**, if the component is not leaded, the strain will be from **PCB Strain**.

– **Delta Strain Event**: This column indicates the Event during which the results for the part were calculated. Remember, the Event which caused maximum delta strain for one part may not be the same Event for another part.

– **Material**: This column shows the material of the part. This comes from the part's **Material** property which you can view or modify in the **Part Editor**.

– **PCB Element ID**: This column displays the ID (from the finite element model of the PCB) of the element that cause the maximum surface strain for the part.

– **Weight**: This column gives the weight of the Part. This comes from the part's **Weight** property which you can view or modify in the **Part Editor**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Parts Tab**: The table in the **Parts** tab lists three strains for all the parts on the board: 1) **PCB Strain**, 2) **Part Strain**, and (3) **Lead Strain**. If the part is not leaded, the **Lead Strain** column will be empty.

  For a Thermal Mech analysis, each part will have two entries: one at the minimum temperature of the Event causing maximum delta strain and the second one at the maximum temperature of the same Event.

  Note:

  For information on how Sherlock calculates these values, refer to Thermal Mech Strain in the Sherlock Theory Reference.

  Tip:

  Remember, results tables in Sherlock are customizable. To add, remove, or reorder columns, right-click any cell and select **Customize**. The customization interface is identical to the one for customizing Parts tables (p. 239).

  - **Temperature**: This column provides the temperature step name at which the part strain values were calculated. For Events, this will be consistent with the Event min and max temperatures. If Custom Temperature points were used, the step name will be consistent with what you entered in **Custom Uniform Temps** (p. 714).
  - **PCB Strain**: This column provides the PCB surface strain for the part as maximum elemental surface equivalent strain at the component's footprint for the Event causing maximum delta strain.
  - **Part Strain**: This column displays the part strain as the maximum equivalent element strain for the Event causing maximum delta strain.
  - **Lead Strain**: This column provides the maximum lead strain for the part as maximum equivalent element strain for the Event causing maximum delta strain.
  - **Lead Element ID**: This column provides the element ID from the FE model corresponding to the **Lead Strain** column.
  - **Lead ID**: This column provides the lead ID number for the lead strain in Lead Strain column.
  - **PCB Element ID**: This column provides the element ID from the FE model corresponding to the strain value in the **PCB Strain** column.

    Part Element ID: This column provides the element ID from the FE model corresponding to the strain value in **Part Strain** column.

- **Leads Tab**: This table lists three strains for a given lead, namely, **PCB Strain**, **Part Strain**, and **Max Lead Strain**. If there are no leads for any component on the board, the **Leads** tab will not be available. For Thermal Mech, each lead has two entries, one at the minimum temperature of

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the Event causing maximum delta strain and the second one at the maximum temperature of the same Event.

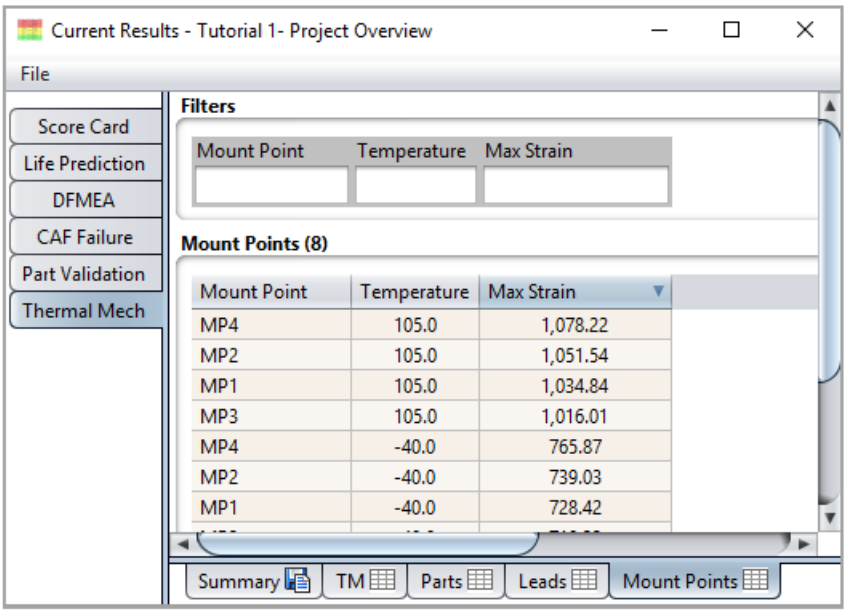
Note:

For information on how Sherlock calculates these values, refer to Thermal Mech Strain in the Sherlock Theory Reference.

- **RefDes**: This column provides the reference designator of the parts that have leads modelled. Parts with leads *not* modelled as a finite element model, are listed in the **Results Table**.
- **Lead ID**: This column provides the lead ID number.
- **Temperature**: This column provides the step name at which the part strain values are calculated. This will be consistent with the Event min and max temperatures. If Custom Temperature points were used, the step name will be consistent with what you entered in **Custom Uniform Temps** (p. 714).
- **Max PCB Strain**: This column will provide maximum equivalent strain of the lead elements of the Lead ID which are attached to the PCB.
- **Max Part Strain**: This column will provide maximum equivalent strain among the lead elements attached to the part.
- **Max Lead Strain**: This column displays the maximum equivalent strain across the given Lead ID.
- **Delta Strain Event**: The column displays the Event causing the maximum delta strain. This is also the Event where Max PCB strain, Max Part Strain, and Max Lead Strain are computed.
- **Lead Element ID**: This column provides the element ID from the FE model corresponding to the lead identified in the Max Lead Strain column.
- **Lead Part Element ID**: This column provides the element ID from the FE lead model corresponding to the **Max Part Strain** column.
- **Lead PCB Element ID**: This column provides the element ID from the FE PCB model corresponding to the **Max PCB Strain** column.

- **Other Tabs**: You will find additional tabs in the **Current Results Table** for further exploration. Which tabs appear depends on the unique features of your board and whether you chose to model them for the analysis. In the example below, there is a Mount Points tab because four mount points were added to the board and **Mount Point Modeling** was ENABLED in the **Thermal Mech Properties** dialog.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



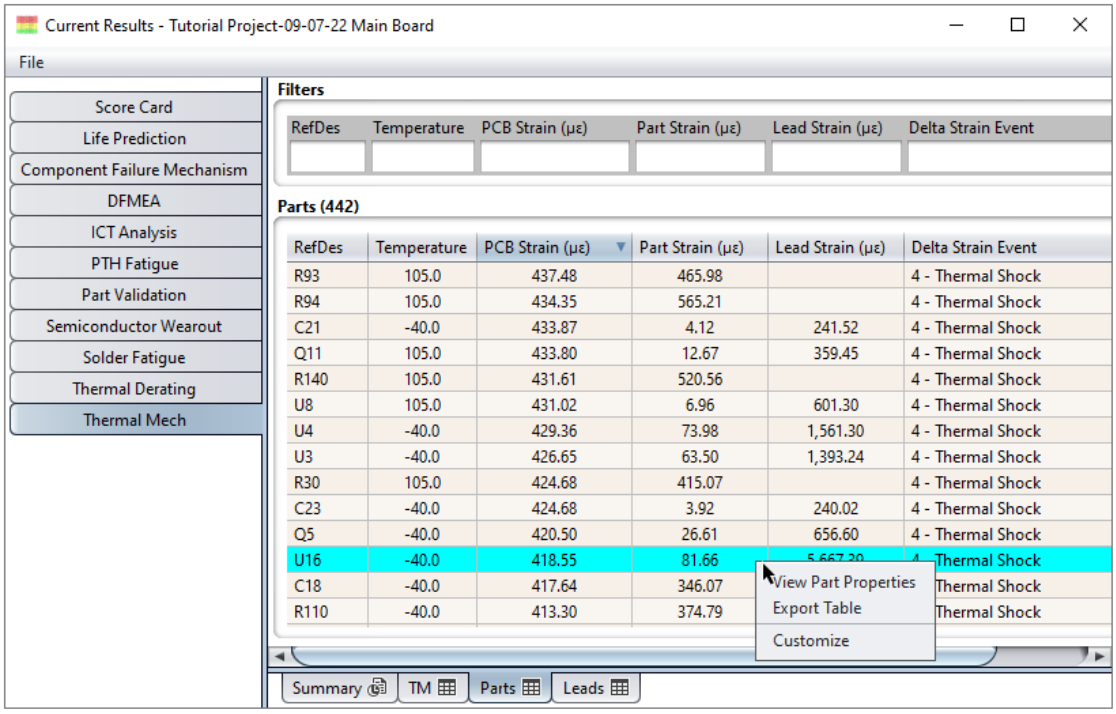
As shown below, you can right-click any row to view a part's properties or to export the table. Double-clicking a row brings up the **Part Properties** window for that part.

As mentioned above if you selected multiple Events for the Thermal Mech analysis (see **Thermal Events** tab (p. 715)), the results displayed for a given part are based on the Event which caused the largest delta strain. In fact, the table includes a column which displays the Event which caused the reported delta strain (see the last column in the table below).

Tip:

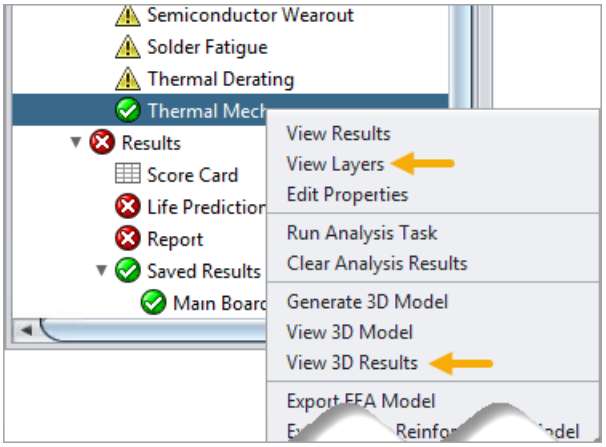
If the **Delta Strain Event** column does not appear in the table, remember, results tables in Sherlock are customizable. To add, remove, or reorder columns, right-click any cell and select **Customize**. The customization interface is identical to the one for customizing Parts tables (p. 239).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.6.10.2. Layer Viewer and 3D Results Viewer

Sherlock provides graphical representations of the results which you can access using the **2D Layer Viewer** and the **3D Viewer**. To open the viewers, right-click **Thermal Mech** in the **Project Tree**. In the context menu, select **View Layers** or **View 3D Results**. The results provided in the **2D Layer Viewer** are only for the PCB and not for the parts on the board. Use the Sherlock **3D Viewer** for viewing parts and leads. For a complete guide to using the viewers, refer to Viewing and Managing Analysis Results (p. 785).



Layer Viewer

The **Layer Viewer** in Sherlock provides the ability to plot Displacement, Stress and Strain on the top and bottom surfaces of the PCB at both the minimum and maximum temperature of a given Event. It should be noted that the results in the Layer Viewer are associated with the PCB and to get information about Parts and Leads, the User should explore Sherlock **3D Viewer** and/or **Results Table**. The displacement plots are displayed in mm, stress and strain on the other hand are displayed

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

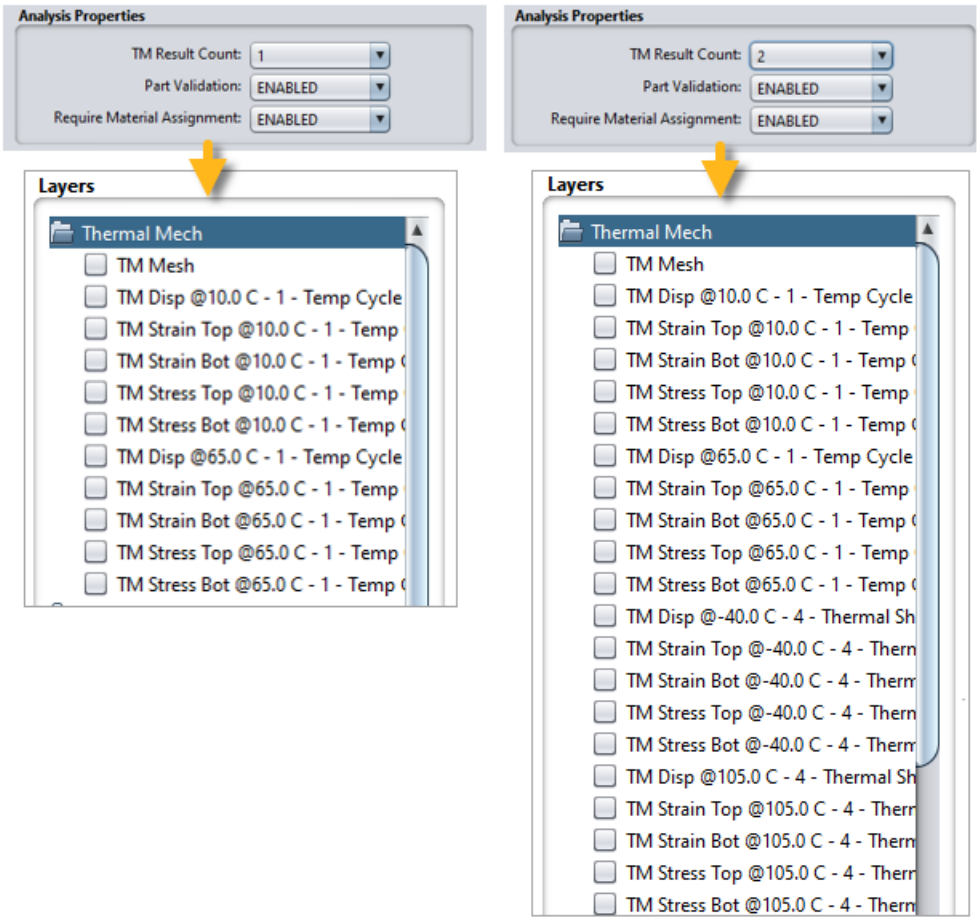
in MPa and micro-strain (με). For Thermal Mech, when the analysis includes conduction (Part Temp Rise or Thermal Map), temperature distribution on the board can be plotted as well.

The naming convention used to represent results in layer viewer is 'TM {PlotName} @ {StepName} – {EventName}. Some examples are shown below :

Plot Name	Plot	Step	Event Name
TM Disp @ 10.0°C – 1 – Temp Cycle	Displacement	10.0°C	1-Temp Cycle
TM Strain Top @ 65.0°C – 1 Temp Cycle	Strain Top	65.0°C	1-Temp Cycle

In the case of multiple Thermal Events, the **Layer Viewer** will display results for the Event(s) that caused maximum displacement. How many Events are displayed is limited by what you entered for the **TM Result Count** (p. 716) (as illustrated in the image below). Also, to avoid cluttering the display, Sherlock displays a maximum of four Thermal Events in the **Layer Viewer**

Because of the upper limit of four Thermal Events in the **Layer Viewer**, if there is a part that has a maximum strain caused by an Event which is not in the top four, the corresponding part's strain cannot be viewed in **Layer Viewer**. This is only a plotting limitation. The results tables still account for all Thermal Events.



3D Result Viewer

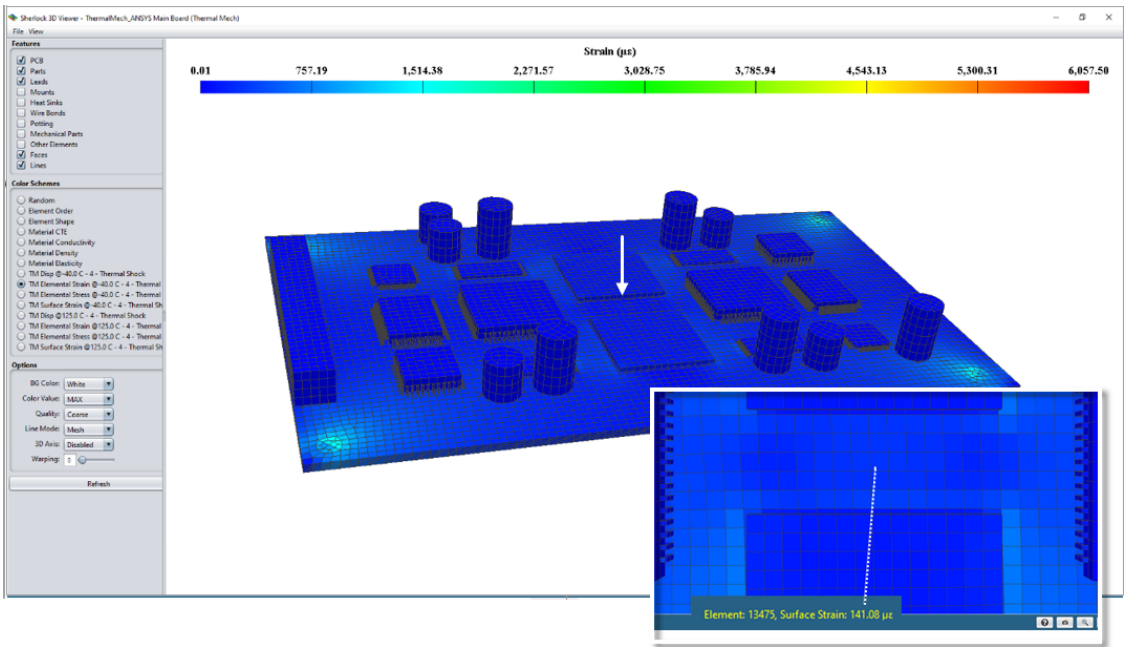
You can view elemental stress and strain in using Sherlock's **3D Viewer**. Sherlock also provides options to view displacement and surface strains in the **3D Viewer** to compare the contour with

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

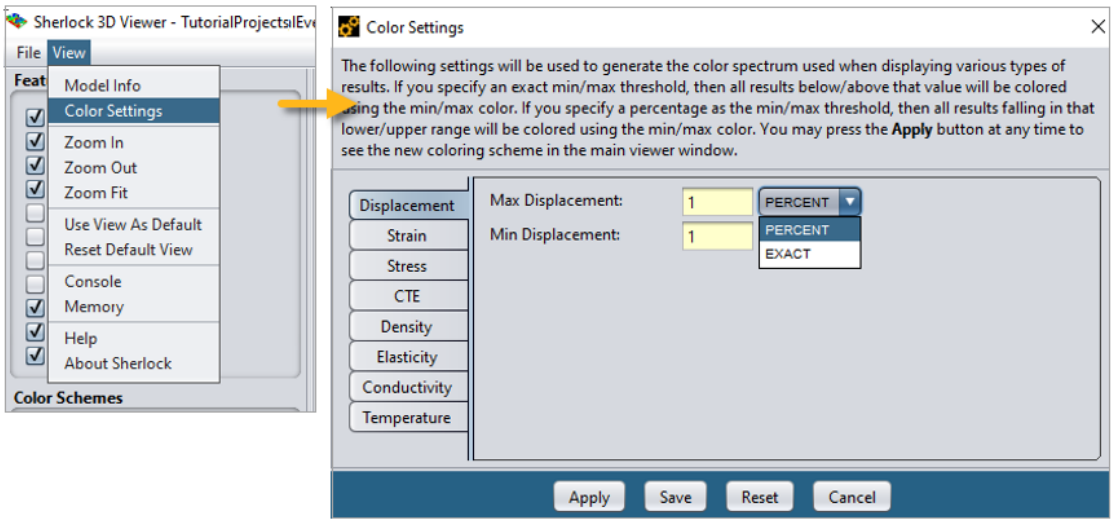
that of the **Layer Viewer**. The inset below illustrates how the **3D Viewer** allows you to view surface strain by simply hovering the mouse over a specific element. (See *Viewing 3D Results* (p. 792) for details.)

Note:

In the **3D Viewer**, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at *reference temperature* only.



In the **3D Viewer**, you can adjust the min and max of the legend, and you can modify the plot based on custom limits. To do this, click **View > Color Settings** to open the **Color Settings** dialog. You can set limits for the parameters as a **PERCENT** or as an **EXACT** value.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.7. FEA- Virtual Accelerometer

Much like real accelerometers are attached to circuit cards to measure loads in real-world and lab testing, Ansys Sherlock allows you to add one or more virtual accelerometers to a circuit card to measure loads during simulated testing. Response curves for each accelerometer shown the displacement and/or acceleration at the accelerometer location at a given time or frequency during the analysis. This document describes how to add virtual accelerometers to a circuit card and the types of results provided by various analysis tasks.

In this section, the following topics are covered:

9.7.1. Accelerometer Editor

9.7.2. Accelerometer Results

9.7.1. Accelerometer Editor

Virtual accelerometers are added, modified, and removed by selecting the **Edit > Edit Accelerometers** menu option in the **Layer Viewer** for a selected circuit card. After that menu option is selected, a set of **Accelerometer Editor** buttons will be displayed at the bottom of the **Layer Viewer** and you will remain in the editor until either the **Save** or **Cancel** buttons are pressed.

9.7.1.1. Adding an Accelerometer

To add an accelerometer to a circuit card, right-click anywhere in the circuit card display and select the **Add Accelerometer** option from the context menu.

In the example shown below (A), we're adding an accelerometer to a simple 51 x 51 mm circuit card, that has a single component (the gray square) in the middle and mount points in each of the four corners (the black squares). Such a simple board will allow us to easily see how the board responds to various loads and how the accelerometer measures the response of the board at a given location.

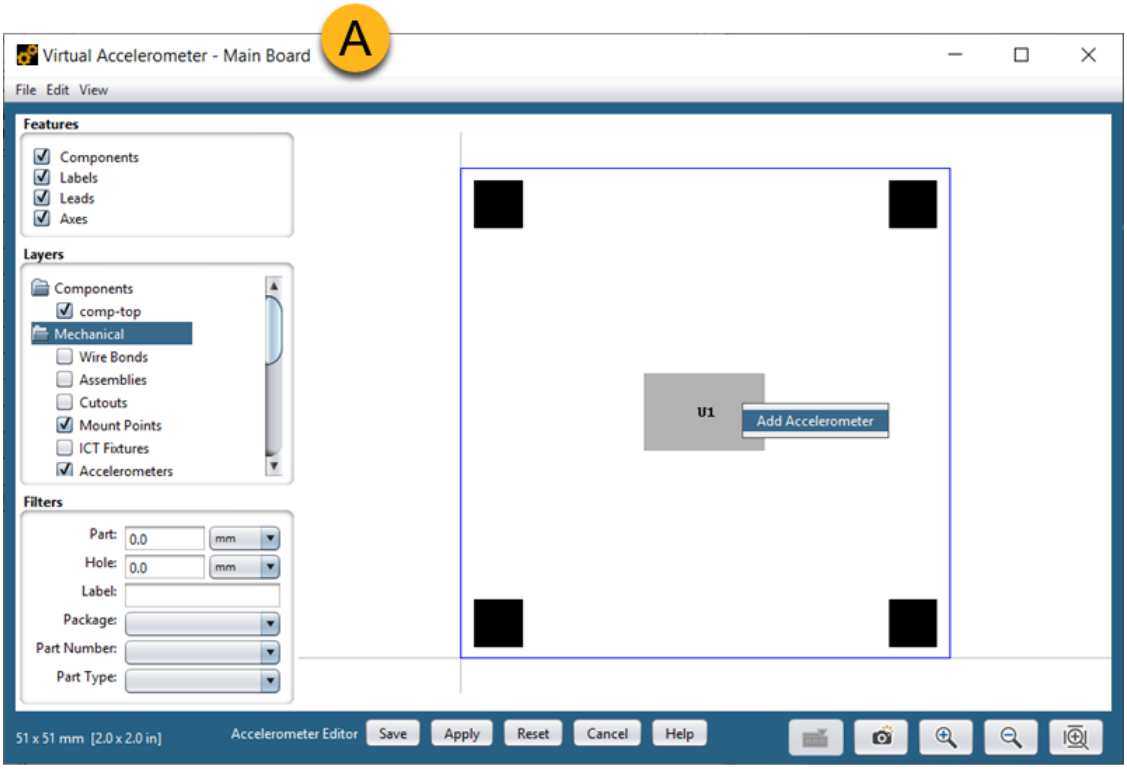
When the **Add Accelerometer** menu option is selected, an edit dialog (B, below) will be displayed allowing us to specify the accelerometer location and properties.

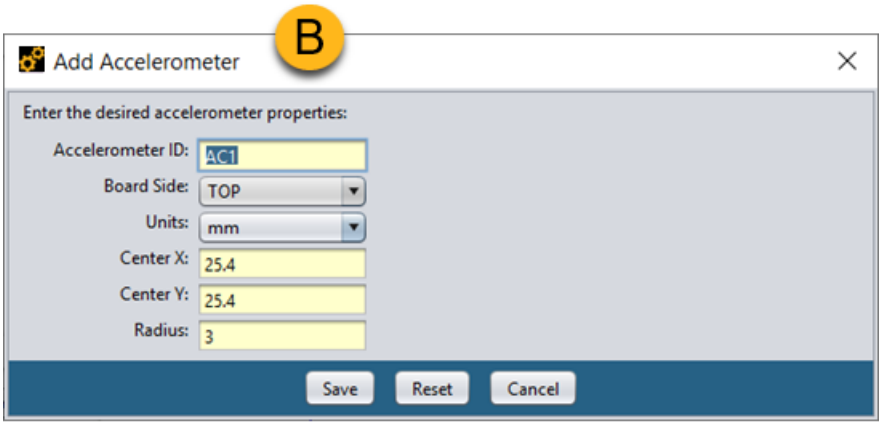
The **Accelerometer ID** is automatically assigned a unique value, which is sufficient for most purposes. You may override that assignment if necessary, provided a unique ID is assigned.

The **Board Side**, **Center X** and **Center Y** properties indicate the board location where the center of the accelerometer will be located, while the **Radius** property indicates the size of the accelerometer.

The accelerometer size determines how many data values will be examined to compute the accelerometer response. Specifically, the maximum value for all simulated FEA nodes lying inside of the boundary of the accelerometer will be used as the value of the accelerometer at a given time or frequency.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

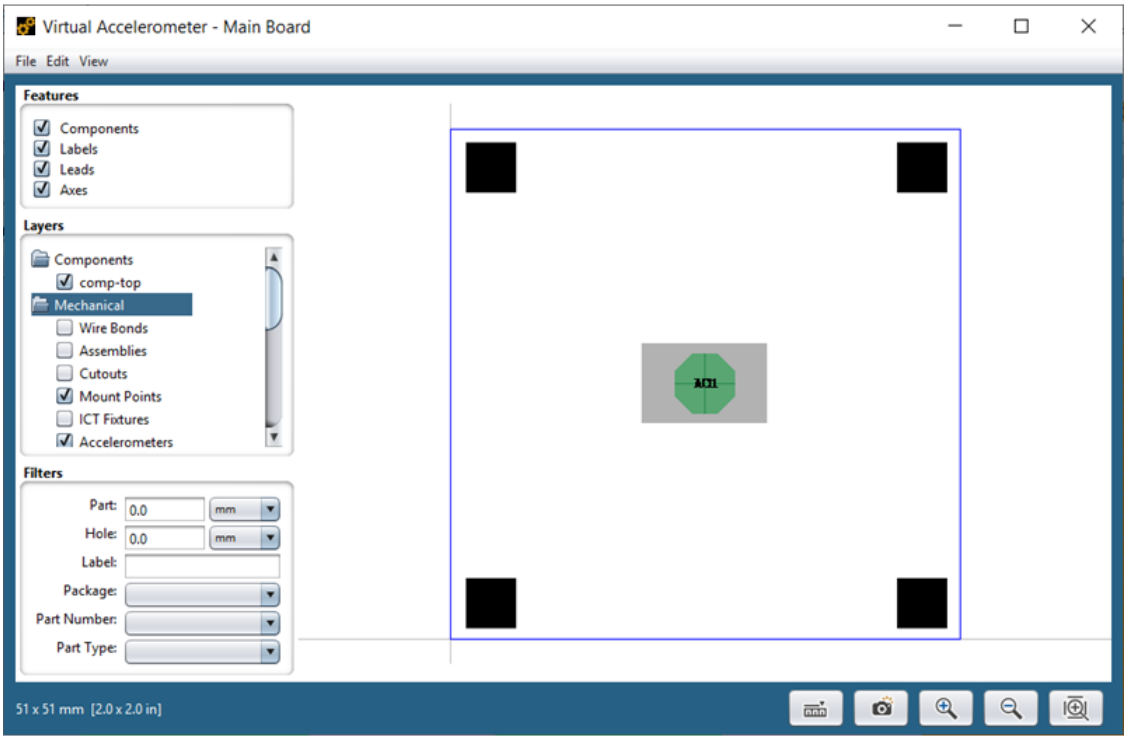




Press the **Save** button in the **Add Accelerometer** dialog to add the virtual accelerometer to the circuit card at the given location.

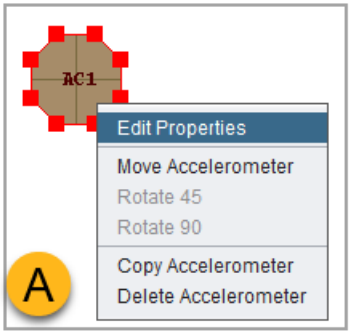
A green octagon will be displayed in the **Layer Viewer** (below) with a crosshair over the center location. In the example below, we've placed the accelerometer in the exact center of the board so that we can see the maximum deflection of the board during a mechanical shock test.

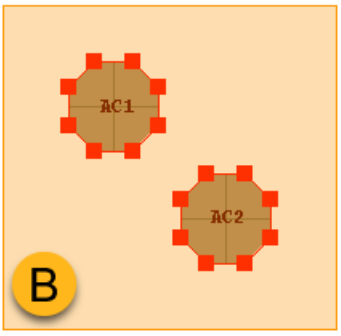
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.7.1.2. Modifying An Accelerometer

Accelerometer properties can be modified by right-clicking inside of the green marker and selecting the **Edit Properties** option from the context menu (A). You may also use the **Copy Accelerometer** menu option to copy an existing accelerometer to create a new accelerometer at another location. In either case, the edit dialog will be displayed allowing you to edit any of the properties assigned to the accelerometer, just like in the previous example.





For convenience, accelerometers can be moved either by selecting the **Move Accelerometer** option in the context menu or by SHIFT-left-clicking the green marker to select it and the dragging it with the left mouse button pressed.

Multiple accelerometers can be selected by holding down the SHIFT key and dragging the mouse with the left button pressed to draw a box around all the desired markers (B). Then, right-click in any of the selected markers to display the context menu with edit options for multiple selections. This provides a way to modify multiple accelerometers quickly.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.7.1.3. Removing an Accelerometer

Virtual accelerometers are removed simply by selecting them and choosing the **Delete Accelerometer** option from the context menu. Any results previously generated for the accelerometer will be deleted when the corresponding analysis results are cleared, or the analysis task is re-executed.

9.7.1.4. Accelerometer Changes

The **Save** button at the bottom of the **Layer Viewer** is used to save all accelerometer changes and close the **Accelerometer Editor**. Alternatively, the **Apply** button can be used to save the current changes, but keep the **Accelerometer Editor** active.

The **Cancel** button can be used to discard all changes made during the most recent edit session and close the **Accelerometer Editor**. Alternatively, the **Reset** button can be used to discard all changes, but keep the **Accelerometer Editor** active.

Note:

The **Save**, **Apply** and **Reset** buttons are only enabled after at least one change has been made in the **Accelerometer Editor**.

9.7.2. Accelerometer Results

Virtual accelerometer results are automatically generated for all Mechanical Shock and Harmonic Vibe tests executed after an accelerometer has been created. That allows users to compare the board response at a given point over the time period or frequencies covered by the test or to compare the board response from one test to another.

Accelerometer results are displayed using the following chart types:

- Displacement vs. Time/Freq
- Acceleration vs. Time/Freq

Displacement charts show the maximum displacement of all FEA nodes covered by the accelerometer over the time period or frequency range covered by the test. Where appropriate, acceleration charts show the maximum acceleration of all FEA nodes covered by the accelerometer over the time period or frequency range covered by the test.

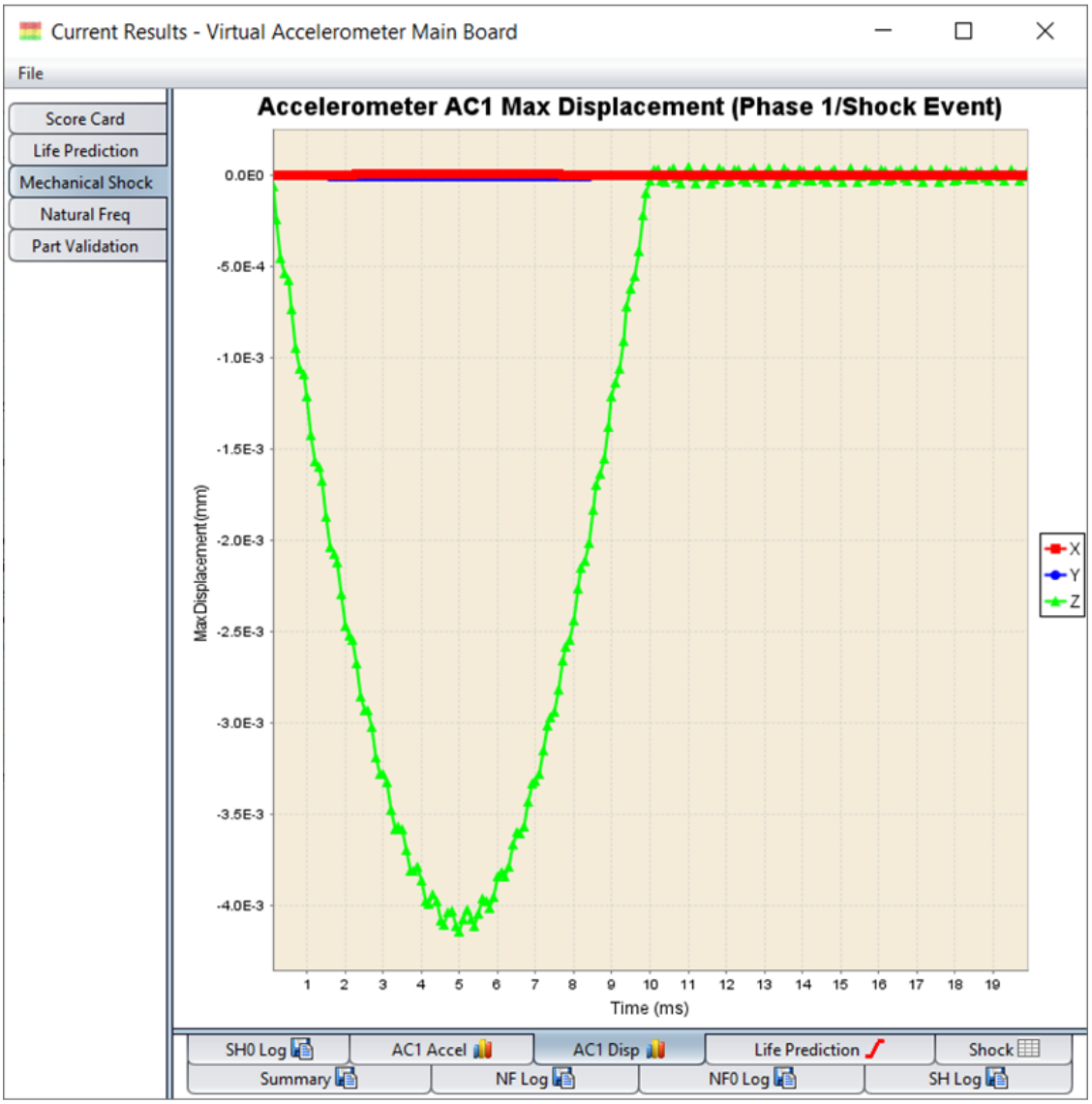
Note:

Acceleration charts are not currently provided for Random Vibration results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.7.2.1. Displacement Results

Virtual accelerometer displacement charts, such as the one shown here, plot the maximum displacement of the circuit card at the accelerometer location over the time period or frequency range covered by the test.

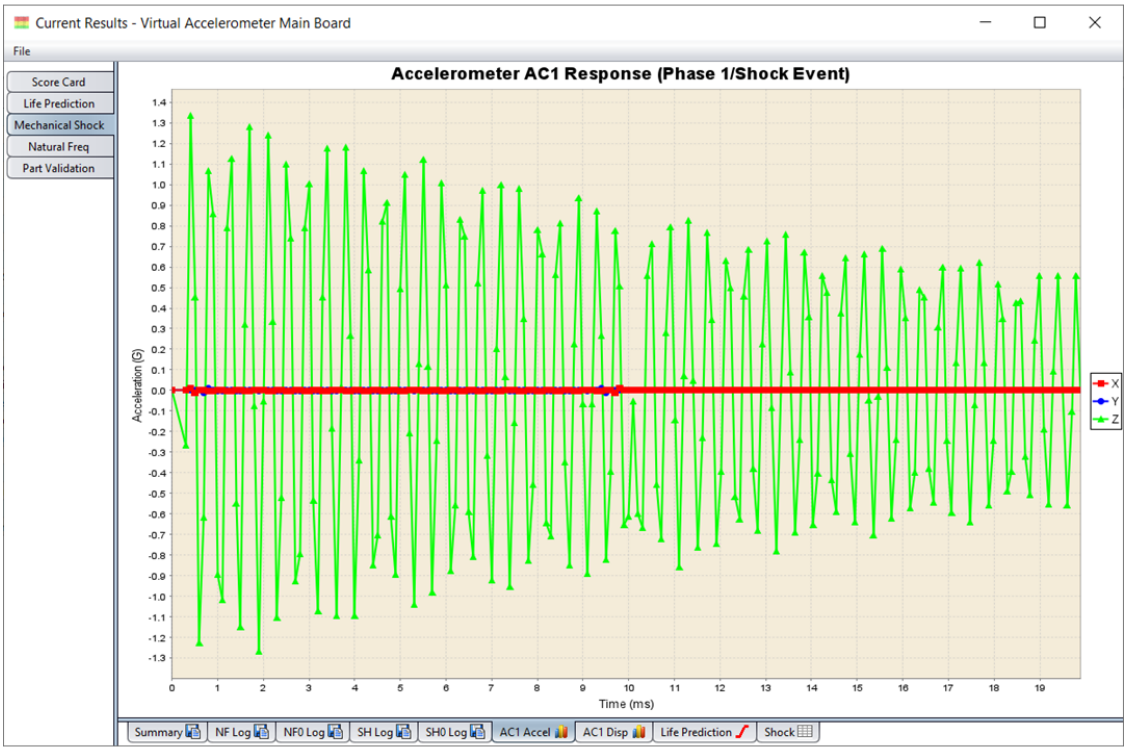


In this example, a Mechanical Shock analysis was performed, so the X-axis shows the time period over which the dynamic load occurred. The Y-Axis shows the maximum displacement, with positive and negative values indicating the direction of the displacement. In this case, the mechanical shock load was exerted in the default downward Z direction. Three curves show the displacement in the X, Y and Z directions. The green curve clearly confirms that the circuit card deflects the most in the Z direction, with minimal deflection in the X and Y directions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.7.2.2. Acceleration Results

Virtual accelerometer acceleration charts, such as the one shown here, plot the maximum acceleration of the circuit card at the accelerometer location over the time period or frequency range covered by the test.



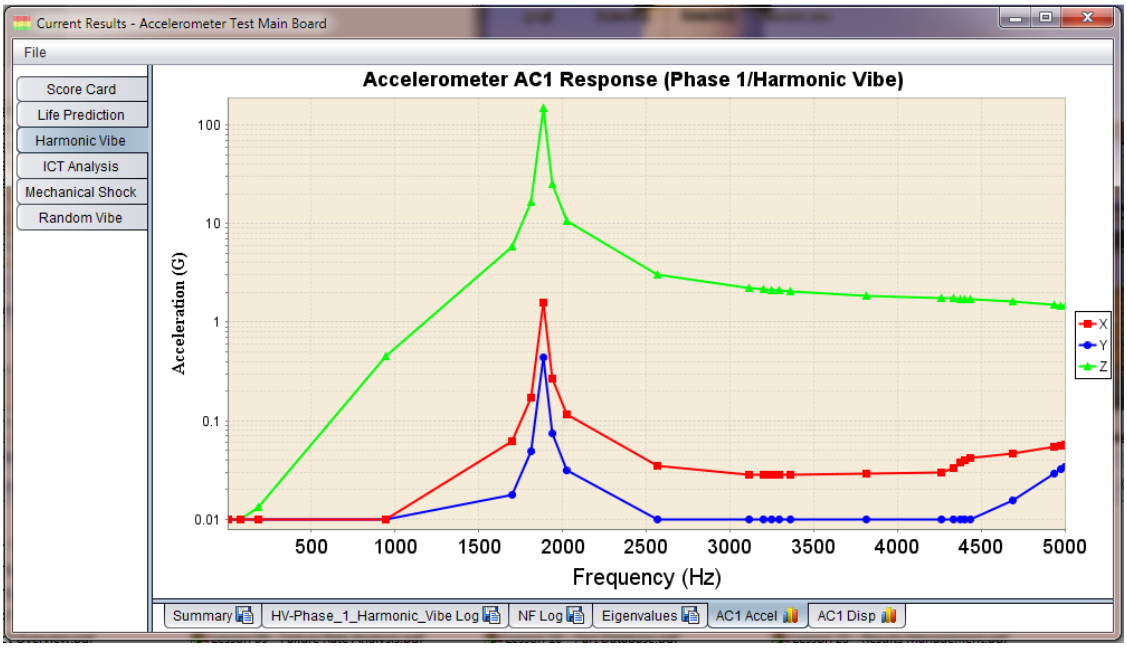
Here again, we see that the primary acceleration is in the Z direction, with a decaying oscillation over time as the circuit card rings from the initial shock load. Accelerations in the X and Y directions were minimal because the shock load was exerted in the downward Z direction.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.7.2.3. Number of Data Points

The number of data points displayed along the curve depends on the number of time steps used by the FEA engine to perform the simulation. For Mechanical Shock Events, as in the above examples, the number of steps is typically determined automatically by the FEA Engine. For Harmonic Vibration Events, however, the number of data points depends on the natural frequencies contained in the frequency range of the test and the **Vibe Data Points** and **Vibration Bias** properties defined in the **Settings > FEA Settings** dialog.

As evident below in a separate example, the data points generated during a harmonic vibration analysis may be biased around the natural frequencies of the circuit card.



In this case, the first three natural frequencies of the circuit card are 1.8kHz, 3.2kHz and 4.4kHz, all of which lie within the 0 Hz to 5kHz frequency sweep specified for the test. With Vibration Bias set to 5 and Vibe Data Points set to 7 the FEA engine automatically generated 5 data points closely straddling each natural frequency and 2 additional data points between each pair of natural frequencies. By increasing the number of bias points, we can see more detail around each natural frequency, but it also increases the overall test time because of the extra simulation steps required.

9.8. FEA- Ansys Workbench™ Integration

Ansys Workbench is a widely used FEA application that combines the ease of a point-and-click interface with the power of Ansys analysis engines. From Sherlock, you can export circuit card data directly into a Workbench analysis stream and then import reliability results back to Sherlock as part of a Workbench solution.

The workflow is summarized below:

1. Export an Ansys Workbench journal file, a STEP file, and a materials Python script from Sherlock for a given CCA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Open the Workbench application and run the exported journal file. Workbench imports the geometry and materials and sets up the appropriate component systems.

   For instructions on using the **Sherlock (Pre)** and **Sherlock (Post)** component systems inside of the Workbench application, see: Sherlock Pre and Post in the *Workbench User's Guide*.

3. If desired, add geometries to the model using Ansys Discovery™.

4. In the Ansys Mechanical application:

   - Assign materials to any new geometries as needed.
   - Define supports and loads.
   - Perform the analysis.

5. Import analysis results into Sherlock

These sections which follow explain each step in further detail:

9.8.1. Exporting a CCA from Sherlock to Workbench
9.8.2. Ansys Workbench Geometry and Materials Import
9.8.3. Performing the Analysis
9.8.4. Importing Analysis Results into Sherlock

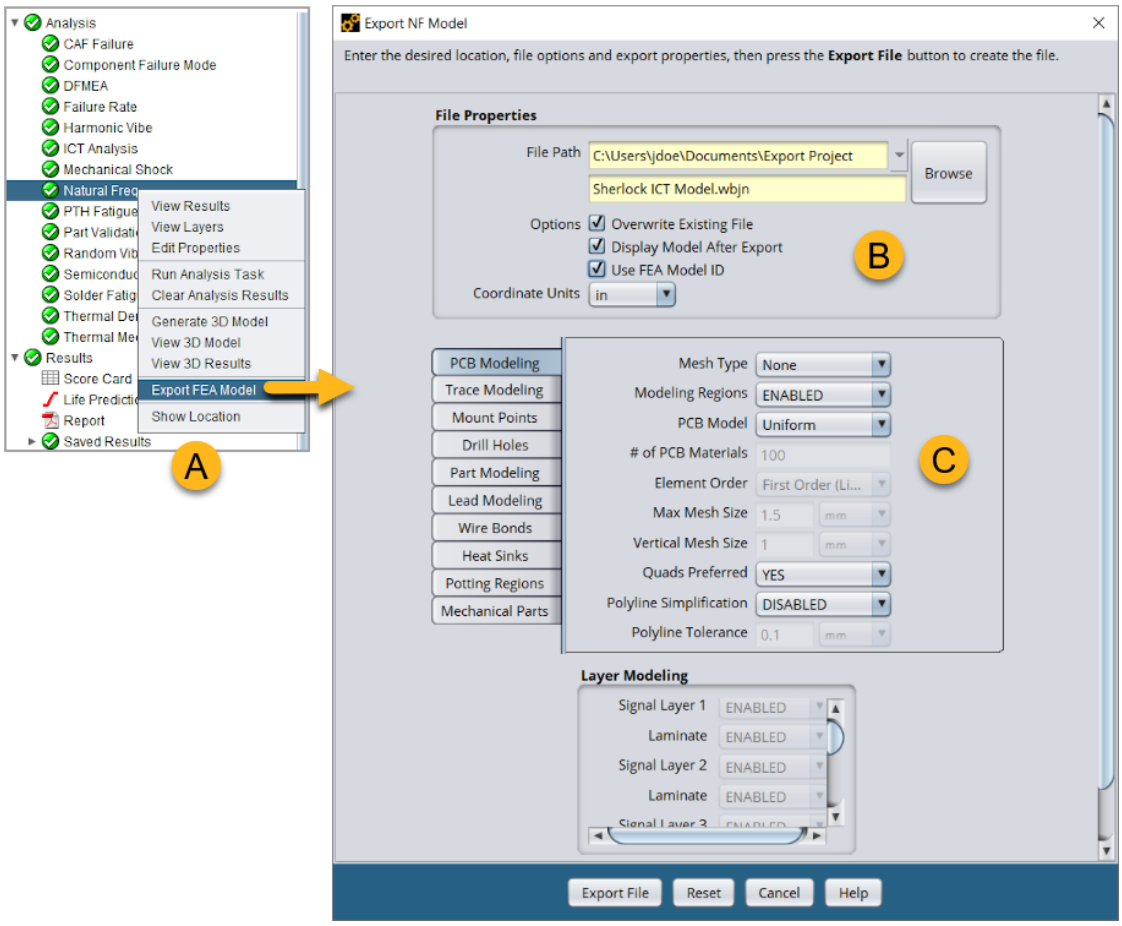
Tip:

Starting with release 2024 R2, Sherlock can take a BGA component from the Parts List and model it in greater detail for export to Ansys Mechanical. See Advanced BGA Modeling (p. 648).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.8.1. Exporting a CCA from Sherlock to Workbench

To use a Sherlock circuit card assembly in Workbench, the 3D model, material definitions, and material assignments must be exported from Sherlock. The model sharing process begins in Sherlock by right-clicking on any FEA-based analysis entry in the **Project Tree** and selecting the **Export FEA Model** option from the context menu (A, in the image below).



In the example shown, the Natural Frequency module has been selected. All FEA-based analysis modules will include the currently defined mount points (except for the ICT module which will include all defined ICT fixtures).

The **Export FEA Model** dialog allows you to specify the location and name of the model file to be generated along with options to include specific types of elements in the exported model.

File Properties (B):

- **Browse**: Click the **Browse** button to modify the file path, file name, and file type. To generate the files required by Workbench, choose **Ansys Workbench Journal (*.wbjn)** as the file type.
- **Overwrite Existing File**: If enabled, Sherlock will automatically overwrite an existing file that has the same file name. Otherwise, Sherlock will ask for confirmation before overwriting the file.
- **Display Model After Export**: If you select the option **Display Model After Export** (E), Sherlock will automatically launch Ansys Workbench and execute the exported Workbench journal file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Use FEA Model ID**: This option enhances Sherlock-Workbench integration in situations where multiple circuit boards from different projects have the same name and would lead to name collisions in Ansys Mechanical. When you select this option, Sherlock adds a unique prefix to all created Named Selections to uniquely identify each circuit board. A board's id can be found on the **Edit Circuit Card Details** properties form (right-click the circuit card in Sherlock's **Project Tree** and select **Circuit Card Properties**).

  Because the feature adds extra characters to the name and Ansys Mechanical has a 32-character limit for names, you should enable this option only when you are importing CCAs with identical names into Mechanical. Sherlock will continue to process previously generated `.rst` files (without the unique id) normally.

- **Coordinate Units**: (in, mm, m, mil, micron, nm) Use the dropdown menu to specify the coordinates to be used in the exported model. This feature replaces the **Scale to Microns** option for Abaqus exports in earlier versions of Sherlock.

PCB Modeling Settings (C):

- The **Mesh Type** should be set to **None** to create a geometric model. See FEA-02a PCB Modeling (p. 473) for more information about geometric models.

- If you select the option **Display Model After Export**, Sherlock will automatically launch Ansys Workbench and execute the exported Workbench journal file.

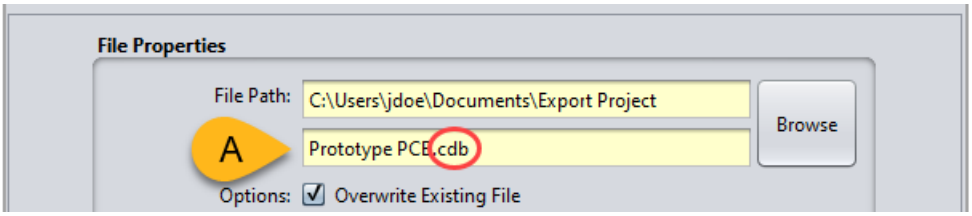
When all desired options have been selected, click the **Export File** button to generate the export.

Sherlock will create three separate files: the Ansys Workbench Journal (`.wbjn`) file, a 3D CAD model file in STEP format (`.stp`), and a Python script file (`.py`). The Python script will be used to assign materials to the 3D geometry.

9.8.1.1. Exporting ECAD to Mechanical CAD Automation

There are special considerations when translating ECAD files to Ansys Mechanical:

- When performing an FEA export, click **Browse** in the **Export** dialog and select the file type **Ansys CDB Script (*.cdb)** as shown below (A).



- If your board has a BGA component, you may select **Shells and Beam** as the **Element Order** for the BGA. See Shells and Beam Element Order (p. 751) below.

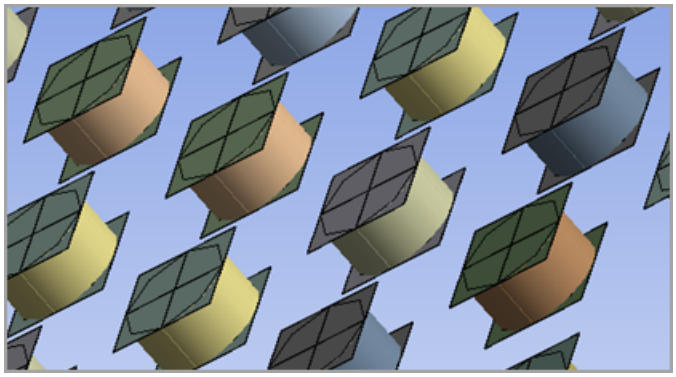
- Sherlock will export the meshed geometry to blocked CDB format and import it into Mechanical via Workbench's **External Model** system. (See External Model in the *Workbench User's Guide*.)

- If an ODB++ ECAD file is associated with the Sherlock CCA board, it will import the file into Mechanical using Workbench's **External Data** system. (See External Data in the *Workbench User's Guide*.)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.8.1.1.1. Shells and Beam Element Order

When you select **Shells and Beam** as the **Element Order**, each solder ball in a BGA is replaced by two patches (shells) and one beam as shown in the image below. This allows for an effective transfer of forces and moments from the solder to the PCB and package.



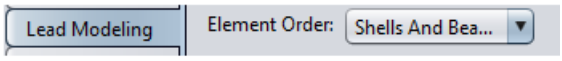
Each patch is a square, its width equal to the ball diameter. Patch thickness equals 1% of the ball height. The radius of each beam equals one half the patch width and has a length equal to the ball height. Even if **Ball Modeling** is set to two elements, only a pair of patches and one beam will be created. (For a complete explanation of solder ball modeling, refer to **BGA Solder Ball Properties** and **Solder Ball Modeling** in FEA- Lead Modeling (p. 563).)

Note:

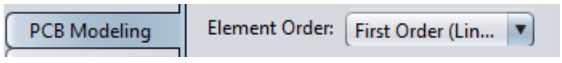
You can only use the **Shells and Beam** element order for *exporting* a model from Sherlock. You cannot reimport that same model back into Sherlock later. For models you intend to reimport into Sherlock, you should select a different element order in Sherlock's **Export Model** dialog.

To use the Shells and Beam option, do the following:

- In the **Lead Modeling** tab
  - Set **Lead Modeling** to **ENABLED**.
  - Set **Element Order** to **First Order (Linear)**.

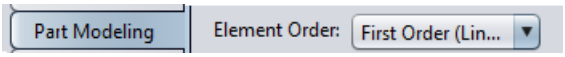


- In the **PCB Modeling** tab
  - Set **PCB Modeling** to **ENABLED**.
  - Set **Element Order** to **First Order (Linear)**.



- In the **Part Modeling** tab
  - Set **Part Modeling** to **ENABLED**.
  - Set **Element Order** to **First Order (Linear)**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Warning:

As explained above, when you select **Shells and Beam** as the **Element Order** in the **Leads** tab, you must also set the **Element Order** to **First Order (Linear)** in *both* the **PCB Modeling** and the **Part Modeling** tabs. Otherwise, the contact/target pairs will not be created correctly in the export.

9.8.2. Ansys Workbench Geometry and Materials Import

If Workbench launches automatically after the export, the `.wbjn` script that was created during the export process will run as well. Ansys Mechanical will then launch with the imported model displayed. At this point, you may proceed to the section below, Add Additional Geometries (p. 753). If Workbench does not automatically launch after the export, you will need to run the exported Workbench journal file manually. See below, Manually Running Workbench Journal File (p. 752).

"Why do I see different materials and material properties in Ansys Mechanical?"

You might notice that the materials and material properties you see in Ansys Mechanical are sometimes different from those you assigned in Sherlock. This is especially obvious when you export a CCA to Mechanical. For example, a component with a package made of EXPOXYENCAPSULATE material in Sherlock might appear as PM001 in Mechanical and seem to have different material properties. This is normal and is a result of how Sherlock internally models components and calculates their effective material properties.

Every component on the CCA is modeled in detail in Sherlock, composed of material properties, lead frame properties, die properties, and so on. Sherlock considers all of this data and generates a simplified component for analysis purposes and assigns to it *effective* materials and material properties. These effective properties reflect how all the constituent parts of the component and their individual material properties perform collectively.

The *effective* materials and material properties are what you see in Mechanical.

9.8.2.1. Manually Running Workbench Journal File

Perform the following steps if Ansys Workbench did not launch after Sherlock completed the model export:

1. Open Ansys Workbench.
2. From the main menu of Workbench, select **File > Scripting > Run Script File**.
3. Using the **Open** file dialog, select the `.wbjn` file exported from Sherlock, and click the **Open** button.

Workbench will open the selected file and execute the commands which will prompt Workbench to do the following:

- Set the project units.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Create an **Engineering Data** component and import required material definitions.
- Create a **Geometry** component and import the exported `.stp` geometry file.
- Create a **Mechanical Model** component and link the previously created **Engineering Data** and **Geometry** components.
- Start Mechanical and assigns materials from **Engineering Data** to the **Mechanical Model** component.

Note:

The `.wbjn` file contains the absolute file path to the exported `.stp` file and `.py` file. If the location of the `.stp` or `.py` file changes after the export, open the `.wbjn` file in a text editor and modify the file path(s) to reflect the changes. If Workbench cannot locate the `.stp` file, it will open a dialog for you to select the file's location.

Note:

If the `.py` file is not run, materials will not be assigned in Mechanical. In this case, you may run the `.py` file as a macro from within Workbench by going to **Automation > Run Macro** and selecting the `.py` file.

9.8.2.2. Add Additional Geometries

Once the Workbench application has run the `.wbjn` script and all the components have been created, you are ready to run an analysis. However, you may wish to modify the imported model by adding additional geometries such as an enclosure around the PCB. To do this, right-click **Geometry** in the **Geometry** component and choose to edit the geometry in the Ansys Discovery application. If you add additional geometries, you must manually assign the proper materials in Ansys Mechanical.

Note:

If you edit the geometry that Sherlock outputs, you may need to rerun the `.py` file. You should also check for duplicate Named Selections in the Mechanical model

---

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.8.3. Performing the Analysis

This section explains the steps to performing analysis tasks in Mechanical and importing the results back into Sherlock. The following topics are covered:

9.8.3.1. When to Use Multiple Models for Multiple Analyses
9.8.3.2. Setting Up the Project Schematic
9.8.3.3. Sherlock-to-Workbench Model Update Notifications
9.8.3.4. Settings and Parameters Required for a Valid Results File
9.8.3.5. Natural Frequency Analysis
9.8.3.6. Settings by Analysis Type

9.8.3.1. When to Use Multiple Models for Multiple Analyses

Before beginning your analysis, decide how many versions of your model you will need. In some circumstances, depending on the combination of analyses you intend to run, Ansys recommends creating multiple versions of the model.

Ansys recommends the following: If you are running the ICT analysis and/or the Thermal Mechanical analysis, you should use a separate model exclusively for each one. As far as all the other analyses are concerned, they can share the same model. So, if you were to run all the Sherlock-compatible analyses on a particular model, you should create three versions of the model:

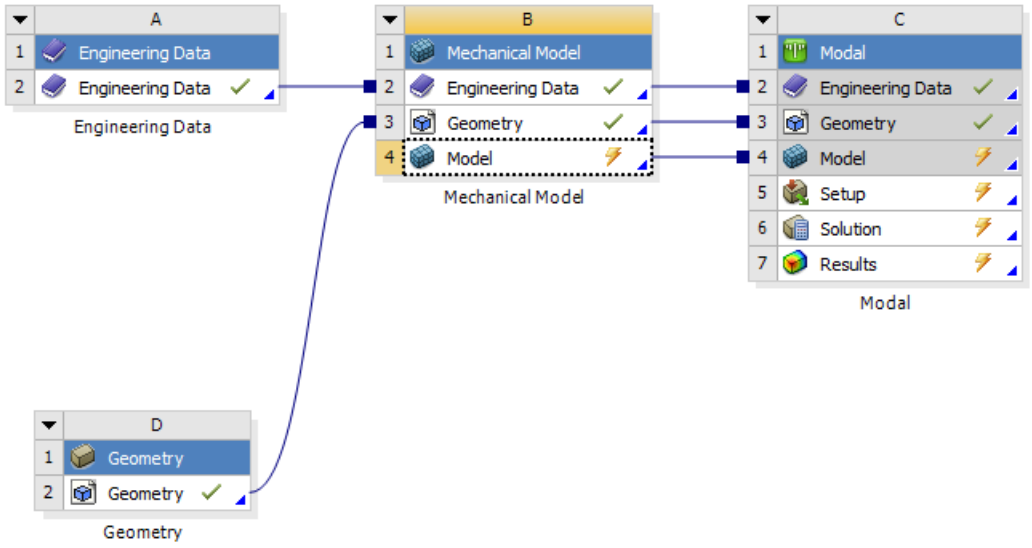
- One model for the ICT analysis.
- A second model for the Thermal Mechanical analysis.
- A third model to be shared by all other analyses.

This recommendation makes sense when you understand the unique requirements of each analysis. For ICT, the generated model includes defined test fixtures not used in the other analyses (which have defined mount points). In the case of the Thermal Mechanical analysis, the generated model is the same one used for other types of analysis, but Thermal Mech requires additional analysis definitions which would conflict with the other analyses.

9.8.3.2. Setting Up the Project Schematic

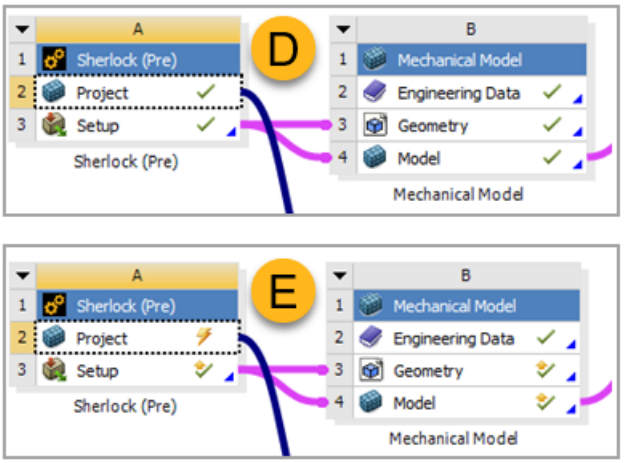
From the Workbench Project Schematic, double-click or drag a **Modal** analysis component onto the project schematic. When adding the **Modal** analysis, link the **Engineering Data**, **Geometry**, and **Model** from the previously created **Mechanical Model** component to the new **Modal** analysis component. Once linked, double-click the **Setup** row of the **Modal** component to open Ansys Mechanical.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.8.3.3. Sherlock-to-Workbench Model Update Notifications

At this point, you may still modify your model within the Sherlock application. When the Workbench application detects the upstream modification, it may change the state of some system cells to indicate you must manually update or refresh them. The images below, for example, show a Workbench system before (D) and after (E) the model was modified in the Sherlock application. See Understanding Cell States in the *Workbench User's Guide*.



The Sherlock application notifies Workbench when the following modifications occur:

- The material property of a component, mount pad, or mechanical part is changed.
- The **PCB Model** type (**PCB Modeling** tab in the **FEA Analysis Properties** dialog) changes.
- The **Filtered Part Results** property (**Part Modeling** tab in the **FEA Analysis Properties** dialog) changes.
- A cutout, mechanical part, or potting region is added or deleted.
- A mount pad or mount hole is added, deleted, or moved.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Changes are made to any stackup/laminate properties (**Stackup Layer** tab).
- Changes are made to a part's properties (anything in the **Edit Parts** dialog).
- Parts/components are added or deleted from the Parts List or Layer Viewer.
- Parts are moved either within the Layer Viewer or by changing location data in the **Edit Parts** dialog.

9.8.3.4. Settings and Parameters Required for a Valid Results File

---

Note:

Read this section carefully. It is important you configure the following settings correctly in Mechanical. Otherwise, Mechanical will likely generate a results file that is not compatible with Sherlock or one that will cause errors when you import it into Sherlock.

Before going further, verify that Ansys Mechanical is configured properly to ensure it generates an export file that Sherlock can use. The following topics are explained in the section:

9.8.3.4.1. Set Uniform Units

9.8.3.4.2. Use Supported Element Types

9.8.3.4.3. Review Mesh Properties

9.8.3.4.4. Update Contacts

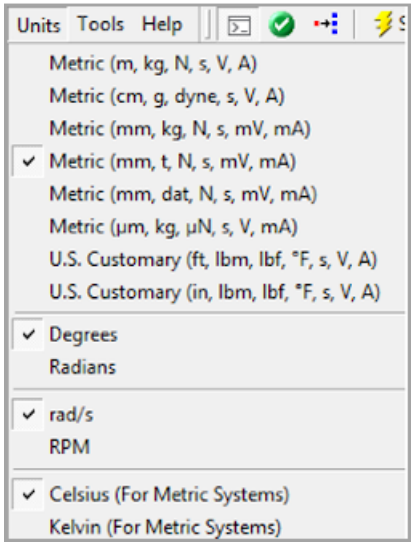
9.8.3.4.5. Preserve Named Selections

9.8.3.4.6. Define Boundary Conditions

9.8.3.4.7. Define Loads

9.8.3.4.1. Set Uniform Units

In Mechanical, select the **Units** menu and verify the selected units match those used in Workbench. This will typically be **Metric (mm, t, N, s, mV, mA)** as shown below.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.8.3.4.2. Use Supported Element Types

Sherlock supports the following element types: wedges, bricks, and tetras. As shown in the table below, each element type requires a specific number of nodes. If other element types exist in the analysis results file, Sherlock will report an error.

Shape	Number of nodes
Tetra	4, 10
Wedge	6, 15
Brick	8, 20

9.8.3.4.3. Review Mesh Properties

Next, review the mesh properties in Ansys Mechanical and make changes as desired. If changes are required, such as modifying the **Element Size**, click the **Mesh** object in the **Tree Outline** and adjust the settings in the **Details** pane. See Mesh Controls Overview in the Ansys Modeling and Meshing Guide for more information.

9.8.3.4.4. Update Contacts

If the model exported from Sherlock is a layered model, it might be necessary to update the contact conditions that were automatically created in Ansys Mechanical. A part on the PCB should only be in contact with the layer to which it is connected. If the default tolerance under **Contacts** is too large, this can result in parts being in contact with other layers as well.

To correct this problem, expand the **Connections** object in Mechanical and click **Contacts**. In the **Details** pane, change the **Tolerance Type** to **Value** and decrease the **Tolerance Value**. As an initial troubleshooting step, consider decreasing the Tolerance Value to 0.01. Then, right-click the **Contacts** object and select **Create Automatic Connections**. Once the new connections are generated, verify that each part is in contact with a single layer.

9.8.3.4.5. Preserve Named Selections

It is important that Mechanical preserves Named Selections that match the component names created by Sherlock. Otherwise, Mechanical will rename objects when creating analysis components, resulting in an export file Sherlock cannot process. To prevent these name changes, turn **Off** the Object Renaming setting in Mechanical.

When adding additional geometries which do not originate from the exported Sherlock model, Ansys recommends you prefix those geometries with the string **0THER_**. This helps Sherlock choose the correct material for any element assigned to multiple Named Selections, which could result in unexpected strain results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.8.3.4.6. Define Boundary Conditions

For all analysis types, you must define boundary conditions or supports for the model. For example, for a Modal analysis, click the **Modal** analysis object in the **Outline** pane in Mechanical. (The specific object you choose, of course, will depend on the type of analysis performed). Next, click **Supports** from the menu bar and select the type of support to add. Define the support as desired. Once all the boundary conditions have been defined, you are ready to perform the Modal analysis. Other types of analyses require external load conditions as well.

If previously defined supports have been added to existing analysis components, and they are the same for the current analysis component, you may copy and reuse them in Mechanical.

9.8.3.4.7. Define Loads

With the exception of the Modal analysis, you must define loads for the various analysis types Sherlock supports. From within Mechanical, click the desired analysis object in the **Outline** pane. Next, click the **Loads** object in from the menu bar and select the type of load you want to add. Define the load within Mechanical. Once all loads and supports have been defined, you are ready to run the selected analysis.

If previously defined loads have been added to existing analysis components and they are the same for the current analysis component, you may copy and reuse them in Mechanical.

Note:

For Mechanical Shock analysis, there are special considerations to setting up load parameters. See the section below, Mechanical Shock Analysis (p. 760).

9.8.3.5. Natural Frequency Analysis

The Natural Frequency analysis in Sherlock is the same as the Modal analysis in Workbench, and the same analysis properties in Sherlock may be defined in Workbench. To modify these settings, click **Analysis Settings** under the selected **Modal** analysis in Mechanical:

- **NF Result Count** in Sherlock is set using the **Max Modes to Find** setting in Mechanical.
- In Mechanical set **Limit Search to Range** to **Yes**.
- **Min Frequency** in Sherlock is set with **Range Minimum** in Mechanical.
- **Max Frequency** in Sherlock is set with **Range Maximum** in Mechanical.

After you choose your settings, click the **Solve** button in Mechanical. Mechanical will mesh all parts as necessary then perform the analysis. Any issues encountered while performing the analysis will be indicated in the Mechanical user interface.

Once the analysis is complete, you can review the results within Mechanical, and/or you can import them into Sherlock to calculate reliability predictions. Importing results back into Sherlock is explained in the section below, Importing Analysis Results into Sherlock (p. 761).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.8.3.6. Settings by Analysis Type

In this section, special considerations and settings for each analysis type are covered.

9.8.3.6.1. Vibration Analysis

9.8.3.6.2. Mechanical Shock Analysis

9.8.3.6.3. ICT Analysis

9.8.3.6.1. Vibration Analysis

The Harmonic Vibration analysis in Sherlock is the same as the Harmonic Response analysis in Mechanical. Likewise, the Random Vibration analysis in Sherlock is the same as the Random Vibration analysis in Mechanical. In Sherlock, loads are defined using the **Harmonic Event** (p. 354) and the **Random Event** (p. 350) definitions for the respective analysis. The load values for the **Harmonic Event** are defined in terms of acceleration (G) with respect to frequency. For the **Random Event**, the loads are defined using power spectral density (PSD) in terms of G^2/Hz with respect to frequency (Hz).

When setting up for vibration analysis, both **Loads** and **Supports** must be defined. See the prior sections on Boundary Conditions (p. 758) and Loads (p. 758) for more information.

To modify the settings for a vibration analysis in Mechanical, click **Analysis Settings** under the **Harmonic Response/ Random Vibration** analysis.

- Modify **Range Minimum** and **Range Maximum** values.
- Modify **Cluster Results** to be a value of **Yes** for Harmonic Analysis.

Note:

In Workbench, when running a *Random Vibration* analysis, be sure to set the following options to **Yes** under **Random Vibration > Analysis Settings > Output Control**: **Keep Modal Results**, **Calculate Velocity**, and **Calculate Acceleration**. If these settings are not set correctly, the upstream **Modal** system's data will not be transferred to the **Sherlock (Post)** system. See Sherlock Pre and Post in the *Workbench User's Guide* for details.

Other settings to consider changing:

- **Options > Modal Frequency Range**: Set to **Manual** if not using a prior Natural Frequency analysis.
  - **Max Modes to Find**: This is the same as **NF Result Count** in Sherlock –

    Modal Range. Minimum: This is the same as **Min Frequency** in Sherlock.
  - **Modal Range Maximum**: This is the same as Max Frequency in Sherlock.
- **Damping Controls > Constant Damping** ratio

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you have configured the settings, click the **Solve** button in the Mechanical menu. Ansys Mechanical will then mesh all parts as necessary and perform the analysis. Any issues encountered while performing the analysis will be indicated in Mechanical's user interface.

Once the analysis is complete, you can review the results within Mechanical and/or import the results into Sherlock to perform reliability calculations. Importing results back into Sherlock is explained in the section below, Importing Analysis Results into Sherlock (p. 761).

9.8.3.6.2. Mechanical Shock Analysis

The Mechanical Shock analysis in Sherlock is the same as Transient Structural analysis in Workbench. You will find the settings for performing the Mechanical Shock analysis in Mechanical by clicking **Analysis Settings** under the selected **Transient Structural** analysis. You will need to define the Step Controls for a Transient Structural analysis.

For the Mechanical Shock analysis, both **Loads** and **Supports** must be defined. See the prior sections on Boundary Conditions (p. 758) and Loads (p. 758) as well as the important note below.

Note:

Mechanical Shock loads require additional attention. In order to capture the response of the PCB, you should add some time to the end of the shock pulse duration during which the load is zero. The additional time should be three times the cycle time of the first natural frequency of the PCB.

After you have configured the settings, click the **Solve** button. Ansys Mechanical will then mesh all parts as necessary and perform the analysis. Any issues encountered while performing analysis will be indicated in Mechanical's user interface.

Once the analysis is complete, you can review the results within Mechanical and/or import the results into Sherlock to perform reliability calculations. Importing results back into Sherlock is explained in the section below, Importing Analysis Results into Sherlock (p. 761).

9.8.3.6.3. ICT Analysis

ICT Analysis in Sherlock is the same as the Static Structural analysis in Workbench. The settings for performing the ICT analysis are found by clicking **Analysis Settings** under the selected **Static Structural Analysis** in Mechanical.

When setting up an ICT analysis, both **Loads** and **Supports** must be defined. See the prior sections on Boundary Conditions (p. 758) and Loads (p. 758) as well as the important note below.

After you have configured the settings, click the **Solve** button. Ansys Mechanical will then mesh all parts as necessary and then perform the analysis. Any issues encountered while performing analysis will be indicated in Mechanical's user interface.

Once the analysis is complete, you can review the results within Mechanical and/or import the results into Sherlock to perform reliability calculations. Importing results back into Sherlock is explained in the section below, Importing Analysis Results into Sherlock (p. 761).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

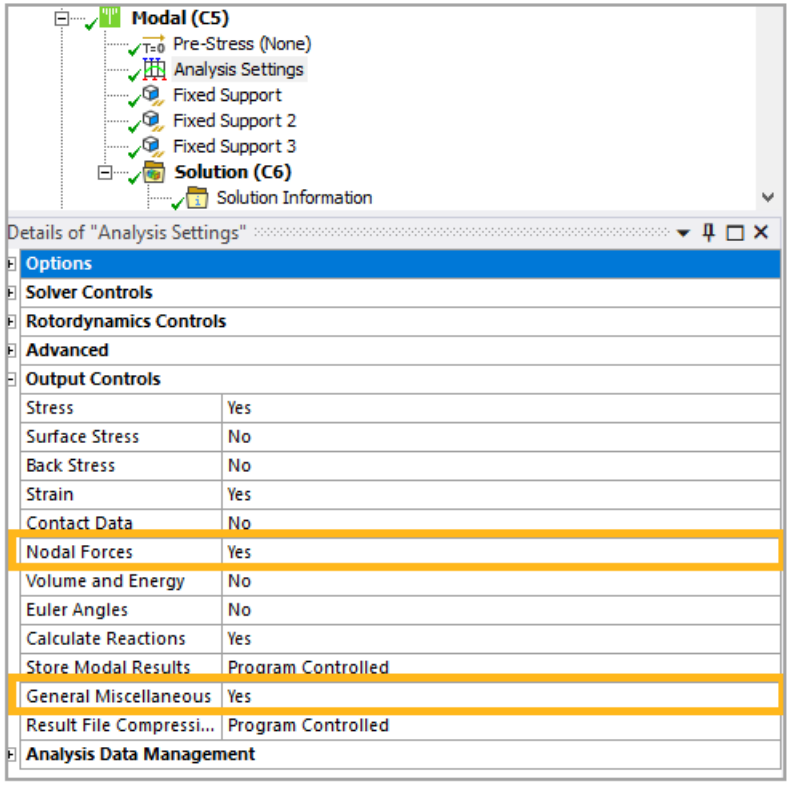
9.8.4. Importing Analysis Results into Sherlock

Once you have completed your analysis in Workbench, you can import the results into Sherlock for review and/or to generate reliability predictions. The import process follows the same basic procedure described earlier (for example, see Imported Data Source (p. 658) under *Natural Frequency Analysis*), but this section includes additional considerations relevant to exports from Ansys Mechanical.

9.8.4.1. Setting Nodal Forces in Workbench

By default. Sherlock does not use nodal lead forces in its calculations. (Refer to the option **Use Nodal Lead Forces** (p. 447) under Engine Properties (p. 447).) If this option is set to ENABLED (which is the more accurate setting), the `.rst` file from Workbench must include nodal forces.

Ansys Mechanical does not store nodal forces by default, so you must change the following settings before solving: In Mechanical under **Analysis Settings > Output Controls**, set **Nodal Forces** to **Yes**. And set **General Miscellaneous** to **Yes**.



Note:

If Sherlock is set to use nodal lead forces but the imported `.rst` file does *not* contain nodal forces, Sherlock will generate a "missing nodes" error when processing the file. See below, Resolving Possible Import Issues (p. 764).

9.8.4.2. Step-by-Step Procedure for Importing Results

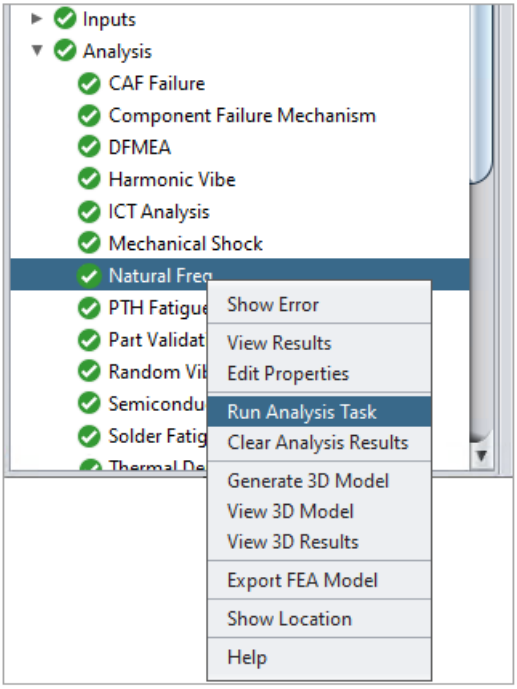
To import the Ansys Mechanical results file into Sherlock, do the following.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. The analysis results and a copy of the model will be stored in an `*.rst` results file. Before importing it into Sherlock, you may wish to make a copy of the file and rename it to associate it with the Sherlock project and analysis type from which it originated.

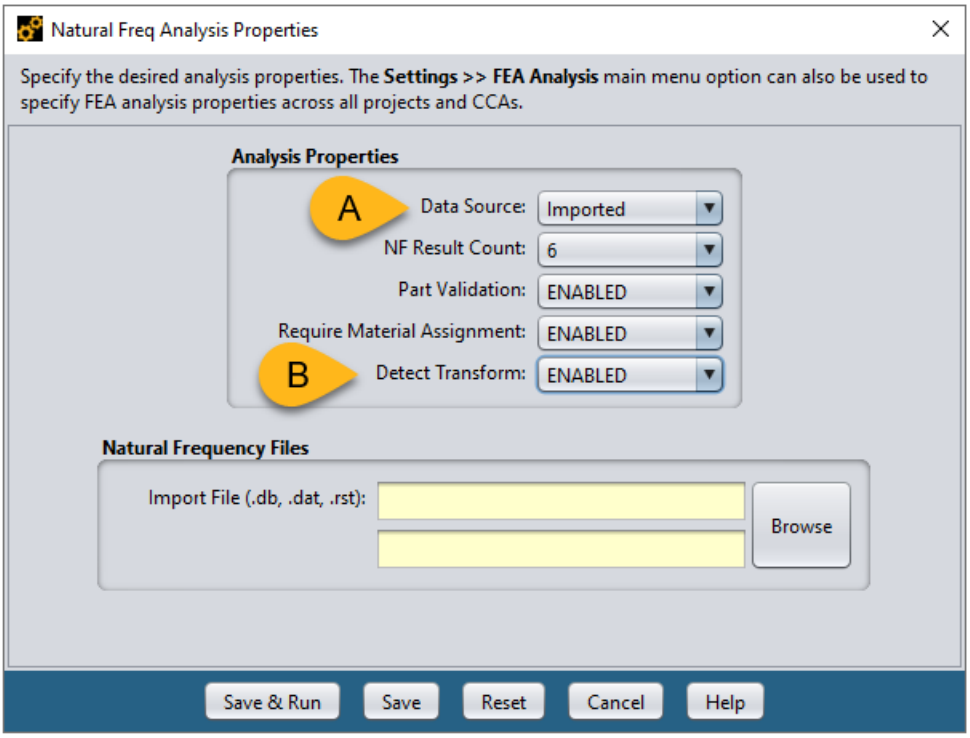
   If the **Save MAPDL db** option was enabled when performing the analysis in Mechanical, then a `*.db` file could also be used for the model. The `*.db` file requires additional processing, so it takes more time for Sherlock to perform the import.

2. In Sherlock, right-click the **Natural Freq** analysis node (or the node of relevant analysis type) on the **Project Tree**, then select **Run Analysis Task**.



3. When the **Analysis Properties** dialog appears (shown below), set the **Data Source** option to **Imported** (A). When **Imported** is selected, Sherlock provides the navigation interface for you to locate the file exported from Mechanical. The file required depends on the specific analysis task. Refer to the list below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Analysis Type	Files Required for Analysis Import
Natural Frequency	Natural frequency analysis results file (`.rst`)
ICT	ICT analysis results file (`.rst`)
Mechanical Shock	Mechanical shock analysis results file (`.rst`)
Thermal Mechanical	Thermal mechanical analysis results file (`.rst`)
Vibration	Natural frequency analysis results file (`.rst`) **Note:** When doing standalone vibration analysis, reuse the vibration `.rst` file for the Natural Frequency model and result. Vibration analysis results file (`.rst`)

In summary, only results (`.rst`) files are needed for the import. Sherlock automatically enables the Native RST Reader. You can manually enable or disable the Native RST Reader by selecting **Settings> FEA Analysis** from Sherlock's main menu. The **Native RST Reader** option is located under **Engine Properties.**

4. **Detect Transform** (B): (ENABLED, MANUAL, or DISABLE) When **Detect Transform** is ENABLED, Sherlock will automatically reorient the imported model to account for any Part Transform object that was applied in Mechanical. Selecting **MANUAL** allows you to enter the **Transform** properties manually.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When running the **Detect Transform** feature, Sherlock uses multiple threads by default. To learn more about this feature and how to adjust it, see **Transform Detection Threading** under FEA Analysis Settings (p. 446).

Note:

If a **Part Transform** has been applied to the model in Ansys Mechanical and **Detect Transform** is DISABLED in Sherlock, the component results will not be correct.

When viewing an imported 3D model with **Transformed** results, the model will be in the un-transformed coordinate space. The 3D results, however, will be in transformed space. **Detecting Transform** does not support Part Transforms defined as a coordinate system Transform, only when it is defined in terms of rotation and translation.

5. Once the files have been selected, click the **Save & Run** button to import the results.
6. When the import is complete, the results may be viewed in the **Sherlock Results Viewer** or **3D Results Viewer**.

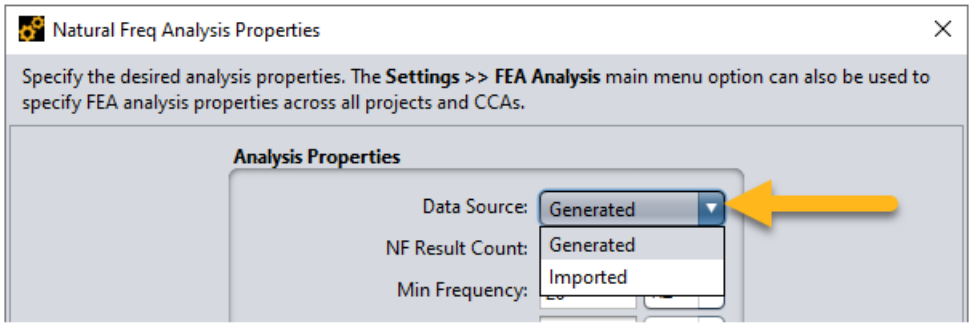
9.8.4.3. Resolving Possible Import Issues

- **Nodes not Found**: After importing results into Sherlock, you might see a message on the **Analysis Issues** tab indicating that result nodes associated with a part could not be found. This likely means that parts in the Sherlock project do not exist as modeled components in the imported results. (See above, Setting Nodal Forces in Workbench (p. 761).)

  It could also mean that the units used in Workbench and Mechanical differ from those used in the Sherlock circuit card assembly. In the latter case, refer back to Set Uniform Units (p. 756).

- **Missing Lead Results**: If the imported results contain leads but Sherlock does not generate lead results, try the following.

  1. In the **Analysis Properties** window in Sherlock, change the **Data Source** to **Generated** as shown below.



  2. Select the **Lead Modeling** tab and set the **Lead Modeling** option to ENABLED.
  3. **Save** the change.
  4. Perform the analysis import again.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Unexpected Strain Values:** After importing results into Sherlock, if some strain values are not what you expect but the stress values are, this usually means elements have been assigned to multiple element sets with different material properties. Review the model to verify that Sherlock's Named Selections have been preserved and that additional geometries have been prefixed with the string **OTHER_**. See Preserve Named Selections (p. 757).

9.9. FEA- Ansys Icepak Integration

You can export your Sherlock projects to Ansys Icepak. Icepak is an industry leading software application which performs thermal and fluid flow analyses of integrated circuits, packages, printed circuit boards, and electronic assemblies- all powered by Ansys Fluent's Computational Fluid Dynamics (CFD) solver. When exporting to Icepak, Sherlock provides the package geometries and material properties with the level of detail needed for highly accurate thermal analysis.

This section will guide through exporting a file from Sherlock and importing it into Icepak. As an overview, the process can be summarized in three steps:

1. From Sherlock, export a CCA (circuit card assembly). Sherlock creates a TCL (`.tcl`) script file and an STL file for each component on the CCA.

2. Using your PC's command-line interface, prompt Icepak to run the exported TCL file. Icepak will then import the geometry and materials data as well as create an Icepak TZR project file (`.tzr`).

3. If required, you can add additional geometries to the CCA from within Icepak.

Once you've completed these steps, you are ready to perform an Icepak analysis in standalone mode or as part of a larger Ansys Workbench analysis flow. In the sections that follow, you will find a detailed explanation of the above steps.

Note:

The workflow discussed in this document is for Classic Icepak, not for AEDT Icepak.

9.9.1. Step 1: Exporting the .tcl File from Sherlock

9.9.2. Step 2: Creating the Icepak Project TZR File

9.9.3. Step 3: Modify or Add Additional Geometries

9.9.1. Step 1: Exporting the .tcl File from Sherlock

A. The model sharing process begins in Sherlock by right-clicking any FEA-based analysis entry in the **Project Tree** and selecting **Export FEA Model** from the context menu. In the example below, the **Natural Frequency** module has been selected (A, below). All FEA-based analysis modules will include the currently defined mount points- except for the ICT module which will include all defined ICT fixtures.

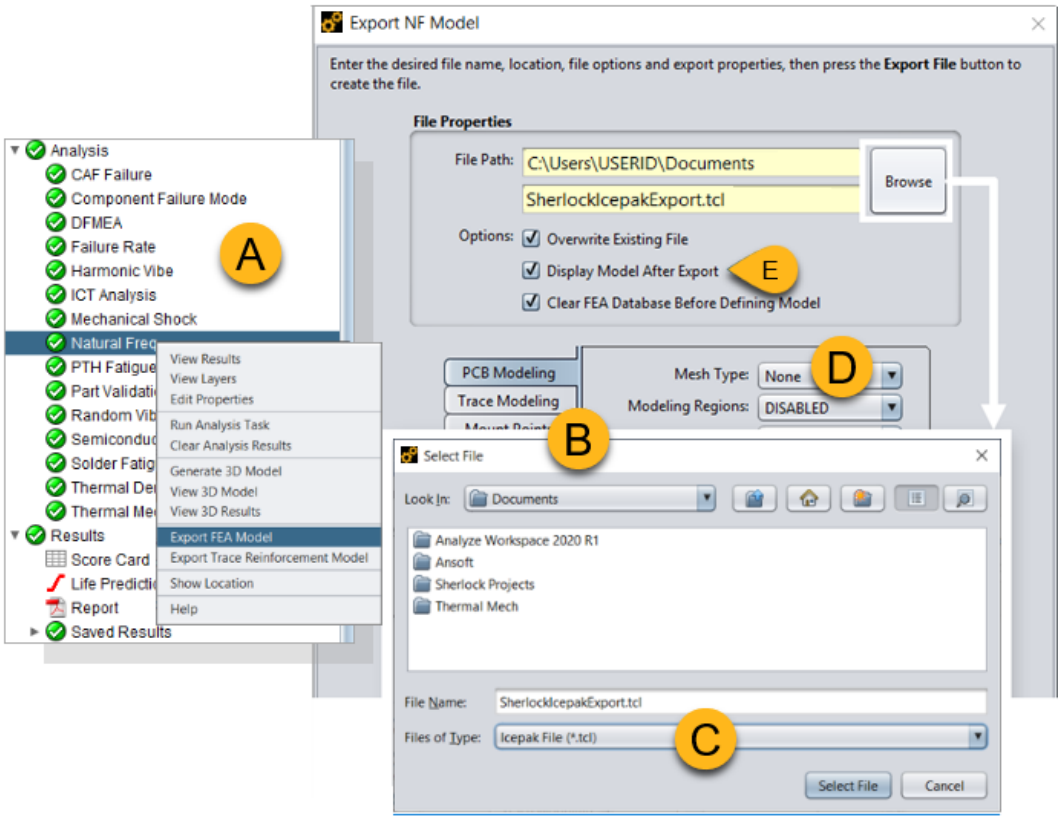
B. In the **Export Model** dialog, click **Browse** to open the **Select File** window (B). Create a **File Name** for the `.tcl` file. (The file name cannot contain spaces.) Choose the location on your PC where

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you want the model saved. Select **Icepak (*.tcl)** as the **File Type** (C). Click **Select File** to return to the **Export Model** dialog.

Note:

As noted above, the file path and name specified for the exported TCL file cannot contain spaces. If spaces are included, the import will not work.



C. In the **PCB Modeling** tab, the **Mesh Type** should be set to **None** (D) to create a geometric model. (See FEA – Overview (p. 443) and PCB Modeling (p. 473) for more information about exporting geometric models.)

Note:

If you want to preview your geometry in Icepak, select **Display Model After Export** (E, above). Once the export is complete, this option will launch a basic version of Icepak with your model displayed. This is a helpful way to catch errors early. In order to run analyses and have access to the full suite of Icepak tools, however, you must create the TZR project file as decribed below in Step 2. (p. 767)

D. Finally, using the **Export Model** dialog, you may select and configure the elements you want to appear in the exported model. When finished, click **Export File**.

E. Sherlock will now create the TCL script, an STL file for each component in the circuit card assembly, and a text file containing any messages pertaining to the export.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

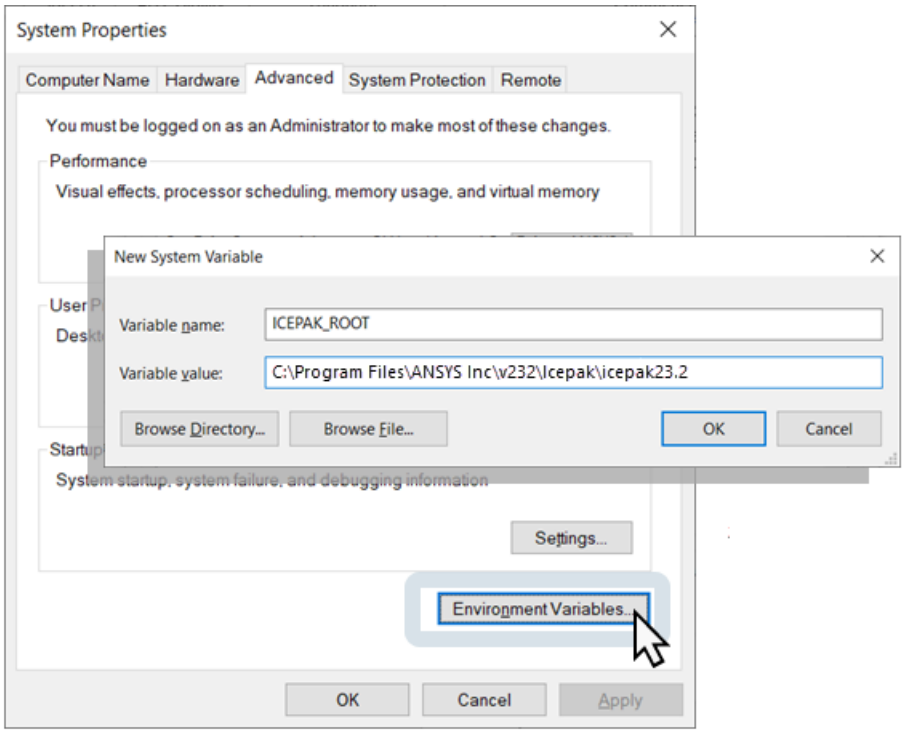
9.9.2. Step 2: Creating the Icepak Project TZR File

You must now convert the TCL file to a TZR file, the standard file-type Icepak uses. This conversion is necessary even if you selected **Display Model after Import** in the **Export Options**. The first step is to create an environment variable which points to the location of your Icepak installation:

A. Open **Windows File Explorer**, right-click **This PC** and select **Properties**. Then click **Advanced System Settings** which opens the **System Properties** window shown below. Click the **Environment Variables** button.

B. In **System Variables**, click **New**. In the example below, we've chosen **`ICEPAK_ROOT`** as the variable, but you may choose whatever name you like. The **Variable value** is the location of Icepak on your computer. To set it, click the **Browse Directory** button and navigate to the location of your Icepak application. Select **OK** to create the environment variable.

C. Restart your computer before continuing.

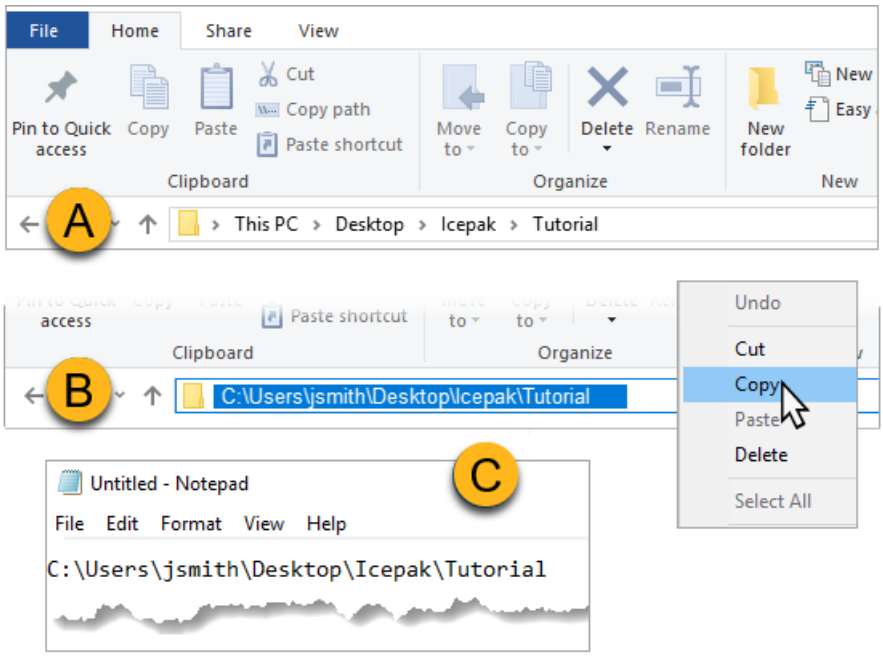


D. You will perform the next few steps using your PC's command-line interface. First, copy to your PC's clipboard the file string that points to the location of your TCL file by doing the following:

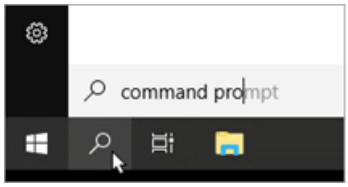
1. Using **Windows File Explorer**, navigate to the folder where your exported TCL file is located. In this example (A, below), the file is located in a folder called **Tutorial**- which is inside an Icepak folder on the **Desktop**. Click the address bar so the file path is displayed (B).

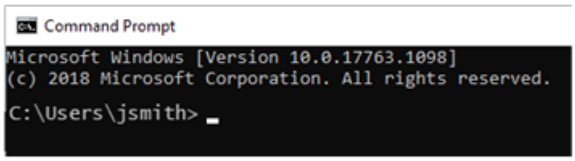
2. Right-click the address bar and choose **Copy**. It's a good idea to paste the file path into a text document for temporary safe keeping (C).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



E. Open the command-line interface on your computer: In Windows, click the **Search** icon in the task bar, type **`command prompt`**, and press **Enter**. This will open the command-line window.





F. In the command-line interface, change the working directory to the folder where your TCL file is stored:

1. At the command prompt, type **`cd`** (change directory) followed by a space. Then paste (**CTRL + V**) the file path you copied earlier. The result should look something like this:

```
cd C:\Users\jsmith\Desktop\Icepak\Tutorial
```

2. Press **Enter**. Your working directory should now be the location of your TCL file. (As a doublecheck, you can type **`dir`** and press **Enter**. This will display a list of files in the current working directory. You should see your TCL file in the list.)

G. Next, create the command which executes the creation of the Icepak Project. (A sample command is shown below.) It consists of the following parts:

1. The filepath *in quotation marks* to the `icepak.exe` file (followed by a space). The filepath on your pc might differ from the example below. Use **Windows File Explorer** to verify the location of `icepak.exe`. If needed, copy and paste from the address bar as shown earlier in Step D. If your Ansys software is installed in the default location, the file path will looking something like this:

```
"C:\Program Files\ANSYS Inc\v251\Icepak\icepak25.1\bin.win64_amd\icepak.exe"
```

2. The command **`-run_script`** (preceded and followed by a space)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The name of the `.tcl` file exported from Sherlock (followed by a space)
4. The name you want to give the new Icepak project file

H. Once you've composed the command, press **Enter**. Icepak will now read the TCL file generated in Step One (p. 765), import the geometry and materials data, create the Icepak project file (`.tzr`), and launch.

Sample Command Line:

```
"C:\Program Files\ANSYS Inc\v251\Icepak\icepak25.1\bin.win64_amd\icepak.exe" -run_script yourexportedfilename
```

9.9.3. Step 3: Modify or Add Additional Geometries

Once Icepak has created the TZR file and has generated all the CCA components, you are ready to begin your thermal analysis.

If you want to modify the properties of your model before proceeding, simply right-click the imported Block item in the **Icepak Tree** and modify the values as required. If desired, you can add additional geometries through the Icepak interface.

9.10. Strain Mapping in Sherlock

The Strain Mapping feature in Sherlock allows you to generate reliability predictions using data from strain images and CSV files— without having to run a finite element (FE) analysis within Sherlock or having to import a results file (`.rst` file) from a prior FE analysis. In fact, you can use strain-distribution images created from external solvers. This feature provides several advantages:

1. You can generate predictions using strain maps derived from any FE solver.
2. The board can be anywhere in space, and Sherlock can generate predictions as long as you provide the image or CSV file of the strain distribution.
3. You can provide custom strain distribution values for the analysis.
4. Because the analysis does not require large files needed for an FE analysis, Sherlock can generate predictions much more quickly.

Tip:

If you wish to experiment with the Strain Map feature, the Sherlock tutorial folder contains sample strain map image files and a CSV file for the circuit board in the first tutorial *Project Overview.* (p. 8) For a standard Ansys installation, the files are in the following location:

`C:\Program Files\Ansys Inc\v251\sherlock\Tutorial Project.zip`

Topics Covered in this section:

9.10.1. Strain Map Feature: Capabilities and Limitations
9.10.2. Adding Strain Map Files (Image or CSV) to Sherlock
9.10.3. Random Vibe Analysis Using Strain Maps

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.10.4. Harmonic Vibe Analysis Using Strain Maps
9.10.5. Mechanical Shock Analysis Using Strain Maps
9.10.6. Running a Strain Map Analysis on an Assembly
9.10.7. Results of the Strain Map Analysis

9.10.1. Strain Map Feature: Capabilities and Limitations

Currently, the Strain Map feature works for the Random Vibration, Harmonic Vibration, and the Mechanical Shock analyses. As you prepare for an analysis, it is important to understand the capabilities and limitations of each analysis and file-type:

- **Image Files**: If Sherlock's reliability prediction is based on an image, strain data is treated as PCB surface strain, whether parts are leaded or not.
- **CSV/Excel Files:** CSV and Excel files support the use of lead strain and PCB strain in Sherlock's prediction analysis.
- Strain mapping in general does not support Part Strain.
- For Harmonic Vibration, strain mapping does not support multiple modes.
- For Mechanical Shock, when using a CSV strain map, Sherlock bases predictions on surface strain only, not lead strain.

9.10.2. Adding Strain Map Files (Image or CSV) to Sherlock

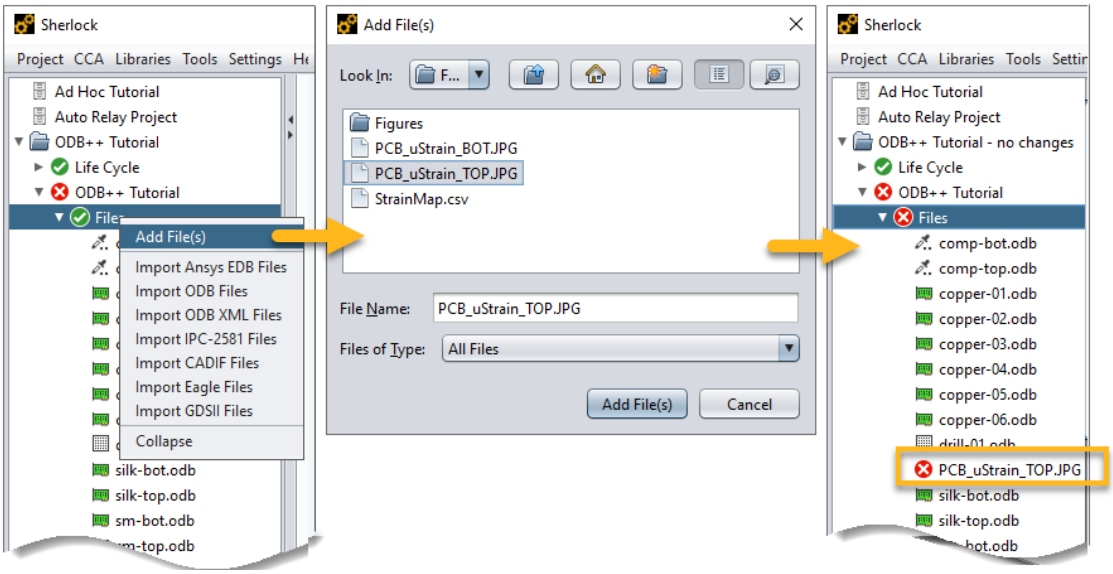
Running a Strain Map analysis begins by adding to the circuit board the file(s) containing the strain map data, either an image file or a CSV/Excel file. The procedure is the same for all analysis types.

9.10.2.1. Adding Strain Map Image Files
9.10.2.2. Adding Strain Map CSV/Excel Files

9.10.2.1. Adding Strain Map Image Files

The procedure for adding a strain map image to Sherlock is similar to adding a thermal map image (p. 722): Right-click **Files** under the CCA as shown below and select **Add File(s)** in the context menu. In the **Add File(s)** dialog, select the strain map image you wish to import. Click the **Add Files(s)** button to add the file to the CCA.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tip:

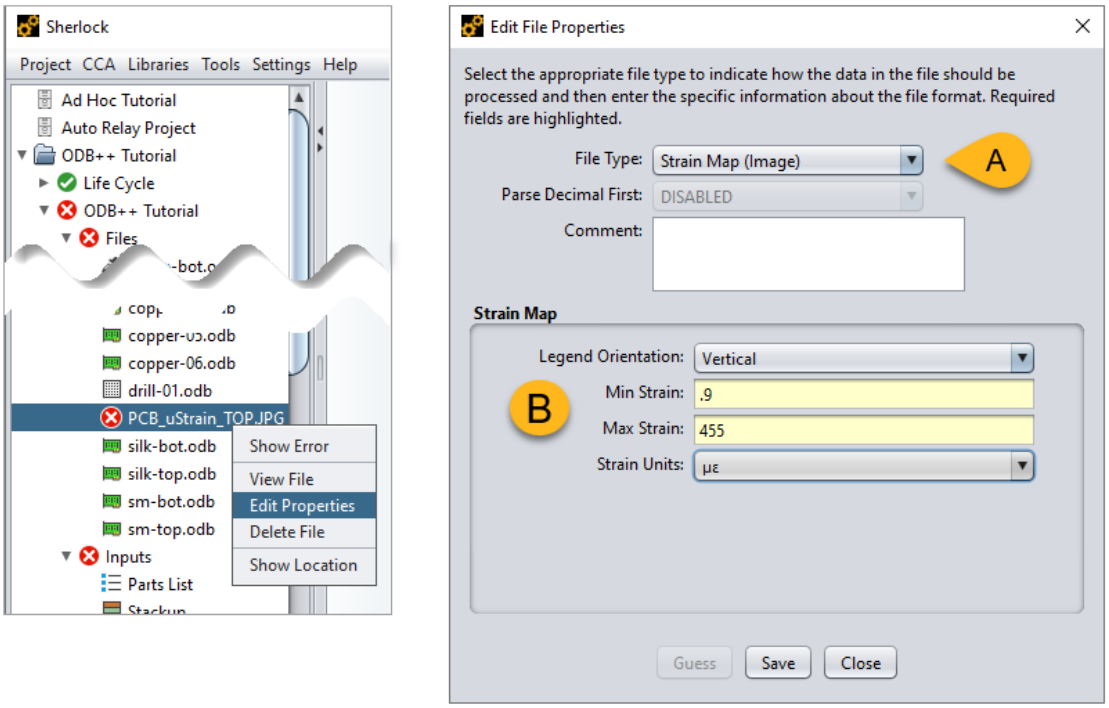
Sherlock allows you to run a strain map analysis on an *assembly* of circuit cards as explained below (see Running a Strain Map Analysis on an Assembly (p. 783)). So, if your project contains more than one circuit card, make sure you add each strain map file under the correct circuit card.

Once you have added the strain map image, you must then edit the file properties so they are configured for the analysis:

1. In the **Project Tree**, right-click the image you wish to edit and select **Edit File Properties** in the context menu as shown below.

2. In the **Edit File Properties** dialog, under **File Type**, select **Strain Map (Image)** from the drop-down menu (A). This populates the **Strain Map** options panel (B).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter the information requested in the **Strain Map** form (B):

   - For **Legend Orientation** select either **Vertical** or **Horizontal**, depending on how the legend is oriented in the image.

   - For **Min Strain** and **Max Strain**, enter the minimum and maximum strain values represented in the legend. In the sample image used below (see E), the scale's maximum value is 455. The minimum value is .9.

   - For **Strain Units**, specify whether the strain values in the image are expressed in **strain** or **με** (microstrain). Selecting the ε option is equivalent to selecting **strain**.

4. Once you have defined these inputs, click the **Save** button.

Next, you must align the image to the board and specify the location of the legend:

1. As shown below, right-click the image file and select **Edit Layer** from the context menu to open the **Mapping** tab.

2. To align the image to the board, click the **Align Board** button. Adjust the boundaries of the BLUE box (C, below)–by grabbing and dragging the nodes– to mark the edges of the circuit board. Clicking **Max Bounds** expands the blue box to the edges of the image. You may click **Reset** to restore the red and blue boxes to their last saved or default configuration. Click **Done** when you are finished sizing the blue border.

Tip:

You may also manually enter the coordinates of each node that defines the box. Right-click any node of the blue box and select **Edit Location**. This opens the **Edit Vertex** dialog where you can enter the X and Y coordinates of the node. While the

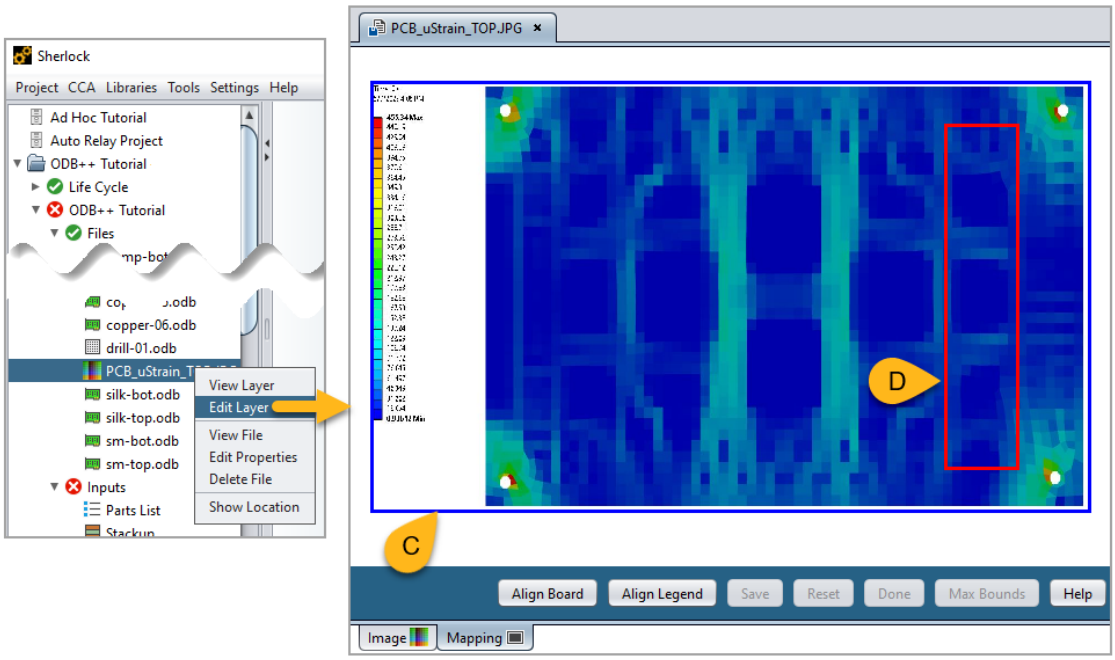
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dialog allows you to enter the coordinates in the units of your choice, Sherlock converts the coordinates to inches when you save and close the dialog. If you reopen the **Edit Vertex** dialog later, Sherlock displays the coordinates in inches.

This tip also applies to aligning the legend (the red box) as explained in step three.

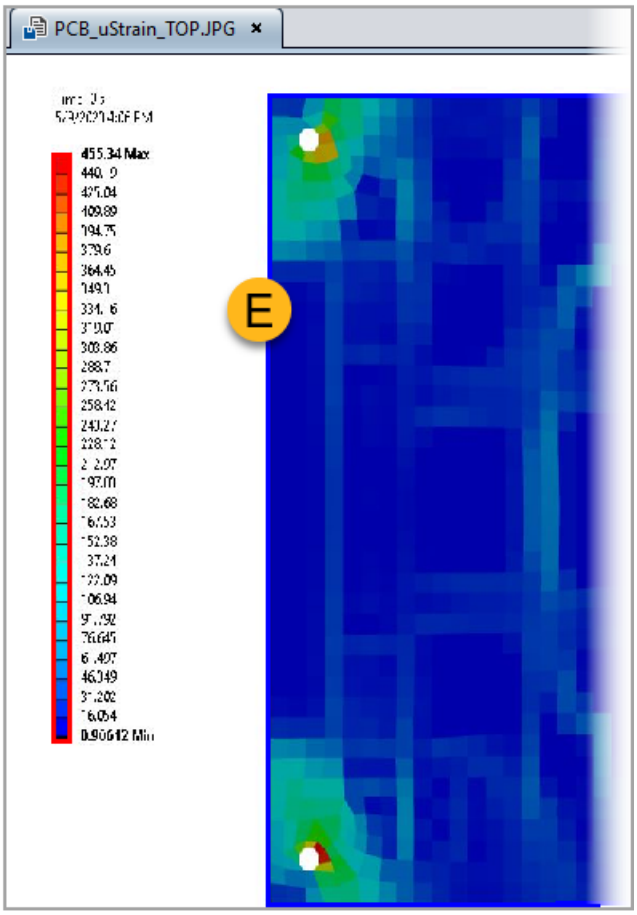
3. To help Sherlock find the color bar of the legend, click **Align Legend** and adjust the RED box (D) so it encloses the color scale. The red box should enclose only the color bar– not the numbers beside the bar.

4. To preserve your changes, you must click **Save** before exiting the **Mapping** tab.



A correctly aligned strain map image is shown here (E). The red box marks the location of the color scale, and the blue box marks the edges of the circuit board.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

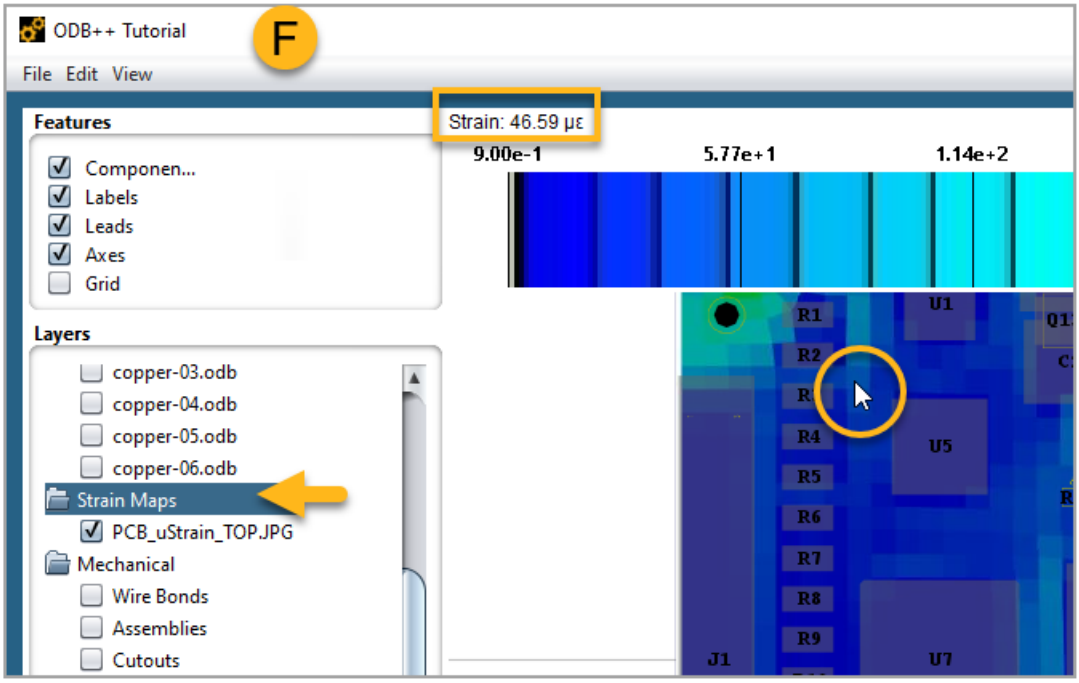


With alignment completed and saved, the image is now ready for Sherlock to run its reliability analysis.

Tip:

Once you have successfully processed the image file, you can view the strain map in the Sherlock Layer Viewer (F, below). Use the mouse to probe any point on the board. The strain value for each location appears in the upper left of the display.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



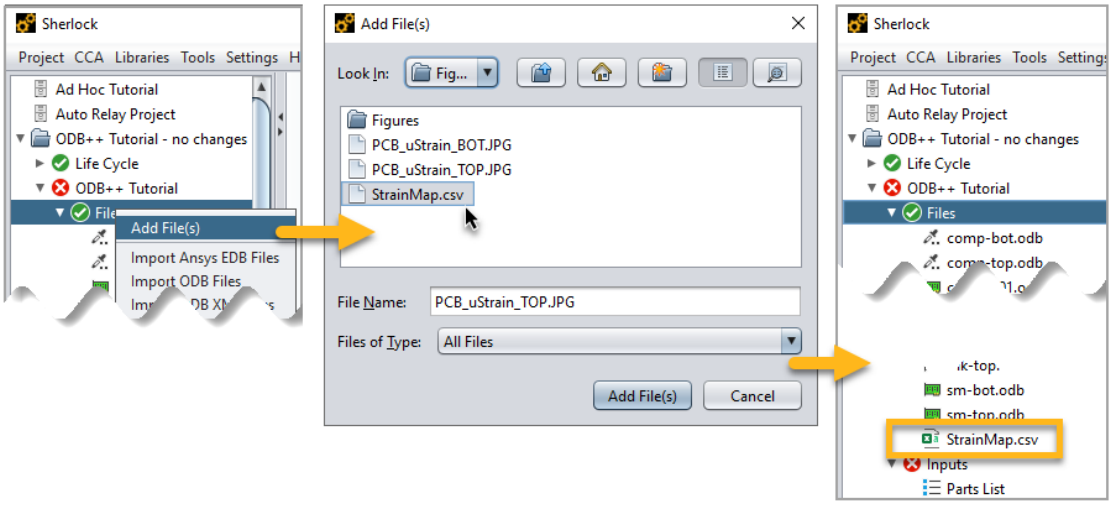
9.10.2.2. Adding Strain Map CSV/Excel Files

You can also add to your project strain information from CSV/Excel files. Strain data can be assigned to Parts, Leads, Mount Points and Wire bonds. The import process is similar to that of adding an image file. The required file format is shown here:

	A	B
1	RefDes	Strain
2	U3	10.39793
3	U4	8.884281
4	U7	3.243746
5	U8	4.252689
6	U9	10.04928
7	J1	9.385937
8	U10	12.15716
9	LEAD_U10_298	2.492278
10	LEAD_U10_299	2.018008
11	LEAD_U10_287	25.56879
12	LEAD_U10_288	14.4245

To add an CSV/Excel file, right-click **Files** under the CCA as shown below and select **Add File(s)** from the context menu. From the **Add File(s)** dialog, select the CSV file you wish to add for your Strain Map analysis. It will then appear in the **Project Tree** under **Files**.

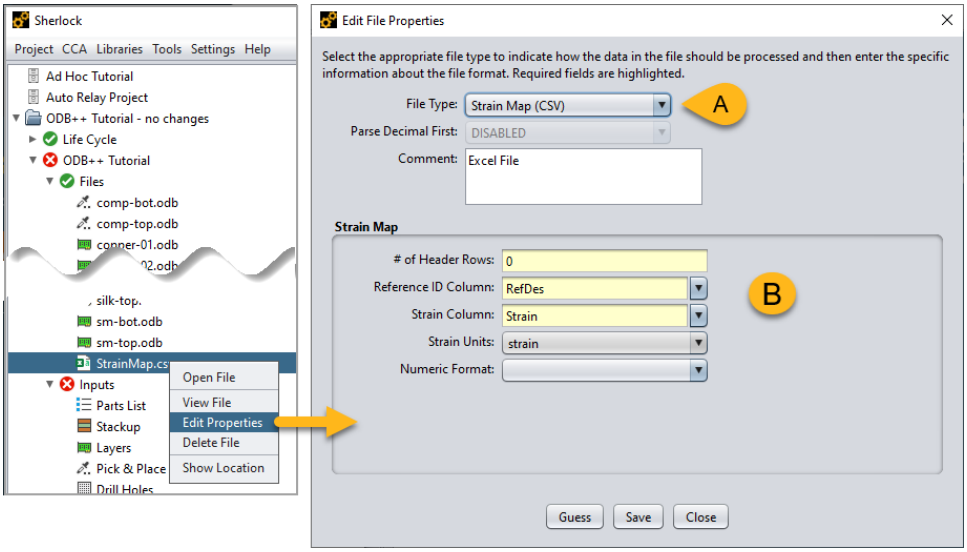
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once it's added to the **Files** list, you must configure the file's properties so Sherlock can process it correctly:

1. Right-click the CSV/Excel file in the **Project Tree** and select **Edit Properties**.

2. In the **Edit File Properties** window, change the **File Type** to **Strain Map (CSV)** or to **Strain Map (Excel)** from the drop-down list (A, below). The Strain Map properties form will appear (B).



3. Set the Strain Map file properties (B):

Tip:

Click the **Guess** button if you want Sherlock to attempt to fill in the fields for you. If needed, you can edit any wrong guesses.

- **# of Header Rows**: The first row of a CSV file is indexed at 0. The example shown below has one header row, so the **# Header Rows** should be set to **0**. If, for example, there were three header rows, it should be set to **2**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	A	B
1	RefDes	Strain
2	U3	10.39793
3	U4	8.884281
4	U7	3.243746
5	U8	4.252689
6	U9	10.04928
7	J1	9.385937
8	U10	12.15716
9	LEAD_U10_298	2.492278
10	LEAD_U10_299	2.018008
11	LEAD_U10_287	25.56879
12	LEAD_U10_288	14.4245

C

- **Reference ID Column**: Specify which column contains component IDs (RefDef in Sherlock Part Properties). In the example above (C), **`RefDes`** should be entered.
- **Strain Column**: Specify which column contains the strain values. In this example, it is **`Strain`**.
- For **Strain Units**, specify whether the strain values are expressed in **strain** or **με** (microstrain). Selecting the ε option is equivalent to selecting **strain**.

4. Click **Save** to save the changes.

Tip:

As with the image file (p. 774), you can view the CSV strain data in the Sherlock Layer Viewer. (Make sure you have selected the CSV file under the **Strain Maps** folder in the Viewer.) You can use the mouse to probe any point on the board. The strain value for each location appears in the upper left of the display.

9.10.3. Random Vibe Analysis Using Strain Maps

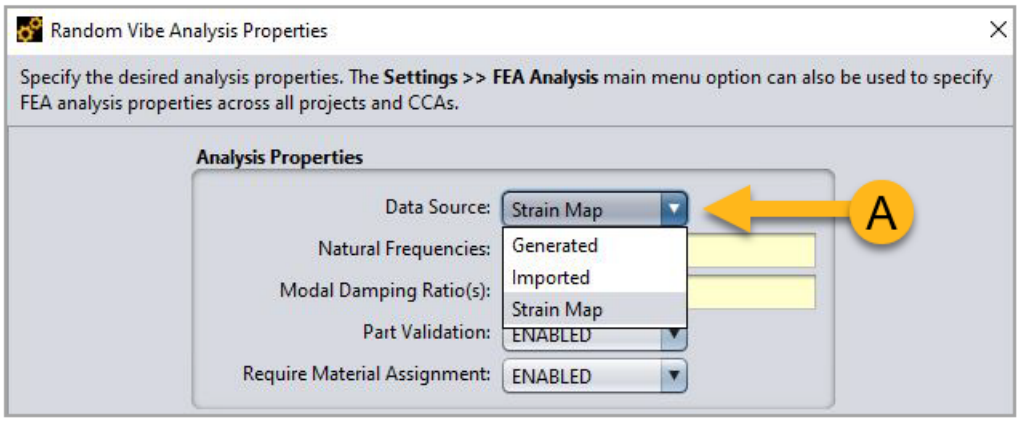
Note:

Before beginning your analysis, review the section above, Strain Map Feature: Capabilities and Limitations (p. 770).

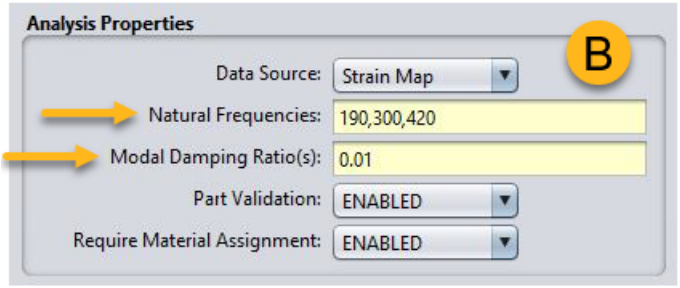
The first step in setting up your Random Vibe analysis is adding the strain map file(s) to the CCA in your Sherlock project. This is explained above in Adding Strain Map Files (Image or CSV) to Sherlock (p. 770). Once the files have been added, do the following to run your analysis:

1. In the Sherlock **Project Tree**, right-click **Random Vibe** under **Analysis**. Select **Edit Properties** from the context menu.
2. In the **Random Vibe Analysis Properties** window, select **Strain Map** as the **Data Source** (A, below).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

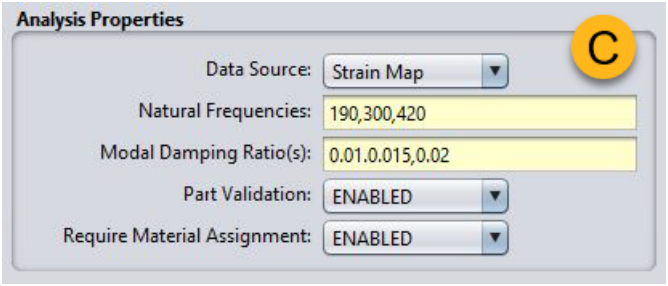


3. Enter the **Natural Frequencies** and the **Damping Ratio(s)** used in the analysis from which the results were extracted (B).



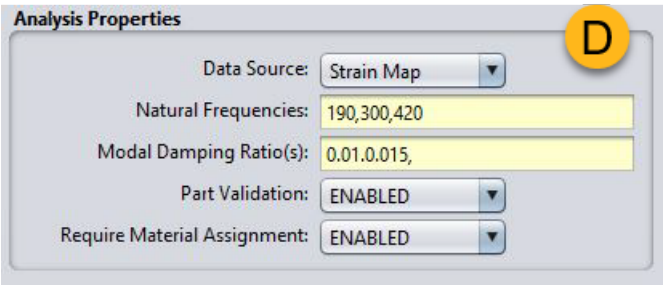
Sherlock uses this information for making cross-over frequency calculations. Both input fields can take multiple values separated by comma. If you provide one damping value and multiple natural frequencies, Sherlock uses the value across all the frequencies. In the example above (B), modal damping of 1% will be used for all frequencies.

If the number of frequencies is the same as the number of damping values, Sherlock associates each frequency with its corresponding damping value. In the example below (C), 1% damping will be used for the first frequency (190Hz), 1.5% will be used for the second frequency (300Hz), and 2% will be used for the third frequency (420Hz).

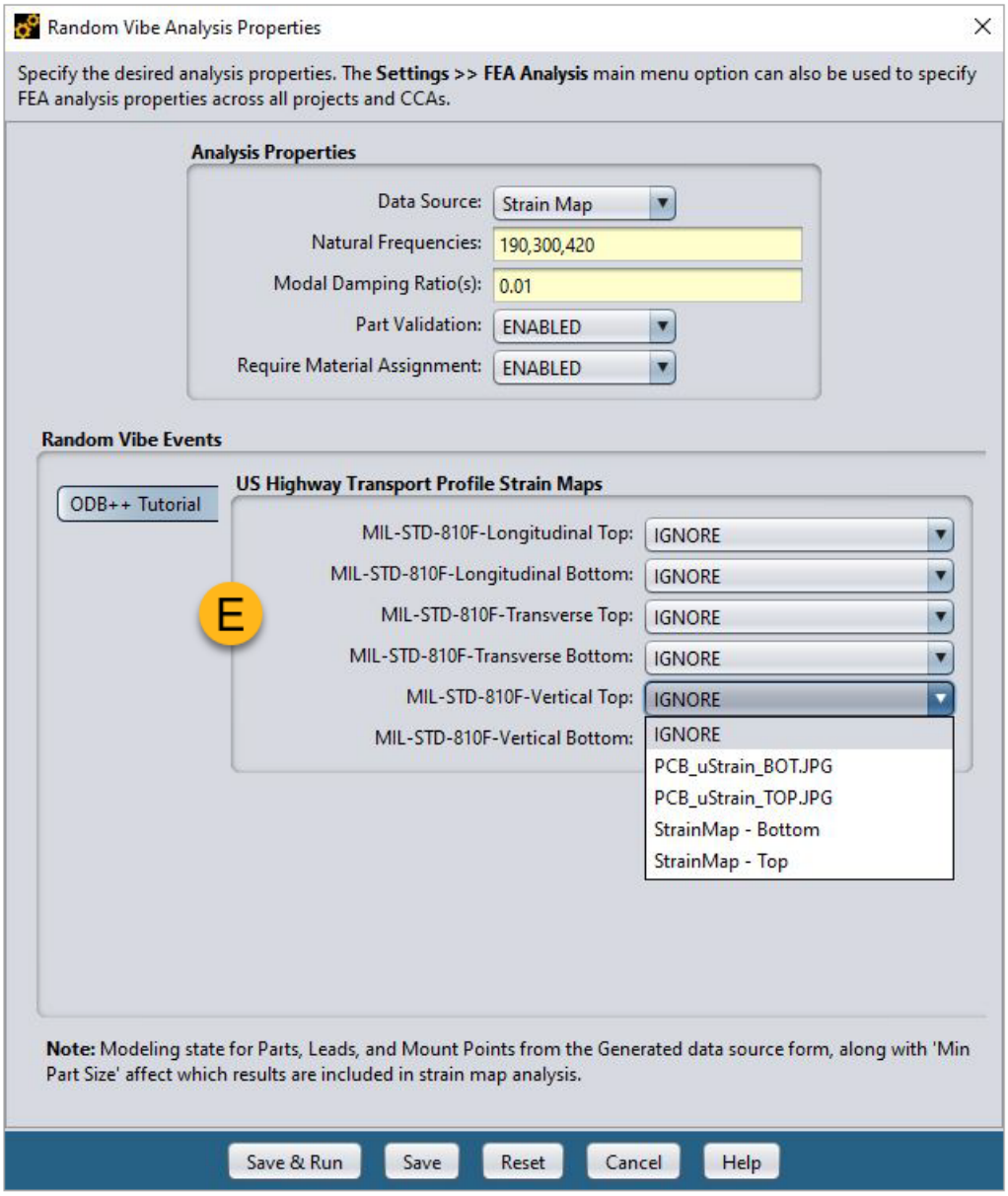


Finally, if the number of damping values is less than the number of frequencies, then the last damping value will be used for the frequencies which do not have corresponding damping values. In the example below (D), the first frequency (190Hz) will use 1% damping, and the other frequencies will use 1.5%.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. When the **Data Source** is set to **Strain Map**, the **Random Vibe Analysis Properties** interface updates so you can associate the strain maps to the Random Vibe Events available in the Sherlock project (see E). Notice, as well, the circuit card assembly tab (**ODB++ Tutorial**) and its corresponding **Random Vibe Events**. For each Event there is a drop-down list so you can choose a strain map available under the CCA.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this specific example, there are six Random Vibe Events defined (E), while in the Sherlock project there are only three. This allows you to associate two separate images for each Event- one image to the top of the board and one to the bottom.

5. Once the correct images have been selected for the top and the bottom surfaces of each Random Vibration Event, click **Save & Run** to start the Random Vibe analysis.

Tip:

If your project contains an assembly of circuit boards, see below Running a Strain Map Analysis on an Assembly (p. 783).

9.10.4. Harmonic Vibe Analysis Using Strain Maps

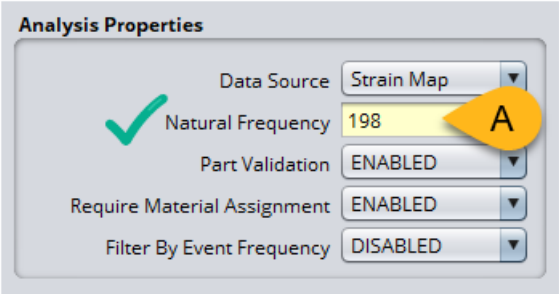
Note:

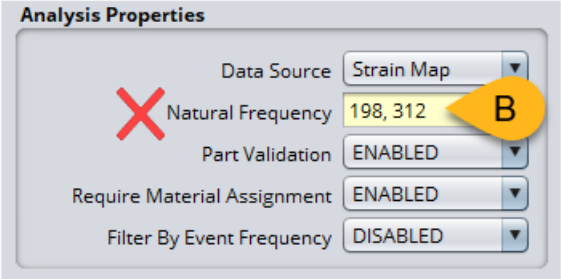
Before beginning your analysis, review the section above, Strain Map Feature: Capabilities and Limitations (p. 770).

Beginning with release 2025 R1, the Sherlock application supports reliability predictions using Strain Maps for Harmonic Vibe. The process is similar to that explained above in Random Vibe Analysis Using Strain Maps (p. 777). Strain maps can be an image and/or in a CSV/Excel file.

Note:

Currently, Strain Mapping for the Harmonic Vibe analysis supports only one mode. You may use strain maps at any frequency, but typically it is best to use the frequency that causes the most damage. Support for multiple modes will be available in a future release. The image below shows an example of a single mode analysis (A) and a multiple mode analysis (B) which is not supported.





Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.10.5. Mechanical Shock Analysis Using Strain Maps

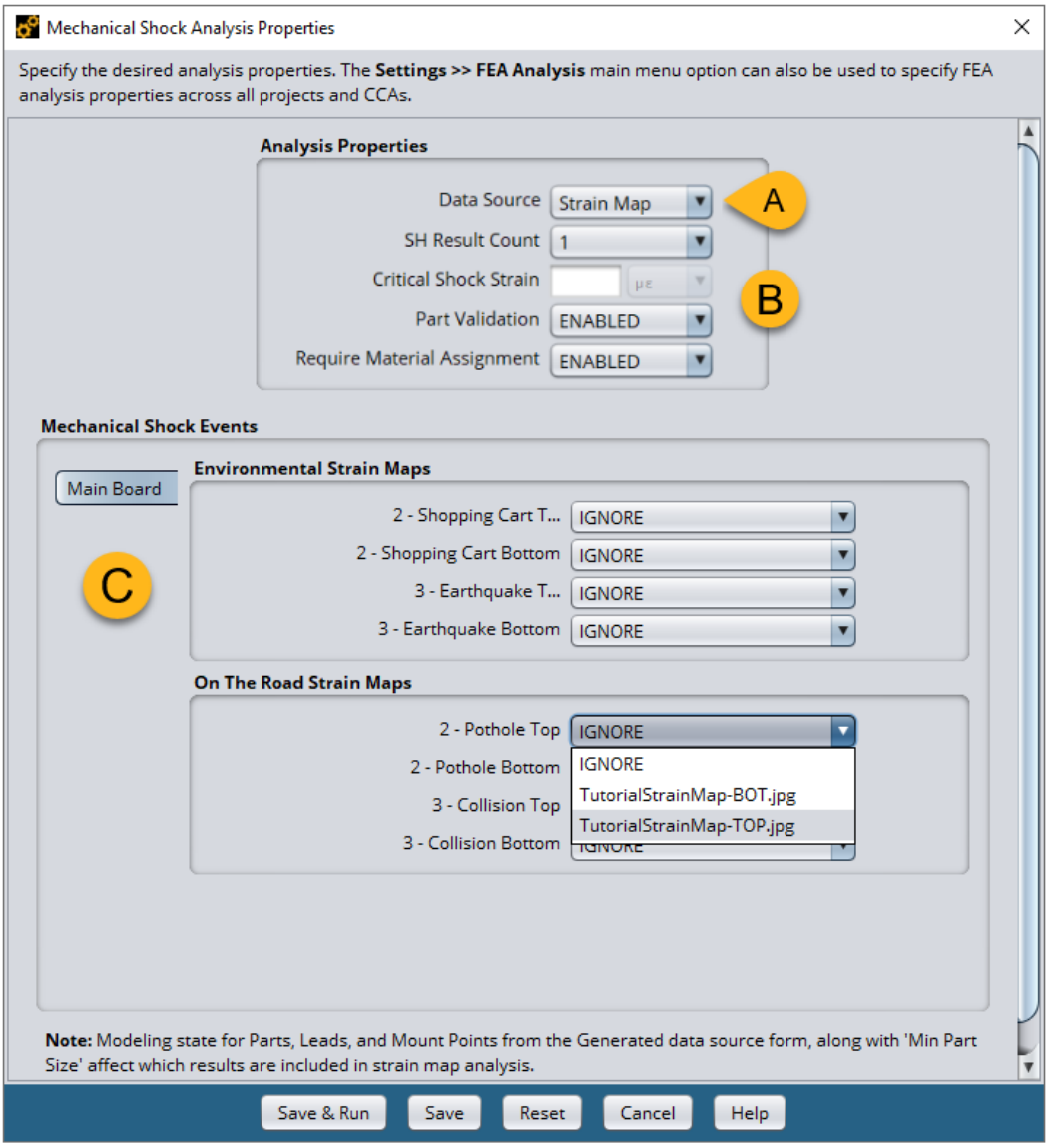
Note:

Before beginning your analysis, review the section above, Strain Map Feature: Capabilities and Limitations (p. 770).

The first step in setting up your Mechanical Shock analysis is adding the strain map file(s) to the CCA in your Sherlock project. This is explained above in Adding Strain Map Files (Image or CSV) to Sherlock (p. 770). Once the files have been added, do the following to run your analysis:

1. In the Sherlock **Project Tree**, right-click **Mechanical Shock** under **Analysis**. Select **Edit Properties** from the context menu.
2. In the **Mechanical Shock Analysis Properties** window, select **Strain Map** as the **Data Source** (A, below).
3. Configure the options in the **Analysis Properties** form (B). You will find an explanation for these options in the *FEA- Mechanical Shock* section under Generated Data Source (p. 676).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. The **Mechanical Shock Events** form (C) displays a list of all Shock Events (p. 347) you have created. (See Shock Event in the *Sherlock User's Guide* (p. 347) for a review on creating Life Cycle Events.) For each Event, the dropdown menu contains a list of all the strain maps you have defined for that board (D). Select the relevant strain map for each Event. Select **IGNORE** for any Event that is not being considered for the analysis.

5. Once the correct images have been selected for the top and the bottom surfaces of the relevant Mechanical Shock Events, click **Save & Run** to start the analysis.

Tip:

If your project contains an assembly of circuit boards, see below Running a Strain Map Analysis on an Assembly (p. 783).

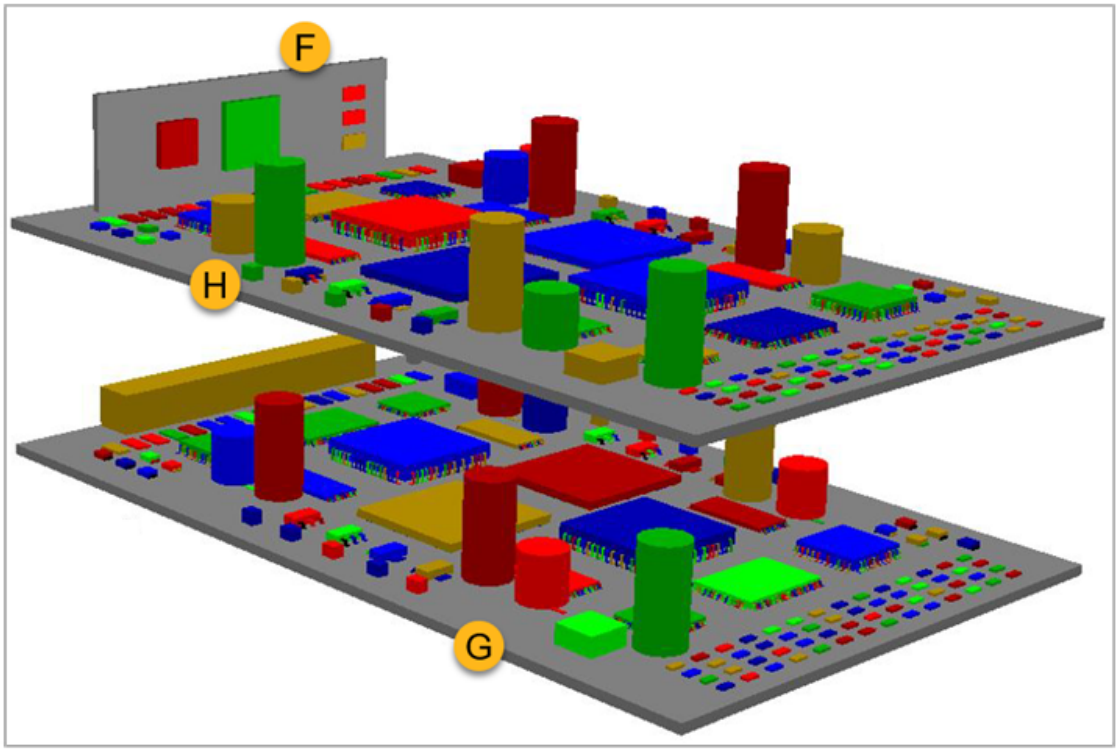
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.10.6. Running a Strain Map Analysis on an Assembly

Sherlock supports running a Strain Map analysis on an assembly of multiple CCAs as explained below. The sample project in the following example is for a Random Vibration analysis, but the steps readily apply to Mechanical Shock as well.

In the sample assembly, there are three circuit cards:

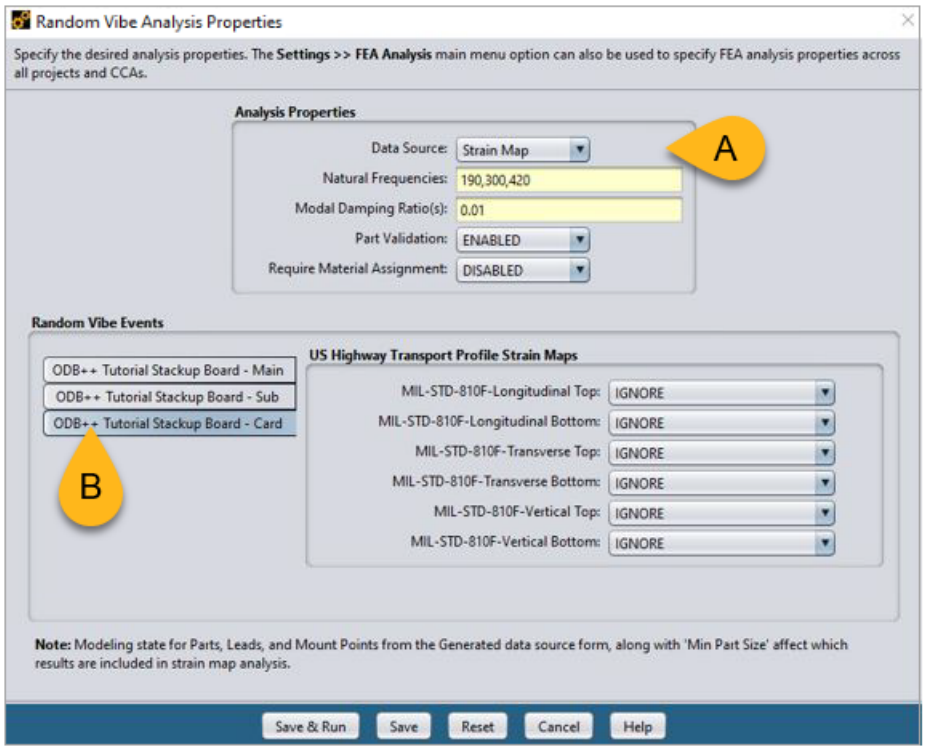
- An ODB++ Tutorial Stackup Board- Card (F)
- An ODB++ Tutorial Stackup Board- Main (G)
- An ODB++ Tutorial Stackup Board- Sub (H)



To run a Strain Map analysis on the assembly, you must add the strain map files to their respective CCAs in the Sherlock **Project Tree**. For instances, strain map images for the CCA-Card (F) should be added under **Files** in the **Project Tree** under the **ODB++ Tutorial Stackup Board – Card**.

Once the strain map files have been added to the correct CCA's, right-click **Random Vibe** in the **Project Tree** and select **Edit Properties**. When the **Random Vibe Analysis Properties** dialog opens, set the **Data Source** to **Strain Map** (A, below). The procedure for selecting the strain maps is similar to that of a standalone board as discussed earlier. The only difference is that in the **Random Vibe Events** panel, you will see a separate tab for each circuit board (B). Select a tab, and then select which strain maps should be applied to that board.

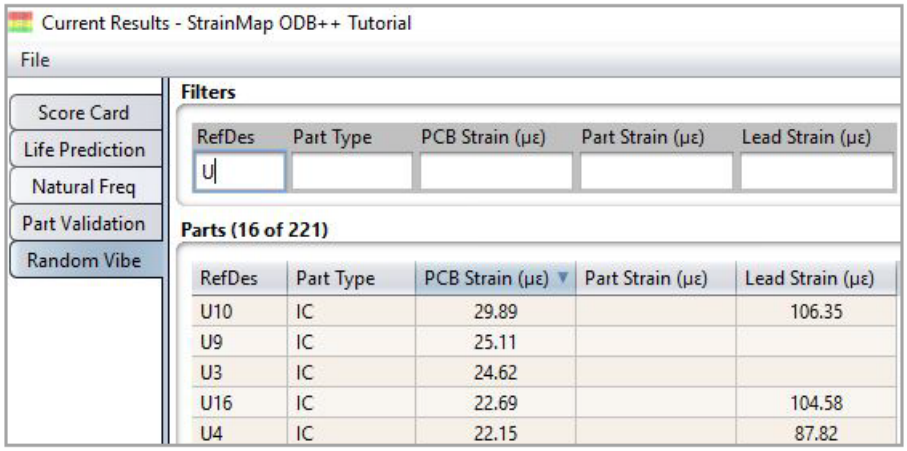
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9.10.7. Results of the Strain Map Analysis

Whether you run a Strain Map analysis on a standalone board or an assembly, you can view the results in the Results Table. The format of the Results Table is similar to tables generated by an FE analysis but with three main differences, as shown in the image below:

- Because Part Strain is not supported in the Strain Map analysis, that column will be empty.
- If strain map images were used, the **Parts** tab will display only **PCB Strain**.
- If a CSV file was used for the strain map, the **Leads** tab will have only a **Lead Strain** column.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 10: Viewing and Managing Analysis Results

Ansys Sherlock produces a wealth of results and information regarding circuit card reliability. During the design life cycle it is usually necessary to catalog and compare results during the process. Sherlock accommodates this need through two results management displays: the Sherlock Results Viewer and the Sherlock 3D Viewer. With the Sherlock Results Viewer you can save and manage past reliability results such as **Analysis Results**, **Parts** and **Layer** information. With the Sherlock 3D Viewer you can interactively view the 3D models and results generated by Sherlock for **ICT** and **Shock/Vibration Analysis**. This section describes the various features provided by these viewers.

Tip:

The sections below give a *general* overview of the tools Sherlock's provides for viewing results. Additionally, you can find analysis-specific content under the relevant analysis in this user's guide. For example, the chapter on the Harmonic Vibe analysis (FEA) has its own results section, Harmonic Vibration Analysis Results (p. 694). Or, if you are running a Thermal Derating analysis, refer to the subsection on interpreting results (p. 437) under *Thermal Derating*.

Section Topics

10.1. Sherlock Results Viewer

10.2. Sherlock 2D Viewer

10.3. Sherlock 3D Viewer

10.4. Reporting and Data Export

10.1. Sherlock Results Viewer

The Sherlock Results Viewer allows you to view the most recent results generated by Sherlock for a given CCA, as well as result snapshots saved to the file system. In Windows, Sherlock Results Viewer can be launched using any of the following methods.

- To view the current analysis results from within Sherlock, double-click any entry in the **Analysis** folder of the Sherlock **Project Tree** or right-click an entry and select the **View Results** menu item.
- To view saved analysis results from within Sherlock, double-click any entry in the **Results** > **Saved Results** folder of the Sherlock **Project Tree** or right-click an entry and select the **Show Results** menu item.
- To view saved analysis results directly from the desktop, locate the saved result file in the file system (saved either automatically in the Sherlock project directory or in a folder designated using the **Save As** menu option discussed below). All analysis result files are assigned the .dfr-result file type which is automatically registered with Windows and associated with the Sherlock Results Viewer during the Sherlock installation process. As such, you may simply double-click an analysis result file with that

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

suffix at any time to launch the Sherlock Results Viewer and automatically open the selected result file.

- To view saved analysis results in a Sherlock Results Viewer that is already running, you may either use the **Results > Saved Results** option in the main Project Tree to open the saved result file or you may simply drag-and-drop the file onto the viewer window. In either case, any currently open analysis results will be automatically closed, and the new analysis results will be displayed.

Linux

To launch the Results Viewer, use the command **`runSherlock -resultViewer`**

Note:

Only a single result file may be displayed in the Sherlock Results Viewer at a given time. If you open another file in the same Results Viewer, the previous results will be automatically closed and removed from the viewer tabs.

10.1.1. The Results Viewer and Memory Management

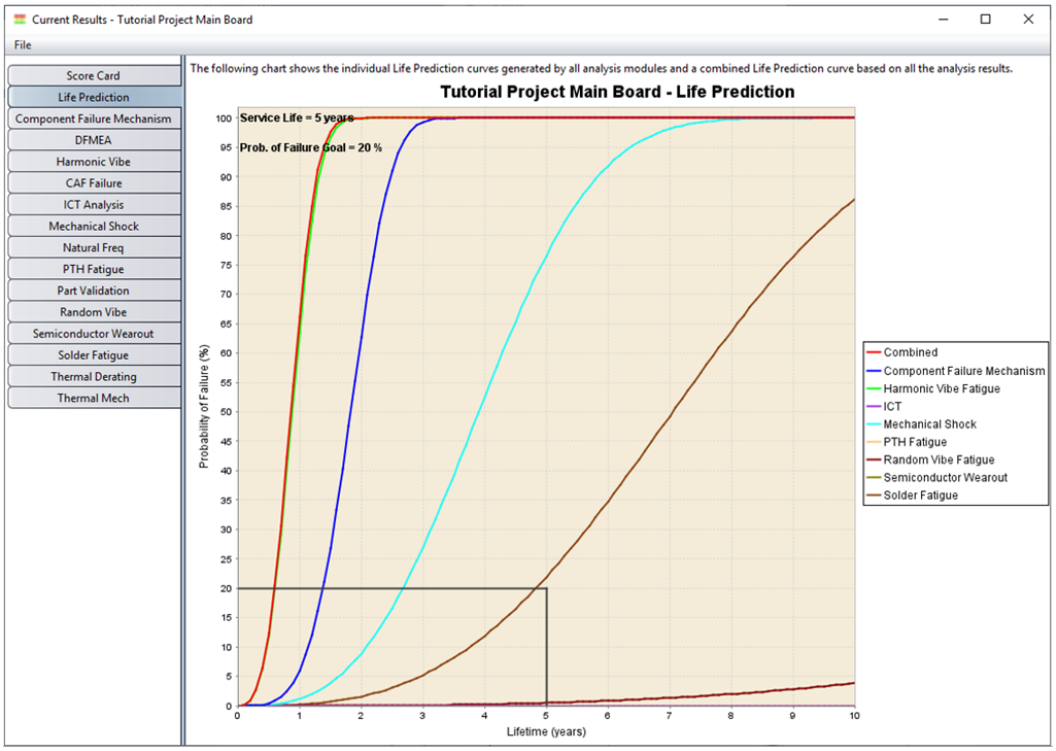
The Sherlock Results Viewer may run slowly when analyzing large projects even if the computer has ample amounts of memory. This is likely a Java issue.

If you encounter an error message that the Results Viewer has run out of memory, you can allocate more memory using the **Launcher Settings**. From Sherlock's **Main Menu**, click **Settings > Launcher**. Refer to Launcher Settings (p. 84) for additional information.

10.1.2. Viewing Current Results

Let's get started by viewing the current analysis results for the Tutorial Main Board. In this case, we'll double-click the **Results > Life Prediction** entry in the Project Tree for the **Main Board** to launch the Sherlock Results Viewer to display the current results:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



As shown here, the Results Viewer consists of a main viewing area that is tabbed to organize the results. The tabs on the left-hand side indicate the types of results that are available. Some analysis tabs also display sub-tabs along the bottom of the viewing panel to allow you to see various analysis details.

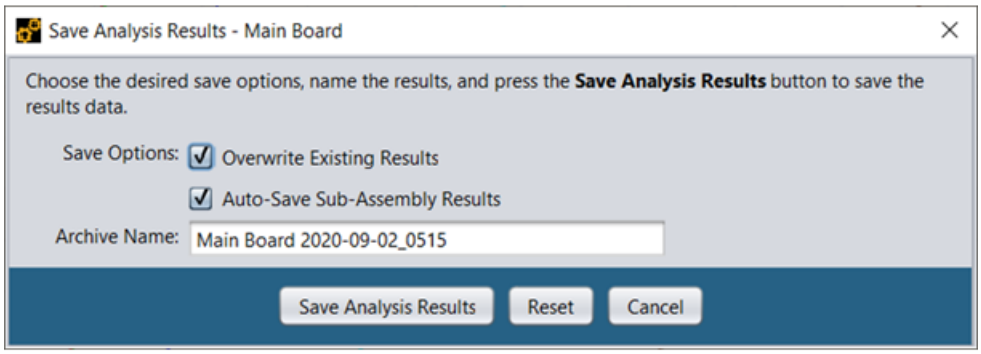
When viewing current analysis results, there are only two menu options available in the Results Viewer: **Save Results** and **Exit**. The **Exit** option closes the Results Viewer window and works the same as pressing the **X** icon of the window itself. You can save results by selecting **File > Save Results** from the upper left-hand corner of the results dialog. The **Save Results** option is a convenient way to save the current results for later review.

10.1.3. Saving Results

If you think the results are worthy of being saved, we can do just that by select **CCA > Save Circuit Card Results** from the main menu. At that point, you will be prompted to name the results (which is used as the file name), with a default name provided based on the CCA name and the date/time. You are free to use any appropriate name for the saved results.

The option **Auto-Save Sub-Assembly Results** will also save for each CCA which is a sub-assembly of the current CCA those results with the name provided.

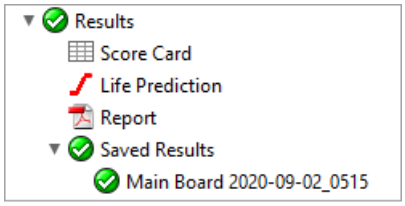
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

In addition to the **CCA > Save Results** menu option available in the Sherlock Results Viewer, you can save results directly from Sherlock using the **Result > Save Circuit Card Results** menu option in the Sherlock **Project Tree**. In either case, the same dialog will appear.

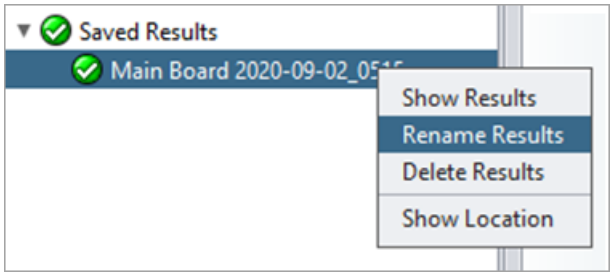
When the **Save Analysis Results** button is pressed, the results will be stored in the file system in a directory associated with the CCA and will appear in the Circuit Card's **Saved Results** folder, as shown here:



Saved results can be opened with the Results Viewer by either double-clicking the name or by right-clicking and selecting **Show Results**.

10.1.4. Renaming / Deleting Results

If you need to rename saved results, you can right-click and select **Rename Results**. Likewise, to remove past results, select **Delete Results**.



10.1.5. Viewing Saved Results

When the Results Viewer is used to view current results, it displays tabs for the following information:

- Score Card

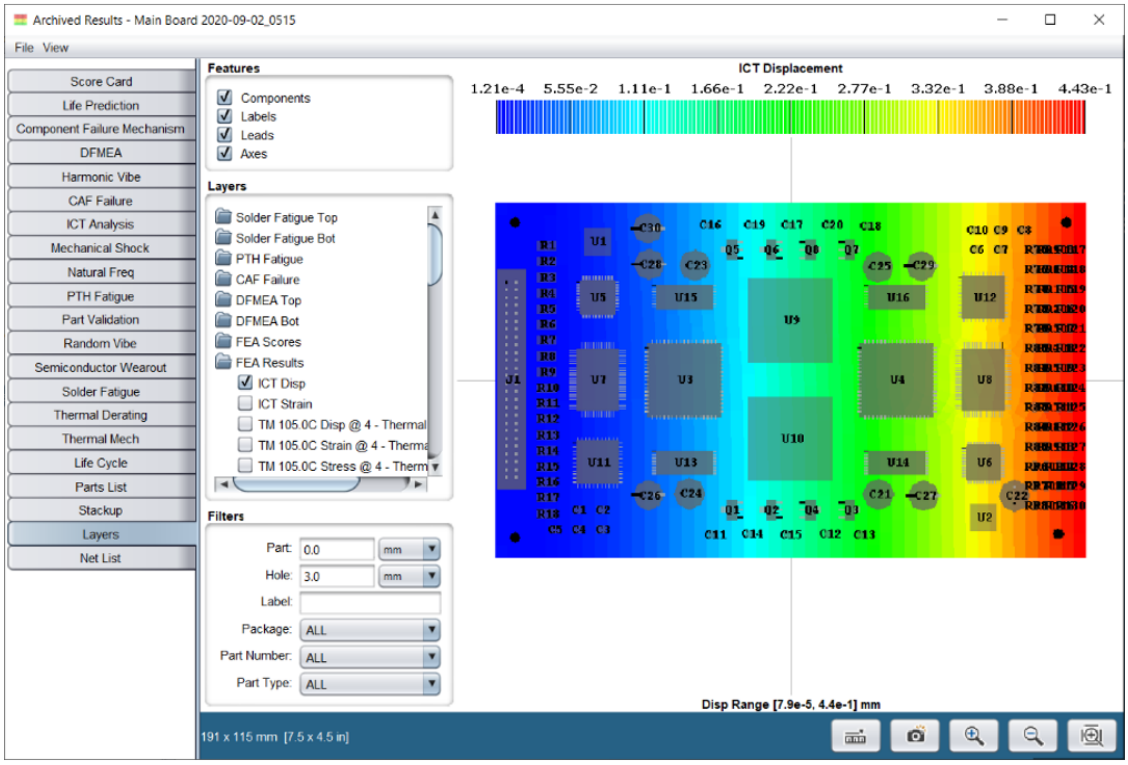
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Life Prediction
- Analysis Results

Other current information, such as the Life Cycle summary and Parts List, can be viewed and/or edited using the various data displays/editors provided by Sherlock. When analysis results are saved, all current CCA data is included in the snapshot to allow users to review both inputs and analysis results at a later time. Specifically, the following information is stored in addition to the results listed above:

- Life Cycle
- Parts List
- Stackup Data
- Result Layers
- 3D Results (if available)

For example, if we double-click the saved result created above the Sherlock Results Viewer will show the additional tabs that display the input data used to generate the corresponding analysis results.



You are free to review any of the information provided, but you will not be allowed to edit any of the data values.

Note:

Sherlock opens all current and saved results in separate Results Viewer windows, making it easy to compare and contrast results.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

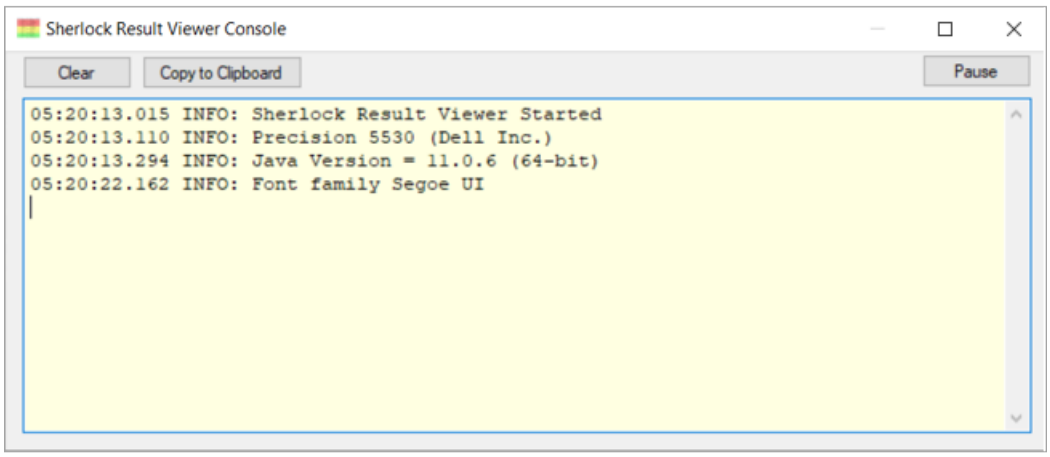
10.1.6. Exporting Saved Results

The Sherlock Results Viewer provides several ways to export and share results data. As mentioned in the Project Overview (p. 8) tutorial, Sherlock exports data in various formats such as PDF Reports, tabular data files, and images. Specifically, the following exports may be performed:

- To generate a PDF Report of the saved CCA results, use the **File > Generate Report** menu item, select the desired options and click **Generate Report**. Once the report has been generated, it can be viewed again by selecting **View Repor**t from the **View** menu. See the Project Overview (p. 8) tutorial for more details.
- To export data from any of the tables displayed in the Sherlock Results Viewer (for example: Parts List, Stackup, Analysis Results), select one or more rows in the table, right-click any entry and select the Export menu item. See the Project Overview (p. 8) tutorial for more details.
- By default, analysis results are automatically saved in the **Sherlock Project Directory**. If you would like to share the results with another user or save them in another file system location use the **File > Save As** menu option in the Sherlock Results Viewer. Simply specify the desired file name and location to create a copy of the analysis results.

10.1.7. Console Window

The Sherlock Results Viewer displays debugging and other trouble-shooting information to a console window. (Windows only. See Linux note below.) Normally, this window is hidden. If you need to view this information, use the **View > Console** menu option to display the **Console Window** as follows:



During normal viewer processing, messages may be continually written to this window. To pause the output, click the **Pause** button. The console text can be conveniently copied to the system clipboard by clicking the **Copy to Clipboard** button, thereby allowing you to easily paste the contents into an email or save them in a text file for future analysis. As indicated by the name, the **Clear** button clears the contents of the **Console Window**.

Linux

On Linux, there there is no separate client console window. For its output, Sherlock uses the terminal window which was used to launch Sherlock.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.2. Sherlock 2D Viewer

You will find a guided explanation of the 2D Layer Viewer in Tutorial: Layer Viewer (p. 43).

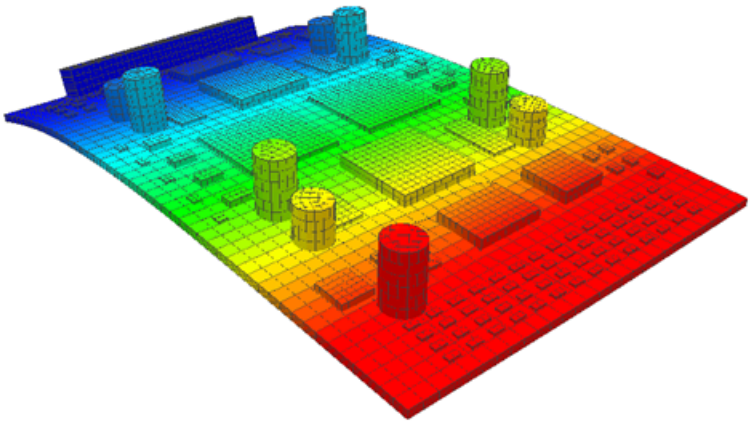
10.3. Sherlock 3D Viewer

The Sherlock 3D Viewer is a powerful tool for studying the results generated by Sherlock's finite element analyses: Harmonic Vibe, ICT, Mechanical Shock, Random Vibe, and Thermal Mech. The Viewer displays the model and the analysis results using an intuitive Color Scheme to convey the results for each element in the model.

You can fully interact with the 3D model and the Color Schemes to create a visual presentation that best suits your needs:

- Choose from a list of Color Schemes to visualize displacement, stress, strain values, composite scores, as well as various material properties.
- Hover over individual elements to view exact strain or stress values.
- Move, rotate, or zoom your view of the model.
- Magnify the level of warping so you can easily see displacement effects.
- Export a publication-ready image of the model or a 3D interactive model for PDF.

10.3.1. Launching the 3D Viewer
10.3.2. Viewing and Configuring the 3D Results
10.3.3. 3D Viewer's Main Menu and Toolbar
10.3.4. Exporting 3D Images
10.3.5. The 3D Viewer and Memory Management

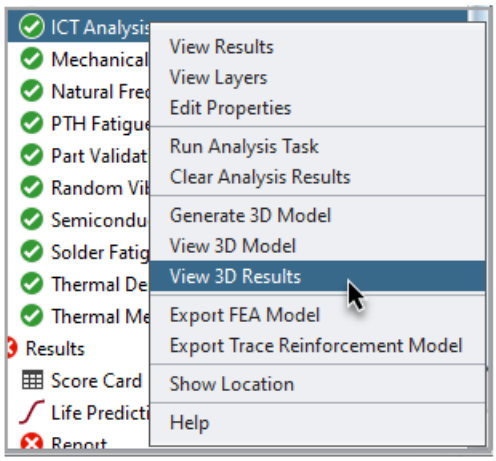


10.3.1. Launching the 3D Viewer

In Windows, you may launch the 3D Viewer several ways:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **From Sherlock's Project Tree:** As an example, if you wish to view the most recent **ICT** results, right-click the **ICT Analysis** entry in the Sherlock **Project Tree** for a given CCA and select the **View 3D Results** menu item.



- **From the Results Viewer:** To view saved 3D results from the Sherlock Results Viewer (p. 785), use the **View** menu to select the desired 3D results (if any) saved with the results snapshot.

- **From your PC Desktop:** To view saved 3D results directly from the desktop, locate the 3D results file in the file system (saved either automatically in the Sherlock project directory or in a folder designated using the **Save As** menu option). All 3D result files are assigned the **.dfr-3d** file type which is automatically registered with Windows and associated with the Sherlock 3D Viewer during the Sherlock installation process. As such, you may simply double-click a 3D results file with that suffix to launch the Sherlock 3D Viewer and automatically open the selected result file.

- **From Sherlock 3D Viewer:** To view saved 3D results in a Sherlock 3D Viewer that is already running, use the **File > Open File** menu option to open the saved results file, or simply drag-and-drop the file onto the viewer window. In either case, any currently open 3D results will close automatically, and the new 3D results will be displayed.

Linux

If Sherlock is running on Linux, launch the 3D Viewer using the command: **`runSherlock -3dViewer`**

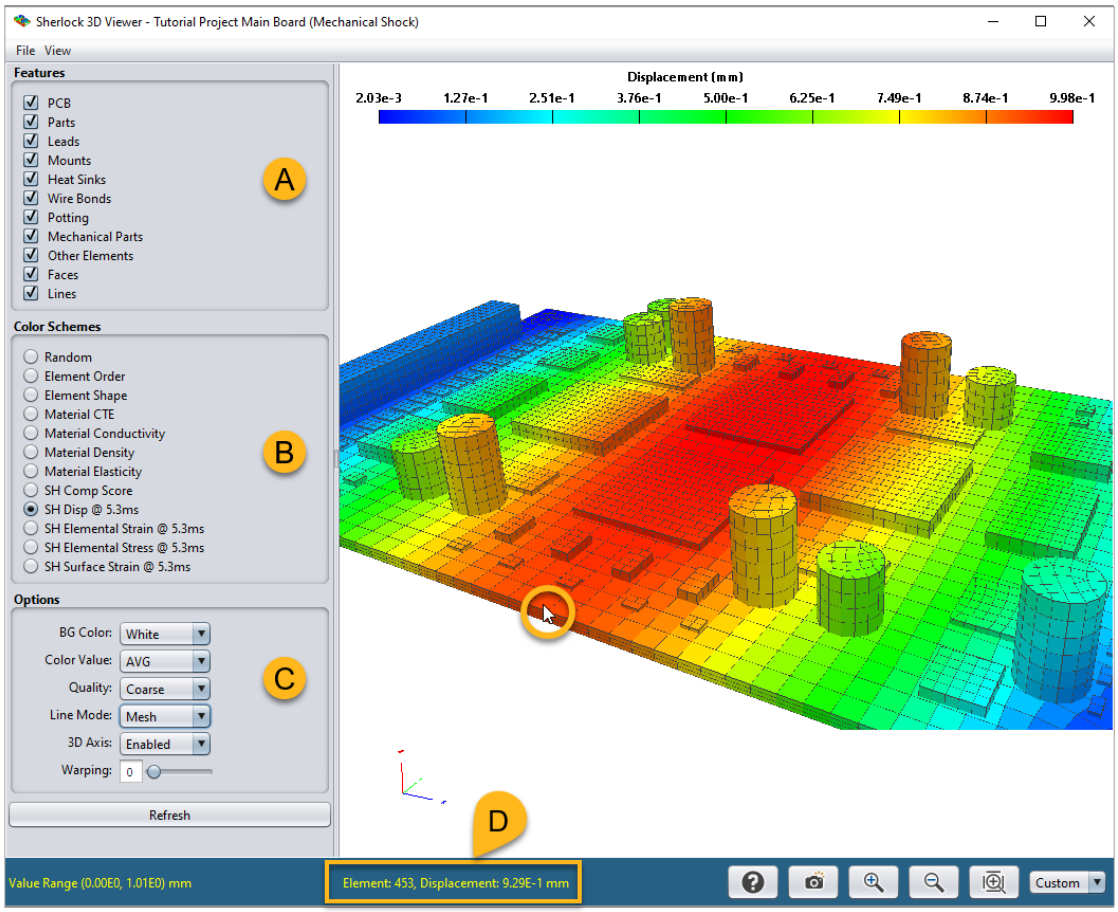
10.3.2. Viewing and Configuring the 3D Results

Sherlock's 3D Viewer provides several convenient ways to view the 3D models and analysis results. The main viewing area shows the 3D model (which includes only the features you have selected) along with the overlaid color scheme which conveys the analysis results. You can pan, rotate, and zoom the 3D display as needed using simple mouse controls.

Tip:

Press the **Help** icon  for a list of view-navigation mouse controls.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Customizing the View Using the 3D Viewer Panels

The Viewer panels (A, B, & C in the image above) provide options for you to customize the view so the results are displayed in a way most helpful to you:

- **Features**: (A, above) Here you select which model features you want displayed- such as the PCB, parts, leads, and so on. Typically, all features are selected, but you are free to choose whichever ones suit your needs. Click the **Refresh** button to apply your selection.

  Tip:

  Remember, after selecting any option(s) in the panels, you must click the **Refresh** button to apply the new settings.

- **Color Schemes**: (B) This panel allows you to select what kind of information (from the results file) you want the Color Scheme to display. In the example above (generated from a Mechanical Shock analysis), the Color Scheme displays the displacement that occurred at the 5.3 ms mark.

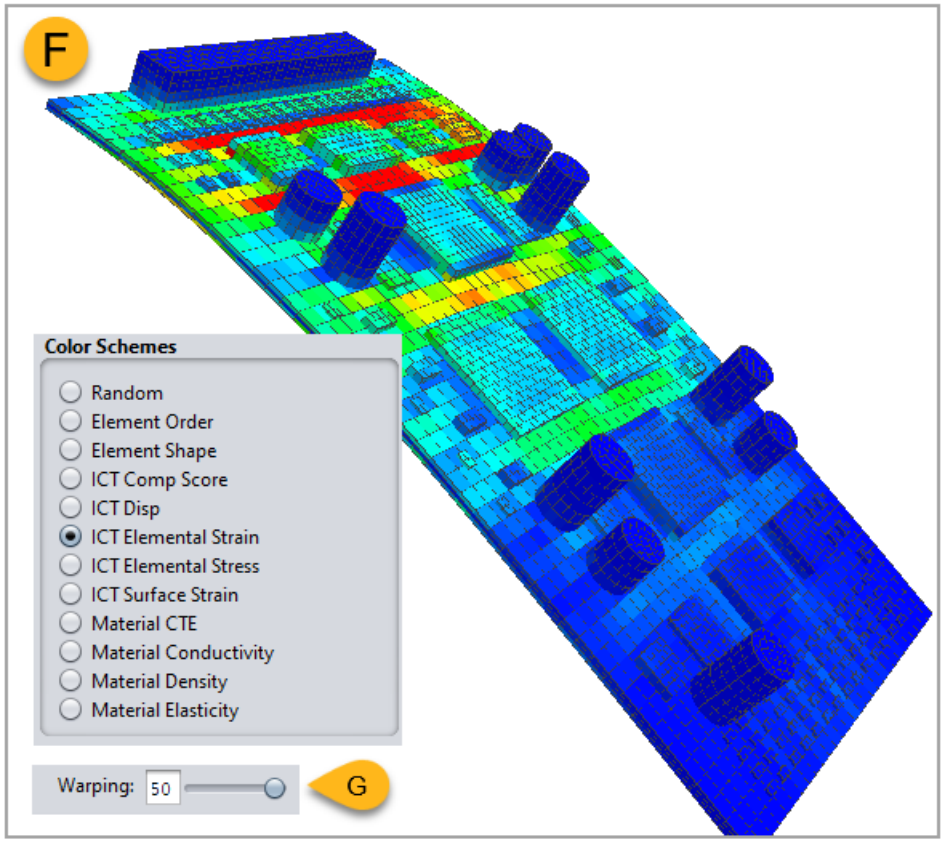
  The list of available Color Schemes varies, depending on the specific analysis type and how the analysis was configured. As always, click the **Refresh** button to apply your selection.

  Once you have selected and applied a Color Scheme, you can hover the mouse over any portion of the board to see the value associated with that color. (This feature works for Color Schemes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

based on element properties, material properties, or FEA results. It will not work when **Random** is selected.) In the image above, the mouse is pointing to Element 453 which has a displacement value of 9.29E-1 mm (see D). By left-clicking that location, you can send that information to the 3D Viewer Console. If the **3D Viewer Console** is not open, simply select **View > Console** in the 3D Viewer window.

By default, the displacement values computed for a specific analysis-type are applied to all nodes in the display model. For example, if you are viewing the results from an ICT analysis and have selected the **ICT Disp** results in the Viewer, then the model will be bent and colored to indicate the computed displacements. If, on the other hand, you select the **ICT Elemental Strain** results, then the model will be bent to indicate the computed displacements, while the colors indicate the computed strains. (See F below.) In this way, a single 3D plot can indicate both displacement and strain results.



Tip:

Another way to customize Sherlock's use of colors is to experiment with the **Color Settings** option in the **View** menu. This determines the results-range of the minimum-results color (blue) and of the maximum-results color (red). In other words, you can adjust the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

width of the blue and red regions at each end of the color spectrum. See Color Settings (p. 796) below.

Note:

In the 3D Viewer, when viewing any of the color schemes related to material properties (**Material CTE**, **Material Conductivity**, **Material Density**, and **Material Elasticity**), the values represented are for the material at 20°C only.

- **Options**: This panel (C, above) provides display options for altering the 3D rendering of the model. You must click the **Refresh** button to apply any changes you make.
  - **BG Color**: (White, Blue, Black, Gray) Set the Viewer's background color.
  - **Color Value**: (AVG, MIN, MAX) Specify whether the color scheme displays the average, minimum, or maximum result for each element. Default is **AVG**.
  - **Quality**: Specifies the number of triangles or quads used to render the results data. The **Coarse** setting displays a single element for each element used in the 3D analysis model and is suitable for most purposes. The **Fine** setting displays 4 or 5 elements for each modeled element, resulting in a more uniform look.

    Tip:

    Typically, **Coarse** is used for general viewing and **Fine** is used for publication-ready images.

  - **Warping**: Since displacement values in the results file can be relatively small, the **Warping** factor is provided (G, above) to amplify displacement, making it easier to see. A warping factor of 0 eliminates all displacement results. A warping factor of 1 shows the actual results, and a warping factor between 2 and 50 multiplies the displacement values by the given amount.
  - **Line Mode**: (Outline, Mesh, or None) If you want black lines to outline the edges of the board only, select **Outline**. Select **Mesh** to mark the outline of each element. Select **None** to turn off this feature altogether.
  - **3D Axis**: (Enabled, Disabled) If you want the X/Y/Z axis indicator to appear in the Viewer, select **Enabled**.

10.3.3. 3D Viewer's Main Menu and Toolbar

3D Viewer Menu

The 3D Viewer's main menu (near the top of the window) contains two items: **File** and **View**.

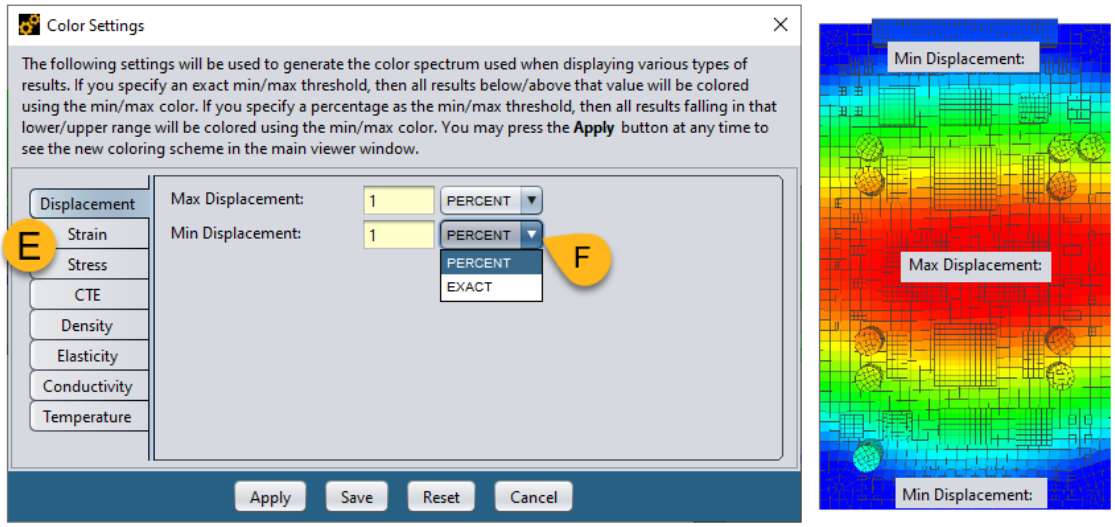
- **File**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Open File**: Open a `.dfr-3d` file (Sherlock 3D Results file) or `.inp` file (Abaqus input file) in the 3D Viewer.
  - **Recent 3D Files**: Select a recently viewed file from the list.
  - **Save As...**: Save `.dfr-3d` file in another location and/or with another name. Only the analysis results and 3D model are saved, not the Viewer settings.
  - **Export Image**: See Exporting 3D Images (p. 798) below.
  - **Close Viewer**: Closes the 3D Viewer.

- **View**
  - **Model Info**: Displays basic information about the 3D model: Number of solids, number elements, number of lines, and number of faces.
  - **Color Settings**: This opens the **Color Settings** window shown below. Here you can set the **Max** and **Min** values which affect how Sherlock displays the Color Scheme. When these values are applied, results above the **Max** value are displayed as red on the model. Results below the **Min** value are blue. The normal color spectrum is applied to values in between. In effect, adjusting the Min and Max values reduces or increases the width of the blue color (minimum) and of the red color (maximum results) when Sherlock displays the results.

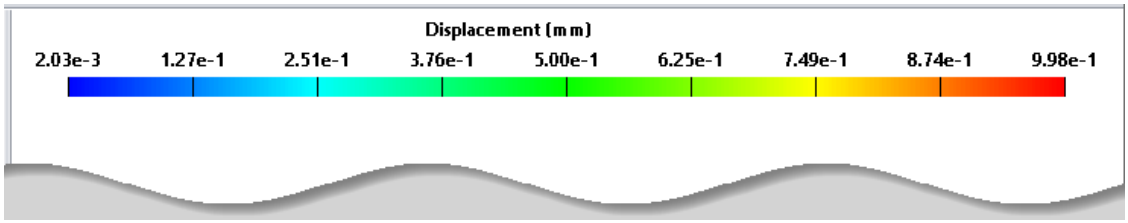
    Each result-type (displacement, strain, stress, and so on) has its own tab (E, below) so you can apply a distinct setting for each.



    Use the dropdown menu (F) to specify whether the **Max**/**Min** value you entered is a **PERCENT** or an **EXACT** number.

    → **PERCENT**: The percentage is based on the number of results, not the *range* of the results. Example: If Sherlock generated ten thousand displacement results for a given analysis and **Max Displacement** is set to 1 PERCENT, the highest 100 results will be displayed as red, regardless of the displacement *range* (2.03e-3 to 9.98e-1 mm in the example below).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



→ **EXACT**: This means the number entered represents an exact value, assuming the same units displayed in Viewer (mm in the example above). If, for example, you enter **1.28e-1 EXACT** in the **Min Displacement** field, any results below 1.28e-1 mm will be displayed as blue.

You may press the **Apply** button at any time to see the new coloring scheme in the main Viewer window.

- **Zoom Commands**: Basic zoom commands: Zoom Out, Zoom In, Zoom Fit. Functions the same as the zoom icons in the **Toolbar**.
- **Use View as Default**: Sets the current view of the model (position, orientation, and level of zoom) as the default view which is used for initial 3D Viewer startup and when you select **Default** in **View Selector** in the **Toolbar**.
- **Reset Default View**: Restores the view to the default setting. See above, **Use View as Default**.
- **Console**: Displays the **Sherlock 3D Viewer Console**.
- **Memory**: Displays the **Memory Monitor** for the 3D Viewer. See The 3D Viewer and Memory Management (p. 799).
- **Help**: Displays basic reminders how to navigate the view with your mouse.

3D Viewer Toolbar



- **H. Help**: Displays basic reminders how to navigate the view with your mouse.
- **I. Export Image**: Opens the **Export Image** dialog. See Exporting 3D Images (p. 798) below.
- **J. Zoom In**
- **K. Zoom Out**
- **L. Zoom to Fit**: Adjusts zoom level so model fits the display size.

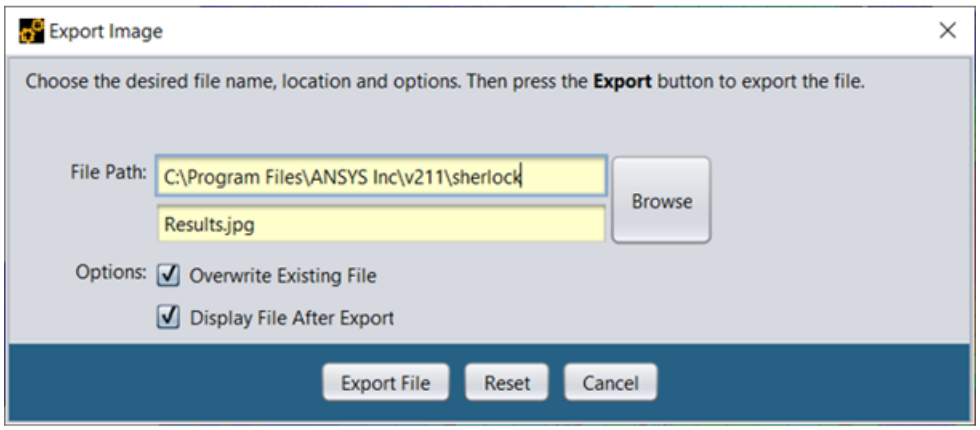
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **M. View Selector**: Choose among several preset view options.

> **Tip:**
>
> To set the current view as the default view, click **View** in the Viewer's main menu and select **Use View as Default**.

10.3.4. Exporting 3D Images

You can create publication-ready image files at any time by clicking the camera icon and designating a destination file. Alternatively, you can select the **File > Export Image** menu item. In either case, the **Export Image** window opens, as shown here, allowing you to specify the location and name of the file to be exported. Click the **Browse** button to find the appropriate location and/or name of the file to be exported.



The filename suffix you enter specifies the export file format. The following formats are supported:

***.jpg** or ***.jpeg** - JPEG image file format

***.png** - Portable Network Graphics file format

***.pdf**- Portable Document Format

The JPEG and PNG file formats provide a 2D representative of the circuit card results that you can incorporate into reports, presentations, and email messages. The PDF file format provides a standard 3D representation of the circuit card and the results you can be view interactively using an Adobe PDF viewer or a compatible PDF viewer that supports the Adobe 3D PDF standard.

Select one or more of the processing options to customize the export process:

- **Overwrite Existing File**: If selected, Sherlock will automatically overwrite any existing file with the same name without prompting you for confirmation.
- **Display File After Export**: If selected, Sherlock will attempt to display the exported file using whatever application is associated with the file name suffix in Windows. For example, if a PDF file is generated, then the default PDF viewer will open to display the exported file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3.5. The 3D Viewer and Memory Management

The Sherlock 3D Viewer may run slowly when analyzing large projects even if the computer has ample amounts of memory. This is likely a Java issue.

If you encounter an error message that the 3D Viewer has run out of memory, you can allocate more memory to the 3D Viewer in the **Launcher Settings**. From Sherlock's **Main Menu**, click **Settings > Launcher**. Refer to Launcher Settings (p. 84) for additional information.

10.4. Reporting and Data Export

In addition to the large amount of data maintained for circuit cards, Ansys Sherlock generates a large amount of data when analysis tasks are executed. In most cases, Sherlock summarizes such data in graphical form to help users determine the real sources of problems. But, given the extensive nature of the analysis performed, even the summary graphics can be overwhelming. More importantly, analysis results might be used for subsequent analysis or reporting tasks performed outside of Sherlock. For these reasons, Sherlock has features designed to allow users to export data in various formats including:

- Printable report files (PDF)
- Graphical chart image files (PNG)
- Tabular data files (CSV)

This document shows how each of these types of files can be exported and integrated with other user data.

Note:

The examples shown in this document are based on the Sample Project provided as part of the Sherlock Tutorial package as discussed in the Project Overview (p. 8) lesson.

In this section, the following topics are covered:

10.4.1. Generating Report Files

10.4.2. Exporting Graphical Data

10.4.3. Exporting Tabular Data

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

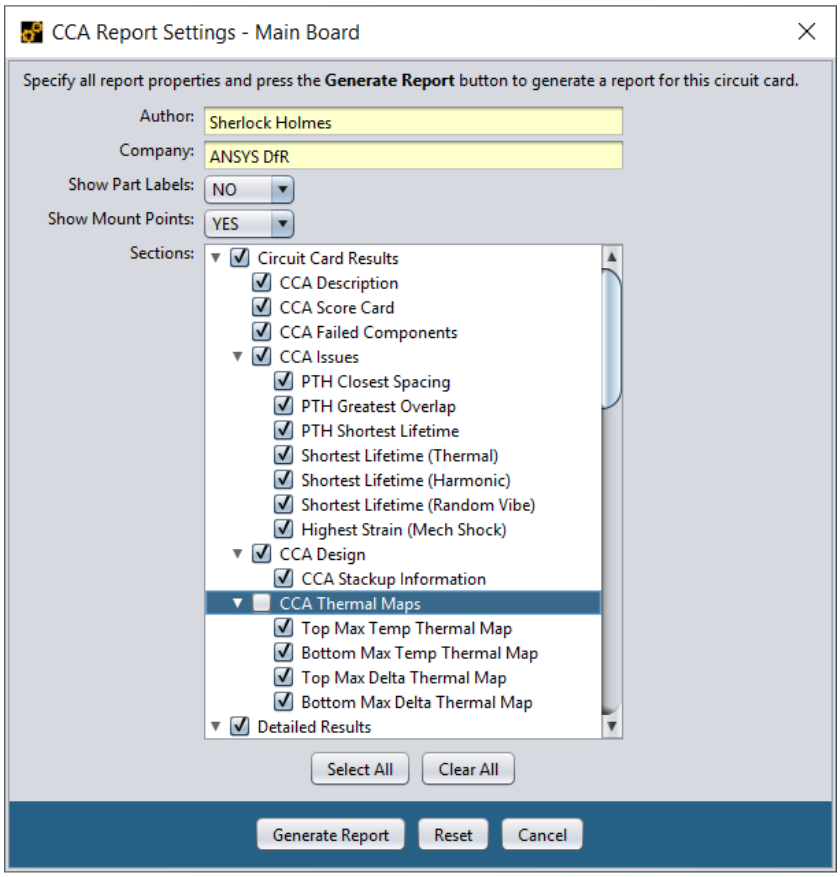
10.4.1. Generating Report Files

The fastest and easiest way to export project data is to generate a report file. Report files are formatted as PDF documents based on a collection of predefined report objects that can be selectively chosen by the user. Reports can be generated for an entire project or for each circuit card assembly. Report objects are defined that can be used to:

- Provide an overview of the analysis tasks performed
- Describe the circuit cards analyzed
- Document the Life Cycle assumptions
- Show graphical results
- List the most important circuit card issues

Sherlock report files are formatted so that they can be used as preliminary, intermediate or final reports, depending on specific user needs.

To generate a Project Report, right-click the **Report** entry located in the **Results Folder** of the **Project Tree** and select the **Generate CCA Report** menu option. After selecting **Generate Report**, the **Report Properties** dialog will appear.



The **Report Properties** dialog allows you to select the report sections to be included in the report. In this project report example, the report will include an initial set of explanatory sections that describe

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the analysis tasks performed by Sherlock, followed by a complete list of the Life Cycle Phases and Events defined for this project.

After those project-level sections, the report will include circuit-card level results for each of the circuit cards defined in the project. Depending on the number of circuit cards contained in the project and your reporting needs, you can select exactly which CCA data should be included for each circuit card.

When generating a circuit card report instead of a project report, on the **Circuit Card Results** and **Circuit Card Details** will be available for selection and the report will be limited to the given circuit card. Whereas a project report will provide details for all circuit cards. The **Circuit Card Details** section found in the circuit card report will include all the select analysis layers generated for each circuit card.

As shown here, the **CCA Thermal Maps** are NOT included in the report

Although the section list is very long, it is relatively easy to navigate and select desired sections. The check box to the left of each entry indicates whether it is included in the report. The only important thing to remember is that if you deselect an item in the tree, then all items below that item will be excluded from the report as well, regardless of their individual settings. This allows you to quickly include or exclude report sections without having to change all the individual settings below that item.

When the **Generate Report** button is pressed Sherlock will generate a PDF report file containing the selected sections, based on the current analysis inputs and results. A copy of the report file will be kept in the Sherlock project or circuit card directory for subsequent viewing but will be automatically cleared if any Sherlock inputs are changed.

Note:

If you want to maintain a collection of reports that address different topics or that cover different inputs or analysis results, then you should save of copy of the PDF report to your own directory and change the file name accordingly. Sherlock only keeps the most recently generated report for a given project or circuit card.

10.4.1.1. Customized Reports

By creating their own **Report Definition File**, advanced users can customize the following report characteristics:

- Logo image, title, keywords
- Section headers, descriptive text, and formatting
- The order that sections are published
- Define additional sections

This allows users to generate reports bearing the logo of their organization, that include organization-specific information, and that present data in a user-defined order.

Report Definition Files are formatted as an XML file, containing a collection of predefined tags, as shown here:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<SherlockReport>
    <Document
        title       = "Sherlock Analysis Results"
        subject     = "Sherlock Analysis Results"
        keywords    = "Sherlock, Reliability"
    />
    <Logo
        image   = "runtime/images/ReportLogo.png"
        width   = "250"
        height  = "150"
    />
    <Introduction title="Introduction" optional="true">
        The Sherlock automated design analysis software is capable of providing a comprehensive
        assessment of early stage circuit card design and its ability to meet quality, reliability,
        and durability expectations.   This assessment consists of a number of validated techniques,
        including physics-of-failure (PoF) based prediction that takes into consideration the
        use environment and the required lifetime. PoF modeling is a science-based approach that
        analyzes the loads and stresses in an application and evaluates the ability of materials
        to endure them from a strength and mechanics of material point of view. The wearout lifetime
        prediction will include a level of detail that provides the ability to understand the
        contributing factors and make targeted improvements.
        <p/>
        The analyzes that this software is capable of performing include:
        <p indent="40" />
        1. Plated through hole (PTH) lifetime<br/>
        2. Risk of conductive anodic filament (CAF) formation <br/>
        3. Solder joint lifetime under thermal cycling<br/>
        4. Solder joint lifetime under vibration<br/>
        5. Risk of interconnect failure during mechanical shock<br/>
        6. Mean time between failure (MTBF) calculation<br/>
```

The easiest way to create a custom Report Definition File is to copy the standard **Report.xml** file for a project report, or the standard **CCAReport.xml** file for a CCA report, or the standard **Results-Report.xml** file for archived results reports, contained in the config sub-directory of the Sherlock installation directory (usually **c:\Program Files\Ansys Inc\v251\sherlock**) to the **Sherlock User Data Directory**. Whenever Sherlock generates a report, any report definition file located in the **Sherlock User Data Directory** will be used instead of the standard report definition file located in the **Sherlock Installation Directory**. This allow users to make custom changes to their reports without fear of causing permanent damage. If the report doesn't turn out the way you like, just remove the local copy of the report definition file and Sherlock will revert to the standard definition file.

The order in which sections are created within the report are the same as they are defined in the XML file. When moving a section, be sure to copy the entire section defined by the beginning and ending XML tags.

Additional report sections for static text may also be included. Create a new unique tag with **title** and **optional** attributes, define the text, then end the section with the appropriate ending tag. Any section that is defined with an **optional** value of **true** will be listed in **Report Settings** dialog. If a given section is not optional, it will not be listed in the dialog but will always appear in the created report.

Example user-defined section:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<UserPage title="Company Introduction" optional="true">
This UserPage tag is being used to define a new report section.  The title of this
section will be <i>Company Introduction</i>.  Since it is defined to be an optional
section, the section title will appear as a selectable section in the Report Settings
dialog.  Sub-sections can also be defined here by creating a new XML tag within the
UserPage tag defined here.  If defined to be optional, then it too will be listed as a
selectable section in the Report Settings dialog.
</UserPage>
```

Each report section supports the following HTML tags to adjust formatting.

- **
** - adds a newline to the report
- **<p/>** - begins a new paragraph. Optionally may specify the attribute **indent** to specify the indent of the paragraph (example: **<p indent="40" />**)
- ****text**** - displays the text between the tags in bold.
- ****text**** - displays the text between the tags in italics.

It is beyond the scope of this lesson to explain how XML files are properly formatted and edited. Users familiar with maintaining XML documents should have no problem modifying the standard report XML files for their own needs. Users new to XML formatted files should contact Ansys Technical Support for help in creating customized reports.

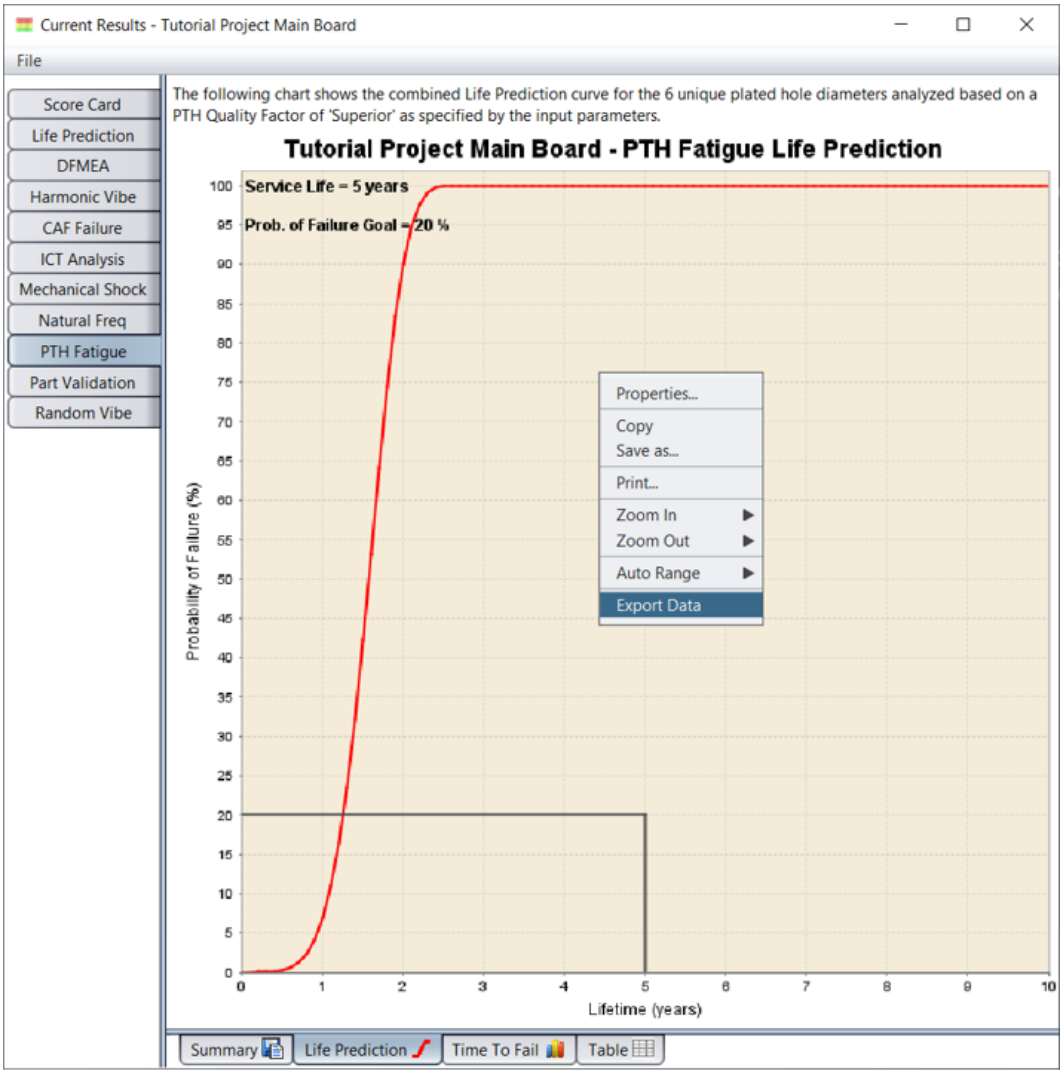
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.4.2. Exporting Graphical Data

Ansys Sherlock generates and displays several graphical layers and charts to show circuit card designs and analysis results. Many of these graphical outputs can be included in the PDF report by selecting the appropriate section. As a convenience, all graphical data shown in Sherlock can also be exported as image files with just a few mouse clicks, as we describe in this section.

10.4.2.1. Exporting Data

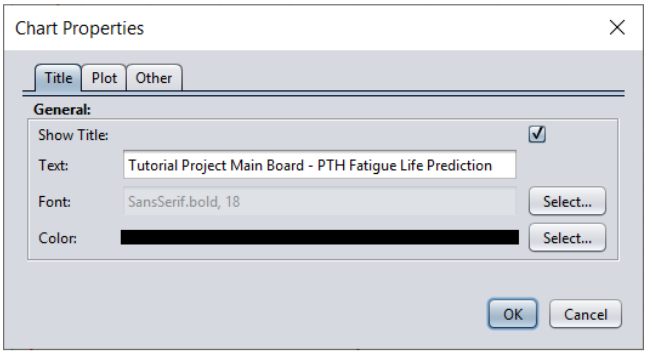
Chart data can be exported as a Comma Separated Value (CSV) file, that can be subsequently imported into any spreadsheet or database application for further review or processing. To do so, simply choose the **Export Data** menu option from the chart's right-mouse menu. Here, a file chooser dialog will appear allowing you to specify the name and location of the export file to be created. By default, the chart title is used as the file name, but it is easily changed to suit your purposes.



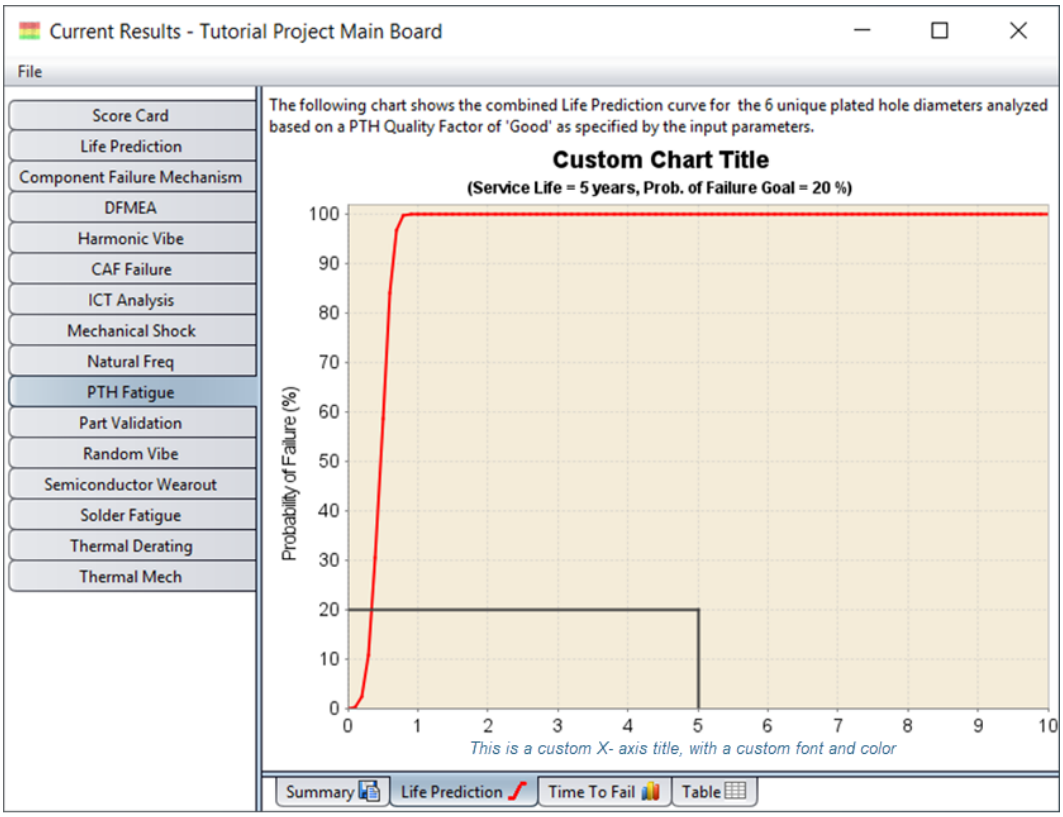
10.4.2.2. Changing Plot Properties

In some cases you may want to modify or customize the chart for a specific purposes, such as including it in a presentation or paper. If so, select the **Properties...** menu option from the chart's right-mouse menu to display the Chart Properties dialog.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Chart Properties** dialog allows you to easily change the chart title, axes titles and axes format. When combined with the built-in zoom and auto-range features, you can easily create a custom chart such as the one shown below in which we modified the chart title, X-axis title and the data ranges.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.4.3. Exporting Tabular Data

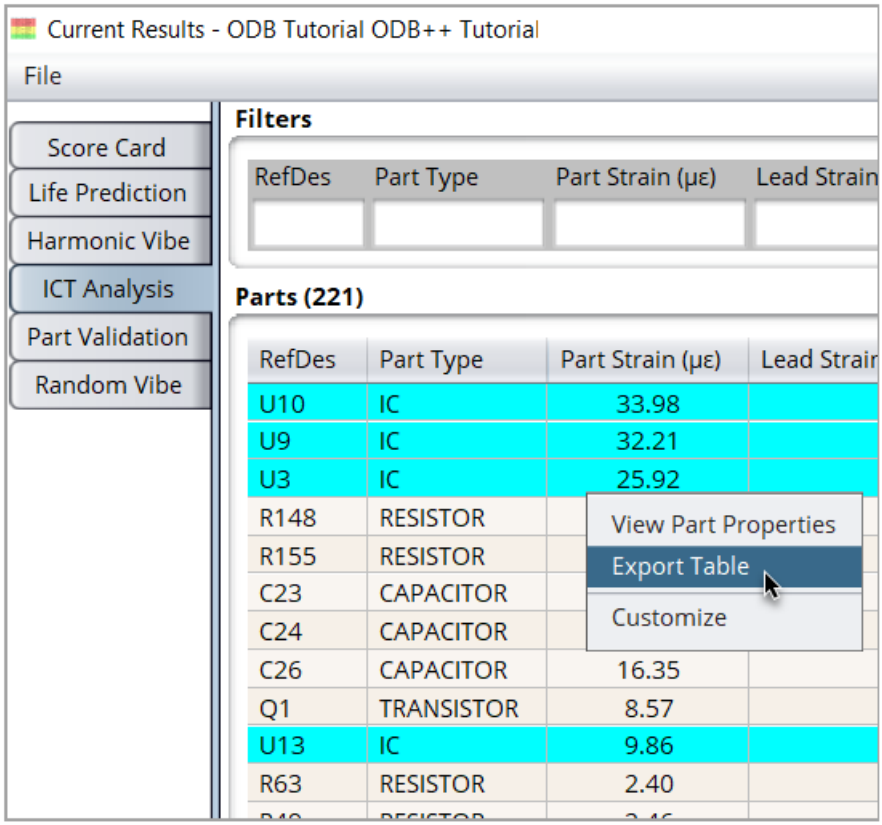
Graphics are very useful for summarizing and analyzing large data sets but there are times when you need to analyze the results in more detail or use them as input for other analysis tasks performed outside of Sherlock. To satisfy such needs, Sherlock provides the ability to export data from the following tables to a CSV file:

- Parts List
- Stackup Data
- Analysis Results

To export table data, open the table of interest in Sherlock and do the following:

1. If you wish to export only a selected set of table rows, highlight the rows now. Hold the **Shift** key and click to select a group of adjacent rows. Or, hold the **Ctrl** key and click to select individual rows.

2. If you have selected multiple rows, right-click any highlighted row and select the appropriate export option from the context menu (as shown below), such as **Export Parts List**, **Export Stackup Data**, or **Export Table**. If you intend to export all rows, simply right-click any row and select the export option in the context menu. The export dialog will open.

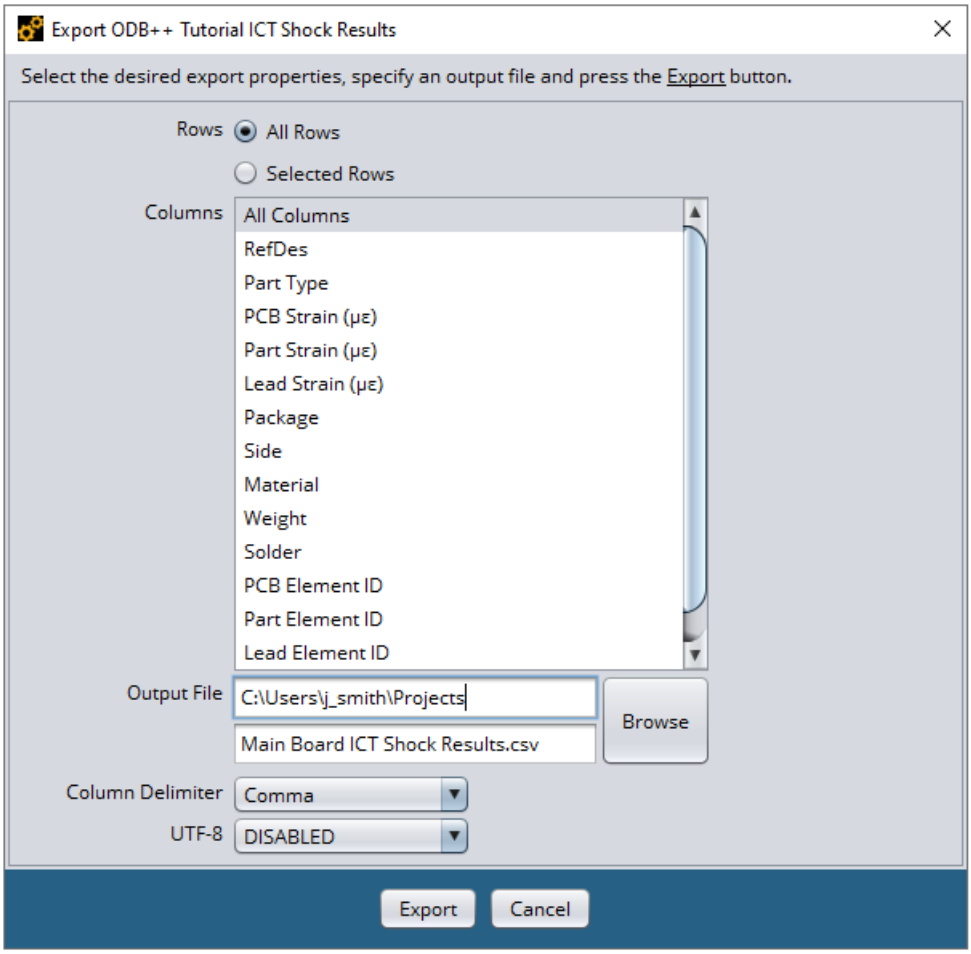


3. In the export dialog shown below, select the export properties:

   - **All Rows**: Select this option to export all table rows.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Columns**: Specify which columns you wish to export. Select **All Columns** or manually select which columns to export by holding the **Shift** or **Ctrl** key while you click specific columns.
- **Selected Row**s: Select this option to export only the rows you selected earlier in step 1.
- **Browse**: Use the **Browse** button to specify the name and location of the file to be created, or enter it directly in the text field.
- **Column Delimiter**: Select the column delimiter for the exported file: **Comma**, **Semicolon**, **Tab**, or **Space**.
- **UTF-8**: Enable this option so that special Unicode characters (like με) are displayed correctly when the file is opened in an external application such as Excel.



4. When you click the **Export** button, Sherlock exports the data as a CSV file in the location you specified.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 11: Integration with other Ansys Products

The following sections in the user's guide provide guidance in integrating other Ansys products into your Sherlock workflow:

FEA Analysis:

- **Ansys Workbench™ / Ansys Mechanical™** applications: The Ansys Workbench application is a simulation integration platform, a widely used FEA tool combining the ease of a point-and-click interface with the power of Ansys analysis engines.

  – From the Sherlock application, you can export circuit card data directly into a Workbench analysis stream and then import reliability results back to the Sherlock application as part of a Workbench solution. In the *FEA Analysis Types* section, see FEA- Ansys Workbench™ Integration (p. 747).

  – Starting with release 2024 R2, Sherlock automates the process of modeling a BGA component in greater detail for export to Ansys Mechanical. See Advanced BGA Modeling (p. 648).

- **Ansys Icepak® electronics cooling simulation software** (the classic version, not the version in Ansys Electronics Desktop™): Icepak is an industry leading software application which performs thermal and fluid flow analyses of integrated circuits, packages, printed circuit boards, and electronic assemblies—all powered by Ansys Fluent®'s Computational Fluid Dynamics (CFD) solver. When exporting to Icepak, Sherlock provides the package geometries and material properties with the level of detail needed for highly accurate thermal analysis. This workflow is described the in the *FEA Analysis Types* section. See FEA- Ansys Icepak Integration (p. 765).

- **Ansys LS-DYNA® nonlinear dynamics structural simulation software**: The LS-DYNA application is a general-purpose finite element program capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. When running an FEA Mechanical Shock Analysis (p. 674) in the Sherlock application, you can import LS-DYNA analysis results by selecting **LS-DYNA** as the **Imported Data Source** as explained here: LS-DYNA Imported Data Source (p. 680).

Solder Fatigue

Ansys AEDT Icepak electronics systems design platform (Ansys Electronics Desktop version): Sherlock's Solder Fatigue Analysis Module (p. 419) allows you to input thermal analysis results generated in Ansys Electronics Desktop (AEDT) - Icepak. This workflow is described in Using AEDT-Icepak Results in Sherlock for Solder Fatigue Analysis (p. 426).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 12: API Services

Beginning with release 2025 R1, Sherlock APIs are now available online at the Ansys DevPortal where you will find:

- **Closed source APIs utilizing gRPC**—a high performance, universal RPC (Remote Procedure Call) framework, allowing those with programming experience to access some of the Sherlock application's internal functions, making it easier, for example, to integrate Sherlock's capabilities with other applications.
- **PySherlock:** PyAnsys is a collection of open source Python libraries designed for engineers to enhance the capabilities of Ansys products, including the Sherlock electronics reliability prediction software. These Python client libraries enable efficient data manipulation, automation, and customization of workflows, empowering engineers to gain deeper insights and to streamline simulations. The project fosters a collaborative environment within the engineering community, encouraging knowledge sharing and community-driven development. With comprehensive documentation and vibrant community support, developers and engineers can maximize capabilities of PyAnsys and contribute to its growth.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.