



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

Ansys CFD-Post User's Guide



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

Preface	xxvii
1. About this Manual	xxvii
2. Document Conventions	xxviii
2.1. Spelling Conventions	xxix
1. Overview of CFD-Post	31
1.1. CFD-Post Features and Functionality	31
1.2. 3Dconnexion Product Support	33
1.3. Compatibility with File Hosting Services	33
1.4. Advanced Features	33
1.5. Next Steps... ..	33
2. Starting CFD-Post	35
2.1. Starting CFD-Post with the Ansys CFX Launcher	35
2.1.1. Valid Syntax in CFD-Post	36
2.2. Starting CFD-Post from the Command Line	37
2.2.1. Optional Command Line Arguments	37
2.3. Setting CFD-Post Operation Through Environment Variables	39
2.4. Running in Batch Mode	42
2.4.1. Example: Pressure Calculation on Multiple Files using Batch Mode	42
3. CFD-Post Graphical Interface	45
3.1. Graphical Objects	46
3.1.1. Creating and Editing New Objects	47
3.1.2. Selecting Objects	47
3.1.3. Object Visibility	48
3.2. Common Tree View Shortcuts	49
3.3. Details Views	49
3.4. Outline Workspace	50
3.4.1. Outline Tree View Shortcuts	51
3.4.2. Outline Details View	52
3.4.2.1. Geometry Tab	52
3.4.2.1.1. Selecting Domains	52
3.4.2.2. Color Tab	52
3.4.2.2.1. Mode: Constant	52
3.4.2.2.2. Mode: Variable and Use Plot Variable	52
3.4.2.2.3. Range	53
3.4.2.2.4. Hybrid/Conservative	53
3.4.2.2.5. Color Scale	53
3.4.2.2.6. Color Map	53
3.4.2.2.6.1. Accessing the CFD-Post Color Map Editor	54
3.4.2.2.7. Contour	54
3.4.2.2.8. Undefined Color	55
3.4.2.3. Symbol Tab	55
3.4.2.3.1. Symbol	55
3.4.2.3.2. Symbol size	55
3.4.2.4. Render Tab	55
3.4.2.4.1. Use Post Region Rendering	55
3.4.2.4.2. Show Faces	56
3.4.2.4.3. Show Faces: Transparency	56
3.4.2.4.4. Show Faces: Draw Mode	56
3.4.2.4.5. Show Faces: Face Culling	57

3.4.2.4.6. Show Faces: Lighting	58
3.4.2.4.7. Show Faces: Specular Lighting	58
3.4.2.4.8. Show Mesh Lines	58
3.4.2.4.9. Show Mesh Lines: Edge Angle	58
3.4.2.4.10. Show Mesh Lines: Line Width	58
3.4.2.4.11. Show Mesh Lines: Color Mode	58
3.4.2.4.12. Show Mesh Lines: Line Color	58
3.4.2.4.13. Apply Texture	59
3.4.2.4.14. Apply Texture: Predefined Textures	59
3.4.2.4.15. Apply Texture: Custom Textures	59
3.4.2.4.16. Apply Texture: Texture Examples	59
3.4.2.5. View Tab	60
3.4.2.5.1. Apply Rotation Check Box	60
3.4.2.5.1.1. Method, Axis, From, To	60
3.4.2.5.1.2. Angle	60
3.4.2.5.2. Apply Translation Check Box	60
3.4.2.5.3. Apply Reflection/Mirroring Check Box	60
3.4.2.5.4. Apply Scale Check Box	61
3.4.2.5.5. Apply Instancing Transform Check Box	61
3.4.3. Case Branch	61
3.4.3.1. Domain Details View	62
3.4.3.1.1. Instancing Tab	62
3.4.3.1.2. Info Tab	62
3.4.3.1.3. Data Instancing Tab	63
3.4.3.2. Boundary and Subdomain	65
3.4.3.3. Injection Regions	65
3.4.3.4. Other Locations	67
3.4.3.5. Spray	67
3.4.3.5.1. Spray: Geometry Tab	67
3.4.3.5.1.1. Domains	67
3.4.3.5.1.2. Reduction Type	67
3.4.3.5.2. Spray: Color Tab	67
3.4.3.5.3. Spray: Symbol Tab	67
3.4.3.5.4. Spray: Render Tab	68
3.4.3.5.5. Spray: View Tab	68
3.4.3.6. Operating Maps	68
3.4.3.6.1. Operating Map: Chart Data Tab	68
3.4.3.6.2. Operating Map: Chart Display Tab	70
3.4.3.6.3. Operating Map: General Tab	70
3.4.3.6.4. Limitations of Operating Maps based on Response Points	70
3.4.3.7. Mesh Regions	71
3.4.4. User Locations and Plots	71
3.4.4.1. Wireframe	71
3.4.4.1.1. Wireframe: Definition Tab	71
3.4.4.1.2. Wireframe: View Tab	72
3.4.5. Report	72
3.4.5.1. Omitting Default Report Sections	75
3.4.5.2. Changing the Default Report Sections	76
3.4.5.3. Adding New Sections to a Report	76
3.4.5.4. Report Templates	76
3.4.5.4.1. Turbo Report Templates	77

3.4.5.4.1.1. Procedures for Using Turbo Reports when Turbomachinery Data is Missing	80
3.4.5.4.2. Choosing a Turbo Report	81
3.4.5.5. Creating, Viewing, and Publishing Reports	83
3.4.5.5.1. Report Object	83
3.4.5.5.1.1. Figures: File Type	84
3.4.5.5.1.2. Figures: Figure Size	84
3.4.5.5.1.3. Figures: Width and Height	84
3.4.5.5.1.4. Figures: Fit all figures in the viewport before generation Check Box	84
3.4.5.5.1.5. Charts: File Type	84
3.4.5.5.1.6. Charts: Chart Size	84
3.4.5.5.1.7. Charts: Width and Height	84
3.4.5.5.2. Title Page Object	84
3.4.5.5.2.1. Custom Logo Check Box	84
3.4.5.5.2.2. Custom Logo	84
3.4.5.5.2.3. Ansys Logo Check Box	84
3.4.5.5.2.4. Title	84
3.4.5.5.2.5. Author	85
3.4.5.5.2.6. Current Date Check Box	85
3.4.5.5.2.7. Table of Contents Check Box	85
3.4.5.5.2.8. Table of Contents Check Box: Captions in Table of Contents Check Box	85
3.4.5.5.3. File Report Object	85
3.4.5.5.4. Mesh Report Object	85
3.4.5.5.5. Physics Report Object	85
3.4.5.5.6. Solution Report Object	85
3.4.5.5.7. Operating Points Report Object	86
3.4.5.5.8. Adding Objects to the Report	86
3.4.5.5.9. Controlling the Content in the Report	87
3.4.5.5.10. Refreshing the Report	87
3.4.5.5.11. Viewing the Report	87
3.4.5.5.12. Publishing the Report	87
3.4.5.5.12.1. Format	88
3.4.5.5.12.2. File	88
3.4.5.5.12.3. Figures	88
3.4.5.5.12.4. Save images in separate folder Check Box	88
3.4.5.5.12.5. More Options Button	89
3.4.6. Display Properties and Defaults	89
3.5. Variables Workspace	89
3.5.1. Variables Tree View	89
3.5.2. Variables Details View	91
3.5.2.1. Fundamental Variables	92
3.5.2.1.1. Saving Variables Back to the Results File	92
3.5.2.2. Radius and Theta	93
3.5.2.3. Boundary-Value-Only Variables	93
3.5.2.4. User Variables	93
3.5.3. Variables: Example	95
3.6. Expressions Workspace	96
3.6.1. Expressions Tree View	97
3.6.2. Expressions Workspace: Expressions Details View	97
3.6.2.1. Expression Definition Tab	97
3.6.2.2. Plot Expression Tab	98

3.6.2.3. Evaluate Expression Tab	98
3.6.3. Expressions Workspace: Example	99
3.6.3.1. Further Expressions	100
3.7. Calculators Workspace	100
3.8. Turbo Workspace	100
4. CFD-Post in Ansys Workbench	101
4.1. The Ansys Workbench Interface	101
4.1.1. Toolbox	102
4.1.2. Project Schematic: Introduction	103
4.1.3. Workspace Tabs	104
4.1.4. View Menu	104
4.1.5. Properties Pane	104
4.1.6. Files Pane	106
4.1.7. Sidebar Help	107
4.1.8. Shortcuts (Context Menu Options)	107
4.2. File Operation Differences	107
4.3. An Introduction to Workflow within Ansys CFX in Ansys Workbench	108
4.4. Using Ansys Workbench Journaling and Scripting with CFD-Post	111
4.4.1. Acquiring a Journal File with CFD-Post in Ansys Workbench	111
4.4.1.1. Journal of an Operation That Creates a Plane in CFD-Post	111
4.4.2. Scripting	113
4.4.2.1. Example: Using a Script to Change an Existing Locator	113
4.5. Tips on Using Ansys Workbench	113
4.5.1. General Tips	114
4.5.1.1. Ansys Workbench Interface	114
4.5.1.2. Setting Units	114
4.5.1.3. Files Pane	114
4.5.1.4. Ansys Workbench Connections	114
4.5.2. Tips for Results Systems	114
4.5.2.1. Changes in Behavior	114
4.5.2.2. Duplicating Systems	115
4.5.2.3. Renaming Systems	115
4.5.2.4. Results Cell	115
4.5.2.5. Recovering After Deleting Files	115
4.5.2.6. License Sharing	116
4.6. Limitations When Using Ansys CFD-Post in Ansys Workbench	116
5. CFD-Post 3D Viewer	117
5.1. Object Visibility	117
5.2. 3D Viewer Modes and Commands	119
5.2.1. 3D Viewer Toolbar	119
5.2.2. CFD-Post 3D Viewer Shortcut Menus	121
5.2.2.1. Shortcuts for CFD-Post (Viewer Background)	121
5.2.2.2. Shortcuts for CFD-Post (Viewer Object)	122
5.2.3. Viewer Hotkeys	123
5.2.4. Mouse Button Mapping	124
5.2.5. Picking Mode	126
5.2.5.1. Selecting Objects	126
5.2.5.2. Moving Objects	127
5.3. Views and Figures	127
5.3.1. Creating a Figure	127
5.3.1.1. Copying Objects for Figures	128

5.3.2. Switching to a View or Figure	128
5.3.3. Changing the Definition of a View or Figure	128
5.3.4. Deleting a Figure	128
5.3.5. Views	128
5.3.5.1. Object Visibility	129
5.3.5.2. Legends	130
5.4. Stereo Viewer	130
6. CFD-Post Operating Points Viewer	131
6.1. Adding Filter Rules	133
6.2. Operating Point Parameter Table	135
7. CFD-Post Workflow	137
7.1. Loading and Viewing the Solver Results	137
7.2. Qualitative Displays of Variables	137
7.3. Analysis	138
7.4. Quantitative Analysis of Results	138
7.5. Sharing the Analysis	138
7.6. Typical Workflow	139
8. CFD-Post File Menu	141
8.1. Load Results Command	141
8.2. Close Command	146
8.3. Load State Command	146
8.4. Save State Command and Save State As Command	147
8.5. Save Project Command	147
8.6. Refresh Command (Ansys Workbench only)	147
8.7. Import Commands	147
8.7.1. Import Surface, Line or Point Data into CFD-Post	147
8.7.2. Import Fluent Particle Track File	149
8.7.3. Import Mechanical CDB Surface	149
8.7.3.1. File	150
8.7.3.2. Length Units	150
8.7.3.3. Specify Associated Boundary Check Box	150
8.7.3.3.1. Boundary	150
8.7.3.4. Maintain Conservative Heat Flows Check Box	150
8.7.3.5. Read Mid-Side Nodes Check Box	150
8.7.3.6. Mapping Success Label	151
8.8. Export Commands	151
8.8.1. Export	151
8.8.1.1. Export: Options Tab	152
8.8.1.1.1. File	152
8.8.1.1.2. Type	152
8.8.1.1.3. Locations	152
8.8.1.1.4. Name Aliases	152
8.8.1.1.5. Coord Frame	153
8.8.1.1.6. Unit System	153
8.8.1.1.7. Boundary Data	153
8.8.1.1.8. Export Geometry Information Check Box	153
8.8.1.1.8.1. Line and Face Connectivity Check Box	153
8.8.1.1.8.2. Node Numbers Check Box	153
8.8.1.1.9. Profile Type	153
8.8.1.1.10. Spatial Fields List Box	154
8.8.1.1.11. Select Variable(s) List Box	154

8.8.1.2. Export: Formatting Tab	154
8.8.1.2.1. Vector Variables	154
8.8.1.2.1.1. Vector Display Options	154
8.8.1.2.1.2. Brackets	154
8.8.1.2.2. Include Nodes With Undefined Variable Check Box	154
8.8.1.2.2.1. Null Token	154
8.8.1.2.3. Precision	155
8.8.1.2.4. Separator	155
8.8.1.2.5. Include File Info Header Check Box	155
8.8.1.2.6. Include Header Check Box	155
8.8.1.3. Exporting Polyline Data	155
8.8.1.3.1. POLYLINE Data Format	156
8.8.1.4. Exporting Boundary Profile/Surface Data	156
8.8.1.4.1. USER SURFACE Data Format	156
8.8.2. Export External Data File	157
8.8.2.1. Options Tab	157
8.8.2.1.1. File	157
8.8.2.1.2. Location	157
8.8.2.1.3. Unit System	158
8.8.2.1.4. Boundary Data	158
8.8.2.1.5. Select Recommended Variables	158
8.8.2.1.6. Select Additional Variables	159
8.8.2.2. Formatting Tab	160
8.8.3. Export Mechanical Load File	160
8.8.3.1. Options Tab	161
8.8.3.1.1. File	161
8.8.3.1.2. Location	161
8.8.3.1.3. Unit System	161
8.8.3.1.4. Boundary Data	161
8.8.3.1.5. Export Data	161
8.8.3.1.6. Fluids	162
8.8.3.1.7. Specify Reference Temperature	162
8.8.3.2. Formatting Tab	162
8.9. Mechanical Import/Export Commands	163
8.9.1. Mechanical Import/Export Example: One-Way FSI Data Transfer	163
8.10. FSI with Mechanical APDL and CFX: Manual One-way Mapping	163
8.11. Report Command	165
8.12. Save Picture Command	165
8.13. Loading Recently Accessed Files	169
8.14. Quit Command	169
8.15. File Types Used and Produced by CFD-Post	169
8.15.1. Ansys CFX Files	170
8.15.1.1. Transient Blade Row Postprocessing	172
8.15.1.2. Limitations with Ansys CFX Files	172
8.15.2. Ansys Meshing Files	173
8.15.3. Ansys Mechanical Results Files	173
8.15.3.1. Limitations with Ansys Mechanical Results Files	173
8.15.4. Ansys Icepak Files	175
8.15.5. CGNS Files	176
8.15.6. Common Fluids Format (CFF) Files	176
8.15.6.1. Supported Functionality	177

8.15.6.1.1. Physical Objects (Domains, Boundary Conditions and Subdomains)	177
8.15.6.1.2. Topological Regions	178
8.15.6.1.3. Meshes	178
8.15.6.1.4. Mesh Zones	178
8.15.6.1.5. Steady State and Transient simulations	178
8.15.6.1.6. Multi-configuration (Modified Mesh/Modified Physics)	179
8.15.6.1.7. Single-phase and Multi-phase Solutions	179
8.15.6.1.8. Naming of Variables	179
8.15.6.2. Limitations with Ansys CFF Files	179
8.15.6.2.1. Name Limitations in CFD-Post	181
8.15.6.2.2. Solver Model Limitations in CFD-Post	181
8.15.6.2.3. Transient limitations	181
8.15.6.2.4. Solution Limitations in CFD-Post	181
8.15.6.2.5. Solver Specific Limitations in CFD-Post	181
8.15.6.2.6. CFF-Post File Limitations	182
8.15.7. Ansys Fluent Files	182
8.15.7.1. Ansys Fluent Mesh Files	182
8.15.7.2. Fluent Restart Files	182
8.15.7.3. Fluent Postprocessing Files	183
8.15.7.4. Ansys Fluent Project Files	183
8.15.7.5. Ansys Fluent Particle Track Files	184
8.15.7.6. Summary of Ansys Fluent File Extensions	184
8.15.7.7. Considerations When Naming Geometries in Fluent	184
8.15.7.8. Limitations with Ansys Fluent Files	185
8.15.7.9. Quantitative Differences Between CFD-Post and Fluent	186
8.15.7.10. Limitations in CFD-Post using Fluent Files	192
8.15.8. Ansys Forte Files	194
8.15.9. Ansys FENSAP-ICE Files	194
8.15.9.1. Grid Files	195
8.15.9.2. Solution Files	195
8.15.9.3. Solution Files - Icing	195
8.15.9.4. View Set-up File	195
8.15.9.4.1. Examples	196
8.15.9.5. Limitations	197
8.15.10. CFX-4 Dump Files	197
8.15.10.1. Limitation with CFX-4 Files	197
8.15.10.2. Interpolation of Results	197
8.15.11. CFX-TASCflow Results Files	198
8.15.11.1. Limitations with CFX-TASCflow Files	198
8.15.11.2. Variable Translation	199
9. CFD-Post Edit Menu and Options (Preferences)	201
9.1. Undo and Redo Commands	201
9.2. Options Command	202
9.2.1. CFD-Post Options	203
9.2.1.1. General	203
9.2.1.1.1. Beta Options	204
9.2.1.1.2. Load Options	204
9.2.1.1.3. Advanced	204
9.2.1.2. Files	205
9.2.1.2.1. CFX	205
9.2.1.2.2. FLUENT	206

9.2.1.2.3. CGNS	206
9.2.1.2.4. Variables	207
9.2.1.2.5. CFD-Post Solution Units	207
9.2.1.3. Turbo	208
9.2.1.4. Viewer	208
9.2.1.4.1. Object Highlighting	208
9.2.1.4.2. Background	209
9.2.1.4.2.1. Color	209
9.2.1.4.2.2. Image	209
9.2.1.4.3. Ansys Logo	209
9.2.1.4.4. Text/Edge Color	209
9.2.1.4.5. Axis/Ruler Visibility	209
9.2.1.4.6. Stereo	209
9.2.2. Common Options	209
9.2.2.1. Appearance	210
9.2.2.2. Threading	210
9.2.2.3. Setting the Display Units	210
9.2.2.4. Viewer Setup	211
9.2.2.4.1. Mouse Mapping	212
10. CFD-Post Monitor Menu	213
10.1. Monitor Run in Progress	213
10.2. Start Auto Update	213
10.3. Stop Auto Update	214
10.4. Update Once	214
11. CFD-Post Session Menu	215
11.1. New Session Command	215
11.2. Start Recording and Stop Recording Commands	216
11.3. Play Session Command	216
12. CFD-Post Insert Menu	219
12.1. Location Submenu	220
12.1.1. Point Command	220
12.1.1.1. Point: Geometry Tab	221
12.1.1.1.1. Domains	221
12.1.1.1.2. Definition	221
12.1.1.1.2.1. Method	221
12.1.1.1.2.2. Point	222
12.1.1.1.2.3. Node Number	222
12.1.1.1.2.4. Location	222
12.1.1.1.2.5. Variable	222
12.1.1.1.3. Nearest Node Value	223
12.1.1.2. Point: Color Tab	223
12.1.1.3. Point: Symbol Tab	223
12.1.1.3.1. Symbol	223
12.1.1.3.2. Symbol Size	223
12.1.1.4. Point: Render Tab	223
12.1.1.5. Point: View Tab	223
12.1.2. Point Cloud Command	223
12.1.2.1. Point Cloud: Geometry Tab	224
12.1.2.1.1. Domains	224
12.1.2.1.2. Method	224
12.1.2.1.3. Definition	224

12.1.2.1.3.1. Locations	224
12.1.2.1.3.2. Sampling	225
12.1.2.1.3.3. # of Points	225
12.1.2.1.3.4. Spacing	225
12.1.2.1.3.5. Aspect Ratio	225
12.1.2.1.3.6. Grid Angle	225
12.1.2.1.3.7. Reduction	226
12.1.2.1.3.8. Max Points	226
12.1.2.1.3.9. Factor	226
12.1.2.1.3.10. Seed	226
12.1.2.2. Point Cloud: Color Tab	226
12.1.2.3. Point Cloud: Symbol Tab	226
12.1.2.4. Point Cloud: Render Tab	226
12.1.2.5. Point Cloud: View Tab	227
12.1.3. Line Command	227
12.1.3.1. Line: Geometry Tab	227
12.1.3.1.1. Domains	227
12.1.3.1.2. Definition	227
12.1.3.1.2.1. Method	227
12.1.3.1.2.2. Point 1	227
12.1.3.1.2.3. Point 2	227
12.1.3.1.3. Line Type	228
12.1.3.1.3.1. Cut/Sample Options	228
12.1.3.1.3.2. Samples	228
12.1.3.1.4. Line Translation Using Picking Mode	228
12.1.3.2. Line: Color Tab	228
12.1.3.3. Line: Render Tab	228
12.1.3.4. Line: View Tab	228
12.1.4. Plane Command	229
12.1.4.1. Plane: Geometry Tab	229
12.1.4.1.1. Domains	229
12.1.4.1.2. Definition	229
12.1.4.1.2.1. Method	229
12.1.4.1.2.2. X	230
12.1.4.1.2.3. Y	230
12.1.4.1.2.4. Z	230
12.1.4.1.2.5. Point	230
12.1.4.1.2.6. Normal	230
12.1.4.1.2.7. Point 1, Point 2, and Point 3	230
12.1.4.1.3. Plane Bounds	230
12.1.4.1.3.1. Type	230
12.1.4.1.3.2. Radius	231
12.1.4.1.3.3. X/Y/Z Size	231
12.1.4.1.3.4. X/Y/Z Angle	231
12.1.4.1.3.5. Invert Plane Bound Check Box	231
12.1.4.1.4. Plane Type	231
12.1.4.1.4.1. Slice Option	231
12.1.4.1.4.2. Sample Option	232
12.1.4.1.5. Plane Translation using Picking Mode	232
12.1.4.2. Plane: Color Tab	232
12.1.4.3. Plane: Render Tab	232

12.1.4.4. Plane: View Tab	232
12.1.5. Volume Command	232
12.1.5.1. Volume: Geometry Tab	233
12.1.5.1.1. Domains	233
12.1.5.1.2. Element Types	233
12.1.5.1.3. Definition	233
12.1.5.1.3.1. Method	233
12.1.5.1.3.2. Point	234
12.1.5.1.3.3. Radius	234
12.1.5.1.3.4. Location	234
12.1.5.1.3.5. Variable	234
12.1.5.1.3.6. Hybrid/Conservative Options	234
12.1.5.1.3.7. Mode (for the Sphere and From Surface options)	234
12.1.5.1.3.8. Mode (for the Isovolum option)	235
12.1.5.1.3.9. Value Fields	235
12.1.5.1.3.10. Inclusive Check Box	235
12.1.5.1.3.11. How CFD-Post Calculates Isovolumes	235
12.1.5.1.4. Remove Internal Faces	237
12.1.5.2. Volume: Color Tab	237
12.1.5.3. Volume: Render Tab	237
12.1.5.4. Volume: View Tab	237
12.1.6. Isosurface Command	237
12.1.6.1. Isosurface: Geometry Tab	239
12.1.6.1.1. Domains	239
12.1.6.1.2. Definition	239
12.1.6.1.2.1. Variable	239
12.1.6.1.2.2. Hybrid/Conservative Option	239
12.1.6.1.2.3. Value	239
12.1.6.2. Isosurface: Color Tab	239
12.1.6.3. Isosurface: Render Tab	239
12.1.6.4. Isosurface: View Tab	239
12.1.7. Iso Clip Command	240
12.1.7.1. Iso Clip: Geometry Tab	240
12.1.7.1.1. Domains	240
12.1.7.1.2. Location	240
12.1.7.1.3. Visibility Parameters	241
12.1.7.2. Iso Clip: Color Tab	241
12.1.7.3. Iso Clip: Render Tab	241
12.1.7.4. Iso Clip: View Tab	241
12.1.8. Vortex Core Region	241
12.1.8.1. Vortex Core Region: Geometry Tab	242
12.1.8.1.1. Domains	242
12.1.8.1.2. Definition Area	242
12.1.8.1.2.1. Method	242
12.1.8.1.2.1.1. Vortex Core Mathematics	243
12.1.8.1.2.1.2. Vortex Core References	246
12.1.8.1.2.2. Level	247
12.1.8.1.2.3. Actual Value	247
12.1.8.2. Vortex Core Region: Color Tab	248
12.1.8.3. Vortex Core Region: Render Tab	248
12.1.8.4. Vortex Core Region: View Tab	248

12.1.9. Surface of Revolution Command	248
12.1.9.1. Surface of Revolution: Geometry Tab	249
12.1.9.1.1. Domains	249
12.1.9.1.2. Definition	249
12.1.9.1.2.1. Method	249
12.1.9.1.2.2. Point 1 (a,r) and Point 2 (a,r)	249
12.1.9.1.2.3. Line	249
12.1.9.1.2.4. # of Samples	250
12.1.9.1.2.5. Theta Samples	250
12.1.9.1.2.6. Project to AR Plane Check Box	250
12.1.9.1.3. Rotation Axis	250
12.1.9.1.3.1. Method	250
12.1.9.1.3.2. Axis	250
12.1.9.1.3.3. From/To Text Boxes	250
12.1.9.1.4. Angle Range Check Box	250
12.1.9.1.4.1. Min./Max. Angle	251
12.1.9.1.5. Axial/Radial Offset	251
12.1.9.1.5.1. Start/End A	251
12.1.9.1.5.2. Start/End R	251
12.1.9.2. Surface of Revolution: Color Tab	251
12.1.9.3. Surface of Revolution: Render Tab	251
12.1.9.4. Surface of Revolution: View Tab	251
12.1.10. Polyline Command	252
12.1.10.1. Polyline: Geometry Tab	252
12.1.10.1.1. Method	252
12.1.10.1.2. File	253
12.1.10.1.3. Domains	253
12.1.10.1.4. Boundary List	253
12.1.10.1.5. Intersect With	253
12.1.10.1.6. Contour Name	253
12.1.10.1.7. Contour Level	254
12.1.10.2. Polyline: Color Tab	254
12.1.10.3. Polyline: Render Tab	254
12.1.10.4. Polyline: View Tab	254
12.1.11. User Surface Command	254
12.1.11.1. User Surface: Geometry Tab	255
12.1.11.1.1. Method	255
12.1.11.1.2. File	256
12.1.11.1.3. Domains/Boundary List/Intersect With	256
12.1.11.1.4. Contour Name/Contour Level	257
12.1.11.1.5. Surface Name	257
12.1.11.1.6. Rotation Check Box	257
12.1.11.1.7. Translation Check Box	257
12.1.11.1.8. Scale Check Box	257
12.1.11.1.9. Type	257
12.1.11.1.10. Mode	257
12.1.11.1.11. Distance	258
12.1.11.1.12. Variable	258
12.1.11.1.13. Direction	258
12.1.11.1.14. Specify Associated Boundary Check Box	258
12.1.11.1.15. Length Units	259

12.1.11.2. User Surface: Color Tab	259
12.1.11.3. User Surface: Render Tab	259
12.1.11.4. User Surface: View Tab	259
12.1.12. Surface Group Command	259
12.1.12.1. Surface Group: Geometry Tab	260
12.1.12.1.1. Domains	260
12.1.12.1.2. Locations	260
12.1.12.2. Surface Group: Color Tab	260
12.1.12.3. Surface Group: Render Tab	260
12.1.12.4. Surface Group: View Tab	260
12.1.13. Turbo Surface Command	260
12.1.14. Turbo Line Command	261
12.2. Vector Command	261
12.2.1. Vector: Geometry Tab	262
12.2.1.1. Domains	262
12.2.1.2. Definition	262
12.2.1.2.1. Locations	262
12.2.1.2.2. Sampling	262
12.2.1.2.3. Variable	262
12.2.1.2.4. Hybrid/Conservative Options	262
12.2.1.2.5. Projection	262
12.2.1.2.6. Direction	263
12.2.2. Vector: Color Tab	263
12.2.3. Vector: Symbol Tab	263
12.2.3.1. Symbol	263
12.2.3.2. Symbol Size	264
12.2.3.3. Normalize Symbols Check Box	264
12.2.4. Vector: Render Tab	264
12.2.5. Vector: View Tab	264
12.3. Contour Command	264
12.3.1. Contour: Geometry Tab	265
12.3.1.1. Domains	265
12.3.1.2. Locations	265
12.3.1.3. Variable	265
12.3.1.4. Range	265
12.3.1.5. Variable Location: Vertex and Face Options	265
12.3.1.6. Boundary Data: Hybrid and Conservative Options	266
12.3.1.7. Color Scale	267
12.3.1.8. Color Map	267
12.3.1.9. # of Contours	267
12.3.1.10. Clip to Range Check Box	267
12.3.2. Contour: Labels Tab	267
12.3.2.1. Show Numbers Check Box	267
12.3.2.1.1. Text Height	267
12.3.2.1.2. Text Font	267
12.3.2.1.3. Color Mode	267
12.3.2.1.4. Text Color	268
12.3.3. Contour: Render Tab	268
12.3.4. Contour: View Tab	268
12.4. Streamline Command	268
12.4.1. Streamline: Geometry Tab	269

12.4.1.1. Type	269
12.4.1.2. Definition	269
12.4.1.2.1. Domains	269
12.4.1.2.2. Start From (3D Streamline)	269
12.4.1.2.3. Surfaces	270
12.4.1.2.4. Start From (Surface Streamline)	270
12.4.1.2.5. Locations	270
12.4.1.2.6. Sampling	270
12.4.1.2.7. Preview Seeds Button	270
12.4.1.2.8. Variable	270
12.4.1.2.9. Hybrid/Conservative Options	270
12.4.1.2.10. Direction	270
12.4.1.3. Cross Periodics Check Box	271
12.4.1.4. Simplify Streamline Geometry Check Box	271
12.4.2. Streamline: Color Tab	271
12.4.3. Streamline: Symbol Tab	272
12.4.3.1. Show Symbols Check Box	272
12.4.3.1.1. Min Time	272
12.4.3.1.2. Max Time	272
12.4.3.1.3. Interval	272
12.4.3.1.4. Symbol	273
12.4.3.1.5. Symbol Size	273
12.4.3.2. Show Streams Check Box	273
12.4.3.2.1. Stream Type	273
12.4.3.2.2. Line Width/Tube Width/Ribbon Width	273
12.4.3.2.3. # of Sides	273
12.4.3.2.4. Initial Direction	273
12.4.4. Streamline: Limits Tab	273
12.4.4.1. Step Tolerance	273
12.4.4.1.1. Mode	274
12.4.4.1.2. Tolerance	274
12.4.4.2. Upper Limits	274
12.4.4.2.1. Max Segments	274
12.4.4.2.2. Max Time	274
12.4.4.2.3. Max Periods	274
12.4.5. Streamline: Render Tab	274
12.4.6. Streamline: View Tab	274
12.5. Particle Track Command	275
12.5.1. Particle Track: Geometry Tab	276
12.5.1.1. Method	276
12.5.1.1.1. Domains	276
12.5.1.1.2. Material	276
12.5.1.2. File	276
12.5.1.3. Injections	276
12.5.1.4. Reduction Type	276
12.5.1.4.1. Reduction	277
12.5.1.4.2. Max Tracks	277
12.5.1.5. Limits Option	277
12.5.1.5.1. Limit Type and Start/End <variable>	277
12.5.1.6. Filter Check Box	277
12.5.1.6.1. Start/End Region Check Boxes	278

12.5.1.6.2. Diameter Check Box	278
12.5.1.6.3. Track Check Box	278
12.5.1.6.4. Match ALL/Match ANY Options	278
12.5.2. Particle Track: Color Tab	278
12.5.3. Particle Track: Symbol Tab	278
12.5.3.1. Show Symbols Check Box	278
12.5.3.1.1. Max Time is	279
12.5.3.2. Show Tracks Check Box	279
12.5.3.3. Show Track Numbers Check Box	279
12.5.4. Particle Track: Render Tab	279
12.5.5. Particle Track: View Tab	279
12.5.6. Particle Track: Info Tab	280
12.6. Volume Rendering Command	280
12.6.1. Volume Rendering: Geometry Tab	281
12.6.2. Volume Rendering: Color Tab	282
12.6.3. Volume Rendering: Render Tab	282
12.6.4. Volume Rendering: View Tab	283
12.7. Text Command	283
12.7.1. Text: Definition Tab	283
12.7.1.1. Text String	283
12.7.1.2. Embed Auto Annotation Check Box	284
12.7.1.2.1. Type	284
12.7.1.2.2. Expression	284
12.7.1.2.3. Format (for Filename option)	284
12.7.1.2.4. Format (for the File Date and File Time options)	284
12.7.1.2.5. Determine the number formatting automatically Check Box	284
12.7.1.3. More/Fewer Buttons	285
12.7.2. Text: Location Tab	285
12.7.2.1. Location	285
12.7.2.1.1. Position Mode	285
12.7.2.1.2. X Justification	285
12.7.2.1.3. Y Justification	285
12.7.2.1.4. Position (for Two Coords option)	285
12.7.2.1.5. Position (for Three Coords option)	285
12.7.2.1.6. Rotation	285
12.7.3. Text: Appearance Tab	285
12.7.3.1. Height	285
12.7.3.2. Color Mode	286
12.7.3.3. Font	286
12.8. Coordinate Frame Command	286
12.8.1. Coordinate Frame: Definition Tab	286
12.8.1.1. Type	286
12.8.1.2. Origin	286
12.8.1.3. Z Axis Point	286
12.8.1.4. X-Z Plane Pt	287
12.8.1.5. Symbol Size	287
12.8.1.6. Coordinate Frame Details	287
12.9. Legend Command	288
12.9.1. Default Legends	289
12.9.2. User-defined Legends	289
12.9.3. Legend: Definition Tab	289

12.9.3.1. Plot	289
12.9.3.2. Title Mode	289
12.9.3.3. Title	289
12.9.3.4. Show Legend Units Check Box	290
12.9.3.5. Vertical / Horizontal Options	290
12.9.3.6. Location	290
12.9.3.6.1. X Justification	290
12.9.3.6.2. Y Justification	290
12.9.3.6.3. Position	290
12.9.4. Legend: Appearance Tab	291
12.9.4.1. Sizing Parameters	291
12.9.4.1.1. Size	291
12.9.4.1.2. Aspect	291
12.9.4.2. Text Parameters	291
12.9.4.2.1. Precision	291
12.9.4.2.2. Value Ticks	291
12.9.4.2.3. Font	291
12.9.4.2.4. Color Mode	291
12.9.4.2.5. Color	291
12.9.4.2.6. Text Rotation	291
12.9.4.2.7. Text Height	292
12.10. Instance Transform Command	292
12.10.1. Default Transform Object	292
12.10.2. Instance Transform: Definition Tab	293
12.10.2.1. Instancing Info From Domain Check Box	293
12.10.2.2. Number of Graphical Instances	293
12.10.2.3. Apply Rotation Check Box	293
12.10.2.3.1. Method	293
12.10.2.3.2. Axis	293
12.10.2.3.3. From/To Fields	293
12.10.2.3.4. Full Circle Check Box	294
12.10.2.3.5. Determine Angle From	294
12.10.2.3.6. Number of Passages	294
12.10.2.3.7. Passages per Component	294
12.10.2.3.8. Angle	294
12.10.2.4. Apply Translation Check Box	294
12.10.2.4.1. Translation	294
12.10.2.5. Apply Reflection Check Box	294
12.10.2.5.1. Method	294
12.10.2.5.2. X/Y/Z	295
12.10.2.5.3. Plane	295
12.10.3. Instance Transform: Example	295
12.11. Clip Plane Command	296
12.11.1. Clip Plane: Geometry Tab	297
12.11.1.1. Definition	297
12.11.1.1.1. Method	297
12.11.1.1.2. Slice Plane	297
12.11.1.2. Flip Normal Check Box	297
12.12. Color Map Command	298
12.13. Variable Command	300
12.14. Expression Command	300

12.15. Table Command	300
12.15.1. Editing in the Table Viewer	300
12.15.1.1. Shortcut Menu	301
12.15.1.2. Expressions	302
12.16. Chart Command	305
12.16.1. Creating a Chart Object	305
12.16.1.1. Chart: General Tab	306
12.16.1.1.1. Type	306
12.16.1.1.2. Display Title: Title	306
12.16.1.1.3. Report: Caption	307
12.16.1.1.4. Fast Fourier Transform	307
12.16.1.1.4.1. Fast Fourier Transform (FFT) Theory	307
12.16.1.1.4.1.1. Windowing in Fast Fourier Transforms	308
12.16.1.1.4.1.2. Using Fast Fourier Transforms	309
12.16.1.1.5. Refresh Settings	309
12.16.1.2. Chart: Data Series Tab	310
12.16.1.2.1. Name Controls	310
12.16.1.2.2. Data Source	311
12.16.1.2.2.1. Data Source File Format	311
12.16.1.2.3. File Variable Selection	312
12.16.1.2.4. Custom Data Selection Controls	313
12.16.1.2.5. Monitor Variable Selection	313
12.16.1.3. Chart: X Axis Tab	314
12.16.1.3.1. X Axis Data Selection	314
12.16.1.3.1.1. Specifying an X Function	314
12.16.1.3.2. Category Divisions	315
12.16.1.3.3. Axis Range	315
12.16.1.3.4. Axis Number Formatting	316
12.16.1.3.5. Axis Labels	316
12.16.1.4. Chart: Y Axis Tab	316
12.16.1.4.1. Y Axis: Data Selection	316
12.16.1.5. Chart: Line Display Tab	319
12.16.1.5.1. Line Type and FFT Line Type Options	320
12.16.1.5.2. Fill Area Controls	323
12.16.1.6. Chart: Chart Display Tab	323
12.16.1.6.1. Display Legend Area	324
12.16.1.6.2. Sizes Area	324
12.16.1.6.3. Fonts Area	324
12.16.1.6.4. Grid Area	324
12.16.2. Viewing a Chart	324
12.16.3. Example: Charting a Velocity Profile	325
12.16.4. Example: Comparing Differences Between Two Files	326
12.17. Comment Command	327
12.18. Figure Command	328
13. CFD-Post Tools Menu	329
13.1. Timestep Selector	329
13.1.1. Adding Timesteps	331
13.1.2. Using the Timestep Selector with Transient Blade Row Cases	331
13.1.3. Multiple Files	333
13.2. Animation	334
13.2.1. Animation Types	334

13.2.1.1. Sweep Animation	334
13.2.1.1.1. Animating Planes	335
13.2.1.1.2. Animating Isosurfaces	335
13.2.1.1.3. Animating Turbo Surfaces	336
13.2.1.1.4. Animating Streamlines and Particle Tracks	336
13.2.1.1.5. Animating Mesh Deformation Scaling	336
13.2.1.2. Timestep Animation	336
13.2.1.3. GPU Accelerated Animation	337
13.2.1.4. Keyframe Animation	340
13.2.1.4.1. Creating an Animation	340
13.2.1.4.2. Animating Expressions	341
13.2.2. Animation Dialog Box	341
13.2.2.1. Animation Options Dialog Box: Options Tab	342
13.2.2.1.1. Animation Speed	342
13.2.2.1.2. Transient Case	343
13.2.2.1.3. Print Options	343
13.2.2.1.3.1. Image Format	343
13.2.2.1.3.2. White Background	343
13.2.2.1.3.3. Enhanced Output (Smooth Edges)	343
13.2.2.1.3.4. Image Size	343
13.2.2.1.3.5. Tolerance	343
13.2.2.2. Animation Options Dialog Box: Advanced Tab	343
13.2.2.2.1. Save Frames As Image Files	343
13.2.2.2.2. Output To User Directory	344
13.2.2.2.3. Frame Rate	344
13.2.2.2.4. Quality	344
13.2.2.2.5. Don't Encode Last MPEG Frame	344
13.2.3. Saving an Animation	344
13.2.4. Saving the Animation State (*.can file)	344
13.3. Quick Editor	345
13.4. Probe	345
13.5. Function Calculator	347
13.5.1. Function Selection	348
13.6. Macro Calculator	350
13.6.1. Running Macros from the Macro Calculator	351
13.6.2. Macro Availability	352
13.6.3. Predefined Macros	352
13.6.3.1. Comfort Factors Macro	353
13.6.3.2. Cp Polar Plot Macro	354
13.6.3.3. Gas Compressor Performance Macro	355
13.6.3.4. Gas Turbine Performance Macro	356
13.6.3.5. Liquid Pump Performance Macro	357
13.6.3.6. Liquid Turbine Performance Macro	358
13.6.3.7. Fan Noise Macro	359
13.6.3.7.1. Using the Fan Noise Macro	361
13.6.3.7.1.1. Fan Noise Theory in Brief	361
13.6.3.7.1.2. Fan Noise Macro Input	364
13.6.3.7.1.3. Fan Noise Output (Reports)	364
13.6.3.7.1.4. Fan Noise Examples	365
13.6.4. User-defined Macros	366
13.6.4.1. Writing a Macro	366

13.6.4.2. Macro GUI Definition	368
13.7. Mesh Calculator	371
13.7.1. Mesh Visualization Advice	372
13.8. Case Comparison	373
13.8.1. Calculating Difference Variables	376
13.9. Command Editor	377
14. CFD-Post Help Menu	379
15. Turbo Workspace	381
15.1. Visual Representation of Initialization Status	382
15.2. Define/Modify Global Rotation Axis	382
15.3. Turbo Initialization	382
15.3.1. Requirements for Initialization	383
15.3.2. Initialize All Components	383
15.3.3. Uninitializing Components	384
15.3.4. Individual Component Initialization (Advanced Feature)	384
15.3.5. Details View for Individual Component Initialization	385
15.3.5.1. Definition Tab	385
15.3.5.1.1. Turbo Regions Frame	385
15.3.5.1.2. Background Mesh Frame	386
15.3.5.1.2.1. Purpose of Background Mesh	386
15.3.5.1.2.2. Requirements for Setting Up a Background Mesh	386
15.3.5.1.2.3. Types of Background Mesh	386
15.3.5.1.2.4. Density of the Background Mesh	387
15.3.5.2. Instancing Tab	387
15.4. Turbo View Shortcuts	388
15.5. Turbo Surface	388
15.5.1. Turbo Surface: Geometry	389
15.5.1.1. Domains	389
15.5.1.2. Definition	389
15.5.1.3. Bounds	390
15.5.1.4. Type	391
15.5.2. Turbo Surface: Common Tabs	391
15.5.2.1. Turbo Surface: Color	391
15.5.2.2. Turbo Surface: Render	391
15.5.2.3. Turbo Surface: View	391
15.6. Turbo Line	391
15.6.1. Turbo Line: Geometry	391
15.6.2. Turbo Line: Common Tabs	392
15.6.2.1. Turbo Line: Color	392
15.6.2.2. Turbo Line: Render	392
15.6.2.3. Turbo Line: View	392
15.7. Turbo Plots	393
15.7.1. Introduction to Turbo Plots	393
15.7.1.1. Show Faces/Show Mesh Lines	393
15.7.1.2. Graphical Instancing	393
15.7.1.3. Turbo Measurements	393
15.7.1.3.1. Span	393
15.7.1.3.2. Span Direction	393
15.7.1.3.3. Span Normalized	393
15.7.1.3.4. Streamwise Location	393
15.7.1.3.5. Theta	394

15.7.1.3.6. Advanced: Position of Zero Theta	394
15.7.2. Initialization Three Views	394
15.7.3. 3D View Object	395
15.7.4. Blade-to-Blade Object	395
15.7.4.1. Span	395
15.7.4.2. Angular Shift	395
15.7.4.3. Plot Type	396
15.7.4.3.1. Color	396
15.7.4.3.2. Contour	396
15.7.4.3.3. Vector	396
15.7.4.3.4. Stream	396
15.7.5. Meridional Object	396
15.7.6. Turbo Charts	397
15.7.6.1. Blade Loading Turbo Charts	397
15.7.6.2. Circumferential Turbo Charts	397
15.7.6.3. Hub to Shroud Turbo Charts	398
15.7.6.3.1. Single Line vs. Two Lines	398
15.7.6.3.2. Display	398
15.7.6.3.3. Mode	398
15.7.6.3.4. Point Type	399
15.7.6.3.5. Theta	400
15.7.6.3.6. Samples	400
15.7.6.3.7. Streamwise	400
15.7.6.3.8. Distribution	400
15.7.6.3.9. X/Y Variable	404
15.7.6.3.10. Circumferential Averaging by Length: Hub to Shroud Turbo Chart	404
15.7.6.3.11. Circumferential Averaging by Area: Hub to Shroud Turbo Chart	405
15.7.6.3.12. Circumferential Averaging by Mass Flow: Hub to Shroud Turbo Chart	405
15.7.6.3.13. Constant Blade Aligned Linear Coordinates	405
15.7.6.3.14. Constant Blade Aligned Coordinates	406
15.7.6.4. Inlet to Outlet Turbo Charts	407
15.7.6.4.1. Circumferential Averaging by Length: Inlet to Outlet Turbo Chart	407
15.7.6.4.2. Circumferential Averaging by Area or Mass: Inlet to Outlet Turbo Chart	407
15.8. Turbo Macros	407
15.9. Calculate Velocity Components	408
15.9.1. Calculating Cylindrical Velocity Components for Non-turbo Cases	414
16. CFX Command Language (CCL) in CFD-Post	417
16.1. Object Creation and Deletion	417
17. CFX Expression Language (CEL) in CFD-Post	419
17.1. Variables Created by CFD-Post	420
17.2. User Functions in CFD-Post	420
18. Command Actions	421
18.1. Overview of Command Actions	421
18.2. File Operations from the Command Editor Dialog Box	422
18.2.1. Loading a Results File	422
18.2.1.1. load Command Examples	423
18.2.2. Reading Session Files	423
18.2.2.1. readsession Command Examples	423
18.2.3. Saving State Files	424
18.2.3.1. savestate Command Examples	424
18.2.4. Reading State Files	425

18.2.4.1. readstate Option Actions	426
18.2.4.2. readstate Command Examples	426
18.2.5. Solution Monitoring	427
18.2.6. Creating a Hardcopy	428
18.2.7. Importing External File Formats	428
18.2.8. Exporting Data	428
18.2.9. Controlling the Viewer	429
18.3. Quantitative Calculations in the Command Editor Dialog Box	430
18.4. Other Commands	430
18.4.1. Deleting Objects	430
18.4.2. Viewing a Chart	430
18.4.3. Turbo Post CCL Command Actions	431
18.4.3.1. Calculating Velocity Components	431
18.4.3.2. Initializing all Turbo Components	431
19. Line Interface Mode	433
19.1. Features Available in Line Interface Mode	434
20. Fluent Field Variables Listed by Category	437

List of Figures

3.1. Sample CFD-Post Interface	45
3.2. A Sample Report, Part 1	73
3.3. A Sample Report, Part 2	74
3.4. A Sample Report, Part 3	75
5.1. Mouse Mapping using Workbench Defaults	125
5.2. Viewport Control	127
6.1. Operating Points Viewer	132
12.1. Sample Table Formatting	304
12.2. A-, B-, and C-weighting Functions	317
12.3. Chart made with lines	321
12.4. Chart made with bars	322
12.5. Chart made with steps	323
13.1. Relative position of the source and the observer	362
15.1. Sampling Point Distribution with Include Boundary Nodes Option	404
15.2. Circumferential Averaging by Length	405
15.3. Blade Aligned Linear Coordinates	406
15.4. Blade Aligned Coordinates	406
15.5. Inlet to Outlet Sample Points	407
15.6. Axial, Radial, Circumferential, and Meridional Velocity Components	410
15.7. Velocity Components in Meridional Plane	411
15.8. Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components	412
15.9. Velocity Components in Blade-To-Blade Plane	413
15.10. Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane	414

List of Tables

5.1. Mouse Operations and Shortcuts	125
12.1. Shortcut Menus Toolbar	301
12.2. Table Viewer Tools Toolbar	303
12.3. Octave Band Frequencies and Weightings	315
15.1. Generated Variables	408
17.1. Variables Created by CFD-Post	420
20.1. Pressure and Density Categories	439
20.2. Velocity Category	439
20.3. Temperature, Radiation, and Solidification/Melting Categories	440
20.4. Turbulence Category	441
20.5. Species, Reactions, Pdf, and Premixed Combustion Categories	442
20.6. NOx, Soot, and Unsteady Statistics Categories	443
20.7. Phases, Discrete Phase Model, Granular Pressure, and Granular Temperature Categories	444
20.8. Properties, Wall Fluxes, User Defined Scalars, and User Defined Memory Categories	445
20.9. Cell Info, Grid, and Adaption Categories	446
20.10. Grid Category (Turbomachinery-Specific Variables) and Adaption Category	447
20.11. Residuals Category	448
20.12. Derivatives Category	448
20.13. Acoustics Category	449
20.14. Sensitivities Category	450
20.15. Film Category	450

Preface

The following topics are discussed:

1. About this Manual
2. Document Conventions

1. About this Manual

This manual contains the following chapters:

- [Overview of CFD-Post \(p. 31\)](#) describes CFD-Post functionality and advanced features.
- [Starting CFD-Post \(p. 35\)](#) describes how to start CFD-Post and the environment variables that affect how CFD-Post operates.
- [CFD-Post Graphical Interface \(p. 45\)](#) describes the CFD-Post interface.
- [CFD-Post in Ansys Workbench \(p. 101\)](#) describes how to run CFD-Post in Workbench, including workflow, journaling and scripting, tips, and limitations.
- [CFD-Post 3D Viewer \(p. 117\)](#) describes how to use the CFD-Post **3D Viewer**.
- [CFD-Post Workflow \(p. 137\)](#) describes common ways to use CFD-Post.
- [CFD-Post File Menu \(p. 141\)](#) describes the functionality available from the **File** menu and the file types that CFD-Post supports.
- [CFD-Post Edit Menu and Options \(Preferences\) \(p. 201\)](#) describes the functionality available from the **Edit** menu, such as customizing CFD-Post by setting your preferences on the **Options** dialog box.
- [CFD-Post Monitor Menu \(p. 213\)](#) describes how to view solution data while the solver is running.
- [CFD-Post Session Menu \(p. 215\)](#) describes how to record and replay session files. (Session files contain a record of the commands issued during a CFD-Post session.)
- [CFD-Post Insert Menu \(p. 219\)](#) describes how to create new objects (such as locators, tables, charts, and so on), variables, and expressions by using the **Insert** menu.
- [CFD-Post Tools Menu \(p. 329\)](#) describes how to use the CFD-Post **Tools** menu, which offers access to quantitative analysis utilities, the animation editor and the timestep selector.
- [CFD-Post Help Menu \(p. 379\)](#) describes the commands available in the **Help** menu.
- [Turbo Workspace \(p. 381\)](#) describes how to use the CFD-Post **Turbo** workspace, which improves and speeds up post-processing for turbomachinery simulations.
- [CFX Command Language \(CCL\) in CFD-Post \(p. 417\)](#) describes how to use the CFX Command Language (CCL) and the CFX Expression Language (CEL).
- [CFX Expression Language \(CEL\) in CFD-Post \(p. 419\)](#) describes the CFX Expression Language (CEL) in detail.

- [Command Actions \(p. 421\)](#) describes how to use commands in CFD-Post to edit or create graphic objects and to perform basic input and output operations.
- [Line Interface Mode \(p. 433\)](#) describes running CFD-Post in line-interface mode (that is, without using a user interface, but with a viewer that shows objects you create on the command line) and in batch mode (where a viewer is not provided and you cannot enter commands at a command prompt).
- [Fluent Field Variables Listed by Category \(p. 437\)](#) describes how to convert variable names in a Fluent file to CFX variable names for use in CFD-Post.

2. Document Conventions

This section describes the conventions used in this document to distinguish between text, filenames, system messages, and input that you need to type.

File and Directory Names

Filenames and directory names appear in a plain fixed-width font (for example, `/usr/lib`). On Linux, directory names are separated by forward slashes (/), but on Windows, backslashes are used (\). For example, a directory name on Linux might be `/CFX/bin` whereas on a Windows system, the same directory would be named `\CFX\bin`.

User Input

Input to be typed verbatim is shown using the following convention:

```
mkdir /usr/local/cfx
```

Input Substitution

Input substitution is shown using the following convention:

```
cfx5post -batch <batch_file>
```

you should actually type `cfx5post -batch` and substitute a batch-file name for `<batch_file>`.

Optional Arguments

Optional arguments are shown using square brackets:

```
cfx5export -cgns [-verbose] <file>
```

Here the argument `-verbose` is optional, but you must specify a suitable filename.

Long Commands

Commands that are too long to display on a printed page are shown with "\" characters at the ends of intermediate lines:

```
cfx5export -cgns [-boundary] [-corrected] [-C] \  
[-domain <number>] [-geometry] [-help] [-name <file>] \  
[-summary] [-timestep <number>] [-user <level>] [-norotate] \  
[-boundaries-as-nodes] [-boundaries-as-faces] [-verbose] <file>
```


On a Linux system, you may type the "\" characters, pressing **Enter** after each. However, on a Windows machine you must enter the whole command without the "\" characters; continue typing if the command is too long to fit in the command prompt window and press **Enter** only at the end of the complete command.

<CFDPOSTROOT>

The installation directory for CFD-Post, which differs depending on whether it installed with Ansys CFX. The default installation directory for CFD-Post without Ansys CFX is:

```
C:\Program Files\ANSYS Inc\v251\CFD-Post
```

Operating System Names

When we refer to objects that depend on the type of system being used, we will use one of the following symbols in the text:

<os> refers to the short form of the name that CFX uses to identify the operating system in question. <os> will generally be used for directory names where the contents of the directory depend on the operating system but do not depend on the release of the operating system or on the processor type. Wherever you see <os> in the text you should substitute with the operating system name. The correct value can be determined by running:

```
<CFDPOSTROOT>/bin/cfx5info -os
```

<arch> refers to the long form of the name that CFX uses to identify the system architecture in question. <arch> will generally be used for directory names where the contents of the directory depend on the operating system and on the release of the operating system or the processor type. Wherever you see <arch> in the text you should substitute the appropriate value for your system, which can be determined by running the command:

```
<CFDPOSTROOT>/bin/cfx5info -arch
```

2.1. Spelling Conventions

Ansys CFX documentation uses American spelling:

- atomization rather than atomisation
- color rather than colour
- customization rather than customisation
- discretization rather than discretisation
- initialization rather than initialisation
- meter rather than metre
- normalization rather than normalisation
- vapor rather than vapour
- vaporization rather than vaporisation

When searching, use American spellings:

For:	Search for:
Colour Map	Color Map (or try Color Map Command (p. 298))
Colour Mode	Color Mode (or try Color Mode (p. 267))
Colour Scale	Color Scale (or try Color Scale (p. 53))
Colour Tab	Color Tab (or try Color Tab (p. 52))
Turbo Initialisation	Turbo Initialization (or try Turbo Initialization (p. 382))
Auto-initialise	Auto-initialize (or try Requirements for Initialization (p. 383))
Uninitialise	Uninitialize (or try Uninitializing Components (p. 384))
Initialise All Components	Initialize All Components (or try Initialize All Components (p. 383))
Undefined Colour	Undefined Color (or try Undefined Color (p. 55))
Synchronise Camera	Synchronize Camera (or try Case Comparison (p. 373))

Chapter 1: Overview of CFD-Post

CFD-Post is a flexible, state-of-the-art postprocessor. It is designed to enable easy visualization and quantitative analysis of the results of CFD simulations.

This chapter describes:

- 1.1. CFD-Post Features and Functionality
- 1.2. 3Dconnexion Product Support
- 1.3. Compatibility with File Hosting Services
- 1.4. Advanced Features
- 1.5. Next Steps...

1.1. CFD-Post Features and Functionality

CFD-Post has the following features:

- A graphical user interface that includes a viewer pane in which all graphical output from CFD-Post is plotted. For details, see [CFD-Post Graphical Interface \(p. 45\)](#) and [CFD-Post 3D Viewer \(p. 117\)](#).
- Support for a variety of graphical and geometric objects used to create postprocessing plots, to visualize the mesh, and to define locations for quantitative calculation. For details, see [CFD-Post Insert Menu \(p. 219\)](#).

You can perform a variety of exact quantitative calculations over objects; for details, see [Quantitative Calculations in the Command Editor Dialog Box \(p. 430\)](#).

- Scalar and vector user-defined variables.
- Variable freezing (for comparison with other files).
- Postprocessing capability for turbomachinery applications. For details, see [Turbo Workspace \(p. 381\)](#).
- Standard interactive viewer controls (rotate, zoom, pan, zoom box), multiple viewports, stored views/figures.
- Extensive reports, including charting (XY, time plots). For details, see [Report \(p. 72\)](#).

CFD-Post includes the following functionality:

- Reads:
 - CFX-Solver results files (* .res, * .mres)
 - CFX-Solver transient results files (* .trn)

- CFX-Solver backup results files (*.bak)
- CFX-Solver error results files (*.res, *.err)
- CFX-Solver input files (*.def, *.mdef)
- CFX case files (*.cfx)
- CFX-Mesh files (*.gtm)
- CFD-Post session files (*.cse)
- CFD-Post state files (*.cst)
- Fluent files (*.flprj, *.cas.h5, *.cas, *.cas.gz, *.dat.h5, *.dat, *.dat.gz, *.cdat, *.cdat.gz)
- Fluent mesh files (*.msh, *.msh.gz)
- Icepak files (*.cas, *.dat)
- FENSAP-ICE mesh and solution files (*.grd, *.soln, *.droplet, *.crystal, *.swimsol)
- FENSAP-ICE view set-up files (*.fsp)
- Ansys files (*.rst (deprecated), *.rth (deprecated), *.rmg (deprecated), *.inn, *.inp, *.cdb)
- Ansys Meshing files (*.cldb, *.dsdb)
- Forte files (*.ftind)
- CGNS files (*.cgns, *.cgs)
- CFX-4 dump files (*.d*mp*)
- CFX-TASCflow files (*.lun, *.grd, *.rso)

Note:

CFX-Solver results files are necessary to access some of the quantitative functionality that CFD-Post can provide.

The supported file types are described in [File Types Used and Produced by CFD-Post \(p. 169\)](#).

- Supports transient data, including moving mesh. Node locations are repositioned based on the position for the current timestep.
- Imports/exports Ansys data, generic data, and generic geometry.
- Supports macros through an embedded user interface (see [Macro Calculator \(p. 350\)](#)).
- Outputs to various image types, including PostScript, JPEG, PNG, PPM, VRM, as well as animation (keyframe) and MPEG files. For details, see [Sweep Animation \(p. 334\)](#).

1.2. 3Dconnexion Product Support

See the [Platform Support section of the Ansys Website](#) for a complete list of 3Dconnexion products certified with the current release of Ansys applications.

Note:

Note: The **3D Viewer** in CFX/CFD-Post/TurboGrid does not support the buttons of a 3Dconnexion device. However it might be possible to use the software provided with the device to configure the buttons so that they send key sequences that trigger **3D Viewer** actions.

1.3. Compatibility with File Hosting Services

Files written by Ansys products do not support synchronization with Microsoft's OneDrive file hosting service.

1.4. Advanced Features

CFD-Post also contains advanced features:

CFX Command Language (CCL)

CCL is the internal command language used within CFD-Post. CCL is used to create objects or perform actions. CFD-Post enables command line, session file, or state file input through the CFX Command Language (CCL). For details, see [CFX Command Language \(CCL\) in CFD-Post \(p. 417\)](#).

CFX Expression Language (CEL)

CEL is a powerful expression language used to create user-defined variables, expressions, and so on. For details, see [CFX Expression Language \(CEL\)](#).

Power Syntax

Power Syntax provides integration of the Perl programming language with CCL to enable the creation of advanced subroutines. For details, see [Power Syntax in Ansys CFX](#).

Batch Mode

CFD-Post can be run in batch mode (often using a session file as the basis for a series of actions that will be executed). For details, see [Running in Batch Mode \(p. 42\)](#).

1.5. Next Steps...

Now that you have an overview of the capabilities of CFD-Post, you may want to explore:

- [Starting CFD-Post \(p. 35\)](#)
- [CFD-Post Graphical Interface \(p. 45\)](#).

Chapter 2: Starting CFD-Post

This chapter describes how to start CFD-Post and the environment variables that affect how CFD-Post operates:

- 2.1. Starting CFD-Post with the Ansys CFX Launcher
- 2.2. Starting CFD-Post from the Command Line
- 2.3. Setting CFD-Post Operation Through Environment Variables
- 2.4. Running in Batch Mode

Note:

- You can also start CFD-Post from other Ansys products; for details, refer to the documentation that comes with those products.
- For CFX, TurboGrid and CFD-Post applications, the graphics viewer, which is based on OpenGL 4.5, will work with Microsoft Windows Remote Desktop Connection only if you are using Nvidia Quadro graphics cards with appropriate drivers for Windows 10. If you see an empty (black) viewer, try setting the environment variable `CFX_QT_OPENGL=desktop` on the remote machine, *in the local environment of the intended application* (for example, in a shell or script that launches the application), before starting the application. Consider adding this environment variable to the `cfx5rc.txt` user's configuration file. To work around this issue you may launch the CFX application prior to making the remote connection, or use one of the other Ansys-supported remote display methods.

2.1. Starting CFD-Post with the Ansys CFX Launcher

CFD-Post is installed with the Ansys CFX Launcher, which makes it easy to run CFD-Post. The launcher enables you to:

- Set the working directory for your project.
- Launch CFD-Post and, if available, other Ansys products.
- Access various other tools, including a command window that enables you to run Ansys CFX utilities without having to type the path to the executable.
- Access the online help and other useful information.
- Customize the behavior of the launcher to start your own applications.

You can run the launcher in any of the following ways:

- On Windows:

- From the **Start** menu, select **Ansys 2025 R1 > CFX 2025 R1**.
- In a Command Prompt that has its path set up correctly to run Ansys CFX, enter: `cfx5`

If the path has not been set, you need to type the full path to the `cfx5` command; typically this is:

```
C:\Program Files\ANSYS Inc\v251\CFX\bin\cfx5.exe
```

- On Linux, open a terminal window that has its path set up to run Ansys CFX and enter: `cfx5`

If the path has not been set, you need to type the full path to the `cfx5` command; typically this is:

```
/usr/ansys_inc/v251/CFX/bin/cfx5.exe
```

When the launcher starts, set your working directory and click the **CFD-Post** icon.

Note:

The launcher automatically searches for CFD-Post and other Ansys products, including the license manager.

2.1.1. Valid Syntax in CFD-Post

Valid Syntax for Named Objects

The names of objects must be no more than 80 characters in length. Any of the following characters are allowed to name new objects: A-Z a-z 0-9 <space> (however, the first character must be A-Z or a-z). Multiple spaces are treated as single space characters, and spaces at the end of a name are ignored. In general, object names must be unique within the physics setup.

Valid Decimal Separator

In Ansys CFD-Post, only a period is allowed to be used as a decimal delimiter in fields that accept floating-point input. Depending on your system configuration, fields that accept numeric input will either accept a comma but return an error, or not accept a comma at all.

Ansys Workbench accepts commas as decimal delimiters, but translates these to periods when passing data to CFD-Post.

Note:

CFD-Post will not function correctly on a Linux system with the environment variable "LC_ALL" set to a locale which uses a comma delimiter. The environment variable "LANG" can usually be used to set the locale instead.

Valid Network Path

UNC paths are not supported in CFD-Post. You should use drive letters when opening CFD-Post over a network installation.

2.2. Starting CFD-Post from the Command Line

You may want to start CFD-Post from the command line rather than by clicking the appropriate button on the Ansys CFX Launcher for the following reasons:

- You may want to specify certain command-line arguments so that CFD-Post starts up in a particular configuration. For details, see [Optional Command Line Arguments \(p. 37\)](#).
- CFX contains some utilities (for example, a parameter editor) that can be run only from the command line.
- If you are having problems with CFD-Post, you may be able to get a more detailed error message by starting it from the command line than you would get if you started it from the launcher. When you start CFD-Post from the command line, any error messages produced are written to the command-line window.

To start CFD-Post from the command line, enter:

Windows	<CFDPOSTROOT>\bin\cfdpst
Linux	<CFDPOSTROOT>/bin/cfdpst

2.2.1. Optional Command Line Arguments

The table that follows summarizes the most common of the optional command line arguments:

Argument	Description
-batch-gpu -batch-gpu-rendering -batch -batch-software-rendering <file name>.cse [<results file 1>] [<results file 2> ...]	Starts CFD-Post in batch mode, running the session file you enter as an argument. Options -batch-gpu and -batch-gpu-rendering are equivalent. With either of these options, GPU rendering is used unless compatible GPU hardware is not found, in which case all saved pictures are instead software rendered (by the CPU). GPU rendering uses compatible graphics hardware to produce images quickly with no loss of quality. These options apply to all saved pictures produced by playing the session file. Options -batch and -batch-software-rendering are equivalent. With either of these options, software rendering is used. Software rendering produces images without requiring graphics hardware. Software rendering can be slower than GPU rendering and does not correctly render some graphics objects (that is, OpenGL shader graphics). These options apply to all saved pictures produced by playing the session file. Note that software rendering does not support transparent backgrounds in saved PNG images.
-gui	Starts CFD-Post in graphical user interface (GUI) mode (the default).
-line	Starts CFD-Post in line interface (CFD-Post command line) mode. This interface will start a command line prompt where you can type CCL commands. Typically you would create a session file using the user interface mode, then make modifications to that file as required.

Argument	Description
	To start a CCL section, type "e". When done typing CCL commands, type ".e" to process the CCL. The ability to write and execute CCL is also available in user interface mode through the Command Editor. For details, see Overview of Command Actions (p. 421).
-remote <host> -port <number> -viewerport <number>	-remote specifies a remote host to run on. -port specifies the port number for user interface-engine communication. -viewerport specifies the port for the viewer. This option also requires the host machine to be running CFD-Post with the -server option.
-report <template> [-name <report name>] [-outdir <dir name>] [<res ults file 1>] [<results file 2> ...]	Starts CFD-Post in batch mode, loads the results files, then produces a report and exits. Here, <template> may be one of the following: • The word "auto". If you use the word "auto" for a template, then CFD-Post will attempt to find the most suitable built-in template. • The name of a registered template, wrapped in quotes. Register a template by running CFD-Post in user interface mode. For details, see Report Templates (p. 76). • The name of a state or session file. If you provide a state file as a template, the results file indicated in the state file, if there is one, will be used when no results file name is provided on the command line.
-local-root <path>	Specify the file path of the CFD-Post installation.
-t <file>.cst Alternative form: -state <file>.cst	Start CFD-Post and load the state file <file>.cst.
-results <file>.res Alternative form:-res <file>.res	Start CFD-Post and load the results file <file>.res.
-monitor <file>.dir	Start CFD-Post and begin monitoring the solution in progress.

Argument	Description
-multiconfig single separate last Alternative form: -m single separate last	Select a multi-configuration load option to control how you load a multi-configuration (.mres) file or a results file (.res) that contains a run history (that is, a file that was produced from a definition file that had its initial values specified from a results file from a previous run and saved to the results file that you are loading). Choose: • Single Case to load all configurations of a multi-configuration run as a single case, or all of the results history from a results file that contains a run history. In either case, only one set of results will appear in the viewer, but you can use the timestep selector to move between results. This option is not fully supported. • Separate Cases to load all configurations from a multi-configuration run into separate cases. If a results file with run history is loaded, CFD-Post loads the results from this file and the results for any results file in its run history as separate cases. Each result appears as a separate entry in the tree. • Last Case to load only the last configuration of a multi-configuration results file, or only the last results from a results file that contains a run history.
-s <file>.cse Alternative form: -session <file>.cse	Start CFD-Post and load the session file <file>.cse.
-v Alternative form: -verbose	Display a summary of the currently set environment variables.
-h Alternative form: -help	Display a full list of all the possible arguments with short descriptions.

2.3. Setting CFD-Post Operation Through Environment Variables

There are a number of environment variables that can be used to change how CFD-Post behaves:

Environment Variable	Description/Usage
CFX POST_USER_MACROS	Allows user-defined macros to load at start-up. CFXPOST_USER_MACROS='macro1, macro2, ' If the macros contain user interface commands, the appropriate panels will be added to the Macro Evaluator user interface.

Environment Variable	Description/Usage
	Example: CFXPOST_USER_MACROS='myMacro1.cse, /home/bob/macros/myMacro3.cse'
CFXPOST_ZERO_THETA	Enables adjusting the zero-theta location in single-domain cases. Linux: CFXPOST_ZERO_THETA='x, y, z' Windows: CFXPOST_ZERO_THETA=x, y, z (where x,y,z is a point not on the rotation axis) This will be used in turbo cases to determine at which position the Theta variable will be equal to zero. By default, CFD-Post will set Theta such that the Theta values in the first encountered domain range from zero to some positive value. Example for Linux: CFXPOST_ZERO_THETA='1, 0, 0.5' Example for Windows (no quotes): CFXPOST_ZERO_THETA=1, 0, 0.5
CFX_USER_IMAGE_DATA	Enables you to display a custom logo image in the viewer. <pre>CFX_USER_IMAGE_DATA='filepath xLoc yLoc xAttach yAttach scale alphaR alphaG alphaB transparency'</pre> filepath: path to the image file Only ppm, png, and jpeg files are currently supported. xLoc, yLoc: horizontal and vertical location of the image in the viewer (0-1) xAttach: left, center, right or none. If set to none, xLoc is used. yAttach: top, center, bottom or none. If set to none, yLoc is used. scale: image size relative to viewer size (0-1) If set to 0, original pixels are shown regardless of the viewer size. alphaR, alphaG, alphaB: red/green/blue components (normalized to 0-255) of alpha (the color that will represent 100% transparency) transparency: overall bitmap transparency (0-1)

Environment Variable	Description/Usage
	Example: To display image myImage.ppm in the right-bottom corner, occupying 12% of the viewer size, making the pure green color represent 100% transparent, and setting the overall transparency to 60%, use: <pre>CFX_USER_IMAGE_DATA= '/logos/myImage.ppm 0 0 right bottom 0.12 0 255 0 0.6'</pre>
VIEWER_EYE_POINT	Allows placing the viewer camera to left/right eye position. It can be used for composing stereo images and movies VIEWER_EYE_POINT='cameraZ eyeDist mode' cameraZ: Z location of the camera (must be less than -1.0; -5.0 is optimum) Smaller numbers bring the camera closer to the scene (and also widen the camera angle), larger numbers move it further. eyeDist: distance between the eyes (0.1 is optimum) mode: 0 = normal, 1 = left eye, 2 = right eye, 3 = left/right eye (two viewports), 4 = right/left (two viewports), 5 = stereo Example: VIEWER_EYE_POINT='-5.0 0.1 0'
CFX_BACKGROUND_ROTATE	Applicable to spherical backgrounds only. CFX_BACKGROUND_ROTATE='x y z angle' x,y,z: specifies a direction vector (in the global coordinate system) about which to apply a rotation to the background image angle: specifies the rotation angle, in degrees, of the background image The rotation angle is clockwise looking in the direction of the specified direction vector. Example: If you start CFD-Post with the mountain scenery background, the background will appear upright when the Y axis is "up". You may find that the geometry of your CFD mesh has its "top" side pointing in the X axis direction. You can rotate the background image so that it appears upright when the X axis is "up" by rotating the image about the Z axis by -90 degrees, as follows: CFX_BACKGROUND_ROTATE='0 0 1 -90'

2.4. Running in Batch Mode

Almost all of the functionality of CFD-Post can be accessed when running in batch mode.

When running in batch mode, a Viewer is not provided and you cannot enter commands at a command prompt. Instead, commands are issued via a CFD-Post session file (*.cse), the name of which is specified when executing the command to start batch mode. The session file can be created using a text editor, or, more easily, by recording a session while running in line interface or user interface mode. You can leave a session file recording while you quit from user interface or line interface mode to write the >quit command to a session file. Alternatively, you can use a text editor to add this command to the end of the session file.

When launching CFD-Post in batch mode on a remote Linux machine, the DISPLAY variable on the remote machine must be set to a valid X display before running in batch mode if CHARTS are to be generated (viewer images can still be generated in batch mode without the DISPLAY variable being set). Typically the DISPLAY variable is set to be your local Windows or Linux machine. The remote machine must have permission to connect to the display (for example, by use of the xhost command if the X display is on a Linux machine).

To run in batch mode, execute one of the following commands at the command prompt:

Windows	<CFDPOSTROOT>\bin\cfdpost -batch <filename>.cse
Linux	<CFDPOSTROOT>/bin/cfdpost -batch <filename>.cse

You can include the name of a results file in your session file, which is described in the example below. However, you can also pass the name of a results file and a session to CFD-Post from the command line. This allows you to apply a generic session file to a series of different results files. To launch CFD-Post in batch file mode, load a results file and execute the statements in a session file using one of the following commands:

Windows	<CFDPOSTROOT>\bin\cfdpost -batch <filename>.cse <filename>.res
Linux	<CFDPOSTROOT>/bin/cfdpost -batch <filename>.cse <filename>.res

To load multiple files, you may list the filenames at the end. For example, fluid.res fluid1.res

2.4.1. Example: Pressure Calculation on Multiple Files using Batch Mode

This example calculates the value of pressure at a point in each of three results files.

The purpose of this example is for demonstration only. You will deal with only three results files in this case, and it would be faster to produce the output by using the graphical user interface. However, these features can be useful in situations where a large number of results files need to be processed at once.

In order to carry out this procedure, you will make use of session files, power syntax and the **Command Editor** dialog box. You could use the results from any file by making the appropriate substitutions in the following example.

1. Place three results files (<resfile1>, <resfile2>, <resfile3>) in your working directory. For this example, in all three results files the location 0, 0, 0 must be in the solution domain.

2. Create a session file based on a results file.
 - a. Start CFD-Post and select **File > Load Results**. Select the results file and click **Open**.
 - b. Select **Session > New Session** from the main menu.
For details, see [New Session Command \(p. 215\)](#).
 - c. Enter `batchtest.cse` as the session file name and click **Save**.
 - d. Select **Session > Start Recording** from the main menu to begin recording the session file commands.
 - e. Select **Insert > Location > Point** and accept the default name `Point 1`.
 - f. Click **Apply** to create the point at location 0, 0, 0.

You will now use Power Syntax to find the value of pressure at `Point 1`, and print it to the command line. In addition to printing the value of pressure, it would be useful to know the name of the results file. You will make use of the `DATA READER` object to find the name of the current results file.

- g. Select **Tools > Command Editor**.
- h. Enter the following into the command window:

```
! $filePath = getValue("DATA READER","Current Results File");
! $pressureVal = probe("Pressure","Point 1");
! print "\nFor $filePath, Pressure at Point 1 is $pressureVal\n";
```

Note:

If you copy the text above into the Command Editor, ensure that the exclamation points are at the beginning of lines.

- i. Click **Process** to process the commands.
 - j. Check the terminal window to make sure the command worked as desired.
 - k. Select **Session > Stop Recording** from the main menu to stop recording the session file.
3. This completes the first part of the example. You may want to close down CFD-Post at this time.
4. You can now run the session file on any number of results files using the following command:

```
<CFDPOSTROOT>/bin/cfdpost -batch batchtest.cse <resfile>
```

where `<resfile>` is the name of your results file.

To load multiple files, you can list the filenames at the end of the command. For example:

```
cfdpost -batch batchtest.cse <resfile1> <resfile2> <resfile3>
```

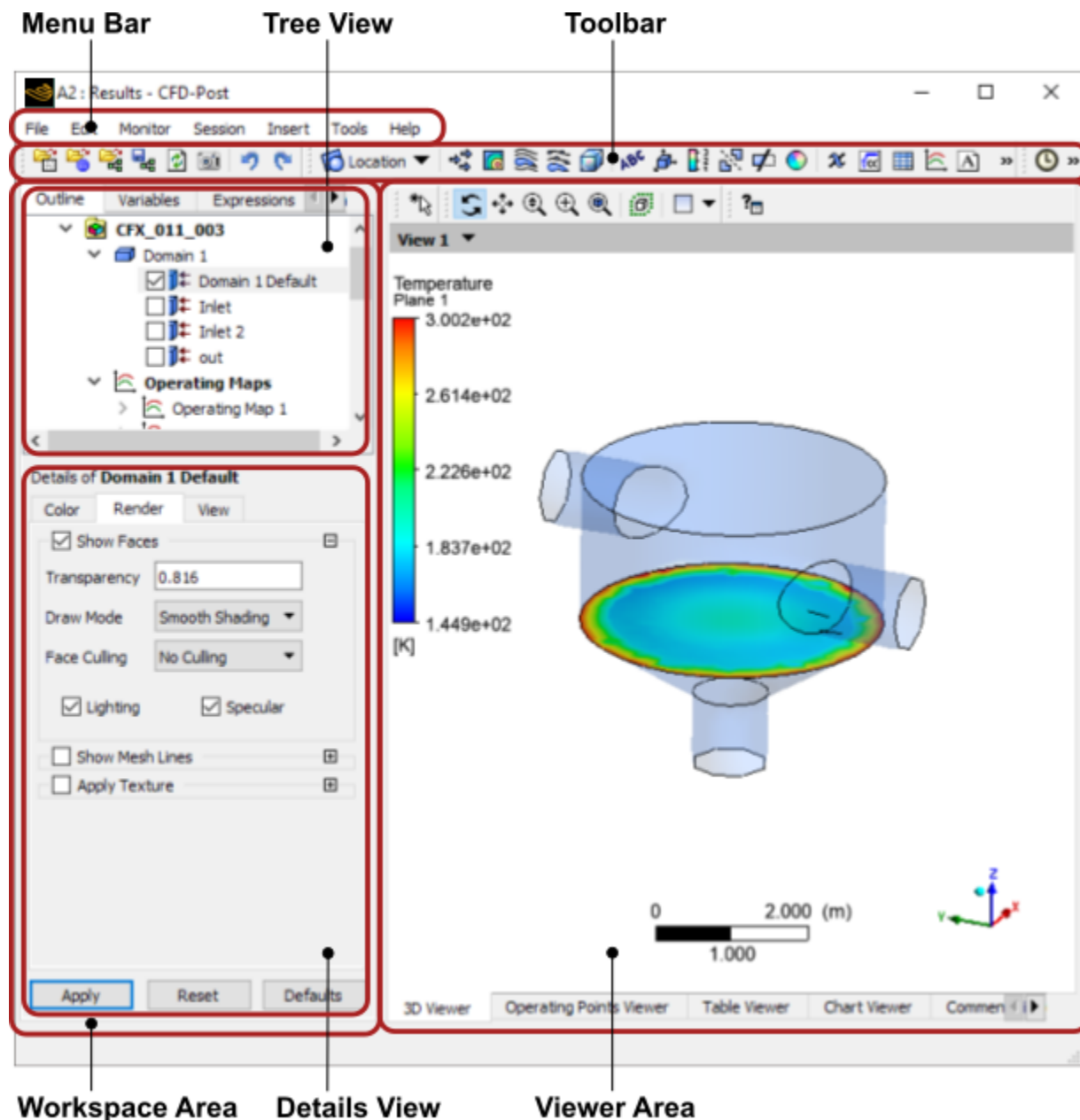
For a very large number of results files, a simple script can be used to pass filenames as command line arguments. As an example, this C shell script would pass arguments as results file names to the CFD-Post command line:

```
#!/bin/csh
foreach file ($argv)
<CFDPOSTROOT>/cfdpost -batch batchtest.cse $file
end
```


Chapter 3: CFD-Post Graphical Interface

The CFD-Post interface contains the following areas: the menu bar, the toolbar, the workspace area, and the viewer area.

Figure 3.1: Sample CFD-Post Interface



When CFD-Post starts, the **Outline** workspace area and the **3D Viewer** are displayed. The top area of the **Outline** workspace is the *tree view* and the bottom area is the *details view* (the details view is populated only after you edit an item, as described in [Details Views \(p. 49\)](#)).

The viewer displays an outline of the geometry and other graphic objects. In addition to the mouse, you can use icons from the viewer toolbar (along the top of the viewer) to manipulate the view.

The main toolbar can be adjusted by right-clicking a blank spot on it (or on the menu bar) then clicking the relevant item in the shortcut menu that appears.

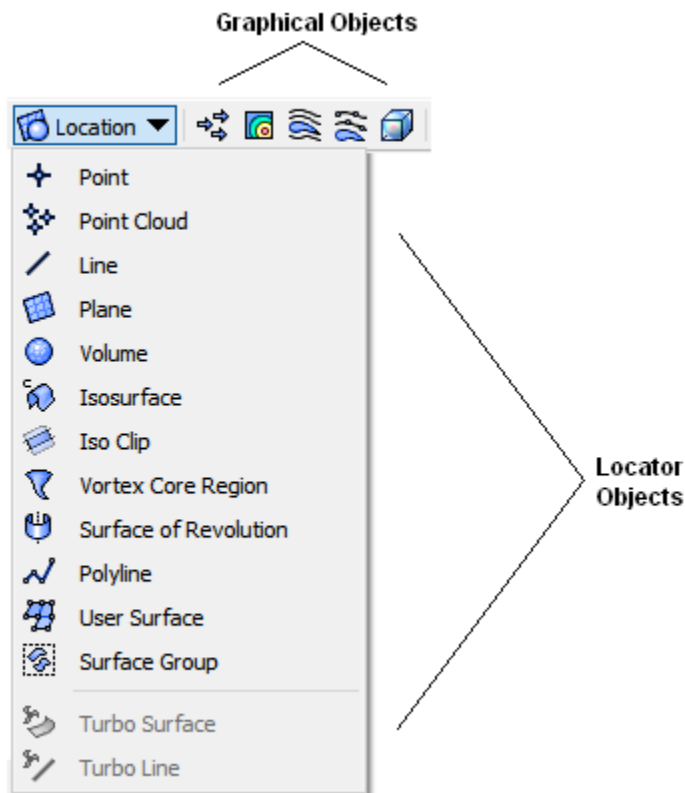
The width of the workspace can be adjusted by dragging its right border to the left or right. The width of the viewer is updated to accommodate the new size of the workspace. The dividing line between the tree view and details view can be dragged vertically to re-size the windows. You may want to do this if the details view contains a large amount of information.

The remainder of this chapter describes:

- [3.1. Graphical Objects](#)
- [3.2. Common Tree View Shortcuts](#)
- [3.3. Details Views](#)
- [3.4. Outline Workspace](#)
- [3.5. Variables Workspace](#)
- [3.6. Expressions Workspace](#)
- [3.7. Calculators Workspace](#)
- [3.8. Turbo Workspace](#)

3.1. Graphical Objects

CFD-Post supports a variety of *graphical objects* and *locator objects* that are used to create postprocessing plots and to define locations for quantitative calculation. In [Figure 3.1: Sample CFD-Post Interface \(p. 45\)](#) a plane has been inserted and configured to display temperature.



The details of all the possible objects and associated parameters that can be defined in CFD-Post are described in the CFD-Post .ccl file available with the installation.

3.1.1. Creating and Editing New Objects

New objects can be created and edited by:

- Right-clicking an object in the tree view area.
- Selecting a command from the **Insert** menu

For details, see [CFD-Post Insert Menu \(p. 219\)](#).

- Right-clicking in the viewer (not applicable for all object types). In many cases, this is the most convenient way to create locators (such as planes). For details, see [CFD-Post 3D Viewer Shortcut Menus \(p. 121\)](#).

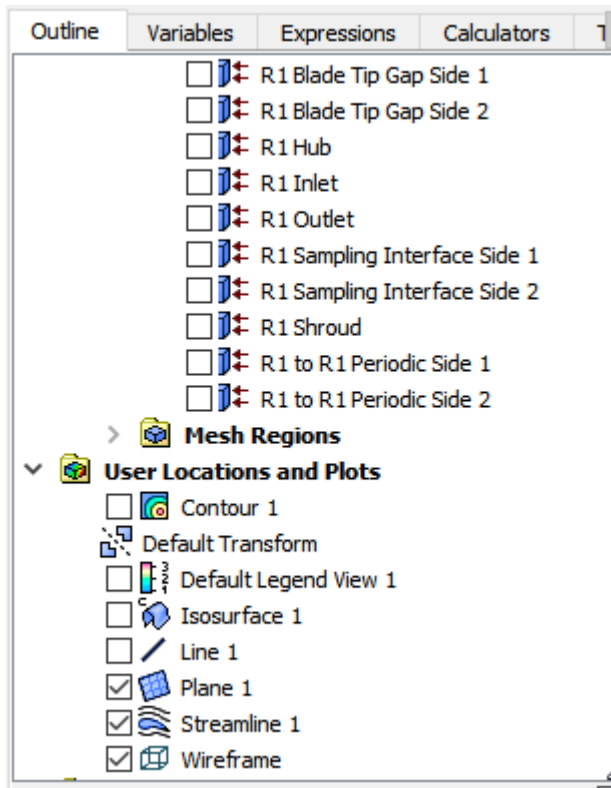
3.1.2. Selecting Objects

You can select multiple objects by holding down the **Ctrl** or **Shift** key as you select each object. Subsequently right-clicking any of the selected objects allows you to perform commands that apply to all of the selected objects (such as **Show** and **Hide**).

3.1.3. Object Visibility

In the **Outline** and **Turbo** workspaces, some objects have a visibility check box beside them. In the graphic that follows, the Plane 1, Streamline 1, and Wireframe objects are set to be visible in the viewer.

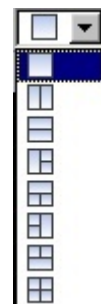
You can change the visibility settings for a group of objects by first selecting a subgroup of objects (using mouse clicks while holding down **Ctrl** (for multiple, independent selections) or **Shift** (to drag over a range of selections)), and then right-clicking on the group and using the appropriate shortcut menu command (for example, **Hide** or **Show**). For details, see [CFD-Post 3D Viewer Shortcut Menus](#) (p. 121).



When multiple viewports are used, the state of each check box is maintained separately for each viewport.

Tip:

You control the number and layout of viewports with the *viewport* icon in the viewer's toolbar.



3.2. Common Tree View Shortcuts

Commonly available tree view shortcuts, which are accessible by right-clicking an object in a tree, include:

Command	Description
New (or Insert)	Inserts an object. In the details view for the new object, the Locations setting, or similar setting as applicable, is preset to the name(s) of the selected object(s).
Edit	Edits the selected object in a details view. ^[a]
Edit in Command Editor	Edits the selected object(s) in the Command Editor dialog box ^[a] . For details, see Command Editor (p. 377) .
Duplicate	Creates a new object of the same type as, and with the same settings as, the selected object.
Delete	Deletes the selected object(s).

^[a] An expression that is set as an input parameter in Ansys Workbench cannot be edited in CFX-Pre or CFD-Post (because the results of such edits are not passed to Ansys Workbench) and will be grayed out. However, the expression can be declared to no longer be an input parameter or it can be deleted.

3.3. Details Views

Details view is a generic term used to describe an editor for the settings of a CCL object. A details view for a particular kind of object (such as a plane object) may be referenced by the name of the type of object being edited, followed by the word "details view" (for example, for the **Wireframe** object, the **Wireframe details view**).

A details view appears after any of the following actions:

- Double-clicking an object in the tree view
- Right-clicking an object in the tree view and selecting **Edit** from the shortcut menu
- Highlighting an object in the tree view and clicking *Edit*  from the shortcut menu
- Clicking **OK** on a dialog box used to begin the creation of a new object
- Clicking an object in the **Viewer** when in pick mode
- Right-clicking on an object in the **Viewer** and selecting **Edit** from the shortcut menu

You use the details view to define the properties of an object. The details view contains one or more tabs, depending on the type of object being defined.

Many properties can be set via a CEL expression. To enter an expression:

1. Click in the field for a property.
2. Click the Enter Expression icon that appears beside the field. This enables the field to accept an expression name.

3. Either enter an expression definition directly, or type the name of an existing expression. You must ensure that the expression evaluates to a value having appropriate units for the property that uses the expression.

For details on CEL expressions, see [Expressions Workspace \(p. 96\)](#).

For CFX components in Ansys Workbench, any CEL expression can be made into a parameterized CEL expression by defining it as a Workbench input parameter. You can do this by creating an expression and parameterizing it by right-clicking it in the Expression editor. You can then use that expression as the value of a property.

You can change a property from being specified by a Workbench input parameter. However, the corresponding CEL expression persists and can be managed by the Expression editor.

Details View Controls

Details views contain the following buttons:

- **Apply** applies the information contained within all the tabs of an editor.
- **OK** is the same as **Apply**, except that the editor automatically closes.
- **Cancel** and **Close** both close the editor without applying or saving any changes.
- **Reset** returns the settings for the object to those stored in the database for all the tabs. The settings are stored in the database each time the **Apply** button is clicked.
- **Defaults** restores the system default settings for all the tabs of the edited object.

3.4. Outline Workspace

The **Outline** workspace consists of objects in a tree view and a details view where you can edit those objects; the tree view appears in the top half of that pane. The details view appears beneath the tree view. For details, see [Details Views \(p. 49\)](#).

You access the **Outline** workspace by clicking the **Outline** tab.

After starting CFD-Post and loading a results file, several special objects will exist in the **Outline** workspace. All of these special objects can have some of their properties edited, but the objects themselves cannot be created or deleted using CFD-Post (without using CCL commands). These objects are described in the following sections:

- [Case Branch \(p. 61\)](#)
- [User Locations and Plots \(p. 71\)](#)
- [Report \(p. 72\)](#)

Objects that do not exist after loading a results file are described in [CFD-Post Insert Menu \(p. 219\)](#).

Shortcuts available to the tree view are described in [Outline Tree View Shortcuts \(p. 51\)](#).

Some of the settings and buttons in a details view of the **Outline** workspace are common for different object types; these are described in [Outline Details View \(p. 52\)](#).

3.4.1. Outline Tree View Shortcuts

Shortcuts for editing and manipulating objects are accessible by right-clicking, in the tree view, on one object or a selection of objects. A selection of multiple objects can be made using **Ctrl** and/or **Shift** keys while clicking objects with the mouse.

The following table shows commands that are specific to the **Outline** tree view. For a list of shortcuts that appear in most tree views, see [Common Tree View Shortcuts \(p. 49\)](#).

Command	Description
Show	Makes the selected objects visible in the viewer.
Hide	Makes the selected objects invisible in the viewer.
Hide All	Makes all objects, except the wireframe object, invisible in the viewer.
Refresh Preview	Refreshes the report. For details, see Refreshing the Report (p. 87) .
Load '<template>' template	Loads the registered template having the name indicated by <template>. For details, see Report Templates (p. 76) .
Report Templates	Allows you to select a report template. For details, see Report Templates (p. 76) .
Add to Report	Sets the selected report objects to appear in the report the next time the report is generated.
Remove from Report	Sets the selected report objects to not appear in the report the next time the report is generated.
Add All to Report	Sets all report objects to appear in the report the next time the report is generated.
Move Up	Moves the selected objects up one level in the report so as to appear closer to the beginning of the report in relation to the other report objects.
Move Down	Moves the selected objects down one level in the report so as to appear closer to the end of the report in relation to the other report objects.
Show in Separate Window	Displays the selected chart in its own window.
Replace results file	Replaces the selected results file with another results file while keeping the state. This is the recommended procedure; reloading the results file through the Load Results panel may not recover the state completely, in particular when Turbo Post is initialized. Note that the Replace results file function will keep the original case name even though the results file has changed.

3.4.2. Outline Details View

A details view appears at the bottom of the **Outline** workspace when you open an object in the tree view for editing (which is described in [Details Views \(p. 49\)](#)). Some **Outline** details views have tabs in common.

3.4.2.1. Geometry Tab

The definition of geometry is unique for each graphic object. The basic procedure for geometry set up involves defining the size and location of the object, with most other properties being object specific. For details, see [CFD-Post Insert Menu \(p. 219\)](#).

3.4.2.1.1. Selecting Domains

For many objects you can select the **Domains** in which the object should exist.

To select the domain, pick a domain name from the drop-down **Domains** menu. To define the object in more than one domain, you can type in the names of the domains separated by commas or click the *Location editor*  icon.

When more than one domain has been used, most plotting functions can be applied to the entire computational domain, or to a specific named domain.

3.4.2.2. Color Tab

The **Color** details tab controls the color of graphic objects in the **Viewer**. The coloring can be either constant or based on a variable, and can be selected from the **Mode** drop-down menu.

The **Color** details tab enables you to view and/or edit the properties of the tree view's Display Properties and Defaults > Color Maps definitions; System colors are view-only but Custom colors can be edited.

The default color map appears in **bold** text.

3.4.2.2.1. Mode: Constant

To specify a single color for an object, select the Constant option from the **Mode** drop-down menu.

To choose a color, click the *Color selector*  icon to the right of the **Color** option and select one of the available colors. Alternatively, click the color bar itself to cycle through ten common colors quickly. Use the left and right mouse buttons to cycle in opposite directions.

3.4.2.2.2. Mode: Variable and Use Plot Variable

You may want to plot a variable on an object, such as temperature on a plane. To do this, you should select the Variable option from the **Mode** drop-down menu. This displays additional options, including the Variable drop-down menu where you can choose the variable you want to plot.

The list of variables contains User Level 1 variables. For a full list of variables, click *More variables* .

For isosurface and vector plots, the **Use Plot Variable** option is also available. This sets the variable used to color your plot to the same as that used to define it.

3.4.2.2.3. Range

Range enables you to plot using the global, local, or a user specified range of a variable. The range affects the variation of color used when plotting the object in the viewer. The lowest values of a variable in the selected range are shown in blue in the viewer; the highest values are shown in red.

- The **Global** range option uses the variable values from the results in all domains (regardless of the domains selected on the **Geometry** tab) and all time steps (when applicable) to determine the minimum and maximum values.
- The **Local** range option uses only the variable values on the current object at the current time step to set the maximum and minimum range values. This option is useful to use the full color range on an object.
- The **User Specified** range option enables you to specify your own maximum and minimum range values. You can use this to concentrate the full color range into a specific variable range.

You should not select the **Local** Range option when coloring an isosurface or turbo surface with the variable used to define it. In this case, the **Local** Range would be zero by definition, and the plot would highlight only round-off errors.

3.4.2.2.4. Hybrid/Conservative

Select whether the object you want to plot will be based on hybrid or conservative values. For details, see [Hybrid and Conservative Variable Values](#).

3.4.2.2.5. Color Scale

The color scale can be mapped using a linear or logarithmic scale. For a linear scale, the color map is divided evenly over the whole variable range. For a logarithmic scale, the color scale is plotted against a log scale of the variable values.

3.4.2.2.6. Color Map

The colors along the color bar in the legend are specified by this option.

Some of the available color maps are described below:

- **Rainbow** uses a standard mapping from blue (minimum) to red (maximum).
- **Rainbow 6** uses an extension of the standard **Rainbow** map from blue (minimum) to magenta (maximum).
- **Divergent Cool to Warm** changes from blue (minimum) through gray (intermediate) to red (maximum).

- Zebra creates six contours over the specified range of values. Between each pair of contour lines, the color scale varies from white (minimum) to black and through to white (maximum) again. The Zebra map can be used to show areas where the gradient of a variable changes most rapidly with a higher resolution (five times greater) than the standard Greyscale color map
- Greyscale changes in color from black (minimum) to white (maximum).
- Blue to White changes in color from blue (minimum) to white (maximum).
- White to Blue changes in color from white (minimum) to blue (maximum).
- FLUENT Rainbow changes in color from blue (minimum) to red (maximum).
- Transparency changes in color from nearly transparent (minimum) to nearly opaque (maximum).

3.4.2.2.6.1. Accessing the CFD-Post Color Map Editor

The CFD-Post Color Map editor enables you to define and name a set of colors (a color map) that you can then apply to an object by using that object's **Color** tab.

To access the Color Map editor, you can:

- From the menu bar, select **Insert > Color Map**.
- From an object's **Color** tab (when the **Mode** is set to **Variable**), click the  icon beside the **Color Map** field and select **Insert**, **Edit**, or **Duplicate**.
- From the **Outline** tree view under **Display Properties and Defaults > Color Maps**, select a **System** or **Custom** color map, right-click, and select either **Insert**, **Edit**, or **Duplicate**.

A **System** color map can be set as the default, but otherwise cannot be edited directly. However, you can duplicate a **System** color map and use that as a basis for a **Custom** color map (which will be completely editable).

Depending on how you access the Color Map editor, it may appear as a dialog box or as a details view.

To learn how to use the Color Map editor, see [Color Map Command \(p. 298\)](#).

3.4.2.2.7. Contour

The **Contour** setting enables coloring with contour bands, provided that user preference **Enable GPU Shader Rendering** (see [Advanced \(p. 204\)](#)) is selected.

This setting can be set to:

- **Smooth**, which produces a smooth color gradient across the color range.

- **Banded**, which produces contour bands in the displayed object and reveals the **Number of Contours** setting, which controls the number of bands for the color range.

Note:

An alternative to coloring an object with contour bands is creating a contour plot. For details, see [Contour Command](#) (p. 264).

3.4.2.2.8. Undefined Color

Undef. Color is the color that is used in areas where the results cannot be plotted because the variable is not defined or variable values do not exist.

For example, a section of an object that lies outside the computational domain will not have any variable value.

Clicking the *Color selector*  icon to the right of this box allows you to change the undefined color. Alternatively, click the color bar itself to cycle through ten common colors quickly. Values written to the results file as zeros are colored as such and will not be undefined. For example, consider results files containing Yplus/Wall Shear values away from a wall boundary.

3.4.2.3. Symbol Tab

Enables you to configure the appearance of a symbol.

3.4.2.3.1. Symbol

Selects the style of the symbol to be displayed.

3.4.2.3.2. Symbol size

Specifies the size of the symbol where 0 is the smallest and 10 is the largest.

3.4.2.4. Render Tab

The appearance of the **Render** tab depends on what type of object is plotted in the viewer.

3.4.2.4.1. Use Post Region Rendering

For visualization purposes, an injection region that uses the **Selected 2D Regions** option consists of all of the mesh faces that meet or overlap the injection holes/slots in the injection region.

To visualize an approximation of the holes/slots of such an injection region:

1. In the details view, on the **Render** tab, select **Use Post Region Rendering**.
2. Set an appropriate value for **Area Fraction**.
3. Click **Apply**.

The injection region is clipped along a contour of the specified value of **Area Fraction**; the portion of the injection region that remains visible is that where the injection region's area fraction variable is greater than the specified value.

3.4.2.4.2. Show Faces

The top half of the tab controls the **Show Faces** options. This toggle is selected by default and draws the faces of the elements that make up an object. The faces are colored using the settings on the **Color** tab. When **Show Faces** is selected, the following options can be set:

3.4.2.4.3. Show Faces: Transparency

Set the Transparency value for the faces of the object by entering a value between 0 (opaque) and 1 (transparent), or use the embedded slider.

Note:

For volume rendering objects, transparency is set on the **Geometry** tab.

3.4.2.4.4. Show Faces: Draw Mode

Shading properties can be None, Flat Shading, or Smooth Shading.

- **Flat Shading:** Each element is colored a constant color. Color interpolation is not used across or between elements.
- **Smooth Shading:** Color interpolation is applied that results in color variation across an element based on the color of surrounding elements.
- **Draw as Lines:** This option draws lines but uses the color settings defined on the **Color** tab. The color settings that are applied to the lines use Smooth Shading. **You must use the Screen Capture feature to print an image or create an animation containing lines drawn using this option.** (See [Animation Options Dialog Box: Options Tab \(p. 342\)](#) for details on the **Screen Capture** feature.)

Note:

- Optionally, you can edit the face you want to show as lines to disable **Show Faces** and to enable **Show Mesh Lines**. The resulting display will be similar to **Draw as Lines**, but in constant-color mode only.
 - Rendering quadrilateral and polygon faces with the option **Draw As Lines** might result in lines being drawn between the first and all other nodes of the face rather than just the lines representing the edges of the face.
-

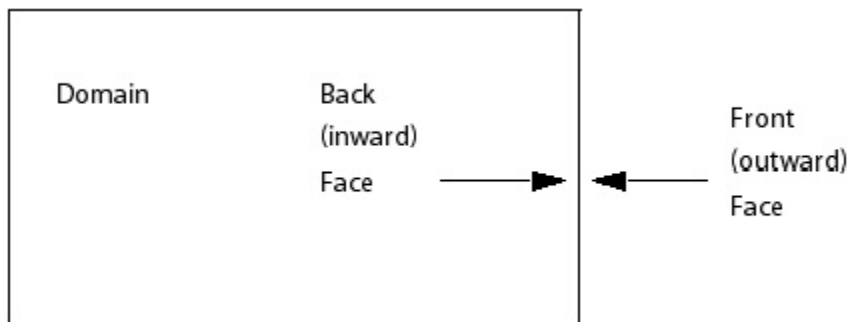
- **Draw as Points:** This option draws points at the intersection of each line, using the color scheme defined on the Color tab. **You must use the Screen Capture feature to print an image containing this option.**

3.4.2.4.5. Show Faces: Face Culling

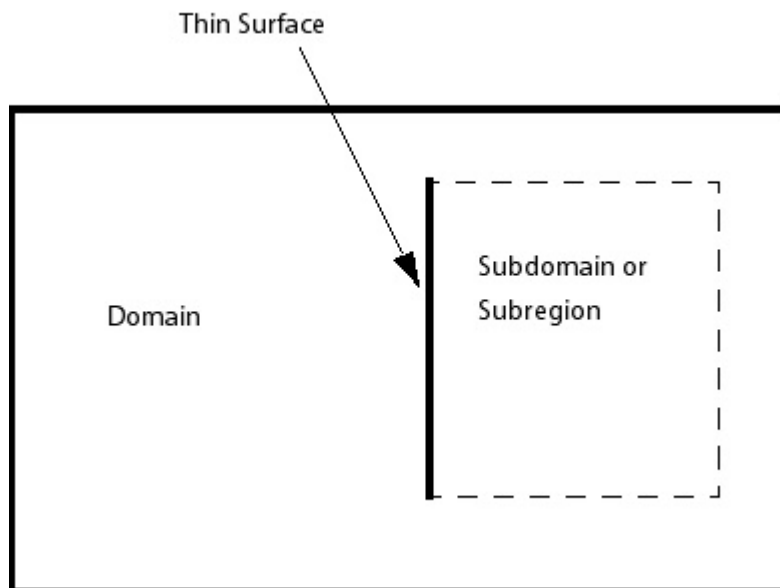
Note:

Face Culling affects only printouts performed using use **Screen Capture** method.

Toggle **Face Culling** (removal) of the front or back faces of the polygons that form the graphic object. This allows you to clear visibility for element faces of objects that either face outwards (front) or inwards (back). Domain boundaries always have a normal vector that points outwards from the domain. The two sides of a thin surface therefore have normal vectors that point towards each other.



- Selecting **Front** clears visibility for all outward-facing element faces. This would, for example, clear visibility for one side of a plane or the outward facing elements of a cylinder locator. When applied to a volume object, the first layer of element faces that point outwards are rendered invisible. You will also generally need to use face culling when viewing values on thin surface boundaries, which are defined using a wall boundary on two 2D regions that occupy the same spatial location.



If you want to plot a variable on a thin surface, you will need to select **Front** Face culling for both 2D regions that make up the thin surface to view the plot correctly. As shown by the

two previous diagrams, viewing only the back faces means that the data for the inward facing surfaces is always visible.

- Selecting **Back** clears visibility for inward-facing element faces (the faces on the opposite side to the normal vector). When applied to volume objects, the effect of back culling is not always visible in the viewer because the object elements that face outward obscure the culled faces. It can, however, reduce the render time when further actions are performed on the object. The effect of this would be most noticeable for large volume objects. In the same way as for **Front Face** culling, it clears visibility of one side of surface locators.
- **No culling** shows element faces when viewed from either side.

3.4.2.4.6. Show Faces: Lighting

When selected, surfaces appear realistically shaded to emphasize shape. Clear this check box when you want to see variable colors that match the legend colors.

3.4.2.4.7. Show Faces: Specular Lighting

When selected, objects appear shiny.

3.4.2.4.8. Show Mesh Lines

Show Mesh Lines can be selected to show the edge lines of the mesh elements in an object. When selected, the following options are available:

3.4.2.4.9. Show Mesh Lines: Edge Angle

This can be altered to enable the same editing features on each object as for the whole wireframe. For details, see [Wireframe](#) (p. 71).

3.4.2.4.10. Show Mesh Lines: Line Width

Set the line width by entering the width of the line in pixels. Set the value between 1 and 11; you can use the graduated arrows, the embedded slider, or type in a value.

3.4.2.4.11. Show Mesh Lines: Color Mode

The line color can be set as **Default** or **User Specified**.

Default sets the line to CFD-Post default color scheme (set using **Options** dialog box in **Edit** menu).

User Specified allows you to pick the color. For details, see [Show Mesh Lines: Line Color](#) (p. 58).

3.4.2.4.12. Show Mesh Lines: Line Color

Line color can be changed by clicking the *Color selector*  icon to the right of the **Color** box and selecting a color. Alternatively, click the color bar itself to cycle through ten common colors quickly. Use the left and right mouse buttons to cycle in opposite directions.

3.4.2.4.13. Apply Texture

Textures are images that are pasted (mapped) onto the faces of an object. They are used to make an object look like it is made of a certain type of material, or to add special labels, logos, or other custom markings on an object.

3.4.2.4.14. Apply Texture: Predefined Textures

To use a predefined texture map, set **Type** to **Predefined** and set **Texture** to the desired material type using the drop-down menu. Options include brick and various types of metal.

Enable **Blend** to blend the texture with the object color specified from the **Color** tab.

Blend allows the colors of the texture to combine with the basic color of the object. For example, if a white object is given the texture **Metal**, the object looks like silver. If the basic color of the object is orange, the object looks like copper. With the **Blend** feature turned off, the basic color of the object has no effect and colors depends only on the texture.

3.4.2.4.15. Apply Texture: Custom Textures

To use a custom texture map, set **Type** to **Custom**.

An **Image File** (for example, a ppm file) must be specified. The dimensions of the image, in pixels, should be powers of two. If the texture image has a number of rows not equal to a power of two, some rows are removed (with an even distribution) until the number remaining is a power of two. The same is true for the number of columns. For example, an image with dimensions 65 by 130 is reduced to an image 64 by 128 before it is applied (the file will not be changed, though).

There are two basic kinds of texture mapping available; textures can either move with the object, as if painted on, or textures can "slide" across objects, producing a "shiny metal" effect. The latter kind of texture mapping is activated by turning on the **Sphere-Map** feature.

When **Sphere-Map** is not used, the following additional features apply:

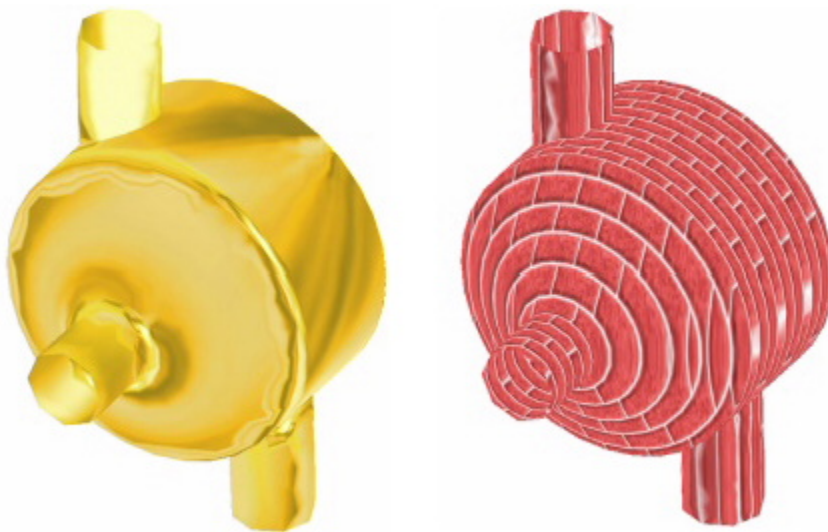
- **Tile**: Causes the texture image to be repeated.
- **Position**: Controls the position of the mapped image relative to the object.
- **Direction**: Controls the direction in which the texture is stamped on the object.

The texture appears undistorted when the object is viewed in this direction.

- **Scale**: Controls the size of the mapped texture relative to the object.
- **Angle**: Controls the texture image orientation about the axis specified by **Direction**.

3.4.2.4.16. Apply Texture: Texture Examples

The next two figures show an object with a gold texture map and an object with a brick texture map.



Note that the brick pattern was applied in the direction of the Y axis, which is roughly going from the lower-left corner to the upper-right corner of the figure. The texture is applied to all faces of the object (locator) ignoring the Y coordinate. This results in the texture becoming smeared in the specified **Direction**.

To avoid this, textures can be applied to smaller locators (that is, ones that cover only a portion of the whole object). The **Direction** setting can then be specified using a direction approximately perpendicular to each of the smaller surfaces. Smaller locators can be found in the tree view (for example, under **Regions**).

3.4.2.5. View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects. For details on Instance Transforms, see [Instance Transform Command \(p. 292\)](#).

3.4.2.5.1. Apply Rotation Check Box

3.4.2.5.1.1. Method, Axis, From, To

These settings specify an axis of rotation. For details, see [Method \(p. 293\)](#).

3.4.2.5.1.2. Angle

The **Angle** setting specifies the angle of rotation about the axis. For details, see [Angle \(p. 294\)](#).

3.4.2.5.2. Apply Translation Check Box

Select the **Apply Translation** check box to move the object in the Viewer. For details, see [Apply Translation Check Box \(p. 294\)](#).

3.4.2.5.3. Apply Reflection/Mirroring Check Box

Select the **Apply Reflection/Mirroring** check box to mirror an object in the Viewer. For details, see [Apply Reflection Check Box \(p. 294\)](#).

3.4.2.5.4. Apply Scale Check Box

Select the **Scale** check box to set values in the X, Y, and Z directions to scale the object about the object's centroid. For example, entering [2, 1, 1] would stretch the object to double its size in the X axis direction.

Note:

If you apply scaling to one or more domains in a multidomain case (including any case that has multiple domains due to the use of data instancing), the resulting domains will generally not be located correctly relative to each other.

3.4.2.5.5. Apply Instancing Transform Check Box

The **Transform** setting specifies a predefined Instancing Transform.

3.4.3. Case Branch

The tree view contains one case branch for each loaded results file or mesh file. The case name is the name of the results file, less the extension.

Tip:

To see the full path to the case file, hover the mouse pointer over the case name.

A case branch contains all domains, subdomains, boundaries, and **Mesh Regions** contained in the corresponding results file. To access the details view of an item, double-click it or right-click it and select **Edit**.

The details view for the case branch name has:

- An **Operating Points** tab (for operating point cases only)

You can use response points instead of operating points to generate smoother looking contours on an operating map.

Set **Number of Response Points** to indicate the approximate number (examples: 100, 1000, 2000) of response points to generate.

Response points are based indirectly on operating point data as follows:

- For each operating point output variable, a corresponding response surface is generated over the two-dimensional input space represented by the **Response Surface Inputs**, which are automatically chosen from the operating point input parameters. If there are more than two operating point input parameters, the remainder are held constant at their mid-range values for the purpose of generating the response surfaces. For details on response surface generation, see [Genetic Aggregation in the DesignXplorer User's Guide](#).
- The response surfaces are sampled at the response point locations in the two-dimensional input space, yielding output variable values for the response points. Note that the response points are uniformly distributed in the input space.

For details on operating maps, see [Operating Maps](#) (p. 68).

- A **View** tab.

Select the **Apply Translation** check box to move the object in the viewer. For details, see [Apply Translation Check Box](#) (p. 294).

The following topics are discussed:

[3.4.3.1. Domain Details View](#)

[3.4.3.2. Boundary and Subdomain](#)

[3.4.3.3. Injection Regions](#)

[3.4.3.4. Other Locations](#)

[3.4.3.5. Spray](#)

[3.4.3.6. Operating Maps](#)

[3.4.3.7. Mesh Regions](#)

3.4.3.1. Domain Details View

A domain object represents each domain loaded from the results file.

3.4.3.1.1. Instancing Tab

Instancing affects the display of objects; it allows multiple copies of objects to be displayed with a specified geometric transformation describing the relative positions. For example, a row of turbine blades can be visualized by applying instancing to an object that shows a single blade.

The **Instancing** tab for a domain is the same as the **Instancing** tab for a turbo component (see [Instancing Tab](#) (p. 387)) and similar to the **Definition** tab for an Instance Transform object (see [Instance Transform: Definition Tab](#) (p. 293)). (The **Definition** tab for an Instance Transform object is different in that its **Axis Definition** settings and **Instance Definition** settings cannot be set from a results file.)

Any viewable object that is associated with one or more domains is, by default, affected by a change to the instancing information (as defined on the **Instancing** tab) of each associated domain, because:

- By default, such an object uses the default transform to control graphical instancing.
- By default, the default transform has the **Instancing Info From Domain** option selected.
- The **Instancing Info From Domain** option causes the graphical instancing information to be taken from each domain (as defined on the **Instancing** tab) that is associated with the object.

3.4.3.1.2. Info Tab

Certain information that CFD-Post reads from the results file is displayed on the Info panel. The units that are shown beside some quantities are the default CFD-Post units, which you can change by selecting **Edit > Options** from the main menu bar.

3.4.3.1.3. Data Instancing Tab

The **Data Instancing** tab is available only for transient blade row cases. On this tab, the **Number of Data Instances** setting can be used to effectively increase the number of blade passages in a given domain. Unlike graphical instancing, data instancing does not, in general, create identical instances of a given domain; instead data instancing creates new instances that can differ in geometry and/or solution data, as appropriate for each instance.

Data instancing alters the geometric representation of the domain mesh and the solution data associated with it. The mesh and solution data that were read from the results file are "expanded" by the number of instances specified such that the domain and its solution data then appear to encompass the original and instanced meshes. Such instancing of the solution data is carried out by using the Fourier coefficients that are stored in the results file.

In this documentation, the term "expanded domain" is used to refer to the domain after instancing has been applied.

The following features of CFD-Post are affected by data instancing:

Feature	Effect of Data Instancing
Wireframe	This object represents the wireframe of all non-expanded domains and expanded domains as defined by the Number of Data Instances setting.
Domain objects	These objects have no graphical representation beyond the tree and are not replicated.
Boundary and Subdomain objects	These objects are not replicated as objects but will be instanced in the viewer and affect evaluation of data when the object is used as a locator.
Mesh regions	These objects are not replicated and only the original definition is represented in the viewer. Data instancing does not affect evaluation of solution data when mesh regions are used as locators.
Solution data	Solution data (which may be used for coloring the locator) is available throughout the expanded domain.
Locations (such as planes, volumes, and isosurfaces)	Locations are not instanced, but act as if the domain has been expanded. For example, the application of data instancing can: • Cause the extension of a plane into the expanded domain • Cause the relocation of a point that is located according to the minimum value of a variable. The relocated point can be anywhere in the expanded domain.
Vector, contour, streamline, particle track, volume rendering	Solution data is available throughout the expanded domain. A plot defined on an expanded domain will be displayed where appropriate in the expanded domain. For example, streamlines will be drawn as continuous lines when continuing from one instance of the original mesh to the next.
Charts (such as Blade Loading, Circumferential, Hub	These objects act as if the domain has been expanded. Charts that involve circumferential averaging (that is, Hub to Shroud,

Feature	Effect of Data Instancing
to Shroud, Inlet to Outlet, and Meridional)	Inlet to Outlet, Meridional) use averaging over only the existing data instances (including the original mesh); therefore for these plots, there should be a sufficient number of data instances to constitute a repeating section of the full wheel.
Mesh Calculator	Mesh calculations return values that are not affected by the Number of Data Instances setting.
Function Calculator	Calculations are performed on the expanded mesh and data.
Turbo Surface, Turbo Line	A turbo surface acts as if the domain has been expanded. It appears only within the expanded domain. A turbo line made using the Inlet to Outlet or Hub to Shroud methods is drawn where specified, but data is available only where the turbo line intersects the expanded domain. The same applies for a turbo line made using the Circumferential method with defined Theta limits. However, a turbo line made using the Circumferential method with Bounds > Type set to None, is drawn according to the theta extent of the expanded domain.
Blade-to-blade plot	Blade-to-blade plots act as if the domain has been expanded. They appear only within the expanded domain.
Instance Transform and Graphical Instancing	Graphical instancing of viewable objects is performed after those objects have been expanded geometrically as defined above. You may find that a combination of data instancing and graphical instancing is appropriate. For example, data instancing can be applied on a portion of the blade row, then graphical instancing can be applied to produce a graphical object (such as a contour plot) that covers the full geometry. Applying both data instancing and graphical instancing can, if not done correctly, produce overlapping graphics (with multiple blade passages plotted in the same space). You should ensure that, for the graphical instancing settings, the number of passages per component is set appropriately; typically, the appropriate value is the number of passages in the expanded domain.
Graphical Scaling	If you apply graphical scaling (Apply Scale Check Box (p. 61)) to a domain that has data instancing applied, the resulting instances of the domain will generally not be located correctly relative to each other.
Case Comparison mode	Case Comparison mode cannot be used when any of the cases being compared involves the use of data instancing.

Note:

When loading a results file via the **Load Results File** dialog box, the **Construct Variables From Fourier Coefficients** option must be selected in order for CFD-Post to

read the Fourier coefficient data, which makes data instancing possible. For details, see [Load Results Command \(p. 141\)](#).

Note:

Global ranges apply to only the set of data instances that you have generated and for only the time steps that you have loaded. Creating and deleting data instances, or loading other time steps, can cause the global range to change.

Note:

Some quantities are time independent and therefore are unchanged for each data instance. For example, the global range of a contour plot of Pressure varies according to the number of data instances but the global range of a contour plot of Pressure .trnavg is unaffected by the number of data instances.

3.4.3.2. Boundary and Subdomain

All boundaries and subdomains associated with a domain are listed under the domain.

The Boundary and Subdomain object types are defined during preprocessing and created in CFD-Post when a file is loaded. You cannot create additional boundary or subdomain objects during postprocessing, or delete the existing ones.

A boundary object exists for each boundary condition defined in the results file. Any mesh regions that were not specifically assigned a boundary condition appear in a default boundary object for each domain.

If you have a complex geometry where many mesh regions are assigned to the default boundary conditions, it may be worth defining named boundary conditions for some of the regions when they are created, even though you still apply the default wall boundary condition to these named regions. You will then have convenient boundary objects created in CFD-Post upon which you can view variables when you come to view the results.

Subdomain objects exist only if subdomains are defined during preprocessing.

You can edit both the **Color** and **Render** properties of Boundary and Subdomain objects. For details, see:

- [Color Tab \(p. 52\)](#)
- [Render Tab \(p. 55\)](#).

3.4.3.3. Injection Regions

All injection regions associated with a domain are listed under the domain.

The Injection Region object type is defined during preprocessing, created in CFX-Solver, and loaded into CFD-Post. You cannot create additional injection region objects during postprocessing, or delete the existing ones.

In CFD-Post, there is an injection region object for each injection region that is stored in the results file and that is specified with the **Selected 2D Regions** option. For details, see [Injection Location Type in the CFX-Pre User's Guide](#).

For more information, see [Injection Regions in the CFX-Pre User's Guide](#).

You can edit both the **Color** and **Render** properties of **Injection Region** objects. For details, see:

- [Color Tab \(p. 52\)](#)
- [Render Tab \(p. 55\)](#).

For visualization purposes, each injection region object in CFD-Post represents the set of all boundary mesh element faces that overlap, or are direct neighbors to, any hole/slot of the same injection region as defined in CFX-Pre.

To help visually resolve the holes/slots, rendering setting **Use Post Region Rendering** (see [Use Post Region Rendering \(p. 55\)](#)), which is on by default, clips the rendered injection region along a contour of the specified value (0.33 by default) of the injection region's associated area fraction solution variable. The latter is named using the injection region name with "Area Fraction" appended, and indicates the local overlap fraction of area. Here, "overlap" refers to mesh face overlap with a hole/slot, and "area" refers to the total mesh face area associated with a mesh vertex.

Note:

The **Use Post Region Rendering** option does not work with results files that are not standard/full results files.

A CEL expression can use an injection region as a locator. An example of such a CEL expression follows:

- To calculate the total mass flow through the injection region's holes/slots, you can evaluate CEL expression

```
massFlow()@[injection region name]
```

where "[injection region name]" represents the name of the injection region.

Note:

In CFD-Post, an injection region locator applies strictly to the hole/slot portion of an injection region; such a locator does not include any mesh faces or parts of mesh faces that do not overlap a hole/slot. Quantitative calculations in CFD-Post that involve an injection region locator's area should not normally require reference to the injection region's associated area fraction variable.

3.4.3.4. Other Locations

Any User Locations that are available are listed (for example, User Surfaces that are specified in the Monitor Surfaces section of CFX-Pre). For more information, see [User Locations in the CFX-Pre User's Guide](#).

You can edit both the **Color** and **Render** properties of User Surface objects. For details, see:

- [Color Tab \(p. 52\)](#)
- [Render Tab \(p. 55\)](#).

3.4.3.5. Spray

The Spray object only becomes available after loading a Forte results file. Spray objects share many of the common features found in CFD-Post, as well some unique features.

3.4.3.5.1. Spray: Geometry Tab

3.4.3.5.1.1. Domains

See [Selecting Domains \(p. 52\)](#).

3.4.3.5.1.2. Reduction Type

Reduction Type enables you to reduce the number of particles present in the Spray object. There are two options.

Option	Setting	Description
Reduction Factor	Reduction	Reduces the number of particles by the factor specified in Reduction .
Maximum Number of Particles	Maximum	Limits the number of particles to no more than specified in Maximum .

3.4.3.5.2. Spray: Color Tab

See [Color Tab \(p. 52\)](#).

3.4.3.5.3. Spray: Symbol Tab

See [Symbol Tab \(p. 55\)](#).

Since, by default, particles generated in a Spray object will have different sizes, you can select **Constant** to make all particles respect the **Symbol Size**. If **Particle Diameter** is selected, the particles will maintain their relative sizes while still scaling with **Symbol Size**.

3.4.3.5.4. Spray: Render Tab

See [Render Tab](#) (p. 55).

3.4.3.5.5. Spray: View Tab

See [View Tab](#) (p. 60).

3.4.3.6. Operating Maps

Defined operating maps are listed under **Operating Maps**, and again under **Report > Operating Points > [case name]**.

For modeling information, see [Operating Maps and Operating Point Cases in the CFX-Solver Modeling Guide](#).

The following topics are discussed:

[3.4.3.6.1. Operating Map: Chart Data Tab](#)

[3.4.3.6.2. Operating Map: Chart Display Tab](#)

[3.4.3.6.3. Operating Map: General Tab](#)

[3.4.3.6.4. Limitations of Operating Maps based on Response Points](#)

3.4.3.6.1. Operating Map: Chart Data Tab

At the top of the **Chart Data** tab is a list of the data series for the operating map. The icons beside the list enable you to:

- Add a data series (*New* )
- Delete a data series (*Delete* )
- View basic statistics (*Statistics* ) , such as minimum and maximum values, for the selected data series. The statistics for the selected series appear in the **Statistics** dialog box.

Corresponding functions are available when you right-click a data series name.

You can set **Type** to:

- Scatter Points

Under **Chart Definition**, set **X Variable** and **Y Variable** to indicate the independent axes of the scatter plot.

Under **Chart Data Source**, set **Source Data** to one of the following options:

- Operating Points

The operating points are plotted.

- Response Points

The response points are plotted. You can set the number of response points in the details view for the case. For details, see [Case Branch \(p. 61\)](#).

- Contour Lines

Under **Contour Definition**, set **Parent Series**, **Variable**, **Range** (with **Min** and **Max** if applicable), and **# of Contours**. Here:

- **Parent Series** is a reference to a series of type Scatter Points from the same operating map. The independent axes for the contour plot are the same as for the parent series.
- **Variable** is the variable for which contour lines of constant value are plotted.
- The **Range** and **# of Contours** settings are the same as the corresponding settings for a regular contour plot, as described in [Range \(p. 265\)](#) and [# of Contours \(p. 267\)](#).

Under **Contour Data Source**, set **Source Data** to one of the following options:

- Operating Points

The operating points are used to generate contours.

- Response Points

The response points are used to generate contours. Contour plots generally look smoother when they are based on response points rather than operating points. You can set the number of response points in the details view for the case. For details, see [Case Branch \(p. 61\)](#).

You can plot contour lines of any output variable. Each contour line of an output variable is initially generated in the same input space as the response surface (that is, the space represented by **Response Surface Inputs**, as listed in the details view for the case, on the **Operating Points** tab). Each contour line is then transformed to the coordinate system of the operating map.

Note that you cannot plot contour lines of either of the response surface input variables when using response points as the data source. However, you can plot contour lines of any operating point variable by using operating points as the data source.

You can add labels for the contours by selecting **Contour Line Label**. Such labels can be controlled by several settings as described next:

- You can choose automatic numerical formatting for the labels by selecting **Determine the number format automatically**. Alternatively, you can specify numerical formatting details manually by specifying the number of significant digits (**Precision**) and the style: Fixed or Scientific.
- You can add a border around each label by selecting **Add label border**.
- The **Position** setting affects how each label is positioned relative to its respective contour line:
 - Head

Each label is positioned near the head endpoint of its respective contour line.

→ Tail

Each label is positioned near the tail endpoint of its respective contour line.

→ Middle

Each label is positioned near the mid-length point of its respective contour line.

→ Random

Labels are arranged using a random number generator that is controlled by a seed value. Changing the seed value changes the label positions.

- You can change the font size for the labels by changing the value of **Size**.
- You can change the **Foreground/Background** color of the labels by clicking (or, to browse in the opposite order, right-clicking) the color bar, or by clicking *Color selector*  and using the **Select Color** dialog box.

3.4.3.6.2. Operating Map: Chart Display Tab

The settings on the **Chart Display** tab of the operating map details view are the same as the corresponding settings of the **Line Display** tab of the chart details view. For details, see [Chart: Line Display Tab \(p. 319\)](#).

3.4.3.6.3. Operating Map: General Tab

The settings on the **General** tab of the operating map details view are the same as the corresponding settings of the **General** tab of the chart details view. For details, see [Chart: General Tab \(p. 306\)](#).

3.4.3.6.4. Limitations of Operating Maps based on Response Points

- Operating point filters have no effect. Filtering is described in [Adding Filter Rules \(p. 133\)](#).
- If there are more than two input parameters in the operating point data:
 - Operating Maps might not look as expected.
 - Each chart axis must be either:
 - An operating point output parameter, or
 - One of the response surface input variables, as listed in the details view for the case, on the **Operating Points** tab, under **Response Surface Inputs**.
- You cannot plot contour lines of either of the response surface input variables when using response points as the data source. However, you can plot contour lines of any operating point variable by using operating points as the data source.

3.4.3.7. Mesh Regions

All of the primitive and composite region names are listed under **Mesh Regions**.

3.4.4. User Locations and Plots

The following objects appear under **User Locations and Plots**:

- User-defined locators

You can define a variety of locators, such as points, lines, planes, and volumes; for details, see [Location Submenu \(p. 220\)](#).

- Transforms

Instance transforms are used to specify how an object should be drawn multiple times. CFD-Post can create instance transforms using rotation, translation, and reflection; for details, see [Instance Transform Command \(p. 292\)](#).

- Legends

Legends can be displayed in the viewer to show the relationship between colors and values for the locators you insert; for details, see [Legend Command \(p. 288\)](#).

- Wireframe

The Wireframe object contains the surface mesh for your geometry; for details, see [Wireframe \(p. 71\)](#).

3.4.4.1. Wireframe

The Wireframe object contains the surface mesh for your geometry and is created as a default object when you load a file into CFD-Post. You can change how much of the surface mesh you want to see by altering the Edge Angle (see the following section), as well as the line thickness and color.

You toggle the visibility of the wireframe on and off by clicking on the **Wireframe** check box in the **Outline** tree view. To change the way the wireframe displays, double-click **Wireframe**.

Note:

You cannot create additional Wireframe objects.

3.4.4.1.1. Wireframe: Definition Tab

The **Definition** tab contains the settings listed below. After making changes, click **Apply** to make those changes visible.

- **Domains** specifies the domains on which the wireframe is displayed. The option **All Domains** refers to all domains *except* immersed solid domains. To include all immersed solid domains, add **All ImmersedSolid Domains** to the selection.

- **Show Surface Mesh** controls whether you see edges and surfaces, or only edges, when the wireframe is visible.
- **Edge Angle** determines how much of the wireframe is drawn. The *edge angle* is the angle between one edge of a mesh face and its neighboring face. Setting an edge angle in CFD-Post defines a minimum angle for drawing parts of the surface mesh. For example, if an edge angle of 30 degrees is chosen, any edges shared by faces with an angle between them of 30 degrees or more is drawn. 30 degrees is the default edge angle; if you want to see more of the wireframe, reduce the edge angle. To change the wireframe's edge angle, set **Edge Angle** to a new value.
- **Color Mode** determines the color of the lines in the wireframe. To change the wireframe's line color, set **Color Mode** to **User Specified** and click the color bar to select a new color.
- **Line Width** determines the thickness of the lines in the wireframe. To change the wireframe's line width, set **Line Width** to a new value.

3.4.4.1.2. Wireframe: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

3.4.5. Report

CFD-Post automatically makes available a report of the output of your simulation. You can control the contents of the report in the **Outline** workspace, see the available sections of the report in the **Report Viewer**, add new sections in the **Comment Viewer**, and publish the report in HTML or in plain text form.

Here is an example of a report that uses the generic template; if you have a RES file loaded in CFD-Post, you can see a similar report by clicking on the **Report Viewer** tab at the bottom of the Viewer area.

Note:

The sample report shown in [Figure 3.2: A Sample Report, Part 1 \(p. 73\)](#), [Figure 3.3: A Sample Report, Part 2 \(p. 74\)](#), and [Figure 3.4: A Sample Report, Part 3 \(p. 75\)](#) is taken from a Report.html file, much like the one that you generate when you click the **Publish** button



Figure 3.2: A Sample Report, Part 1

Figure 3.3: A Sample Report, Part 2

2. Mesh Report

Table 2. Mesh Information for Buoyancy2D_001

Domain	Nodes	Elements
Buoyancy2D	3322	1560

3. Physics Report

Table 3. Domain Physics for Buoyancy2D_001

Domain - Buoyancy2D	
Type	Fluid
Location	Primitive 3D
<i>Materials</i>	
Air at 25 C	
Fluid Definition	Material Library
Morphology	Continuous Fluid
<i>Settings</i>	
Buoyancy Model	Buoyant
Buoyancy Reference Temperature	40.000 [C]
Gravity X Component	-4.900 [m s ⁻²]
Gravity Y Component	-8.500 [m s ⁻²]
Gravity Z Component	0.000 [m s ⁻²]
Buoyancy Reference Location	Automatic
Domain Motion	Stationary
Reference Pressure	0.000 [Pa]
Heat Transfer Model	Thermal Energy
Turbulence Model	Laminar

Figure 3.4: A Sample Report, Part 3

Table 4. Boundary Physics for Buoyancy2D_001

Domain	Boundaries	
Buoyancy2D	Boundary - Buoyancy2D Default	
	Type	WALL
	Location	Primitive 2D A, Primitive 2D B
	<i>Settings</i>	
	Heat Transfer	Adiabatic
	Wall Influence On Flow	No Slip
	Boundary - SymP	
	Type	SYMMETRY
	Location	SYMMET1, SYMMET2
	Boundary - cold	
	Type	WALL
	Location	WALLCOLD
	<i>Settings</i>	
	Heat Transfer	Fixed Temperature
	Fixed Temperature	5.000 [C]
	Wall Influence On Flow	No Slip
	Boundary - hot	
	Type	WALL
	Location	WALLHOT
	<i>Settings</i>	
	Heat Transfer	Fixed Temperature
	Fixed Temperature	75.000 [C]
	Wall Influence On Flow	No Slip

A report is defined by the Report object and the objects stored under it. The Report object, like other objects, can be saved to, and restored from, a state file. For details, see [File Types Used and Produced by CFD-Post](#) (p. 169). Only one Report object exists in a CFD-Post session.

3.4.5.1. Omitting Default Report Sections

You can remove major sections of the report by expanding the **Report** section in the **Outline** tree view and clearing the check box beside the section to be omitted. To see the results of such operations, right-click **Report** and select **Refresh Preview**.

Note:

The **Title Page** option controls the inclusion of the logo, title, dates, and Table of Contents sections.

3.4.5.2. Changing the Default Report Sections

To change a default section, right-click the section name in the **Outline** tree view and click **Edit**.

For the Title Page, you can:

- Add a new logo (JPG or PNG only)
- Remove the Ansys logo
- Change the report's title
- Add the author's name
- Control the display of the date and the table of contents.

After making changes, click **Apply** and **Refresh Preview** to see the results in the **Report Viewer**.

The other report pages control detailed information about the mesh, physics, and solution. Again, after making changes, click **Apply** and **Refresh Preview** to see the results in the **Report Viewer**.

3.4.5.3. Adding New Sections to a Report

You can add new sections by using the **Comment Viewer** (at the bottom of the viewer pane):

1. In the **Comment Viewer** toolbar, click *New Comment*  to ready the **Comment Viewer** for editing.
2. Add a title for your new section in the **Heading** field.
3. Set the level of the heading in the Level field (use "1" for new sections; "2" for subsections, and so on).
4. Type your text in the large, white text-entry field (HTML code is not accepted because it is generated automatically).
5. When your new section is complete, select its name in the **Outline** tree view under **Report**, then press **Ctrl+Up Arrow** (or **Ctrl+Down Arrow**) to move the new section in the report hierarchy.
6. To see how the report will look, right-click **Report** and select **Refresh Preview**. The updated report appears in the **Report Viewer**.
7. To publish the report so that it can be loaded into a third-party browser or editor, right-click **Report** and select **Publish**. For details, see [Publishing the Report \(p. 87\)](#).
8. To save the report, click **OK**. The report is written to the file you specified.

3.4.5.4. Report Templates

Report templates are available for rapidly setting up application-specific reports. Depending on the information contained in a results file, a report template will be selected automatically, and made available as a command in the following places:

- The **File > Report** menu
- The shortcut menu that appears when you right-click the Report object.

In the same places, there is a **Report Templates** command that invokes the **Report Templates** dialog box. This dialog box allows you to:

- Browse the list of existing templates.
- Add (register) a template.

To do this:

1. Click *Add template*  on the **Report Templates** dialog box to invoke the **Template Properties** dialog box.
 2. Select a state or session file that contains a report; alternatively, you can choose to use the current state of CFD-Post, and provide a filename to which to save the template.
 3. Provide a name and description for the template. You cannot use the name of an existing template.
 4. If you are loading a state or session file, and the filename does not end in .cst or .cse, set the **Execution** setting to either **State** or **Session**, as applicable.
- View and/or edit the properties of a template.

To do this:

1. Click *Edit Properties*  on the **Report Templates** dialog box to invoke the **Template Properties** dialog box.
 2. View and/or edit the name, description, and path to the template file, as applicable. You can edit the properties for templates that were added, but not the standard templates.
- Delete templates from the set of available templates.

You can delete only the templates that were added. To do this:

1. Select a user template in the **Report Templates** dialog box.
2. Click *Delete* .

3.4.5.4.1. Turbo Report Templates

Postprocessing within CFD-Post is fully automated using the turbomachinery report templates. The turbo reports are designed for single-phase fluid analyses, and can only be used with steady-state cases. The turbo reports can be used for individual bladerows of a multi-bladerow analysis by loading each bladerow domain separately into CFD-Post using the domain selector. (To enable

the domain selector, click the **Edit > Options** menu, select **Files** and select the **Show domain selector before load** option.)

Important:

- Turbo reports attempt to auto-initialize Turbo mode. However if auto-initialization fails, you must initialize Turbo mode manually and re-run the turbo report.
- CFD-Post cannot automatically detect a solution that is "360 Case Without Periodics", so you need to set this manually.
- Turbo report templates are not designed for multifile usage or comparison mode. In these cases:
 - User charts that contain local variables will not have plots showing the differences in comparison mode.
 - Tables will not show differences in comparison mode.
 - There will be only one picture of the meridional view of the blades (corresponding to the first loaded results file).
- Because transient blade row results are different in each passage, Turbo Reports are not designed for transient blade row cases, and results may not be what is expected. Plots in Turbo reports that appear to show multiple passages actually show copies of the first passage and not expanded passages. In other words, the turbo report tool follows the same behavior as any other solution method; that is, it makes an instanced copy of the first passage and plots the variables there.

These are the variables required for all Release 2025 R1 turbo reports:

CFX Variables Required for all Release 2025 R1 Turbo Reports

- Density
- Force X
- Force Y
- Force Z
- Pressure
- Total Pressure in Stn Frame
- Rotation Velocity
- Velocity
- Velocity in Stn Frame u
- Velocity in Stn Frame v

- Velocity in Stn Frame w
- Velocity in Stn Frame
- Velocity in Stn Frame Flow Angle
- Velocity Flow Angle
- Velocity in Stn Frame Circumferential
- Velocity Circumferential
- Velocity Meridional

Note:

If all of the turbo components in the results file are 'stationary', then variables having names ending with 'in Stn Frame' are not required.

Fluent Variables Required for all Release 2025 R1 Turbo Reports

- Density
- Static Pressure
- Total Pressure
- X Velocity
- Y Velocity
- Z Velocity

In addition to the variables mentioned above, the following variables are required for compressible flow reports:

CFX Variables Required for all Release 2025 R1 Compressible Flow Turbo Reports

- Temperature
- Total Temperature in Stn Frame
- Static Enthalpy
- Total Enthalpy
- Total Enthalpy in Stn Frame
- Isentropic Total Enthalpy
- Polytropic Total Enthalpy
- Total Density in Stn Frame
- Specific Heat Capacity at Constant Pressure

- Specific Heat Capacity at Constant Volume
- Rothalpy
- Static Entropy
- Mach Number
- Mach Number in Stn Frame
- Isentropic Compression Efficiency
- Isentropic Expansion Efficiency

Fluent Variables Required for all Release 2025 R1 Compressible Flow Turbo Reports

- Static Temperature
- Total Temperature
- Enthalpy
- Total Enthalpy
- Specific Heat (Cp)
- Rothalpy
- Entropy
- Mach Number

When variables are missing, lines in the turbo report tables that depend on these variables will be missing.

3.4.5.4.1.1. Procedures for Using Turbo Reports when Turbomachinery Data is Missing

Results files from Fluent (and from some other sources) will not have all the turbomachinery data that CFD-Post requires. For turbo results files that lack data about the number of passages, you must do the following:

1. For Fluent files, prior to loading a turbo report template, create a new variable that the report expects (but which is not available from Fluent files):
 - a. From the toolbar, click *Variable* . The **Insert Variable** dialog box appears.
 - b. In the **Name** field, type **Rotation Velocity** and click **OK**. The details view for **Rotation Velocity** appears.
 - c. In the **Expression** field, type $\text{Radius} * \text{abs}(\omega) / 1 \text{ [rad]}$ and click **Apply**. This expression calculates the angular speed (in units of length per unit time) as a product of the local radius and the rotational speed.

2. When you load a turbo report for a case that is missing some variables, an error dialog box appears showing warnings and errors. Generally this means that some rows in the turbo report will not appear.

Turbo reports for Fluent files will not display information about absolute Mach number. This causes charts of Mach number to display only the relative Mach number.

3. For any results file that is missing the number of passages (such as Fluent files and CFX results files not set up using the Turbo Mode in CFX-Pre), after you load the turbo report template, do the following for each domain:
 - a. A **<domain_name> Instance Transform** appears in the **Outline** tree view under **User Locations and Plots**. Prior to viewing the report, double-click this name to edit the instance transform. In the **# of Passages** field, ensure that the number of passages matches the number of passages in the domain. If you enter a new number, click **Apply**.
 - b. On the **Expressions** tab, double-click the expression **<domain_name> Components in 360** to edit it. Match the definition to the number of components in the domain. If you enter a new number, click **Apply**.
4. In the **Report Viewer**, click **Refresh** to ensure that the contents are updated.

3.4.5.4.2. Choosing a Turbo Report

If the model was set up using CFX-Pre Turbo Mode, then CFD-Post will automatically be able to determine which report to load based on the machine type and flow type selected, and will prompt you to load it. The report can also be loaded manually by right-clicking on the **Report** item in CFD-Post. To avoid conflicts with the current CFD-Post state, you should load the report in a clean session.

In the Computed Results table, any values that are shown as "N/A" mean that the necessary scalar variables for computing these values were missing from the results file. This might happen if the solution was run using an older version of CFX, or if some of the scalar variables were manually disabled.

The report templates are CFD-Post session files located in the CFX install under the etc/PostReports directory. You can edit these reports or make new versions to add to the list of report templates. The following table shows the correspondence between the machine type, the flow type settings, and the report selection.

Machine Type	Fluid Type	Domain Motion (Single Domains Only)	Report Template
Pump	Any	Rotating	Pump Impeller
		Stationary	Stator
Fan	Any	Rotating	Fan
		Stationary	Stator
Fan	Any	Rotating	Fan Noise
Axial Compressor	Compressible	Rotating	Axial Compressor Rotor

Machine Type	Fluid Type	Domain Motion (Single Domains Only)	Report Template
		Stationary	Stator
Centrifugal Compressor	Compressible	Rotating	Centrifugal Compressor Rotor
		Stationary	Stator
Axial Turbine	Compressible	Rotating	Turbine Rotor
		Stationary	Turbine Stator
Radial Turbine	Compressible	Rotating	Turbine Rotor
		Stationary	Turbine Stator
Hydraulic Turbine	Incompressible	Rotating	Hydraulic Turbine Rotor
		Stationary	Stator
Other	Any	Rotating	Pump Rotor
		Stationary	Stator

Several reports support postprocessing results for multiple components/blade rows. These reports attempt to group the components into stages. You can control how the stages are formed by editing the corresponding report session file. The available session files include:

AxialCompressorReport.cse

Report template for axial compressors

CentrifugalCompressorReport.cse

Report template for centrifugal compressors

CompressibleTurbineReport.cse

Report template for compressible flow turbines.

HydraulicTurbineReport.cse

Report template for incompressible flow turbines.

PumpReport.cse

Report template for incompressible flow pumps.

Machine Type	Fluid Type	Report Template
Pump	Any	Pump
Axial Compressor	Compressible	Axial Compressor
Centrifugal Compressor	Compressible	Centrifugal Compressor
Axial Turbine	Compressible	Turbine
Radial Turbine	Compressible	Turbine

Machine Type	Fluid Type	Report Template
Hydraulic Turbine	Incompressible	Hydraulic Turbine

3.4.5.5. Creating, Viewing, and Publishing Reports

To create or modify a report, do the following:

1. Specify the settings for the report that are contained in the **Report** object.
For details, see [Report Object \(p. 83\)](#).
2. Specify the settings for the title page that are contained in the **Title Page** object.
For details, see [Title Page Object \(p. 84\)](#).
3. Decide which predefined tables to use.
For details, see:
 - [File Report Object \(p. 85\)](#)
 - [Mesh Report Object \(p. 85\)](#)
 - [Physics Report Object \(p. 85\)](#)
 - [Solution Report Object \(p. 85\)](#)
 - [Operating Points Report Object \(p. 86\)](#)
4. Optionally, create objects that give additional content to the report.
For details, see [Adding Objects to the Report \(p. 86\)](#).
5. Control which objects get included in the report, and the order in which they are included.
For details, see [Controlling the Content in the Report \(p. 87\)](#).
6. Refresh the report.
For details, see [Refreshing the Report \(p. 87\)](#).

You may refresh the report at any time to see the effect of changes you make to the report settings and content. The report appears in the **Report Viewer**.

You can publish a report so that it can be loaded into a third-party browser or editor. For details, see [Publishing the Report \(p. 87\)](#).

3.4.5.5.1. Report Object

The settings on the **Appearance** tab of the **Report** object are described next.

3.4.5.5.1.1. Figures: File Type

Choose the image format in which you want the image files to be saved.

3.4.5.5.1.2. Figures: Figure Size

Choose the size for figures that appear in the report. There are preset sizes, and an option for setting a custom size.

3.4.5.5.1.3. Figures: Width and Height

If you set **Figure Size** to Custom, set the figure width and height in pixels.

3.4.5.5.1.4. Figures: Fit all figures in the viewport before generation Check Box

When this option is selected, each figure is produced with the view centered and the zoom level set automatically.

3.4.5.5.1.5. Charts: File Type

Choose the image format in which you want the chart files to be saved.

3.4.5.5.1.6. Charts: Chart Size

Choose the size for charts that are saved as part of the report. There are options that specify preset sizes, an option for using the same size as figures, and an option for setting a custom size.

3.4.5.5.1.7. Charts: Width and Height

If you set **Chart Size** to Custom, set the chart width and height in pixels.

3.4.5.5.2. Title Page Object

The **Title Page** object is automatically generated and listed under the **Report** object. The settings of this object determine the content of the title page, and are described next.

3.4.5.5.2.1. Custom Logo Check Box

The **Custom Logo** check box determines whether or not a custom logo is included in the title page. The logo must be available in a file of compatible format.

3.4.5.5.2.2. Custom Logo

The **Custom Logo** setting indicates the image file to use for the custom logo.

3.4.5.5.2.3. Ansys Logo Check Box

The **Ansys Logo** check box determines whether or not the Ansys logo is included in the title page. The Ansys logo is shown in [Figure 3.2: A Sample Report, Part 1 \(p. 73\)](#).

3.4.5.5.2.4. Title

The **Title** setting holds the title of the report.

3.4.5.5.2.5. Author

The **Author** setting holds the name of the author of the report.

3.4.5.5.2.6. Current Date Check Box

The **Current Date** check box determines whether or not the date and time are included in the title page.

3.4.5.5.2.7. Table of Contents Check Box

The **Table of Contents** check box determines whether or not a table of contents is included in the title page. An example is shown in [Figure 3.2: A Sample Report, Part 1 \(p. 73\)](#).

Each entry in the table of contents is an active link to the corresponding section of the report. To follow a link, click the link using the left mouse button.

3.4.5.5.2.8. Table of Contents Check Box: Captions in Table of Contents Check Box

The **Captions in Table of Contents** check box controls the level of detail in the entries in the table of contents. When this check box is selected, the entries in the table of contents that link to objects in the report contain the titles of the objects.

3.4.5.5.3. File Report Object

A **File Report** object containing a file information table, is automatically generated for each loaded results file, and listed under the **Report** object. There are no user-adjustable settings except the check box in the tree view, which controls whether or not the file information is included in the report. An example of the file information table is shown in [Figure 3.2: A Sample Report, Part 1 \(p. 73\)](#).

3.4.5.5.4. Mesh Report Object

A **Mesh Report** object is automatically generated and listed under the **Report** object. The **Mesh Report** object contains settings for a mesh information table and a mesh statistics table. Examples of the mesh information table and mesh statistics tables are shown in [Figure 3.3: A Sample Report, Part 2 \(p. 74\)](#). The data in these tables are the same as given by the mesh calculator. For details, see [Mesh Calculator \(p. 371\)](#).

3.4.5.5.5. Physics Report Object

A **Physics Report** object is automatically generated and listed under the **Report** object only when you load a CFX-Solver results file. The **Physics Report** object allows you to control the output of physics summary data for domains and boundaries. Examples of the physics summary tables are shown in [Figure 3.3: A Sample Report, Part 2 \(p. 74\)](#).

3.4.5.5.6. Solution Report Object

A **Solution Report** object is automatically generated and listed under the **Report** object only when you load a CFX-Solver results file. The **Solution Report** object allows you to

control the output of boundary flow, force, and torque summaries in the report. CFD-Post uses the summary data contained in the results files.

Note:

- The units of the values displayed in a `Solution Report` object are based on the solution units used by CFX-Solver, rather than those selected in CFD-Post.
 - The `Solution Report` object uses monitor data from the CFX-Solver results file. Only the monitor data from the last time step or iteration will be used to generate the report table, regardless of which time step is loaded into CFD-Post for a transient case.
 - Some CFX-Solver cases, particularly some transient blade row cases, do not contain the necessary monitor data to generate the tables. For these cases, the tables will be omitted even if the `Solution Report` object is enabled.
-

3.4.5.5.7. Operating Points Report Object

An `Operating Points` object is automatically generated and listed under the `Report` object only when you load a CFX-Solver results file from an operating point case (`.mres`).

The operating point parameter table for a given case is listed under `Report > Operating Points > [case name]`.

The details view for the operating point parameter table has no settings.

To see the operating point parameter table in the **Table Viewer**, double-click it in the **Outline** tree view, or right-click it and select **Edit**.

The operating maps for a given case are listed under `Report > Operating Points > [case name]`, and again under `Cases > [case name] > Operating Maps`.

The details view for an operating map is described in [Operating Maps \(p. 68\)](#).

To see an operating map in the **Operating Points Viewer**, double-click it in the **Outline** tree view, or right-click it and select **Edit**.

3.4.5.5.8. Adding Objects to the Report

You can create objects of the following types to add additional content to the report:

- Tables

For details, see [Table Command \(p. 300\)](#).

- Charts

For details, see [Chart Command \(p. 305\)](#).

- Comments

For details, see [Comment Command \(p. 327\)](#).

- Figures

For details, see [Figure Command \(p. 328\)](#).

Such objects are listed beneath the Report object in the tree view.

3.4.5.5.9. Controlling the Content in the Report

Report objects can be shown or hidden in the report by setting the check box next to them in the **Outline** tree view. The changes take effect the next time the report is refreshed or published.

You can control the order of Report objects by selecting one or more, then right-clicking on the selection and using the **Move Up** and **Move Down** shortcut menu commands as necessary. You can also press **Ctrl+Up Arrow** and **Ctrl+Down Arrow** to move selected items.

3.4.5.5.10. Refreshing the Report

To refresh the report, you can do any of the following:

- Right-click the Report object, or any of the report objects under it, then select **Refresh Preview** from the shortcut menu.
- Click the **Refresh Preview** button in the details view for the Report object, or any of the report objects under it that have this button.
- Click the **Refresh** button in the **Report Viewer**.
- Select **File > Report > Refresh Preview**.

Note:

The first time you visit the **Report Viewer** after loading a results file, the report will be refreshed automatically.

3.4.5.5.11. Viewing the Report

After the report preview has been generated, you can view it in the **Report Viewer**.

3.4.5.5.12. Publishing the Report

You can publish a report so that it can be loaded into a third-party browser or editor. To publish a report, click the **Publish** button in the **Report Viewer** toolbar to access the **Publish Report** dialog box, adjust settings as appropriate, and click **OK**. You can also access the same dialog box by doing any of the following:

- Right-click the Report object, or any of the report objects under it, then select **Publish**.
- Select **File > Report > Publish**.

3.4.5.5.12.1. Format

Set **Format** to one of:

- ARZ

Writes the report in ARZ format, an Ansys zipped report format that can be viewed using the Ansys Viewer. For details, see [Ansys Viewer Basics in the Ansys Viewer User's Guide](#).

- HTML

Writes the report in HTML format.

- Text

Writes the report in a plain text format.

Note:

Existing HTML reports previously generated can be converted to .arz format by using an application supplied as part of the CFD-Post installation:

```
<CFDPOSTROOT>/bin/cfx5htmlconvert <html-file>
```

3.4.5.5.12.2. File

Set **File** to the filename to use for saving the report.

3.4.5.5.12.3. Figures

You have the following options:

- 2D and 3D images

Both a 2D and 3D image is generated for each figure in the report. The 3D image can be viewed by clicking the 2D image in the report.

- 2D images only

Only a 2D image is generated for each figure in the report.

3.4.5.5.12.4. Save images in separate folder Check Box

(applicable for HTML and Text formats)

Selecting the **Save images in separate folder** check box causes all image files to be put in a directory that is beside the main output file.

3.4.5.5.12.5. More Options Button

The **More Options** button opens the **Publish Options** dialog box. The **Publish Options** dialog box offers the same settings as the Report object (accessible through the **Outline** tab), and overrides the latter for the purpose of publishing the report.

Note:

The **Publish Options** dialog box settings are overwritten with the settings of the Report object if you change or otherwise apply the settings of the latter.

3.4.6. Display Properties and Defaults

The **Display Properties and Defaults** branch of the **Outline** tree view contains a **Color Maps** area that is divided into **Custom** and **System** area. Initially, only the **System** area has entries; these are the default color map names. You can use any of the default color maps as the basis for color maps that you define, which are stored in the **Custom** color map area.

To learn how customize color maps, see [Color Map Command \(p. 298\)](#).

3.5. Variables Workspace

The **Variables** workspace is used to create new user variables and modify existing variables.

The following topics will be discussed:

- [Variables Tree View \(p. 89\)](#)
- [Variables Details View \(p. 91\)](#)
- [Variables: Example \(p. 95\)](#).

3.5.1. Variables Tree View

The **Variables** tree view displays all variables corresponding to a particular case or pair of cases. Variables are categorized under the following folders:

- The **Derived** folder contains variables that are automatically generated by CFD-Post. The **Vortex Core** subfolder contains any variables related to vortex cores. For details on vortex cores, see [Vortex Core Region in the CFD-Post User's Guide \(p. 241\)](#).
- The **Difference** folder contains differences in variables between two cases. The **Difference** folder becomes available only after **Case Comparison** has been enabled through the **Outline** workspace. For details on **Case Comparison**, see [Case Comparison in the CFD-Post User's Guide \(p. 373\)](#).
- The **Geometric** folder contains mesh statistics such as element volume, edge length ratio and minimum face angle.

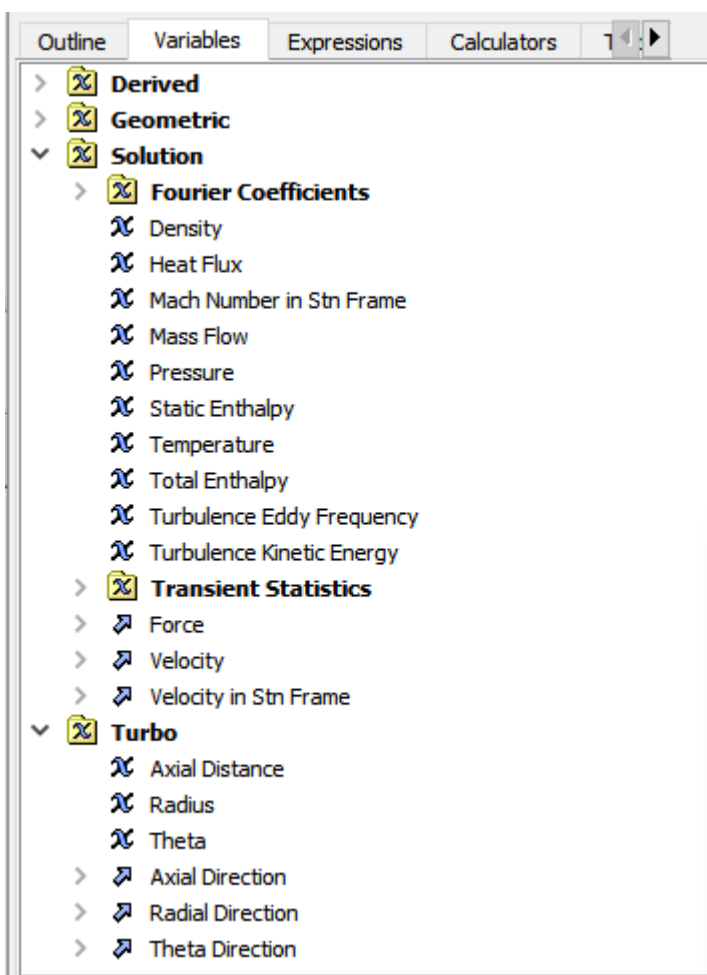
- The **Solution** folder contains variables generated by solver applications. The subfolders **Residuals** and **Corrections** contain variables related to solution quality. The subfolders **Fourier Coefficients** and **Transient Statistics** contain variables related to transient cases.

Note:

When the results file from a transient blade row case is loaded, CFD-Post creates all the variables with available Fourier Coefficients. The timestep selector defaults to the last time step in the simulation. Fourier Coefficient vector variables are available only for the non-expanded domain and are not oriented correctly.

- The **Turbo** folder contains variables related to turbomachinery, particularly those involving cylindrical coordinates. The variable list in the **Turbo** folder expands if you initialize turbo components through the **Turbo** workspace. For details on the **Turbo** workspace, see [Turbo Workspace \(p. 381\)](#).
- The **User Defined** folder contains any new variables created by the user. For details, see [User Variables \(p. 93\)](#).
- The **User Locations and Plots** folder contains variables related to streamlines, particle tracks and user locations from external files.

Variables prefixed by a particular material type are grouped in subfolders. If the variable belongs to another subfolder, such as **Vortex Core** or **Fourier Coefficients**, the material type takes priority. For example, `gas.velocity.helicity` appears under **Derived > Gas > Vortex Core**.



Variables Tree View Shortcuts

The following table shows commands that are specific to the **Variables** tree view. For a description of how to access these shortcuts, and a list of commands that appear in most tree views, see [Common Tree View Shortcuts](#) (p. 49).

Command	Description
All to Conservative	Makes all variables assume conservative values. For details, see Hybrid and Conservative Variable Values .
All to Hybrid	Makes all variables assume hybrid values. For details, see Hybrid and Conservative Variable Values .
Calculate Velocity Components	Calculates velocity components using the global rotation axis. This can also be done in the Turbo workspace. For details, see Calculate Velocity Components (p. 408).

3.5.2. Variables Details View

The **Variable** details view is used to change the definition of fundamental (system) variables, and to create and edit user variables.

To edit an existing variable, either:

- Double-click the variable in the tree view
- Right-click the variable, then choose **Edit** from the shortcut menu.

The above actions cause the **Variables** details view to appear.

3.5.2.1. Fundamental Variables

Fundamental variables (variables provided by the solver) can have their units changed. This would enable you to create a legend that uses alternative temperature units (such as degrees Celsius).

Note:

These settings override the global units setting (defined in the **Options** dialog box, accessible from the **Edit** menu).

1. Toggle between **Hybrid** and **Conservative** values.

This affects any dependent variables and expressions as well. For details, see [Hybrid and Conservative Variable Values](#).

2. Select the units.
3. Click **Reset** to restore the variable settings stored in the database.

Use this to undo changes if you have not yet clicked **Apply**.

3.5.2.1.1. Saving Variables Back to the Results File

In cases other than transient blade row cases, fundamental variables can be redefined using expressions and then saved back into the results file for later use. To do this, select the **Replace with expression (write to results)** check box, enter one or more expressions, then click **Apply**. To change a vector variable, you must write three expressions: one expression for each direction (X, Y, and Z). The result file is updated when you click **Apply**. To restore a fundamental variable to its original state, clear the **Replace with expression (write to results)** check box and click **Apply**.

One reason for modifying the variables in a results file is to modify the initial conditions for a new solver run. In this case, you must modify the principal variable for each affected equation.

In CFX-Pre, in most cases, the principal variable for a given equation is the same as the variable used to specify initial conditions, but there are some exceptions, as shown in the table below:

Equation	CFX-Pre Variable	Principal Variable
Thermal Energy	Temperature	Static Enthalpy
Total Energy	Temperature	Total Enthalpy
Mass Fractions	Mass Fraction	Conservative Mass Fraction
Volume Fractions	Mass Fraction	Conservative Volume Fraction
Continuity (with cavitation activated)	Pressure	Solver Pressure

For example, to initialize the mass fractions equation in CFX-Pre, you would set `Mass Fraction`. In order to modify the initial conditions for the same equation in a results file, you would set `Conservative Mass Fraction` instead.

Note:

- For the thermal energy and total energy equations, you must set `Temperature` as well as the principal variable.
 - When overwriting the mesh `Total Mesh Displacement`, the locations of the mesh nodes in CFD-Post will not be affected, only the variable values.
 - For transient blade row cases, solution variables will not be overwritten because the variables displayed in CFD-Post are not directly loaded from the results file, but are reconstructed from the available Fourier coefficients stored in the results file.
 - The variable ranges stored in the results file are not updated to reflect the modified variable values. This implies that if the new variable data is used to modify the initial conditions for a restart, the "Variable Range Information" displayed in the CFX-Solver Output file at the start of the run will not reflect the new variable values.
-

3.5.2.2. Radius and Theta

The variables `Radius` and `Theta` are available only when the rotational axis has been defined. The rotational axis can either be defined in the results file or in CFD-Post through the **Initialization** panel in the **Turbo** workspace.

3.5.2.3. Boundary-Value-Only Variables

Some variables in the CFX results file take meaningful values only on the boundaries of the geometry. Examples of this sort of variable are `Yplus`, `Wall Shear`, `Heat Transfer Coefficient`, and `Wall Heat Flux`. For details, see [CFX-Solver Output File in the CFX-Solver Manager User's Guide](#).

To obtain sensible plots when using these variables, use them to color only boundary objects. If, for example, you try to color a slice plane through the center of the geometry with one of these variables, you will see a large area of color that is meaningless; only at the very edges of the geometry will there be useful coloration.

For boundary-value-only variables, only hybrid values exist (as they are undefined away from a boundary).

3.5.2.4. User Variables

To create a new user variable, click **Insert > Variable**, or right-click a variable in the tree view and select **New** from the shortcut menu.

There are three basic types of user variables, depending on the value of the **Method** setting:

- Expression

The **Expression** user variable is defined by one or three expressions, depending on whether the **Scalar** or **Vector** option is selected. You can enter new expressions or select existing expressions. For details, see [Expressions Workspace](#) (p. 96).

If **Calculate Global Range** is selected then, after you click **Apply**, the range data is computed and displayed in the details view. If **Calculate Global Range** is not selected when you click **Apply** then you still have the option of selecting this option and clicking **Apply** again in order to compute and display the range data.

For an **Expression** user variable, *and any user variable that depends on it*, the **Boundary Data** setting (which can normally be set to **Hybrid** or **Conservative**) is not applicable. Whenever the defining expression is evaluated, the values of any underlying independent variables are hybrid and/or conservative in accordance with the **Boundary Data** settings (at the time of evaluation) of those independent variables. The defining expression of an **Expression** user variable is automatically re-evaluated whenever you change the **Boundary Data** setting of any underlying independent variable.

- **Frozen Copy**

At the time you click **Apply**, the **Frozen Copy** user variable is defined by copying the current values of an existing *scalar* variable (which is specified by the **Copy From** setting). Hybrid and conservative values are copied as available. If both hybrid and conservative values are available, then after you click **Apply** to create the **Frozen Copy** user variable, the **Boundary Data** setting becomes available, enabling you to select between hybrid or conservative values; your selection affects all objects and expressions that depend on the **Frozen Copy** user variable. Toggling between hybrid and conservative selects data within the copy. It does not cause data to be copied again from the **Copy From** variable.

The **Frozen Copy** user variable values remain constant even if the variable from which the copy was made subsequently changes in any way, for example, by switching to a different time step.

After you create a **Frozen Copy** user variable, changing the **Copy From** variable and clicking **Apply** causes the **Frozen Copy** user variable to be redefined at that time.

Note:

If a **Frozen Copy** is created from another user variable of the **Expression** type, the plotting or calculation from data on element faces is not supported.

- **Gradient**

The **Gradient** user variable is defined as a vector variable that represents the gradient of the selected scalar variable, or the gradient of the magnitude of the selected vector variable.

If both hybrid and conservative values are available for the selected variable, then after you click **Apply** to create the **Gradient** user variable, the **Boundary Data** setting becomes available,

enabling you to select between hybrid or conservative values; your selection affects all objects and expressions that depend on the Gradient user variable.

Note:

- You cannot create a variable with the same name as an existing expression or object.
 - To preserve Frozen Copy user variables between sessions, you can use the [New Session Command \(p. 215\)](#) to record your current session in a session file. Note that state files will not preserve your Frozen Copy user variables.
-

3.5.3. Variables: Example

In this example, you will use an expression to create an Isosurface that is a fixed radial distance from an axis or point. For details, see [Expressions Workspace: Example \(p. 99\)](#). Before trying this example, you must first create the expression in the aforementioned example.

1. Copy the StaticMixer_001.res file (provided with a tutorial) to your working directory and load it into CFD-Post.
2. Click the **Variables** tab.
3. Click  in the **Variables** details view to create a new variable.
4. When the **New Object** window appears, type the name Radial Distance, and then click **OK**.
5. In the variable details view, set **Expression** to radial (which is the expression you created earlier).
6. Click **Apply** to create the new variable.

This variable appears in the tree view and can be used like any other variable. Notice that the variable is listed as **User Defined**.

You can now create an Isosurface using this variable as follows:

1. Select **Insert > Location > Isosurface**.
2. In the **New Isosurface** dialog box, enter a name and then click **OK**.
3. On the **Geometry** tab for the Isosurface:
 - a. Set **Variable** to Radial Distance.
 - b. Set **Value** to 1 [m].

This is a suitable value for results from the StaticMixer_001.res file. You may need to alter this value to something sensible depending on the results you are viewing.

4. Click the **Color** tab and set the **Mode** option to **Variable**. Select a sensible variable (such as, Temperature or Velocity) with which to color the isosurface.
5. Set the **Range** option to **Local** so that the full color range is used on the Isosurface.
6. Click **Apply** to create the isosurface.

You should now see a cylindrical Isosurface centered about the Z-axis. All points on the Isosurface are a distance of 1 m (or the value you used in the **Value** box) from the Z-axis. Note that a cylinder can also be created as a surface of revolution. For details, see [Surface of Revolution Command \(p. 248\)](#). Additional information on expressions is available; for details, see [Further Expressions \(p. 100\)](#).

3.6. Expressions Workspace

The **Expressions** workspace is used to select and generate expressions using the CFX Expression Language (CEL), which you can then use in CFD-Post in place of almost any numeric value (as long as the correct units are returned by the expression).

Note:

- When a setting is defined by an expression, and the latter evaluates to a quantity that has no units, the software internally applies the default units for that setting.
 - In an expression, a term that has no units can be added to a term that has angular units, in which case the software internally applies radians to the term that has no units.
-

The following topics will be discussed:

- [Expressions Tree View \(p. 97\)](#)
- [Expressions Workspace: Expressions Details View \(p. 97\)](#)
- [Expressions Workspace: Example \(p. 99\)](#)

You should be aware of the guidelines regarding expressions:

- When using expressions in multifile and case-comparison situations, a specific expression syntax is employed. For further information, see [Examples of the Calling Syntax for an Expression in the CFX Reference Guide](#).
- You cannot create an expression with the same name as an object or variable.
- Within the CFX Expression Language, some variables are known by short names to save typing the full variable name. For example, **p** refers to **Pressure**. Although it is possible to create an expression with the same name as an abbreviated variable, it is ignored. For example, if you define an expression named **p** with the definition **5 [K]**, an expression defined as **2*p** represents **2*Pressure**, not **10 [K]**.

- You must always provide units inside square brackets for constant values typed into an expression.

Note:

CFD-Post and the CFX-Solver evaluate expressions differently:

- CFD-Post evaluates expressions on slice planes by first interpolating the variables in the expression to the "plane points" (that is, the places where the plane is cut by mesh edges), and then evaluates the expression.
- The CFX-Solver evaluates expressions on the vertices and then interpolates to the plane points.

The results given by these two approaches (evaluate and then interpolate vs. interpolate and then evaluate) will differ most significantly where the variable gradients are large.

3.6.1. Expressions Tree View

The following table shows commands that are specific to the **Expressions** tree view and are accessed by right-clicking an expression in the tree view. For a list of shortcuts that appear in most tree views, see [Common Tree View Shortcuts](#) (p. 49).

Command	Description
Use as Workbench Input Parameter	Specifies the expressions that are to be used as parameters in a Design Exploration session. These parameterized expressions are saved to the CFD-Post state file. To parameterize an expression, right-click the expression and select Use as Workbench Input Parameter or Use as Workbench Output Parameter . The icon next to the expression changes to help identify it as a parameterized expression.
Use as Workbench Output Parameter	

3.6.2. Expressions Workspace: Expressions Details View

The **Expressions** details view contains the following tabs:

- [Expression Definition Tab](#) (p. 97)
- [Plot Expression Tab](#) (p. 98)
- [Evaluate Expression Tab](#) (p. 98)

3.6.2.1. Expression Definition Tab

You can access lists of variables, expressions, locators, functions and constants by right-clicking in the definition window when defining an expression. Although valid values can be chosen from each of the various lists, the validity of the expression itself is not checked until you click **Apply**. For details, see [CEL Operators, Constants, and Expressions](#) and [CFX Expression Language \(CEL\) in CFD-Post](#) (p. 419).

Any expressions not containing variables are evaluated when you click **Apply**.

1. Enter the definition of a new expression or edit the definition of an existing expression in the **Definition** text field.

For details, see [CFX Expression Language \(CEL\)](#).

2. The value of the expression is shown in the **Value** field.
3. Click **Reset** to restore the expression to the definition stored in the database.
Use this to undo changes that have not yet been applied.
4. Click **Apply** to commit any changes or entries made in the **Definition** box.

After you have defined an expression, you can right-click it to make it a parameter for use in a Design Exploration:

- You may choose **Use as Workbench output parameter**.
- If the expression *will not* influence CFX-Pre, you may choose **Use as Workbench input parameter**. Note that this is not a common situation.
- If the expression *will* influence CFX-Pre, you must use the **Expression** shortcut menu in CFX-Pre to make the expression an Ansys Workbench input parameter.

3.6.2.2. Plot Expression Tab

The **Plot** tab enables you to plot an expression for a range of one of its variables with the other variables (if there are any others) held constant.

1. If you have multiple cases loaded and an expression that applies to only one case highlighted, specify the **Case**.
2. Choose the number of sample data points (**# of Points**) of the expression that you would like plotted.
3. Select the independent variable (X) of the expression for use in the plot.
4. Specify a **Range** for this variable in the plot.

All other values are constant (their check boxes cannot also be checked). Enter fixed values for them.

5. Click **Plot Expression** to view the plot.

After viewing the chart, you may click **Define Plot** to return the **Plot** tab to its previous state (which shows the plot settings).

3.6.2.3. Evaluate Expression Tab

The **Evaluate** tab is provided to help you verify that the expression highlighted in the **Expression** tree view is set up correctly. To evaluate an expression:

1. If you have multiple cases loaded and a locator-based function (such as "**areaAve**(Pressure)@**outlet**") highlighted, specify the **Case** in which you want the expression evaluated.
2. If the expression requires that you provide values, type them in.
3. Click **Evaluate Expression**.

The value of the expression is displayed in the **Value** field.

3.6.3. Expressions Workspace: Example

In this example, you will create an expression that you can use to define a new **User Variable**. For details, see [Variables: Example \(p. 95\)](#).

1. Select **Insert > Expression** to create a new expression.

The **Insert Expression** dialog box appears.

2. In the **Insert Expression** dialog box, type a name for the expression and click **OK**.
3. In the **Definition** area of the **Expression** details view, enter the expression: $\sqrt{X^2 + Y^2}$

This expression gives the distance of a point from the Z-axis.

4. Click **Apply** to create the expression.

Note that the **Value** field shows that the variable has units of meters. The value is variable so a single number cannot be shown, as indicated by the placeholder: <variable>.

5. Click the **Plot** tab.

Here, you can define a simple 2D plot. Because the function has two independent variables^[1], you must select a constant value for one of the variables.

6. Check the check box beside **X**.

This selects X as the variable that varies. All other variables requires a fixed value (for plotting).

7. Leave **Start of Range** and **End of Range** at their default values.
8. For **Y**, set **Fixed Value** to 3 [m].
9. Click **Plot Expression**.

A plot shows the variation in the expression with values of X ranging from 0 to 1 [m] and the value of Y held constant at 3 [m].

10. Click the **Evaluate** tab.
11. Set **X** to 0.55 [m] and **Y** to 3 [m].
12. Click **Evaluate Expression**.

^[1] CFD-Post automatically finds the variables associated with an expression, even if the expression depends on another expression.

The value 3.05 [m] appears in the **Value** field. This is consistent with the plot and can easily be verified.

3.6.3.1. Further Expressions

After completing the variable editor example, you can try modifying this expression. You may want to try $\text{sqrt}(X^2+Z^2)$ to define a distance from the Y-axis or $\text{sqrt}(X^2+Y^2+Z^2)$ to define a sphere. Try moving the location of the sphere by adding values to the X, Y, or Z components; for example, $\text{sqrt}(X^2+Y^2+(Z-0.5[m])^2)$ moves the sphere a distance of 0.5 m in the positive Z direction.

3.7. Calculators Workspace

The **Calculators** workspace offers access to the function, macro, and mesh calculators. To access the **Calculators** workspace, click the **Calculators** tab.

For details on the functions available from the **Calculators** workspace, see:

- [Function Calculator \(p. 347\)](#)
- [Macro Calculator \(p. 350\)](#)
- [Mesh Calculator \(p. 371\)](#).

3.8. Turbo Workspace

The **Turbo** workspace improves and speeds up postprocessing for turbomachinery simulations. To access the **Turbo** workspace, click the **Turbo** tab.

For details about using the **Turbo** workspace, see [Turbo Workspace \(p. 381\)](#).

Chapter 4: CFD-Post in Ansys Workbench

CFD-Post can be run in two modes:

- As a stand-alone application started from the Ansys CFX Launcher and independent of the Ansys Workbench software
- As a component launched from Ansys Workbench.

This chapter describes using CFD-Post in Ansys Workbench:

[4.1. The Ansys Workbench Interface](#)

[4.2. File Operation Differences](#)

[4.3. An Introduction to Workflow within Ansys CFX in Ansys Workbench](#)

[4.4. Using Ansys Workbench Journaling and Scripting with CFD-Post](#)

[4.5. Tips on Using Ansys Workbench](#)

[4.6. Limitations When Using Ansys CFD-Post in Ansys Workbench](#)

Note:

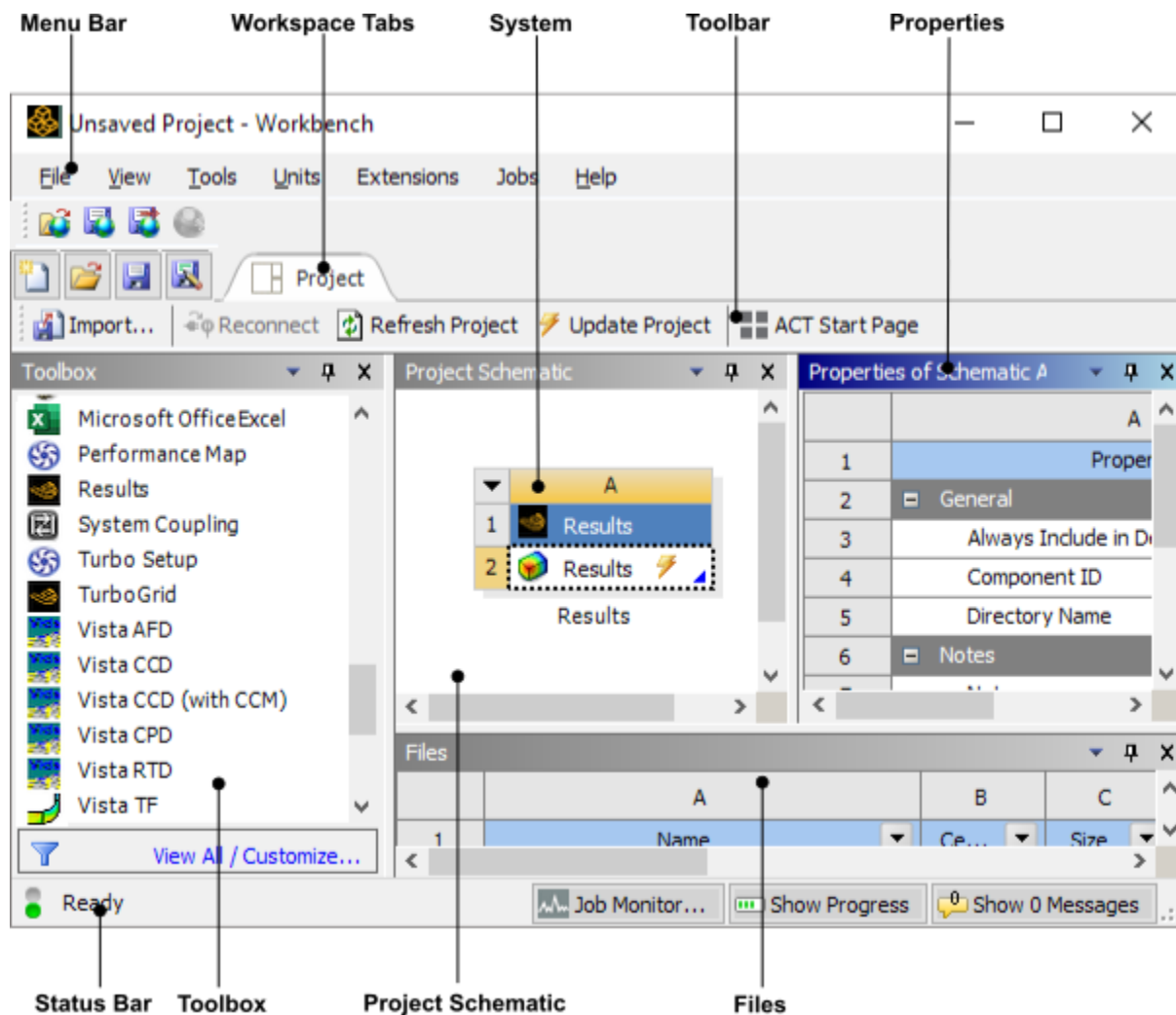
This chapter assumes that you are familiar with using CFD-Post in stand-alone mode. You should consult the Ansys Workbench help for more detailed information on Ansys Workbench.

4.1. The Ansys Workbench Interface

To launch Ansys Workbench on Windows, click the **Start** menu, then select **All Programs > Ansys 2025 R1 > Workbench 2025 R1**.

To launch Ansys Workbench on Linux, open a command line interface, type the path to "runwb2" (for example, "~/ansys_inc/v251/Framework/bin/Linux64/runwb2"), then press **Enter**.

The Ansys Workbench interface is organized to make it easy to choose the tool set that will enable you to solve particular types of problems. Once you have chosen a system from the **Toolbox** and moved it into the **Project Schematic**, supporting features such as Properties and Messages provide orienting information. These features and the status indicators in the system cells guide you through the completion of the System steps.



The following sections describe the main Ansys Workbench features.

4.1.1. Toolbox

The **Toolbox** shows the systems available to you:

Analysis Systems

Systems that match the workflow required to solve particular types of problems. For example, the **Fluid Flow (CFX)** system contains tools for creating the geometry, performing the meshing, setting up the solver, using the solver to derive the solution, and viewing the results.

Component Systems

Systems based on software or software sets. For example, the **CFX** component system contains **Setup** (CFX-Pre), **Solution** (CFX-Solver Manager), and **Results** (CFD-Post). The **Results** component system contains only **Results** (CFD-Post).

Custom Systems

Systems that combine separate analysis systems. For example, the **FSI: Fluid Flow (CFX) > Static Structural** system combines Ansys CFX and the Mechanical application to perform a unidirectional (that is, one-way) Fluid Structure Interaction (FSI) analysis.

Design Exploration

Systems that enable you to see how changes to parameters affect the performance of the system.

Note:

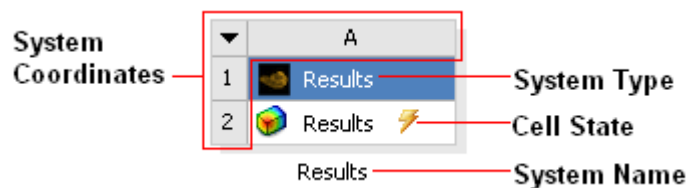
Which systems are shown in the **Toolbox** depends on the licenses that exist on your system. You can hide systems by enabling **View > Toolbox Customization** and clearing the check box beside the name of the system you want to hide.

To begin using a system, drag it into the **Project Schematic** area.

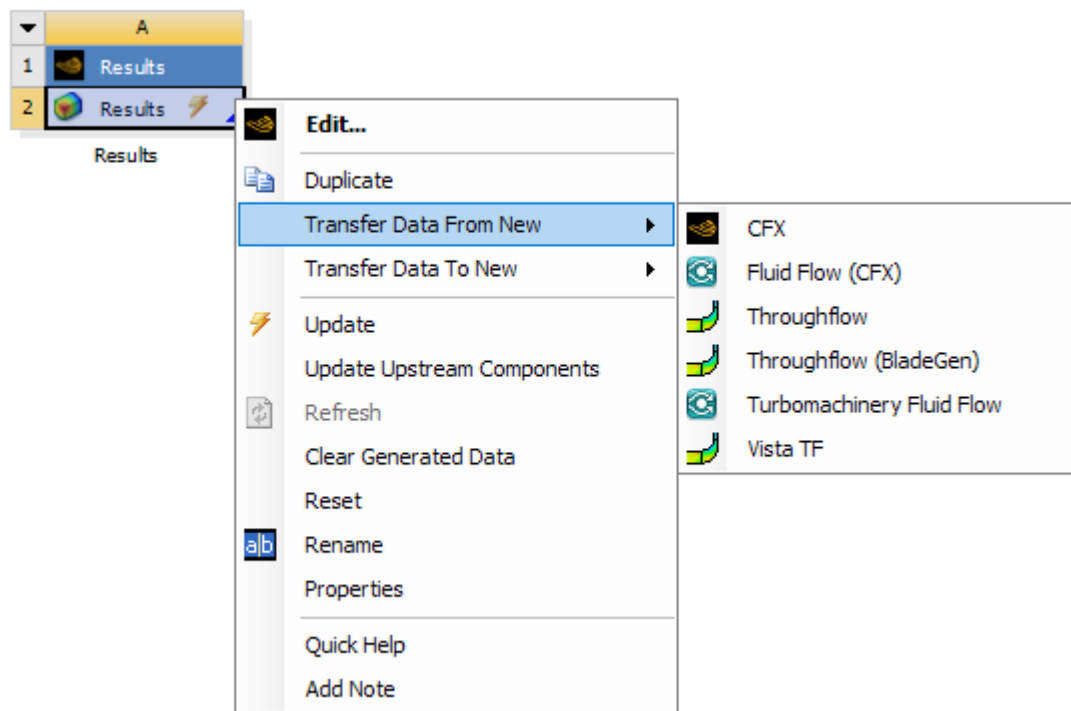
4.1.2. Project Schematic: Introduction

The **Project Schematic** enables you to manage the process of solving your CFD problem. It keeps track of your files and shows the actions available as you work on a project. At each step you can select the operations that process or modify the case you are solving.

When you move a system from the **Component Systems** toolbox to the **Project Schematic**, you will see a system similar to the following:



Each white cell represents a step in solving a problem. Right-click the cell to see what options are available for you to complete a step.



Selecting **Edit** in the example above launches CFD-Post.

4.1.3. Workspace Tabs

Systems such as Engineering Data, DesignXplorer, and Parameters, can be placed in the **Project Schematic** and can be opened in arrangements of views, called *workspaces*. Native workspaces are edited directly within Workbench. Each native workspace is shown in its own tab with its Outline, Properties, Table, and Chart panes displayed when appropriate. Tabs can be opened by editing a system cell from the **Project Schematic**. You can switch between workspaces by selecting their respective tabs. For more details on workspaces and tabs, see [Tabs in Workbench in the Workbench User's Guide](#) and [Panels within Tabs in the Workbench User's Guide](#).

4.1.4. View Menu

You control which panes are displayed by opening the **View** menu and setting a check mark beside the pane you want to display. If you minimize that pane, it appears as a tab above the Status Bar and the check box is cleared from the **View** menu.

4.1.5. Properties Pane

The **Properties** pane is a table whose entries describe the status of a system. These entries vary between system cells and are affected by the status of the cell. Some entries in the **Properties** pane are writable; others are for information only.

To display the **Properties** for a particular cell, right-click the cell and select **Properties**. Once the **Properties** pane is open, simply selecting a cell in the **Project Schematic** will display its properties.

General	Multi-configuration Post Processor Load Options	This is a display-only value. The way that multi-configuration files and transient files open in CFD-Post must be set beforehand in CFD-Post or in the Properties settings for each Solution cell; you cannot configure these settings from the Properties pane of a Results cell.
Update Options	Clear State	When you select this check box, CFD-Post clears the existing state when the Results cell is updated or modified. When you update the Results cell, when CFD-Post is already open, the existing state is cleared before CFD-Post reloads the upstream data and performs any postprocessing.
	Load Report	Loads either a predefined report template or a custom template after CFD-Post reloads the upstream data. The None option has no effect when the Results cell is updated or edited.
	Custom Report Template	Sets the name of the report template when you select Custom under Load Report .
	Publish Report	Select this check box to automatically publish a HTML report. The location of the report is displayed in the Files pane.
	Report Output Filename	Sets the name of the report being published when you select the Publish Report option. The default name is set to Report .html.
	Report Location	Sets the directory where the report is published, when you select the Publish Report option. If you leave this field undefined, the report is saved in the directory associated with the Results cell. Note: When using Custom Report Templates with the RSM, it is strongly recommended that you create a directory named CustomReportTemplates within the user_files

		project directory to store any custom report templates. Workbench will search for custom report templates in this directory and consequently only the filename then needs to be specified. This is particularly convenient when transferring projects between different machines or using the RSM where the absolute path to the report template file may otherwise differ.

4.1.6. Files Pane

The **Files** pane shows the files that are in the current project. The project files are updated constantly, and any "save" operation from a component will save all files associated with the project.

Important:

Although the Files pane reveals the data files that make up a project, you should not attempt to manipulate these files directly, as project data management will proceed unaware of your changes and with unpredictable results.

Ansys Workbench associates data with system cells. This data may be stored in different ways, including as part of the Ansys Workbench project file or as separate files. When files are generated, they appear in the **Files** pane. This pane can be used to identify which files are associated with each cell.

The table that follows associates cell types with file types and gives typical extensions for those file types.

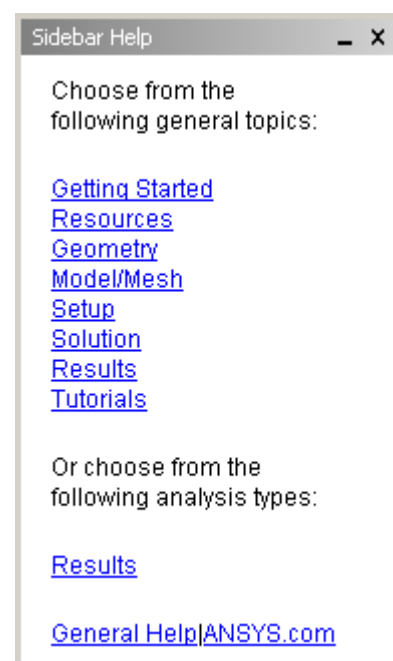
System Cell	File Type	File Extension Examples
Results	CFD-Post State File	.cst
	CFD-Post Output Files ^[a]	AnsysReportLogo.png ^[b] Report.html ^[b]

^[a] Does not include animation files or the output of **Save Picture** commands.

^[b] Generated file (Generated files are not copied when you duplicate a system and are removed when you run the **Clear Generated Data** command.)

4.1.7. Sidebar Help

In addition to having a visual layout that guides you through completing your project, you can also access Sidebar Help by pressing **F1** while the mouse focus is anywhere on Ansys Workbench. Sidebar Help is a dynamically generated set of links to information appropriate for helping you with questions you have about any of the tools and systems you currently have open.



4.1.8. Shortcuts (Context Menu Options)

You can access commonly used commands by right-clicking in most areas of Ansys Workbench. These commands are described in [Context Menus in the Workbench User's Guide](#).

4.2. File Operation Differences

CFD-Post launched from Ansys Workbench has default locations for file operations that are appropriate for Ansys Workbench:

- Save operations default to the user_files directory. The user_files directory appears under the directory that holds the Project file (*<projectfile_name>/user_files/*).
- Open operations default to the permanent files directory. The permanent files directory holds the Project file.
- Export operations initially default to the user_files directory, but change to the last directory used for an export operation during a session.

In addition, there is an icon in the directory tree that takes you to the user_files, and all recent directory selections are available from the directory path drop-down selector.

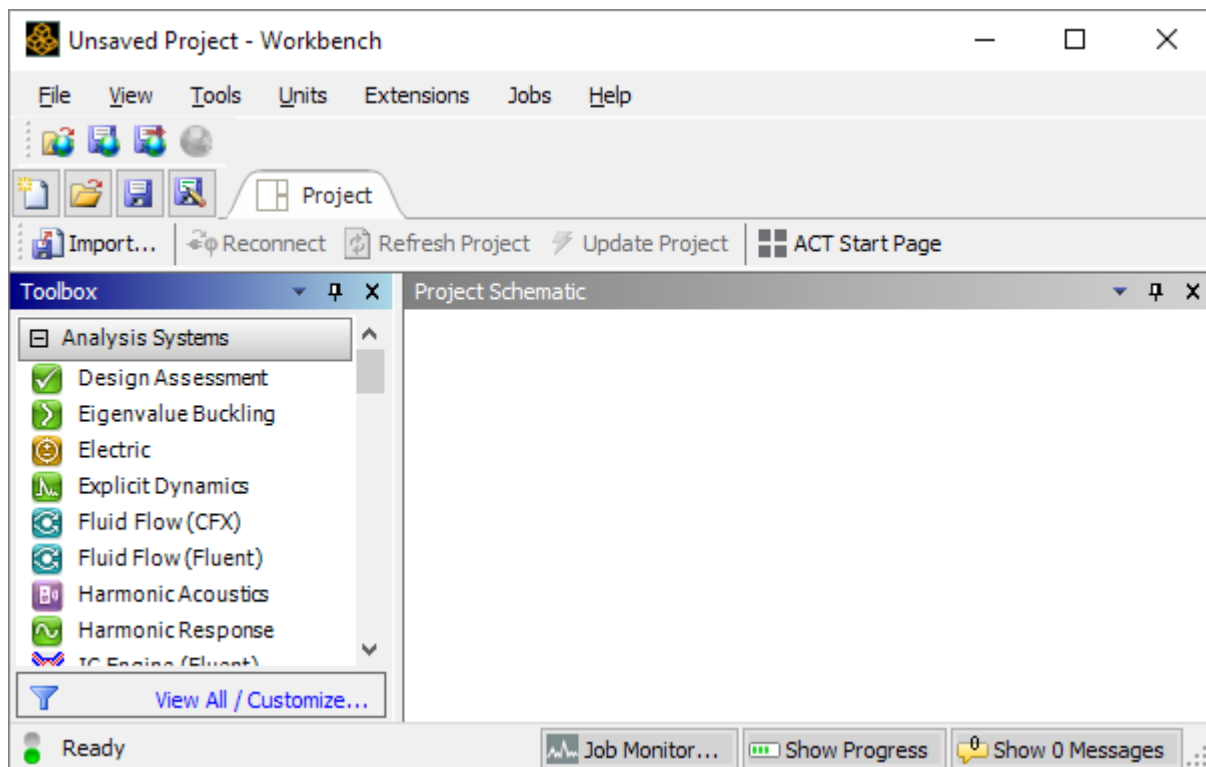
4.3. An Introduction to Workflow within Ansys CFX in Ansys Workbench

This section walks through an example of using Ansys CFX in Ansys Workbench to perform a fluid-flow analysis. This walkthrough assumes familiarity with the basic Ansys Workbench and Ansys CFX applications and does not discuss the details of the steps within each application.

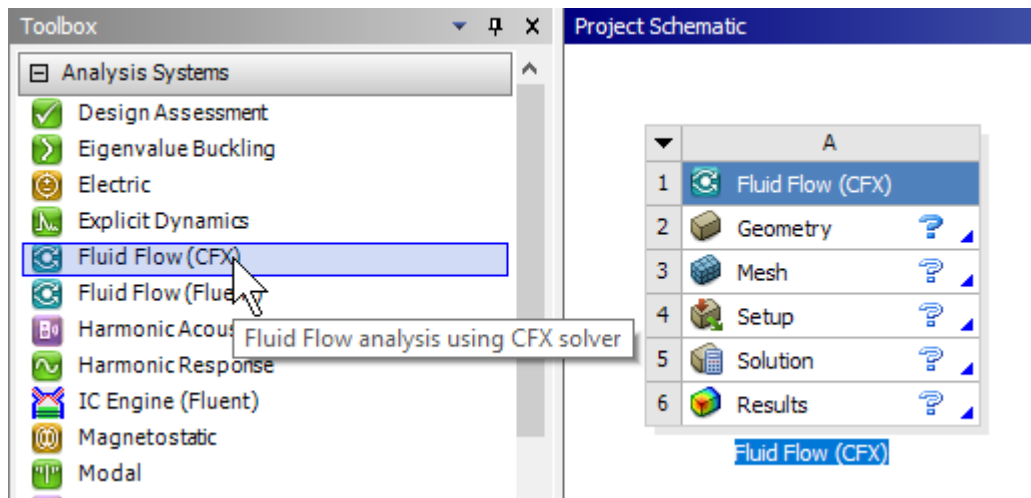
Note:

Although this example uses a Fluid Flow (CFX) analysis system to show workflow, CFD-Post is the results viewing program for a variety of Analysis and Custom systems such as the Fluid Flow (Fluent) analysis system. CFD-Post can also be launched from a Results component system.

1. You begin by launching Ansys Workbench, which opens as an unsaved project and displays the available analysis systems.

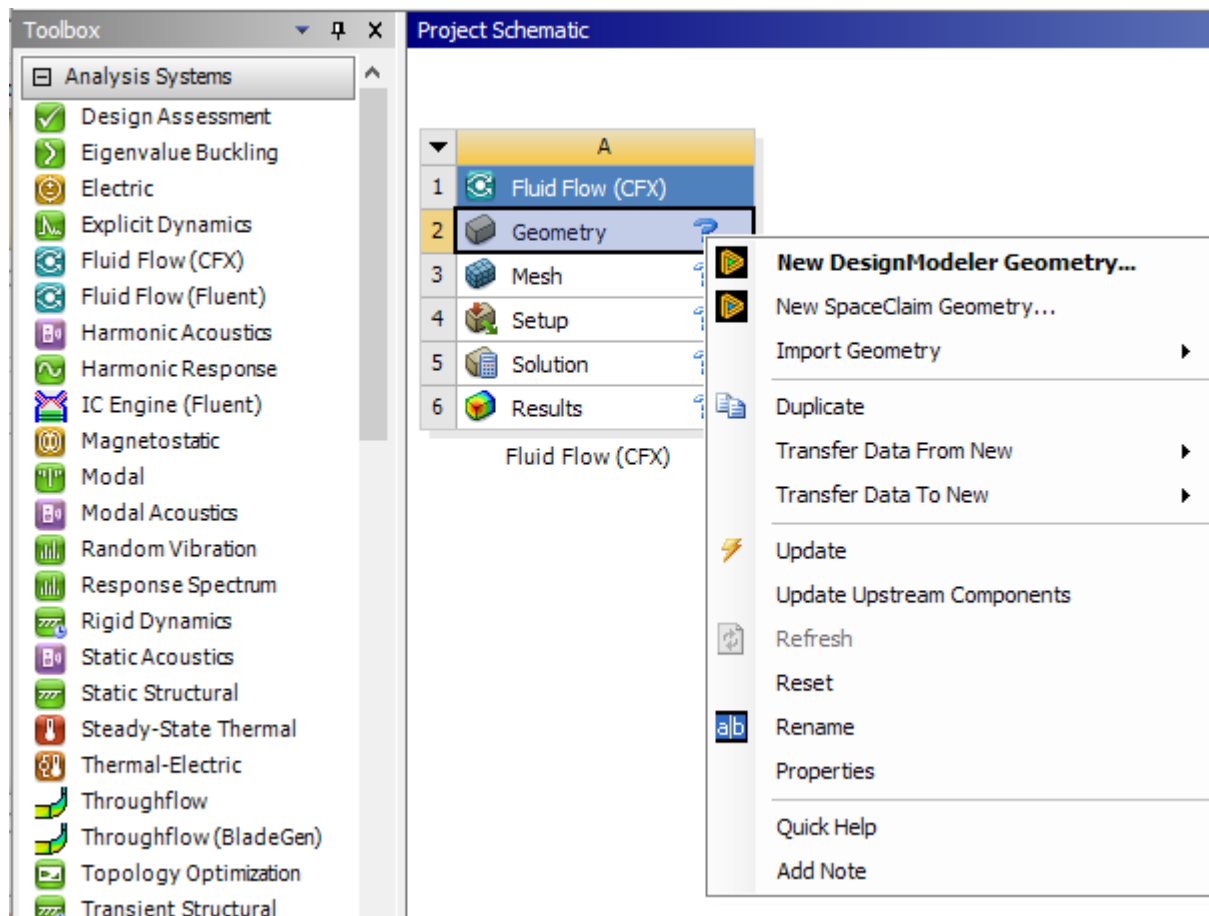


2. In your file system you create a directory in which to store your project files. You then select **File** > **Save As** and save your new project to that directory. This automatically sets your working directory for this project.
3. In the **Analysis Systems** toolbox, double-click **Fluid Flow (CFX)** to create a fluid-analysis system in the **Project Schematic**. (Notice that if you "hover" over systems in the **Toolbox**, a tooltip appears.)



The fluid-analysis system in the **Project Schematic** shows the steps in performing a fluid analysis:

1. Create or import a geometry.
 2. Create a mesh for the geometry.
 3. Set up the analysis that will be sent to the solver.
 4. Control and monitor the solver to achieve a solution.
 5. Visualize the results in a post-processor and create a report.
4. In addition to showing those steps in appropriately named cells, each cell can launch a tool that will enable you to perform the task it names. Right-click the **Geometry** cell to see your options for adding a geometry to your project:



5. As you move through the cells from **Geometry** to **Results**, you can choose to launch the tool that will enable you to complete the cell's step: create a new geometry with Ansys DesignModeler, create a new mesh with Ansys Meshing, edit the case with Ansys CFX-Pre, control the solver's solution with Ansys CFX-Solver Manager, and control the display of the results with CFD-Post.

Note:

You could open a Fluid Flow (CFX) system and go immediately to the **Setup** cell to import an existing case. When the case is loaded, the now-unnecessary **Geometry** and **Mesh** cells disappear.

6. When the analysis is complete and the project is finished, you save the project (and therefore the associated files). Once a project has been saved, it can be re-opened at a later date for review or modification of any aspect of the simulation.

Important:

Saving a project enables you to re-open the project on the machine that originally created it. To make the project available on another machine, you need to use **File > Archive** to create a project archive. To open the project on a different machine, run **File > Restore Archive** on that machine.

4.4. Using Ansys Workbench Journaling and Scripting with CFD-Post

Journaling is the capturing of Ansys Workbench actions (creating a project, opening a system, and so on) to a file. For Ansys CFX applications, CCL and command actions are embedded within Ansys Workbench actions. *Scripting* refers to the processes of editing and running a journal file in Ansys Workbench. With scripting, you could, for example, implement a prescribed workflow.

This section describes how to acquire, edit, and run script files that have commands that affect CFD-Post. For more general information on journal files as well as scripting, refer to the Ansys Workbench online help.

Note:

- Journal actions such as a CFD-Post Export or the loading of a static .res file record the path of the file. You may need to manually adjust this filepath before attempting to rerun the journal, particularly if you have created the journal using an unsaved project. More generally, when you create a project, you should save the project immediately to set file paths that Ansys Workbench uses (rather than require Ansys Workbench to use file paths that have temporary directories, as happens before the project is saved).
 - The handling of file paths described in [File Path Handling in Ansys Workbench in the Workbench Scripting Guide](#) applies to file references that are made outside of CCL and command actions.
 - Journal files must not contain an Undo command from CFD-Post.
-

4.4.1. Acquiring a Journal File with CFD-Post in Ansys Workbench

The basic workflow for acquiring a journal file with CFD-Post in Ansys Workbench is as follows:

1. Start Ansys Workbench.
2. Start journaling: Select **File > Scripting > Record Session** and set a name for the journal file.
3. From **Toolbox** panel, open a system that has a Results cell with an available solution.
4. Edit the Results cell. The actions you perform are captured by the journaling process and written to a .wbjn file.
5. Stop journaling: **File > Scripting > Stop Recording Session**.
6. Optionally, edit the journal file (this is the process of *scripting*).
7. Run **File > Scripting > Run Script File** and select a .wbjn file.

4.4.1.1. Journal of an Operation That Creates a Plane in CFD-Post

In the following incomplete snippet, a user has created a Results system, edited the Results cell, loaded a CFX-Solver Results file (StaticMixer_001.res) and then created a plane named "Plane 1":

Create the Results system

```
template1 = GetTemplate(TemplateName="Results")
system1 = template1.CreateSystem(Position="Default")
```

Edit the Results cell and load the Results file (StaticMixer_001.res)

```
results1 = system1.GetContainer(ComponentName="Results")
results1.Edit()
results1.SendCommand(Command=r"""DATA READER:
  Clear All Objects = false
  Append Results = true
  Edit Case Names = false
  Open to Compare = false
  Multi Configuration File Load Option = Separate Cases
  Open in New View = true
  Keep Camera Position = true
  Load Particle Tracks = true
  Files to Compare =
END
DATA READER:
Domains to Load=
END
> load filename=C:\StaticMixer_001.res, multifile=append""")
```

Set the camera and define a plane colored with a constant color

```
CFX.SendCommand(
  Container="Results",
  Command=""VIEW:View 1
  Camera Mode = User Specified
  CAMERA:
    Option = Pivot Point and Quaternion
    Pivot Point = 0, 0, 0
    Scale = 0.226146
    Pan = 0, 0
    Rotation Quaternion = 0.279848, -0.364705, -0.115917, 0.880476
    Send To Viewer = False
  END
END

> autolegend plot=/PLANE:Plane 1, view=VIEW:View 1""")
CFX.SendCommand(
  Container="Results",
  Command=""PLANE:Plane 1
  Apply Instancing Transform = On
  Apply Texture = Off
  Blend Texture = On
  Bound Radius = 0.5 [m]
  Colour = 0.75, 0.75, 0.75
  Colour Map = Default Colour Map
  Colour Mode = Constant
  Colour Scale = Linear
  Colour Variable = Pressure

  # ...
  # (Lines omitted for brevity)
  # ...

END""")

results1.SendCommand(Command=""# Sending visibility action from View...
>show /PLANE:Plane 1, view=/VIEW:View 1""")
```

Save the project

```
Save(
    FilePath=r"C:\SaveJou.wbpj",
    Overwrite=True)
```

The commands in the script above are the default values for a plane.

4.4.2. Scripting

Scripting refers to the processes of editing and running a journal file in Ansys Workbench. You can create your own scripts and include the power of Python to implement high-level programming constructs for input, output, variables, and logic. The example that follows illustrates this for CFD-Post.

4.4.2.1. Example: Using a Script to Change an Existing Locator

If you have an Ansys Workbench project currently open, you can run a script to change how the results of the simulation are post-processed. For example, if you have opened CFD-Post from an Ansys Workbench system and CFD-Post is displaying a plane named "Plane 1", you can run the following script to change the plane to be colored by the variable *Velocity* or *Pressure*.

Before running this script, you would have to first open the **Command Window** dialog box (by selecting **File > Scripting > Open Command Window** from the Ansys Workbench main menu). To run the script, you would select **File > Scripting > Run Script File** from the Ansys Workbench main menu and then use the browser to open the file containing the script.

```
x = int(raw_input("Enter an integer: 1=Velocity, 2=Pressure: "))

if x == 1:
    print 'Velocity'
    CFX.SendCommand(
        Container="Results",
        Command="""PLANE:Plane 1
        Colour Mode = Variable
        Colour Variable = Velocity
        END""")
elif x == 2:
    print 'Pressure'
    CFX.SendCommand(
        Container="Results",
        Command="""PLANE:Plane 1
        Colour Mode = Variable
        Colour Variable = Pressure
        END""")
```

Depending on the value of *x* you input in the **Command Window**, the script includes the CCL in the appropriate `CFX.SendCommand` argument to set the values for `Colour Mode` and `Colour Variable` in the `PLANE:Plane 1` object for either the *Velocity* or *Pressure* variable.

4.5. Tips on Using Ansys Workbench

This section highlights helpful tips on using Ansys Workbench.

4.5.1. General Tips

4.5.2. Tips for Results Systems

4.5.1. General Tips

The following are useful tips for the general use of Ansys CFX in Ansys Workbench:

4.5.1.1. Ansys Workbench Interface

A lot of important functionality is available in the shortcut menu (cells, parameter bar, and so on). Also, you should enable the **View > Properties** pane and investigate options for each cell.

4.5.1.2. Setting Units

Ansys Workbench units and options are not passed to CFD-Post; this could require you to set units twice.

4.5.1.3. Files Pane

Use the **Files** pane to determine which files were created for each cell/system. This can be very useful if you need to do some runs or change some settings outside of Ansys Workbench, or if you want to manually delete some but not all files associated with a particular cell. It is easiest to find files associated with a specific cell by sorting the pane by Cell ID. This will sort the list by system and then by cell.

4.5.1.4. Ansys Workbench Connections

When selecting a system in the toolbox, Ansys Workbench will highlight the cells in any systems already in the **Project Schematic** to which a valid connection can be made.

4.5.2. Tips for Results Systems

The following are useful tips for the use of Results systems in Ansys Workbench:

4.5.2.1. Changes in Behavior

The ability to play session files is missing in Ansys Workbench for CFD-Post.

The undo stack is cleared in CFD-Post after the application receives commands from Ansys Workbench.

You cannot launch Ansys CFX products from one another in Ansys Workbench; you must use the system cells.

Ansys Workbench "remembers" previous locations of imported files / projects. CFD-Post, however, displays different behavior for loading or saving any files, always using the directory specified in the **Tools > Options > Default Folder for Permanent Files** in Ansys Workbench.

4.5.2.2. Duplicating Systems

Duplication normally involves only user files (files for which you have specified settings). For CFD-Post, this would include the `.cst` file. Other files, which are considered to be "generated" (for instance, the `.html` files), are not duplicated.

4.5.2.3. Renaming Systems

Rename all your CFX and Fluid Flow (CFX) systems to something unique and meaningful that reflects the contents of the system, especially if there are multiple systems. The names of the files associated with the system cells will incorporate this system name when the files are first created, making it easier for you to identify the files in the **Files** pane. In particular, CFD-Post will take the system name (by default "Fluid Flow" for a Fluid Flow system) as the case name of the results in CFD-Post. Note that it is best to rename the systems as soon as they are placed on the **Project Schematic**, as the generated file names and/or the CFD-Post case names will not necessarily be updated if a system is renamed after the appropriate cells already have associated data (for example, a `.cfx` file with the Setup cell). It may be useful to reset the Results cell to update the CFD-Post case name if the system is renamed, but you will lose any existing CFD-Post settings and objects by doing this.

4.5.2.4. Results Cell

CFX-Solver Results files (in particular the `.res` files) are associated with the Solution cell, not the Results cell. This means that a CFX-Solver Results file cannot be imported onto a Results cell; it can be imported onto a Solution cell of a Fluid Flow or CFX system. Similarly, resetting the Results cell will not remove the CFX-Solver Results file.

In Ansys Workbench, the state of CFD-Post is associated with the Results cell. To maintain multiple states, you must generate multiple Results systems. For your convenience, you can provide a unique name for each system.

To perform a file comparison in CFD-Post, drag a Solution cell from another system to the Results cell.

You can have CFD-Post generate report output at every update (by setting Generate Reports in Results cell Properties pane). The `.html` file is visible in the **Files** pane: right-click it, select **Open containing folder**, and double-click the file in the explorer to see the report in a browser.

When updating existing Results cell data (with CFD-Post open) where a turbo chart with an averaged variable was used (for example, turbo reports), a warning dialog box may appear reporting that "No data exists for variable ...". This warning can be ignored.

You can change the CFD-Post multi-configuration load options (available on the **Load Results File** dialog box of CFD-Post when in stand-alone mode) by editing the properties of the Solution cell. This is a property of the Solution cell, rather than the Results cell.

4.5.2.5. Recovering After Deleting Files

If you accidentally delete the current `.def`, `.res` or `.out` files for a CFX system and the Solution cell status is up-to-date, you may get errors when trying to display the solution monitor or edit the Results cell. In this case you will need to replace the files in the File Manager, or **Reset** the Solution

cell, and update the system. If the .def file is missing, you may also need to **Clear Generated Data** for the Setup cell before updating the system.

4.5.2.6. License Sharing

If you are using license sharing in Ansys Workbench, you can use only one license for CFX-Pre/CFD-Post even if you have more available. This has implications if, for example, you want to run a long animation in CFD-Post and use CFX-Pre at the same time. If you know you are going to be working with CFX-Pre and CFD-Post at the same time, you need to change the license-sharing setting before starting your project.

4.6. Limitations When Using Ansys CFD-Post in Ansys Workbench

- In a design points study, when you have some up-to-date design points, it is recommended that you avoid editing the **Results** cell and running CFD-Post interactively. Most editing and viewing actions result in an underlying state change and therefore make the design points appear out-of-date, requiring another update.

- When using Remote Solve Manager:

If you are using custom report templates in CFD-Post, you should avoid using UNC paths (paths that begin with two backslashes rather than a drive letter) when using Remote Solve Manager. You can use one of the following methods to avoid using UNC paths:

- Instead of using a shared cluster directory on the machine that has the head node of the compute cluster, set up a scratch directory on each machine that has an execution node. Each machine will then refer to its own local directory, which does not require access via a network path.
- Use a shared directory on the machine that has the head node of the compute cluster. On each machine that has an execution node, map the network path to that shared directory (or any parent directory of it) as a drive letter. Use that drive letter when specifying the path to the shared directory.

Chapter 5: CFD-Post 3D Viewer

In CFD-Post, the **3D Viewer** is accessible by clicking the **3D Viewer** tab at the bottom of the panel on the right side of the interface.

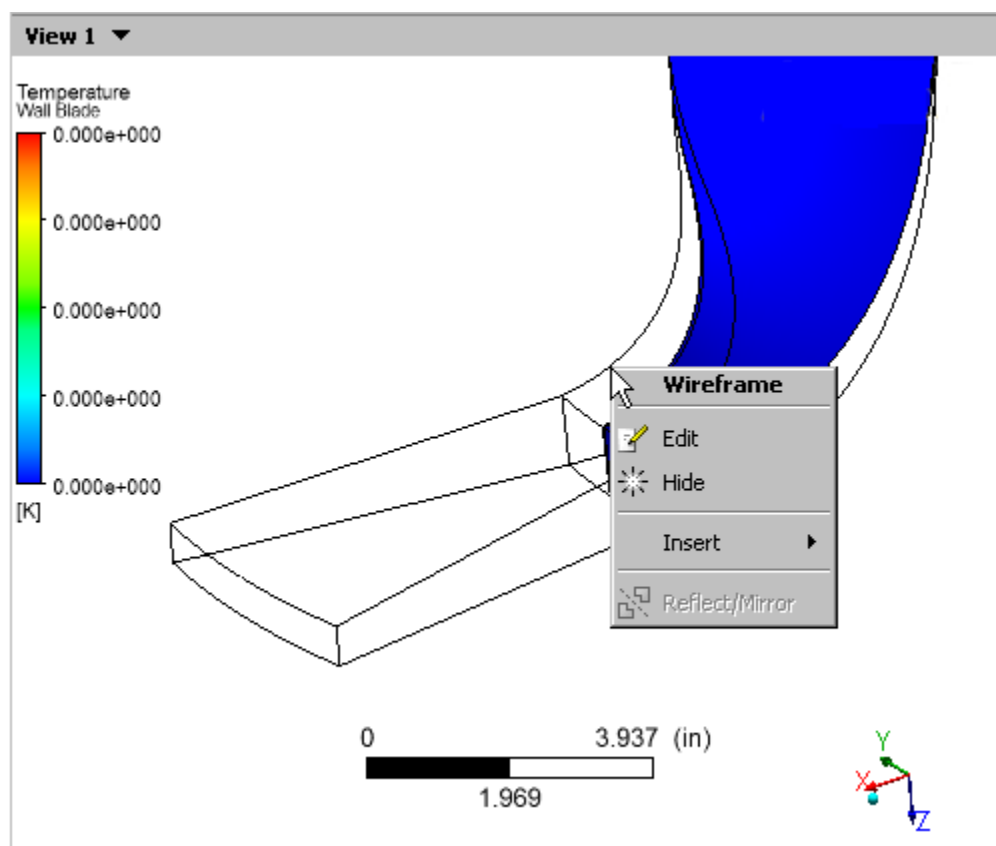
After loading a results file into CFD-Post, you can see a visual representation of the geometry in the **3D Viewer**. You can create various other objects that can be viewed in the **3D Viewer**. For details, see [CFD-Post Insert Menu \(p. 219\)](#).

Descriptions of the various viewing modes and **3D Viewer** commands, including toolbars, shortcut menus, and hotkeys, are given in [3D Viewer Modes and Commands \(p. 119\)](#).

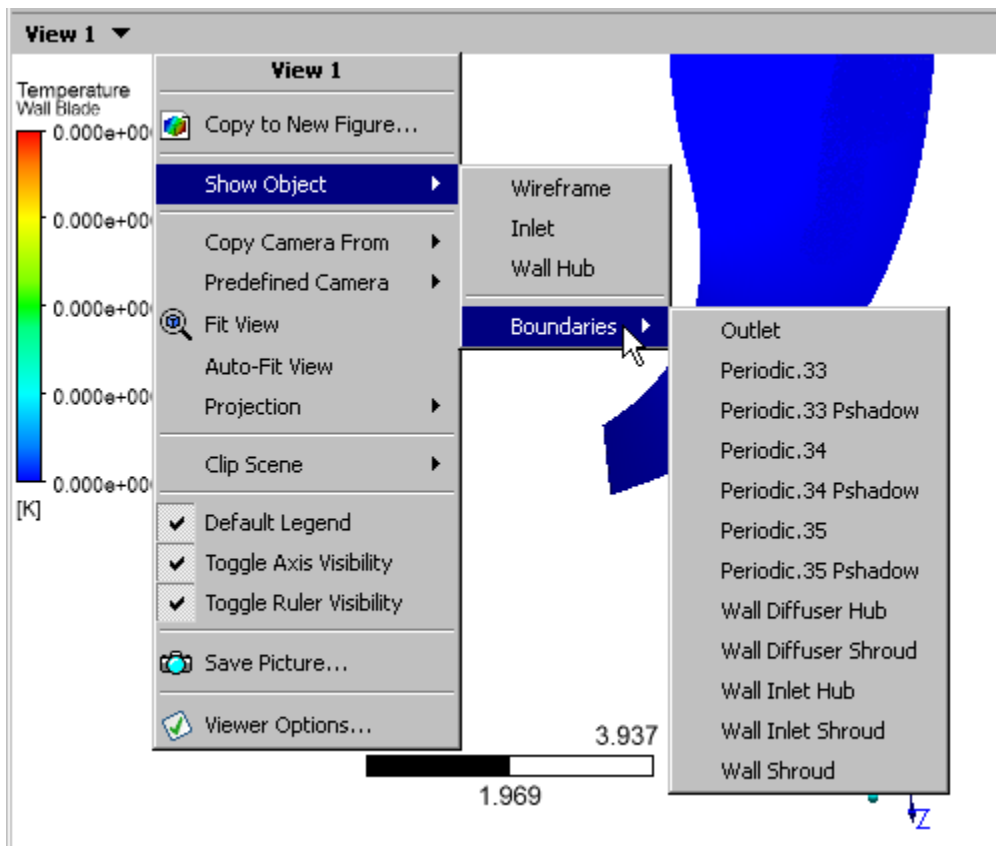
You can switch between four adjustable "views" that each remember the camera angle and state of visibility of all objects. CFD-Post has "figures", which are similar to views except that they can be included in reports. For details, see [Views and Figures \(p. 127\)](#).

5.1. Object Visibility

The visibility of each object can be turned on and off using the check boxes in the tree view, as described in [Object Visibility \(p. 48\)](#). However, you can also hide objects by right-clicking on them and selecting **Hide**. The right-click menu has a title that indicates the object that will be acted upon (**Wireframe** in the figure that follows) so that you do not accidentally hide the wrong object.



Once an object has been hidden, you can show it again by right-clicking on the background of the Viewer and selecting **Show Object**:



5.2. 3D Viewer Modes and Commands

This section describes:

- [3D Viewer Toolbar \(p. 119\)](#)
- [CFD-Post 3D Viewer Shortcut Menus \(p. 121\)](#)
- [Viewer Hotkeys \(p. 123\)](#)
- [Mouse Button Mapping \(p. 124\)](#)
- [Picking Mode \(p. 126\)](#)

5.2.1. 3D Viewer Toolbar

The **3D Viewer** toolbar has the following tools:

Tool	Description
	Activates one of the three picking tools (shown below).
	Selects objects. You can use this tool to drag line, point, plane, and isosurface objects to new locations.
	Selects objects using a box. Drag a box around the objects you want to select.

Tool	Description
	Selects objects using an enclosed polygon. Click to drop points around the objects. Double-click to complete the selection. Note: Polygon Select mode will not allow you to create an invalid region, such as would occur if you attempted to move a point such that the resulting line would cross an existing line in the polygon.
	Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view.
	Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down Ctrl and the middle mouse button.
	Adjusts the zoom level as you drag with the mouse vertically. Alternatively, you can zoom the view by holding down Shift and the middle mouse button.
	Zooms to the area enclosed in a box that you create by dragging with the mouse. Alternatively, you can drag and zoom the view by holding down the right mouse button.
	Centers all visible objects in the viewer.
	When enabled, clicking on an object in the tree view causes that object to be highlighted in the 3D Viewer. The style of highlighting is controlled by Edit > Options > CFD-Post > Viewer > Object Highlighting > Type.
	Selects the viewport arrangement. You can perform Independent zoom, rotation and translate options in each viewport.
	Toggles between locking and unlocking the views of all viewports. When the views are locked, the camera orientation and zoom level of the non-selected viewports are continuously synchronized with the selected viewport. Locking the view for the viewports in this way can be a useful technique for comparing different sets of visible objects between the viewports. This tool is available only when all viewports are using the Cartesian (X-Y-Z) transformation.
	Toggles between synchronizing the visibility of objects in all viewports. When active, any subsequent action to hide or display an object affects all viewports; activating this feature does not affect any existing show/hide states. Note: This toggle will not synchronize the visibility of objects in different cases that have the same name. However, in file comparison mode CFD-Post <i>does</i> synchronize the visibility of objects that have the same name.

Tool	Description
	Displays the Viewer Key Mapping dialog box. See Viewer Hotkeys (p. 123) for details.

5.2.2. CFD-Post 3D Viewer Shortcut Menus

You can access the shortcut menu by right-clicking anywhere on the viewer. The shortcut menu is different depending on where you right-click.

5.2.2.1. Shortcuts for CFD-Post (Viewer Background)

The following commands are available in CFD-Post when you right-click the viewer background:

Command	Description
Deformation	Specifies the deformation scale to be viewed. This option is only available when the Total Mesh Displacement variable exists. When an option is selected, it will be applied to all objects in every view and figure. Select from the following: • Undeformed Shows all objects as if they were not deformed • True Scale Displays all objects with their regular deformation values • 0.5x Auto Shows all objects with half of the optimal (Auto) scale • Auto Adjusts the deformation scaling for optimal viewing. Internally, the deformation is scaled so that the maximum deformation results in a viewable displacement of a percentage of the domain extents, regardless of the problem size. • 2x Auto Adjusts the deformation to be double that of regular deformation • 5x Auto Shows all objects with 5 times their regular deformation value. • Custom... Opens the Deformation Scale dialog box and displays the currently applied scale value for the deformation. Specify a new value to change the scale.

Command	Description
	• Animate... Opens the Animation dialog box in Sweep Animation mode. For details, see Animating Mesh Deformation Scaling (p. 336).
Copy to New Figure	Creates a new figure based on the current camera position, zoom level, and object visibility settings. For details, see Views and Figures (p. 127). The figure appears under the Report object, and can be used in a report. For details, see Report (p. 72). The Make copies of objects check box controls how the new figure is made: • When the check box is selected, visible objects are copied for the new figure. Use this option if you want the figure to retain its appearance when the original objects are modified. • When the check box is cleared, only the camera position, zoom level, and the object visibility settings are stored in the definition of the figure. Use this option if you want the figure to automatically update with changes to the original objects.
Show Object	Shows hidden objects, boundaries, and regions. See Object Visibility (p. 117) .
Copy Camera From	If you have set a Predefined Camera angle in another view, selecting Copy Camera From > <view name> will apply that angle to the current view.
Predefined Camera	Displays different views by changing the camera angle to a preset direction.
Fit View	Centers all visible objects in the viewer. This is equivalent to clicking the  icon.
Auto-Fit View	Automatically fits the view while you rotate the camera or resize the 3D Viewer . This disables the manual resizing actions otherwise available from the toolbar or mouse.
Projection	Switches between perspective and orthographic camera angles.
Clip Scene	Controls scene clipping via clip planes. For details, see Clip Plane Command (p. 296) .
Default Legend	Shows or hides the default legend object.
Axis	Shows or hides the axis orientation indicator (known as the triad) in the bottom-right corner of the viewer.
Ruler	Shows or hides the ruler on the bottom of the viewer.
Save Picture	Same as selecting File > Save Picture . For details, see Save Picture Command (p. 165) .
Viewer Options	Opens the Options dialog box with the viewer options displayed. For details, see Viewer (p. 208) .

5.2.2.2. Shortcuts for CFD-Post (Viewer Object)

The following commands are available in CFD-Post when you right-click an object in the viewer:

Command	Description
Edit	Opens the object for editing.
Hide	Hides the selected object in the 3D Viewer .
Animate	Brings up the Animation dialog box and animates the selected object automatically. For details, see Sweep Animation (p. 334) .
Color	Enables you to change the selected object's color.
Render	Enables you to change some of the selected object's render options (such as lighting and face visibility). To change other render options, select Edit and make your changes on the object's Render tab.
Insert	Opens another menu with options to insert planes, contours, streamlines, etc. For details, see CFD-Post Insert Menu (p. 219) .
Set Plane Center	For planes defined using the Point and Normal method, this action moves the point that defines the plane. This changes the focus for plane bounding operations. See Plane Bounds (p. 230) .
Reflect/Mirror	Applies a reflection to the selected domain. To use this command, right-click the corresponding wireframe in the viewer.
Probe Variable	Opens a toolbar at the bottom of the viewer allowing the specification of coordinate points and variable type. After each field is changed, the solution automatically generates to the right of the variable type setting. For details, see Probe (p. 345) .

5.2.3. Viewer Hotkeys

A number of shortcut keys are available to carry out common viewer tasks. These can be carried out by clicking in the viewer window and pressing the associated key.

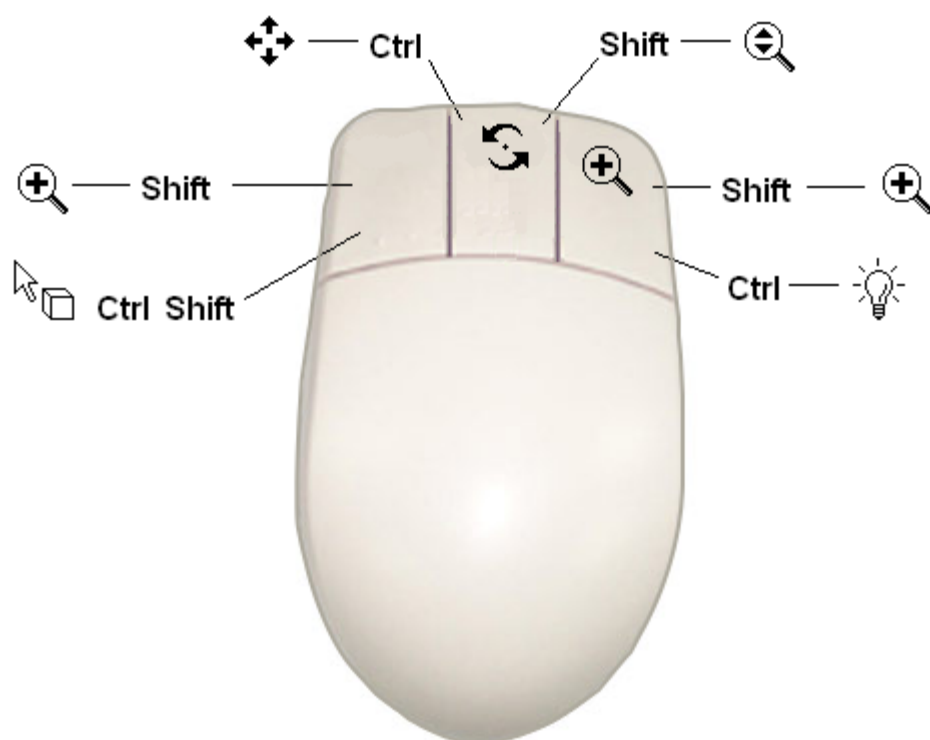
Key	Action
space	Toggles between picking and viewing mode.
arrow keys	Rotates about horizontal and vertical axes.
Ctrl + up/down arrow keys	Rotates about an axis normal to the screen.
Shift + arrow keys	Moves the light source.
1	Switches to one viewport.
2	Switches to two viewports.
3	Switches to three viewports.
4	Switches to four viewports.
C	Centers the graphic object in the viewer window.
N	Toggles the projection between orthographic and perspective.
R	Resets the view to the initial orientation.
S	Toggles the level of detail between auto, off, and on.
U	Undoes transformation.

Key	Action
Shift+U	Redoes transformation.
X	Sets view from +X axis.
Shift+X	Sets view from -X axis.
Y	Sets view from +Y axis.
Shift+Y	Sets view from -Y axis.
Z	Sets view from +Z axis.
Shift+Z	Sets view from -Z axis.

The information in this table is accessible by clicking the *Show Help Dialog*  toolbar icon in the **3D Viewer** toolbar.

5.2.4. Mouse Button Mapping

The mouse mapping options enable you to assign viewer actions to mouse clicks and keyboard/mouse combined clicks. To adjust or view the mouse mapping options, select **Edit > Options**, then **Common > Viewer Setup > Mouse Mapping**.

Figure 5.1: Mouse Mapping using Workbench Defaults**Table 5.1: Mouse Operations and Shortcuts**

Operation	Description	Workbench Mode Shortcuts	CFX Mode Shortcuts
Zoom Object Zoom Camera Zoom	To zoom out, drag the pointer up; to zoom in, drag the pointer down.	Shift + middle mouse button	Middle mouse button Shift + middle mouse button zooms in a step. Shift + right mouse button zooms out a step.
Translate	Drag the object across the viewer.	Ctrl + middle mouse button	Right mouse button
Zoom Box	Draw a rectangle around the area of interest, starting from one corner and ending at the opposite corner. The selected area fills the viewer when the mouse button is released.	Right mouse button Shift + left mouse button Shift + right mouse button	Shift + left mouse button

Operation	Description	Workbench Mode Shortcuts	CFX Mode Shortcuts
Rotate	Rotate the view about the pivot point (if no pivot point is visible, the rotation point will be the center of the object).	Middle mouse button	
Set Pivot Point	Set the point about which the Rotate actions pivot. The point selected must be on an object in the 3D Viewer . When you set the pivot point, it appears as a small red sphere that moves (along with the point on the image where you clicked) to the center of the 3D Viewer . To hide the red dot that represents the pivot point, click a blank area in the 3D Viewer .	Left mouse button when in rotate, pan, zoom, or zoom box mode (as set by the icons in the viewer's toolbar).	Ctrl + middle mouse button
Move Light	Move the lighting angle for the 3D Viewer . Drag the mouse left or right to move the horizontal lighting source and up or down to move the vertical lighting source. The light angle hold two angular values between 0 - 180.	Ctrl + right mouse button	Ctrl + right mouse button
Picking Mode	Select an object in the viewer.	Ctrl + Shift + left mouse button	Ctrl + Shift + left mouse button

5.2.5. Picking Mode

Picking mode is used to select and drag objects in the viewer. The mesh faces must be visible on an object or region to allow it to be picked. Enter picking mode by selecting the *Single Select*  tool in a pull-down menu of the viewer toolbar. If the *Single Select*  icon is already visible, you can simply click the *New Selection*  icon.

You can also pick objects while still in viewing mode by holding down the **Ctrl** and **Shift** keys as you click in the viewer.

5.2.5.1. Selecting Objects

Use the mouse to select objects (for example, points and boundaries) from the viewer. When a number of objects overlap, the one closest to the camera is picked.

You can change the picking mode by selecting one of the toolbar icons:

-  **Single Select**
-  **Box Select**

-  **Polygon Select**

For details, see [3D Viewer Modes and Commands](#) (p. 119).

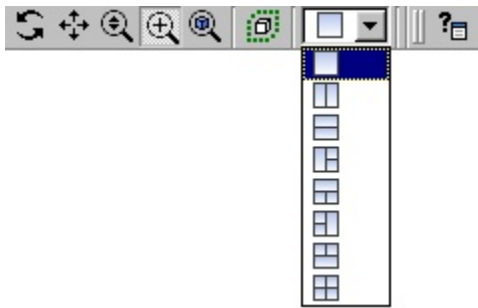
5.2.5.2. Moving Objects

Point, plane and line objects can be moved in the viewer by dragging and dropping the object to a new location. When an object is moved, its definition is updated in the details view. Any other plots that are located on these movable objects are automatically updated.

5.3. Views and Figures

The **3D Viewer** opens with a single *viewport*; you can increase the number of viewports to four by using the viewport icon:

Figure 5.2: Viewport Control



The contents of a viewport are a *view*, which is a CCL object that contains the camera angle, zoom level, lighting, and visibility setting of each object in the tree view.

Each viewport contains a different, independent view. By default, four views exist: **View 1**, **View 2**, **View 3**, **View 4**.

When you select an object in the tree view, its information is applied to the active viewport. When you manipulate an object in the viewport, the view's CCL is updated immediately. However if the focus is on that viewport, you can press **u** to revert your change.

In CFD-Post, you can create figures, which are the same as views, except that they are usable in reports. For details, see [Report](#) (p. 72).

5.3.1. Creating a Figure

In CFD-Post, figures can be created by selecting **Insert > Figure**, or by selecting **Copy to New Figure** from the viewer shortcut menu (after right-clicking a blank area in the **3D Viewer**). The names of views that you create are of the form "Figure *m*" by default, where *m* is an integer that results in a unique name.

A new figure gets its definition from the currently existing view or figure. The latter remains active so that subsequent view manipulations do not affect the new figure.

5.3.1.1. Copying Objects for Figures

A change made to an object will affect all figures that show that object. This can result in an unwanted change to a figure after it has been created. In order to avoid this problem, you may select the **Make copies of objects** option that is available when creating a new figure. This causes all visible objects to be copied, and the new figure to use the copied objects rather than the original ones.

Any copied objects for a figure will appear in the tree view under `User Locations and Plots > Local Objects for FigureName`, where `FigureName` is the name of the figure.

5.3.2. Switching to a View or Figure

To switch to a view or figure, do one of the following:

- Use the drop-down menu in the upper-left corner of the viewport.
- For figures only: Double-click the figure in the tree view (under the Report object).
- For figures only: Right-click the figure in the tree view (under the Report object), then select **Edit** from the shortcut menu.

5.3.3. Changing the Definition of a View or Figure

To change a view or figure:

1. Switch to the view or figure that you want to change.

For details, see [Switching to a View or Figure \(p. 128\)](#).

2. Change the view or figure (for example, rotate the view) either directly, or, in CFD-Post only, select one of the **Copy Camera From** commands from the viewer shortcut menu after right-clicking a blank area of the viewer.

View and figure objects are saved automatically when you switch to a different view or figure.

5.3.4. Deleting a Figure

The figure objects that you have created can be deleted using the tree view or the viewer shortcut menu. To use the viewer shortcut menu:

1. Switch to the figure that you want to delete.
2. Select the **Delete Figure** command from the viewer shortcut menu after right-clicking a blank area of the viewer.

5.3.5. Views

There are four default views that are handled specially. These are named: View 1, View 2, View 3, and View 4. These views will *not* be included in CFD-Post reports. However, any of these views can be viewed in any of the viewports, and you can create new views or figures that will be shown in reports.

5.3.5.1. Object Visibility

The visibility of an object is specified by the VIEW that should display the object, rather than the object specifying whether it is visible. That is, the object is made visible in a certain view--it is no longer a property of the object.

The VIEW object has a parameter named `Object Visibility List` that is set to a comma-separated list of object paths that should be visible in the VIEW object.

Here is an example of the VIEW object CCL to define the visibility for the view:

```
VIEW: View 1 Object Visibility List=/PLANE:Plane 1, /VECTOR:Vector 4END
```

Note:

The `Object Visibility List` parameter should contain only object paths, and not object names.

Setting the `Visibility` parameter on an object has no effect.

For session files, there are command actions that enable you to change the visibility of objects:

- `>show`
- `>hide`
- `>toggle`

Each of these actions take an object name, path, or list of names and paths for which to show, hide, or toggle the visibility. Also, the actions optionally take a parameter that specifies the view to show the object. The visibility action parameters can alternatively take names or entire paths to specify the objects and the views.

Example 1: The following action will show the object `/PLANE:Plane 1` in all existing views, including user figures.

```
>show Plane 1
```

Example 2: The following action will hide both `/PLANE:Plane 1` and `/PLANE:Plane 2` in view `/VIEW:View 1`.

```
>hide Plane 1, /PLANE:Plane 2, view=View 1
```

Example 3: If `Plane 1` is visible, and `Plane 2` is not visible in `/VIEW:View 2`, the following action will make `/PLANE:Plane 1` not visible, and `/PLANE:Plane 2` visible in view `/VIEW:View 2`:

```
>toggle Plane 1, Plane 2, view=/VIEW:View 2
```

5.3.5.2. Legends

There is a default legend for each VIEW object. The default legend is automatically created and deleted along with the view. By default, the default legend is made visible in the view it is associated with.

5.4. Stereo Viewer

If you:

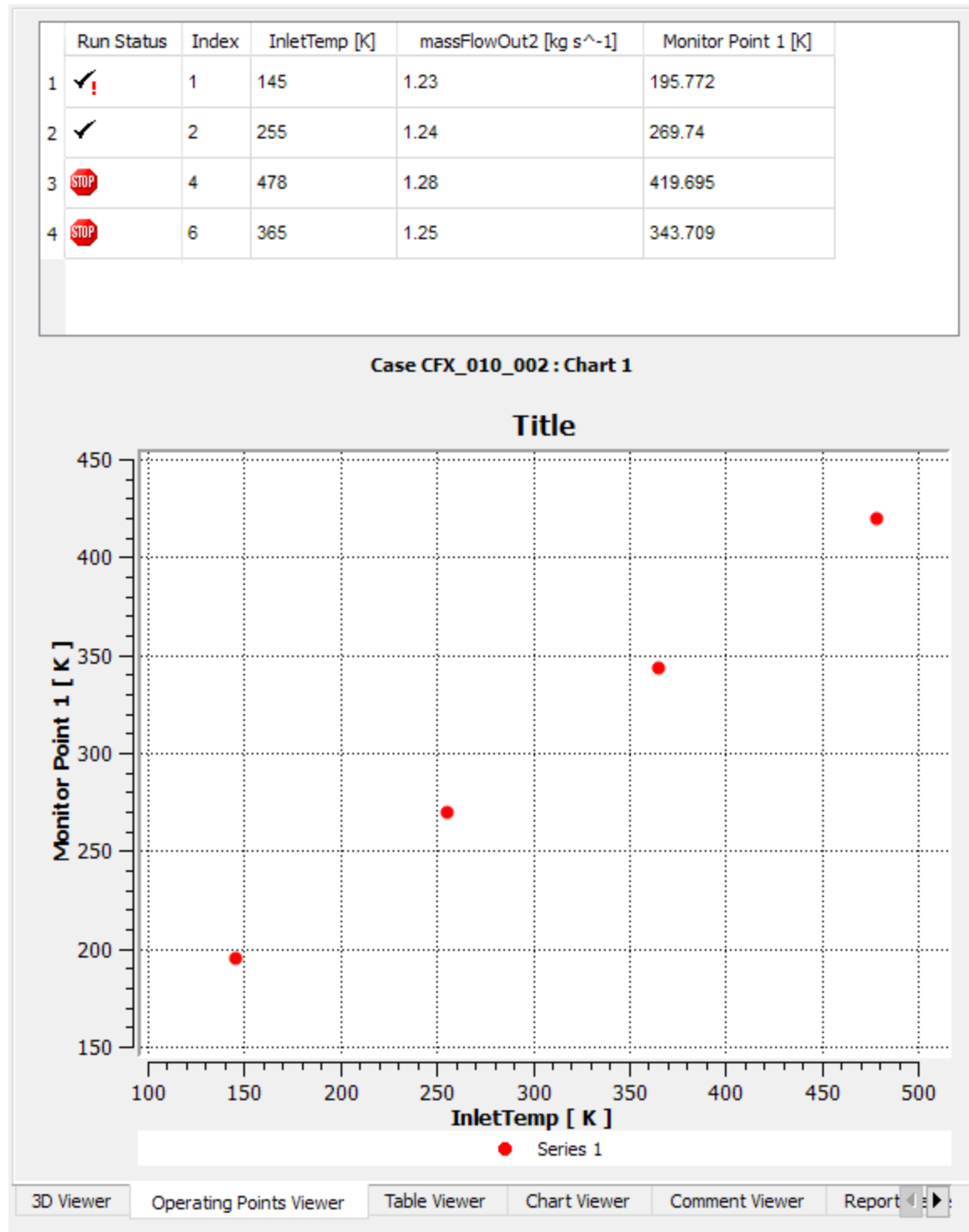
1. Have a standard stereo display
2. Have a graphics card that supports quad buffering OpenGL output
3. Have set your graphics card to "Stereo"
4. Have set your view to Perspective mode (right-click in the Viewer and select **Projection > Perspective**)

...you can view output in stereo. To enable this functionality:

1. Select **Edit > Options**.
2. In the **Options** dialog box, select **CFD-Post > Viewer**.
3. On the **Viewer** panel:
 - a. Set the Stereo **Mode** to **Stereo**.
 - b. Set the **Stereo Effect**. The value of the "stereo effect" that is required is related to the distance between the observer and the display. If the stereo effect is too strong, either move away from the display, or move the slider towards **Weaker**.
4. Click **OK** to save the settings.

Chapter 6: CFD-Post Operating Points Viewer

In CFD-Post, with an operating point case loaded, the **Operating Points Viewer** is accessible by clicking the **Operating Points Viewer** tab at the bottom of the panel on the right side of the interface.

Figure 6.1: Operating Points Viewer

The **Operating Points Viewer** can show, for an Operating Map object listed in the **Outline** tree view, the:

- Operating point parameter table, which shows data for individual operating points.
- Operating map chart, which shows a scatter plot of operating points using the axes defined for the relevant Operating Map object.

- Filter rules, which are criteria for omitting specific operating points from the operating map chart.

If multiple Operating Map objects are listed in the tree view, you can double-click any particular Operating Map object to cause the **Operating Points Viewer** to display data for that object.

The **Operating Points Viewer** has associated icons on the main toolbar:

- *Show/Hide operating points filtering controls*  toggles the visibility of the filter rules. After defining rules, you can hide them to provide more room for displaying the table and/or map chart.
- *Only show operating points table*  makes the operating point parameter table visible and hides the operating map.
- *Only show operating points chart*  makes the operating map chart visible and hides the operating point parameter table.
- *Reset operating points workspace*  makes both the operating point parameter table and operating map chart visible.
- *Apply/Disable operating points filtering*  toggles application of the filter rules (without changing the filter rules).

Also see [Post-processing an Operating Point Case in the CFX-Solver Modeling Guide](#).

The following topics are discussed:

[6.1. Adding Filter Rules](#)

[6.2. Operating Point Parameter Table](#)

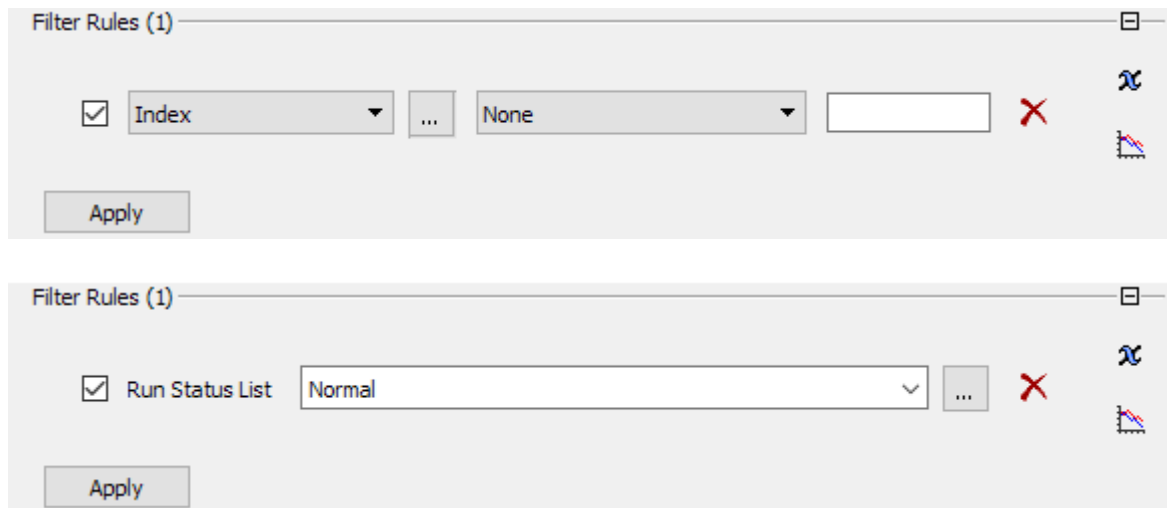
6.1. Adding Filter Rules

You might want to view only a subset of the operating points in the operating point parameter table and operating map chart. You might additionally want to use only a subset of the operating points in computing the response surfaces that underlie the operating map chart. To filter out certain operating points for these purposes, you can define filter rules.

To add a rule:

1. If the **Filter Rules** area of the **Operating Points Viewer** is not shown, click *Show operating points filtering controls* .
2. Click *Add new variable rule*  or click *Add new run status rule* .

A new line appears, which looks like one of the following:



The check box on the left controls whether or not the rule takes effect and the extent of its effect:

- A cleared check box (☐) specifies that the filter rule should be ignored.
- A partially selected check box (☐) specifies that all operating points that do not satisfy the filter rule should be excluded from the operating point parameter table and the operating map chart.
- A fully selected check box (☒) specifies that all operating points that do not satisfy the filter rule should be excluded from the operating point parameter table, the operating map chart, and the generation of all of the response surfaces that underlie the operating map chart.

3. Adjust the settings on the line as necessary.

For a variable rule, note that the first drop-down list contains table column names, the second drop-down list contains relational operators, and the text field (if applicable for the chosen relational operator) holds one value or a comma-separated list of values, as appropriate.

For a run status rule, note that the **Run Status List** options correspond to values that can be shown in the **Run Status** column of the operating point parameter table. For details, see [Operating Point Parameter Table \(p. 135\)](#). The options have the following meanings:

- Normal = run terminated normally
- Error = run failed to terminate normally
- User = run terminated by user request

4. Add more rules as necessary.

- Click **Apply** to apply the selected rules.

Note:

Round-off and limits in numerical precision can affect the evaluation of variable rules. For example, a rule that uses the Greater than or equal relational operator may evaluate to "false" when comparing apparently equal numbers.

6.2. Operating Point Parameter Table

The operating point parameter table is essentially the operating point input parameter table (defined in CFX-Pre) with added columns showing output parameter values (each output parameter corresponding to a monitor defined in CFX-Pre).

The table has columns for run status, index, input parameters, and output parameters. Each row of the table corresponds to an operating point.

In the table:

- The **Run Status** icons and their corresponding text descriptions are the same as those found on the operating point run history page (see [Operating Point Run History Page in the CFX-Solver Manager User's Guide](#)), although, in the **Operating Points Viewer**, the text descriptions are shown in tooltips (when hovering over the icons) as shown below.

	Run Status	Index	MassFlowOut [kg s ⁻¹]	Inlet1Velocity [m s ⁻¹]	Maximum Velocity In2 [m s ⁻¹]
1		1	1000	2	0.92463
2		2	1500	1.5	0.639006
3		3	2500	1	2.76184

Run terminated at user request

Note that a missing **Run Status** icon indicates a missing results file for the corresponding operating point job.

- Double-clicking the header of a table column enables sorting (for the current session of CFD-Post) and sorts the table by that column. Once sorting is enabled, clicking the header of a table column sorts the table by that column. Subsequently clicking the same header reverses the sort order.
- Double-clicking any other cell causes CFD-Post to load the 3D results for, and switch to, the corresponding operating point, subject to the following limitations:
 - CFD-Post cannot switch to an operating point that does not have a mesh included with its results.
 - CFD-Post cannot switch to any operating point if the operating point listed highest in the table does not have a mesh included with its results.
- You can export operating point data from the operating point parameter table in CFD-Post. Simply right-click a table cell and select **Export Table Data to CSV**; the **Export Operating Points Table** dialog box appears.

On the **Options** tab, specify a **File** name, the **Case Name** (if there is more than one case loaded), the **CSV Type**, and the **Unit System**.

The available CSV types are:

- **CSV** — a basic .csv file that can be imported into Microsoft Excel. The file has a single header followed by data.
- **CFX Profile** — a profile file (.csv) with CFX headers.
- **Workbench Design Points** — a Workbench design point .csv file with headers.

On the **Formatting** tab, set **Precision** to the number of digits to be used for numerical data in the exported file.

You can also export operating point data from the operating point parameter table in CFX-Solver Manager. For details, see [Operating Point Run History Page in the CFX-Solver Manager User's Guide](#).

Note:

Operating Point results from Release 19.2 do not show, in the **Run Status** icon text description, the reason for terminating a run.

Chapter 7: CFD-Post Workflow

CFD-Post enables you to qualitatively visualize and quantitatively analyze the processes taking place in a simulation; thus, the general workflow is:

1. Planning steps:
 - a. Decide which variables you want to study (your options are constrained by the variables that were solved in the process of creating the solver results file).
 - b. Determine where in the simulation you want to view those variables.
 - c. Decide how you want to display those variables, either qualitative displays (such as contour plots and charts) or quantitative analysis and displays (such as tables).
2. Production steps:
 - a. Load the solver results file for the simulation into CFD-Post.
 - b. Create any locations, variables, expressions, or functions required.
 - c. Optionally, publish the report, picture, or animation that shows the findings of the study to best advantage.

7.1. Loading and Viewing the Solver Results

When you load a solver results file into CFD-Post, the **3D Viewer** displays the wireframe of the geometry, the mesh, the boundaries, and the domains. You can immediately display on any of the walls or boundaries the values of the variables that were imported with the geometry. You can also use the functions and macros that are supplied.

If those elements do not provide enough resolution, you can make use of CFD-Post's features to add:

- Locations where you can display or analyze variable values (points, point clouds, lines, planes, volumes, isosurfaces, vortex core regions, surfaces of revolution, polylines, user surfaces, and surface groups).
- Using selected locations, generate vectors, contours, streamlines and particle tracks to generate qualitative displays of the results.
- Expressions or macros that you can use to make new variables or to perform quantitative evaluation, integration, and averaging.

7.2. Qualitative Displays of Variables

The display of graphic objects (locations and qualitative displays) occurs in the **3D Viewer** and the **Chart Viewer**.

CFD-Post provides a wide range of control over the **3D Viewer**, such as:

- How the graphical object is to be colored; either prescribed color or by variable. If colored by variable, how the color is mapped over the range of the selected variable.
- Rendering, including transparency, shading, lighting, specularity, and texture.
- Display of lines and faces as well as geometric transformations including rotation, translation, scaling, reflection and instancing.
- Up to four viewports where the orientation of the objects in these miniature viewers can be controlled independently.

The **Chart Viewer** can display data as lines or as symbols.

7.3. Analysis

The quantitative analysis of variables can be displayed in the **Table Viewer** to enable you to display data and expressions.

7.4. Quantitative Analysis of Results

There are a variety of ways you can perform quantitative analysis of results loaded into CFD-Post:

- Use the **Expressions** workspace to make new variables and to numerically process results using a variety of mathematical operations, such as averaging and integration.
- Use the **Calculators** workspace to:
 - Invoke analysis macros supplied for various applications including fan noise, turbomachinery performance, and so on
 - Calculate various measures of mesh quality
 - Probe the value of a function at a given location.
- Use the **Variables** workspace to make new variables.
- Use the **Turbo** workspace to initialize settings for turbomachinery applications.

7.5. Sharing the Analysis

There are a variety of ways to output the results of your analysis:

- Save a picture of the contents of the **3D Viewer** in a variety of formats, including PNG and PostScript.
- Publish a *report*, an HTML publication that includes information about the solver results file, the mesh, and the physics (as well as any other qualitative information, quantitative information, or comments you want to add).
- Produce an animation showing the changes in a variable over a range in the domain.

7.6. Typical Workflow

The following is a typical workflow, which you can simplify, reorganize, or extend suit your work patterns and objectives:

1. Start CFD-Post. ([Starting CFD-Post \(p. 35\)](#))
2. Load one or more results files. ([Load Results Command \(p. 141\)](#))
3. Create expressions ([Expressions Workspace \(p. 96\)](#)) and/or invoke macros ([Predefined Macros \(p. 352\)](#)) to perform the desired numerical processing of results.
4. Create any new variables that will be used for qualitative display. ([Variables Workspace \(p. 89\)](#))
5. Examine the existing locations (wireframe and surface boundaries) and create any additional locators required. ([Location Submenu \(p. 220\)](#))
6. For each locator, select visibility, method of coloring, rendering, and transformation.
7. Create any additional objects (such as lines, vectors, or contours) for quantitative display. ([CFD-Post Insert Menu \(p. 219\)](#))
8. For each object, select visibility, method of coloring, rendering, and transformation.
9. Use the **3D Viewer** to explore the graphic objects and produce animations as required. ([CFD-Post 3D Viewer \(p. 117\)](#))
10. Create tables of data as required and display in the **Table Viewer**. ([Table Command \(p. 300\)](#))
11. Create any desired charts and display in the **Chart Viewer**. ([Chart Command \(p. 305\)](#))
12. Generate or edit any required titles, legends, or labels ([Legend Command \(p. 288\)](#) and [Text Command \(p. 283\)](#))
13. If required, save a picture of the contents in the **3D Viewer**. ([Save Picture Command \(p. 165\)](#))
14. Display the report in the **Report Viewer** and/or modify the report as required. ([Report Command \(p. 165\)](#))
15. Optionally, publish the report to an HTML file. ([Report \(p. 72\)](#))
16. Optionally, save animations. ([Sweep Animation \(p. 334\)](#))

Chapter 8: CFD-Post File Menu

This chapter describes the commands that are available from the **File** menu:

- 8.1. Load Results Command
- 8.2. Close Command
- 8.3. Load State Command
- 8.4. Save State Command and Save State As Command
- 8.5. Save Project Command
- 8.6. Refresh Command (Ansys Workbench only)
- 8.7. Import Commands
- 8.8. Export Commands
- 8.9. Mechanical Import/Export Commands
- 8.10. FSI with Mechanical APDL and CFX: Manual One-way Mapping
- 8.11. Report Command
- 8.12. Save Picture Command
- 8.13. Loading Recently Accessed Files
- 8.14. Quit Command
- 8.15. File Types Used and Produced by CFD-Post

The file types that you can load and display are described in [File Types Used and Produced by CFD-Post \(p. 169\)](#).

8.1. Load Results Command

To load a results file (or files), select **File > Load Results** and browse to the file you want to load. CFX results files and CFX-Solver input files can be loaded from the **Load Results File** dialog box. For information on valid results and CFX-Solver input files, see [File Types Used and Produced by CFD-Post \(p. 169\)](#).

The **Load Results File** dialog box presents you with the following options:

Keep current cases loaded

Controls whether to add to or replace the results that are currently in memory.

If **Keep current cases loaded** is selected, you can choose **Open in new view** to see the two cases side-by-side. If you choose to open the new case the *same* view, the two cases overlap and the title bar of the view displays **All cases**. However, you can use the Viewer's toolbar to manually display two views, then manually display change **All cases** to **Case 1** in **View 1** and **Case 2** in **View 2**. If

you have loaded two cases and you select **Tools > Compare cases**, each case appears in a separate view, with the differences displayed in a third view.

Note:

The **Keep current cases loaded** option is particularly useful to perform simultaneous postprocessing of both fluid (CFX) and solid (Ansys) results when a two-way Fluid-Structure simulation has been performed.

Note:

Before loading any variable from a results file, CFD-Post deletes from its memory any user-defined variable of the same name.

Open in new view

If

- **Keep current cases loaded** is selected, or multiple results files are selected for loading,
- and
- **Open in new view** is selected,

then the new case(s) are opened and displayed in new views, with one view per case, subject to a limit on the number of views that can be displayed together.

Otherwise, no new views are used.

Edit case names

Enables you to change the case name as it appears in the **Outline** tree. The default case name is the filename (without the file type extension). Changing the case name does not affect the filename in the file system.

Clear user state before loading

Loading a results file causes all domain, boundary, and variable objects associated with the results file to automatically be created or updated by default. This would typically include the wireframe model of the geometry and all the boundary conditions created in CFX-Pre. The data associated with a variable is not loaded until the variable is actually used. Any existing objects (such as planes, vector plots) are plotted using the most recently loaded results, if possible. You can disable this behavior by selecting the **Clear user state before loading** check box.

Failing to clear the user state will cause CFD-Post to apply the state of the current file to the results file being loaded. For Turbo cases, it is important to ensure that settings such as the number of instances in 360 degrees is correct (or to adjust the setting to be correct after the file is loaded) as CFD-Post does not automatically check to see if the user settings match between files.

Maintain camera position

Controls the loading behavior when you replace one case with another. When selected, the new case loads in the same orientation and size as the initial case; when cleared, the new case opens to fit into the view.

Load CFX Particle Track data

Controls the loading of the particle tracks that exist in the case.

Construct Variables From Fourier Coefficients

Controls the reading of Fourier coefficients from the results of transient blade row cases. When this option is selected, CFD-Post reads Fourier coefficient data from the results, thereby making it possible to apply data instancing (see [Data Instancing Tab \(p. 63\)](#)), which involves deriving solution variables on an expanded domain; only those variables for which Fourier coefficients exist are available for postprocessing. When this option is not selected, only solved results are available for postprocessing; such results exist only within the simulated passage(s) (for which there was a CFD mesh and a solution provided by the solver).

If you want to postprocess both the derived and solved variables for the same results file, you can use the **Keep current cases loaded** option and load the results twice: once with the **Construct Variables From Fourier Coefficients** option selected, and once with that option not selected.

Run history and multi-configuration options

These settings control how results are loaded for the following types of results:

- A results (.res) file containing a run history.

Such a results file is produced by running the solver using a definition file plus an existing results file (from a previous run) for initial values.

- A multi-configuration results (.mres) file.

Choose **Load only the last results** to load:

- For a results file containing a run history: Only the last results.
- For a multi-configuration results file: The final results file plus all other previous results files belonging to the same configuration.

Choose **Load complete history as > A single case** to load:

- For a results file containing a run history: All of the results history.
- For a multi-configuration results file: All configurations, as a single case.

In either case, only one set of results appears in the viewer, but you can use the timestep selector to switch between results. This option is not fully supported.

Note:

When multi-configuration files are loaded as a single sequence, the solution expressions (Reference Pressure, and so on) represent the last configuration, regardless of which configuration is currently viewed.

Choose **Load complete history as:** > **Separate cases** to load:

- For a results file containing a run history: The results from this file, and the results from any results file in the run history, as separate cases.
- For a multi-configuration results file: All configurations, as separate cases.

In either case, each result appears as a separate entry in the tree.

Note:

When loading a multi-configuration results file, the final results file determines whether all configurations have particle tracks. If the physics for each configuration differs significantly, do not use this method of loading files.

Note:

- You can multi-select results files by holding the **Ctrl** key while you click the filenames.

For transient results, it is recommended that you load only the case file (Fluent results) or the results file (CFX results). CFD-Post automatically loads all of the other data files corresponding to different time steps and creates the time series. Selecting multiple intermediate transient files would lead to multiple sets of results being loaded; each intermediate transient file would load as a separate set of results.

- To unload a set of results, right-click the case name in the tree view and select **Unload**.
 - When a case is unloaded, global variable ranges are not updated.
 - To replace the selected results file with another results file while keeping the state, right-click the case name and select **Replace results file**. (Reloading the results file through the **Load Results** panel may not recover the state completely, in particular when Turbo Post is initialized.) Note that the **Replace results file** function will keep the original case name even though the results file has changed.
-

Domain Selector Dialog Box

If the results file being loaded contains multiple domains, the **Domain Selector** dialog box appears and you are prompted to specify which domains to load. Choosing to load only the domains you require will reduce memory usage and can speed processing time.

If you select the **Don't show this panel again** option, all domains will be loaded automatically on subsequent uses of the Load Results command. Note that you can always re-enable this dialog box from the **Edit > Options > Files** panel (select **Show domain selector before load**).

Run Selector Dialog Box

If you load a fluids project file (extension `.flprj`), the **Run Selector** dialog box appears.

Set **Simulation** to the name of the simulation to load, if applicable.

Set **Run** to the name of the run to load, if applicable.

Set **Data Type** as applicable. The possible choices are:

- FLUENT (CFF-Post)

This option loads the postprocessing data.

- FLUENT (Solution)

This option loads the solution (restart) data.

For details on CFF files, see [Common Fluids Format \(CFF\) Files \(p. 176\)](#).

Solution Units Dialog Box

When you load CFX files into CFD-Post, the solution units that were used by the CFX-Solver are automatically read from the file. When you load a file that does *not* store solution units (such as CFX-4 dump files, CFX-TASC files, Fluent files, or Ansys results files), by default the **Solution Units** dialog box appears and you are prompted to specify the solution units. However, you can enable the **Don't prompt for Solution Units before loading results** toggle to suppress this prompt, in which case the default units of kilograms, meters, seconds, Kelvin, and radians will be used.

Once you have specified the units that were used in the results file, CFD-Post can convert those units to your preferred display units.

You set your preferred display units by selecting **Edit > Options**, then **Common > Units** from the menu bar; for details, see [Setting the Display Units \(p. 210\)](#).

Note:

In CFD-Post, the temperature solution units must be an absolute scale (for example, Kelvin [K] or Rankine [R]); you cannot use Celsius and Fahrenheit. Temperature quantities elsewhere in Ansys CFX can be set in Celsius and Fahrenheit.

Partial Results Files

When a partial results file is loaded, CFD-Post makes available the variables that exist in the full results file, but do not necessarily exist in the partial results file. Variables that do not exist in partial results files are not applicable to the currently loaded time step and are undefined.

You can optionally choose to use variable values that apply to the nearest full results file by changing an option in the **Options** dialog box. For details, see [Turbo \(p. 208\)](#).

8.2. Close Command

The **Close** command closes the currently loaded file, prompting you to save if necessary. CFD-Post remains open. To exit CFD-Post, use the [Quit Command \(p. 169\)](#).

8.3. Load State Command

Selecting **Load State** opens an existing state file.

Note:

Timestep and phase information are not stored in the state file, so loading a state file does not change the timestep or phase.

Overwrite and Append

You can choose to either replace or append to the current state in CFD-Post. You can also choose to load the results file from which the state file was created. The results of these combinations are outlined below.

- **Replace current state** selected and **Load results** selected: The results file used to create the state file is opened, all existing objects are deleted, and new objects that are defined in the state file are created. The results are plotted on the new objects.
- **Replace current state** selected and **Load results** cleared: All existing objects are deleted and new objects that are defined in the state file are created. The results are plotted on the new objects using the existing results.
- **Add to current state** selected and **Load results** selected: The results file used to create the state file is opened. All objects defined in the state file and all existing objects are plotted with the new results. If objects in the state file have the same name as existing objects, the existing objects are replaced by those in the state file.
- **Add to current state** selected and **Load results** cleared: All objects defined in the state file are created and plotted using the current results. Existing objects are not removed unless they have the same name as an object in the state file, in which case they are replaced.

Results files may contain CEL expressions. If you have one or more results files already loaded and you are about to load a state file, you can prevent overwriting these expressions by clearing the **Load results** check box, then selecting the **Preserve current results expressions** check box.

8.4. Save State Command and Save State As Command

When CFD-Post is started from the Ansys CFX Launcher, the **Save State** command produces a CCL file with a .cst file extension. All objects that currently exist in the system are saved to the state file.

Important:

A state file is linked to the results file from which it was created by an absolute path. Therefore, do not change the location of the results file. The state file does not contain the geometry, mesh, or any results; these are loaded from the results file into CFD-Post.

If you have not saved a state file during your current CFD-Post session, selecting **Save State** opens the **Save State** dialog box where you can enter a filename.

If you have already saved a previous state, selecting **Save State** overwrites that file. To save a state to a different filename, you should select **Save State As** from the **File** menu.

When CFD-Post is started from Ansys Workbench, the **Save Project** command writes the current state of the project.

8.5. Save Project Command

When CFD-Post is started from Ansys Workbench, the **Save Project** command writes the current state of the project.

8.6. Refresh Command (Ansys Workbench only)

Reads the upstream data, but does not perform any long-running operation.

8.7. Import Commands

The **Import** menu enables you to import data for:

- A polyline or surface (see [Import Surface, Line or Point Data into CFD-Post \(p. 147\)](#))
- A Fluent particle track file (see [Import Fluent Particle Track File \(p. 149\)](#))
- A Mechanical CDB Surface (see [Import Mechanical CDB Surface \(p. 149\)](#))

Note:

Only CFX-4 and Fluent particle track formats are supported for CFD-Post import.

8.7.1. Import Surface, Line or Point Data into CFD-Post

Using the **Import Surface, Line or Point Data** dialog box, you can read in data for one or more surfaces, polylines, or point clouds.

1. Set **File** to the name of a file of type Comma Separated Values (*.csv) from which to read the data.

You can optionally click *Browse*  to use the **Import Surface, Line or Point Data File** dialog box to select a file.

2. Set **Import As** to one of the following locator types, depending on what you want to create:
 - Surface or Line
 - Point Cloud.
3. Click **OK**.

One locator of the selected type is made for each data set within the CSV file. Each new locator appears in the **Outline** tree view, under **User Locations** and **Plots**. The names of the new locators are based on the data set names provided in the file, and are made unique by numerical suffixes as needed.

Also see [Location Submenu \(p. 220\)](#).

Locator Names

If you import a generic file, the locator that is created is named using the locator name stored in the file, with the prefix **Imported**. If a locator with the same name already exists, the lowest integer greater than 1 that creates a unique name is appended. For example, if the imported file specifies a locator called **Line 1**, the locator that is created is called **Imported Line 1**, unless such a locator already exists, in which case the locator is called **Imported Line 1 1**. If the latter were the case, then importing another file with a locator called **Line 1** would cause the creation of a locator called **Imported Line 1 2**.

Importing Experimental Data in a Customized File

You can import experimental data in a customized file; typically this data will be for a user surface boundary profile or a polyline. The file structures are similar, except that the user surface description requires more information to define the boundary. Refer to [USER SURFACE Data Format \(p. 156\)](#) or [POLYLINE Data Format \(p. 156\)](#) as appropriate.

The example that follows shows experimental data that can be imported into CFD-Post.

Example 8.1: A Surface Data File for CFD-Post

```
[Name]
Experimental Data Set 1
[Data]
Node No., X[m], Y[m], Z[m], Press.[Pa], Vel.[m/s], Temp.[R], ...
0, -0.3, -0.3, -1.0, 0.0, 1.0, 0.224,
1, -1.0, -1.0, 1.0, 1.0, 2.0, 1.35987,
2, -1.0, 1.0, 1.0, 1.0, 3.0, -0.45,
3, -0.3, 0.3, -1.0, 0.0, 4.0, -5.82,
4, 0.3, -0.3, -1.0, 2.0, 5.0, 9.6323,
5, 1.0, -1.0, 1.0, 3.0, 6.0, 7.1859,
6, 1.0, 1.0, 1.0, 3.0, 7.0, -4.656234,
7, 0.3, 0.3, -1.0, 2.0, 8.0, 2.1237,
```



```

8,          0.0,  0.0,  2.0,   5.0,   9.0,   6.456,
[Faces]
# Faces are defined by their points, represented by the point IDs:
#   3 points for a tri-face and 4 points for a quad-face.
# The face normal is defined by the order of the points, so define
# all points in either a clockwise or counterclockwise direction
# to obtain a uniform face normal

0 - 3
# The face above is created from points 0 through 3
7 - 4
4 1 0
# Tri- and quad-faces may be combined
4 5 1
6 3 2
6 7 3
0 3 7 4
2 1 8
6 2 8
5 6 8
1 5 8

```

Importing .stl Files

When importing a .stl file, you select the units in which the selected file was written by choosing the appropriate **Length Units**. This will overwrite any length units already specified in the file.

Only ASCII .stl files are supported.

You can also load STL files from the **Insert** menu in CFD-Post. For details, see [Method \(p. 255\)](#).

8.7.2. Import Fluent Particle Track File

The **Import Fluent Particle Track File** option enables you to import the particle track file associated with the .cas.h5/.cas/.cas.gz/.dat.h5/.dat/.dat.gz/.cdat/.cdat.gz file currently loaded in CFD-Post. The **Import Fluent Particle Track File** dialog box enables you to browse for the appropriate Fluent Particle Track XML file.

For details on creating a particle track file in Ansys Fluent, see [Particle Tracks Dialog Box in the *Fluent User's Guide*](#). For limitations associated with Fluent particle tracks, see [Limitations with Ansys Fluent Files \(p. 185\)](#). For details on configuring the display of a particle track file in Ansys CFD-Post, see [Particle Track Command \(p. 275\)](#).

8.7.3. Import Mechanical CDB Surface

The Import Mechanical CDB Surface feature is fully supported when initiated from the Mechanical side.

The main purpose of the Mechanical import/export facility in CFD-Post is to enable fluid-structure interaction (FSI). The facility enables a mapping of boundary data stored in a CFX results file to a surface stored in an Ansys Mesh (.cdb) file.

Note:

In the volumetric transfer, the temperature at the unmapped nodes will be set to the average temperature of all mapped nodes.

For an example of using this option, see [Mechanical Import/Export Commands](#) (p. 163).

The **Import Mechanical CDB Surface** dialog box has the following options:

8.7.3.1. File

The **File** setting specifies the filename of the file to import. You can type the filepath of the file, or click the *Browse*  icon to search for the file to import.

8.7.3.2. Length Units

The **Length Units** setting specifies what units the imported file will be in.

8.7.3.3. Specify Associated Boundary Check Box

Select the **Specify Associated Boundary** check box to specify an existing boundary to associate with the data in the *.cdb (Ansys mesh) file. When importing Ansys files, you should specify an associated existing boundary. If an export of Ansys data is subsequently performed using the locator from the Ansys file, data from the associated locator is mapped to, and exported with, the Ansys file locator.

8.7.3.3.1. Boundary

The **Boundary** setting specifies the associated boundary for the imported file.

8.7.3.4. Maintain Conservative Heat Flows Check Box

Select the **Maintain Conservative Heat Flows** check box to ensure that the total heat flow for the boundary is equal to that of the imported Ansys surface.

8.7.3.5. Read Mid-Side Nodes Check Box

Select the **Read Mid-Side Nodes** check box to map the Ansys classic geometry for side nodes to CFX geometry. Using this feature can greatly extend the time it takes to load a file as reading the mid-side nodes increases the number of nodes that need to be mapped. Mid-Side nodes are only useful when you perform nodal exports, like Nodal Temperature Export. The mapped mid-side nodes are not used for surface export data calculations.

8.7.3.6. Mapping Success Label

The **Mapping Success** label indicates the percentage of the Ansys surface (.cdb) nodes that have been directly mapped to the CFX boundary surface.

The mapping success is determined in the following order:

1. An Ansys surface node is considered equivalent to the nearest boundary surface node if the node is within a certain tolerance of each other.
2. An Ansys surface node is mapped to the closest face if it is closer to a node than the closest edge.
3. An Ansys surface node is mapped to the closest edge if it is less than a certain tolerance.

Nodes are considered unmapped if none of the above conditions are met, or if the distance from the node to the closest edge is greater than the allowable tolerance. In this case the Ansys surface node is mapped to the nearest boundary surface node regardless of the distance.

Note:

This label only appears in the corresponding **User Surface** panel, after the Ansys surface (.cdb) nodes that have been mapped to the CFX boundary surface.

8.8. Export Commands

The **Export** option has the following sub-options:

[8.8.1. Export](#)

[8.8.2. Export External Data File](#)

[8.8.3. Export Mechanical Load File](#)

8.8.1. Export

The **Export** action enables you to export your results to a data file. You may export results for any available variable in CFD-Post on any defined locator. In the export file, data is written in blocks on a per locator basis in the order given by the locator list. Each block starts with lines listing the values of the selected variables at the locator points (one line corresponds to one point).

The following two examples on how to export data are given at the end of this section:

- [Exporting Polyline Data \(p. 155\)](#)
- [Exporting Boundary Profile/Surface Data \(p. 156\)](#)

8.8.1.1. Export: Options Tab

8.8.1.1.1. File

The **File** setting specifies a file for the data to be exported to. You may type a filename or click

Browse  to search for a file to export the results to, or enter a new filename.

8.8.1.1.2. Type

The **Type** setting has the following options:

Option	Description
Generic	Exports data to a file, writing the data in blocks for every locator. Each block starts with listing the values of the selected variables at the locator points. The Generic option displays the Export Geometry Information check box. For details, see Export Geometry Information Check Box (p. 153) .
BC Profile	Creates a boundary condition profile to be exported. The BC Profile option enables you to select a Profile Type .
Case Summary	Provides a short summary of the results file in xml format.
Geometry Only	Creates a .csv file containing geometric information for the chosen Locations .
STL	Creates a .stl file containing geometric information for the chosen Locations . Only ASCII .stl files are supported.

Note:

If you are using the Geometry Only option to export a surface to set up a User Location in CFX-Pre, ensure that the mesh is in its initial position (not translated, rotated, or deformed). In CFX-Pre, the 2D User Location is independent from the volume mesh, so the mesh used in CFD-Post to create the surface must match the initial mesh found in CFX-Pre.

8.8.1.1.3. Locations

Locations is available only if either the Generic or BC Profile option is selected. The **Locations** setting specifies the locators for which the results of your variable is written. You can hold down the **Ctrl** key to select more than one locator and the **Shift** key to select a block of locators.

8.8.1.1.4. Name Aliases

Name Aliases is available only if either the Generic or BC Profile option is selected. The **Name Aliases** setting specifies custom naming of locators. To change the names of locators that will appear in the output file, insert a comma-separated list of names in the same order as locators.

8.8.1.1.5. Coord Frame

Coord Frame is available only if either the **Generic** or **BC Profile** option is selected. The **Coord Frame** setting specifies the coordinate frame relative to which the data will be exported. Information on creating a custom coordinate frame is available. For details, see [Coordinate Frame Command](#) (p. 286).

8.8.1.1.6. Unit System

The **Unit System** setting determines the units in which the data will be exported. By default, this will use the global units system selected in **Edit > Options**. For details, see [Setting the Display Units](#) (p. 210).

8.8.1.1.7. Boundary Data

Boundary Data is available only if either the **Generic** or the **BC Profile** option is selected. The **Boundary Data** setting enables you to select **Hybrid** or **Conservative** boundary values. For details, see [Hybrid and Conservative Variable Values](#). Setting **Boundary Data** to **Current** will select **Hybrid/Conservative** for each variable depending on the current setting. For details, see [Variables Details View](#) (p. 91).

8.8.1.1.8. Export Geometry Information Check Box

Export Geometry Information is available only if the **Generic** option is selected. Select this check box to export the x, y, z coordinate information of the locator at the beginning of the block.

8.8.1.1.8.1. Line and Face Connectivity Check Box

Line and Face Connectivity is available if the **Generic** or **BC Profile** option is selected. Select this check box to export the connectivity information after the coordinate information in the file.

8.8.1.1.8.2. Node Numbers Check Box

Node Numbers is available only if the **Generic** option is selected. Select this check box to export the node numbers after the coordinate information in the file.

8.8.1.1.9. Profile Type

Profile Type is available only if the **BC Profile** option is selected. The **Profile Type** setting has the following options:

Option	Description
Inlet Velocity	Exports the Velocity Vector variable.
Inlet Total Pressure	Exports the Total Pressure, Total Temperature, and Velocity Direction variables.
Inlet Direction	Exports the Velocity Direction variable.

Option	Description
Inlet Super sonic	Exports the Velocity Vector, Pressure, and Temperature variables.
Outlet Pressure	Exports the Pressure variable.
Wall	Exports the Velocity Vector and Temperature variables.
Custom	Enables you to select custom variables to export from the Select Variable(s) list box.

8.8.1.1.10. Spatial Fields List Box

Spatial Fields is available only if the **BC Profile** option is selected. The **Spatial Fields** list box specifies the coordinate plane axes for the file being exported.

8.8.1.1.11. Select Variable(s) List Box

Select Variable(s) is available only if either the **Generic** or **BC Profile** options are selected. This list box is displayed for the **BC Profile** option only if the **Custom** option is selected for the **Profile Type** setting. This list box selects the variables to export. You can hold down the **Ctrl** key to select more than one variable or use the **Shift** key to select a block of variables.

8.8.1.2. Export: Formatting Tab

8.8.1.2.1. Vector Variables

Vector Variables is available only if either the **Generic** or **Case Summary** options are selected for the **Type** setting in the **Options** tab.

8.8.1.2.1.1. Vector Display Options

The **Vector Display** options enable you to select either **Components** or **Scalar**. The **Components** setting writes each component of a vector to the data file. The components appear inside the selected brackets. The **Scalar** option writes only the magnitude of a vector quantity.

8.8.1.2.1.2. Brackets

Brackets is available only if the **Components** option is selected. The **Brackets** setting selects the type of brackets to wrap around the components.

8.8.1.2.2. Include Nodes With Undefined Variable Check Box

Select the **Include Nodes With Undefined Variable** check box to write **Null Tokens** to the output file. Select the symbol used to denote undefined variable values. For details, see [Null Token](#) (p. 154).

8.8.1.2.2.1. Null Token

Null Token is available only if the **Include Nodes With Undefined Variable** check box is selected. The **Null Token** setting specifies the token to be displayed in the place of an undefined

variable value. You may select the item used as a null token from a predefined list. Examples of a variables with undefined values include **Velocity** in a **Solid Domain** and a variable value at a point outside the solution domain, which can be created using a polyline, sampling plane or surface locator.

Some variables, including **Yplus** and **Wall Shear**, are calculated only on the boundaries of the domain and are assigned **UNDEF** values elsewhere.

If the **Line and Face Connectivity** check box is selected in the **Options** tab, then the **Null Token** is automatically exported.

8.8.1.2.3. Precision

The **Precision** setting specifies the precision with which your results are exported. The data is exported in scientific number format, and **Precision** sets the number of digits that appear after the decimal point. For example, 13490 set to a precision of 2 outputs 1.35e+04. The same number set to a precision of 7 yields 1.3490000e+04.

8.8.1.2.4. Separator

The **Separator** setting specifies the character to separate the numbers in each row.

8.8.1.2.5. Include File Info Header Check Box

Select the **Include File Info Header** check box to export comments at the top of the export file displaying the build date, date and time, and results file from which it is generated.

8.8.1.2.6. Include Header Check Box

Select the **Include Header** check box to include the list locators and a list of variables with their corresponding units. The header should be included for most export applications to ensure successful import into Ansys CFX products.

8.8.1.3. Exporting Polyline Data

To save a polyline or line to a file:

1. Select **File > Export**.

The **Export** dialog box appears.

2. On the **Options** tab:
 - a. Set **Type** to **Generic**.
 - b. Select **Export Geometry Information** and **Export Connectivity**.
3. On the **Formatting** tab, under **Vector Variables**, ensure that the **Vector Display** option is set to **Scalar**.

Note that, on the **Formatting** tab, there is a **Null Token** field. This is used to indicate the string that should be written to represent values that are undefined.

If you want to make your own polyline file with a text editor, follow the format specified below.

For details, see [Polyline Command \(p. 252\)](#).

8.8.1.3.1. POLYLINE Data Format

The following is an abbreviated polyline file:

```
[Name]
Polyline 1
[Data]
X [ m ], Y [ m ], Z [ m ], Area [ m^2 ], Density [ kg m^-3 ]
-1.04539007e-01, 1.68649014e-02, 5.99999987e-02, 0.00000000e+00, ...
-9.89871025e-02, 3.27597000e-02, 5.99999987e-02, 0.00000000e+00,
.
.
.
[Lines]
0, 1
1, 2
.
.
.
[Name]
Polyline 2
.
.
.
```

The name of each locator is listed under the Name heading. Point coordinates and the corresponding variable values are stored in the Data section. Line connectivity data is listed in the Lines section, and references the points in the Data section, where the latter are implicitly numbered, starting with 0.

Comments in the file are preceded by # (or ## for the CFX-5.6 polyline format) and can appear anywhere in the file.

Blank lines are ignored and can appear anywhere in the file (except between the [<data>] and first data line, where <data> is one of the key words in square brackets).

8.8.1.4. Exporting Boundary Profile/Surface Data

Surfaces can be exported and then read into CFX-Pre as a boundary profile (or into CFD-Post as a **User Surface**).

8.8.1.4.1. USER SURFACE Data Format

An abbreviated user surface file, that could be read back into CFD-Post, is shown below:

```
[Name]
Plane 1
[Data]
X [ m ], Y [ m ], Z [ m ], Area [ m^2 ], Density [ kg m^-3 ]
-1.77312009e-02, -5.38203605e-02, 6.00000024e-02, 7.12153496e-06, ...
-1.77312009e-02, -5.79627529e-02, 5.99999949e-02, 5.06326614e-06,
.
.
.
[Faces]
369, 370, 376, 367, 375
350, 374, 367, 368, 351
```



```

.
.
.
[Name]
Plane 2
.
.
.

```

This is similar to the polyline data format described earlier ([POLYLINE Data Format \(p. 156\)](#)), except for the connectivity information. Instead of defining lines, this file defines faces (small surfaces), each by 3 (triangle) to 6 (hexagon) points. The points must be ordered to trace a path going around the face. For proper rendering, the faces should have consistent point ordering, either clockwise or counterclockwise. Each face is automatically closed by connecting the last point to the first point. Face connectivity data is listed in the **FACES** section and references the points in the **Data** section, where the latter are implicitly numbered, starting with 0.

8.8.2. Export External Data File

The **Export External Data File** action enables you to export your results as an Ansys External Data File (.axdt). This file can be imported into the **External Data** system, which can be read into a Mechanical application or System Coupling component system.

The file format for an Ansys External Data File (.axdt) is described in [Ansys External Data File Format in the Workbench User's Guide](#).

8.8.2.1. Options Tab

8.8.2.1.1. File

The **File** setting specifies a file for the data to be exported to. You may type a filename or *Browse*



to search for a file to export the results to. The default filename is `export.axdt` and the default filepath is your current working directory.

8.8.2.1.2. Location

The **Location** setting specifies the locators for which the results of your variables are written.

Note:

For conjugate heat transfer cases, you should make sure that the selected location corresponds to the fluid/solid side of the interface, as intended.

8.8.2.1.3. Unit System

The **Unit System** setting specifies the units for the exported data. By default, this uses the global units system selected in **Edit > Options**. For details, see [Setting the Display Units \(p. 210\)](#).

Note:

The following unit systems in CFD-Post are not supported in the External Data system:

- English Engineering
- British Technical
- US Customary
- Custom

In such cases, the External Data system cannot determine the unit system used in the file. If any of the above systems is selected in CFD-Post, a warning is issued, and the data will be exported in SI units.

8.8.2.1.4. Boundary Data

The **Boundary Data** setting specifies Hybrid or Conservative boundary values. If **Boundary Data** is set to Current, this setting is picked up from each variable. For details, see [Variables Details View \(p. 91\)](#).

8.8.2.1.5. Select Recommended Variables

The **Select Recommended Variables** list box allows you to select the variables to export. You can select from the following options:

- Heat Flow

Note:

For Fluent results, **Heat Flow** is not always defined at the CHT interface boundaries. You should ensure that heat fluxes are available at the boundaries you export.

- HTC and Wall Adjacent Temperature

This option allows you to export the Wall Heat Transfer Coefficient and Wall Adjacent Temperature variables to the external data file.

This option is not available for inviscid flows and Eulerian multiphase cases. For Eulerian multiphase cases, you can select phase-specific temperatures from the **Additional Variables** list.

For a list of Fluent field variables and their equivalent in CFD-Post, see [Fluent Field Variables Listed by Category](#) (p. 437).

Note:

For cases where you have specifically solved for conjugate heat transfer, it is recommended that you export the solid-side Temperature, instead of HTC and Wall Adjacent Temperature.

The HTC and Wall Adjacent Temperature variable is not calculated for laminar flow and will not appear in the **Export External Data File** window for such cases.

- Temperature

For multiphase cases, CFD-Post will calculate and output the **Bulk Temperature**, which is the temperature weighted by the volume fraction of the individual phases. You can select phase-specific temperatures from the **Additional Variables** list.

Note:

For cases where you have specifically solved for conjugate heat transfer, it is recommended that you export the solid-side temperature instead of the fluid-side temperature.

For multiphase cases that include a combined temperature variable instead of phase-specific temperatures, CFD-Post will not calculate the **Bulk Temperature**. However, the combined temperature variable will be written to the output file under **Bulk Temperature**.

Note:

User defined variables named Temperature or Heat Flow will be ignored and a warning message will be issued.

8.8.2.1.6. Select Additional Variables

In addition to the variables listed in the **Select Recommended Variables** list box you can select other field variables you have set to output in Fluent. You can hold down the **Ctrl** key to select more than one variable or use the **Shift** key to select a block of variables.

Note:

User defined variables named Temperature or Heat Flow will be ignored and a warning message will be issued.

8.8.2.2. Formatting Tab

The Formatting tab enables you to specify only a precision value. This setting is the same for the **Export** command. For details, see [Export \(p. 151\)](#).

8.8.3. Export Mechanical Load File

The selected CFX data that is exported by CFD-Post is interpolated onto the Ansys surface from the associated CFX boundary. The interpolated data is then exported to an Ansys load file. For details on how to associate a boundary with an Ansys surface, see [Specify Associated Boundary Check Box \(p. 150\)](#).

When verifying the load applied to the surface in Mechanical, note that the Pressure variable available in CFD-Post is not the same as the element stress representing the static structural load. The element stress variables, Normal Stress, Shear Stress, and Stress (the latter being the combination of Normal Stress and Shear Stress) are vector quantities, whereas the Pressure variable is a scalar quantity.

To compare actual boundary data in CFD-Post with the data that is exported:

1. Load the fluid results file into CFD-Post.
2. Create a new vector variable named Surface Force Density.
3. Specify the components as follows: X => Force X / Area, Y => Force Y / Area, and Z => Force Z / Area.
4. Color the FSI boundary by Surface Force Density.

The plot of Surface Force Density on the FSI boundary should look similar to the plot of Imported Pressure in Mechanical. There will be some minor differences due to interpolation and differences in mesh density.

For an example of using this option, see [Mechanical Import/Export Commands \(p. 163\)](#).

Note:

One-way FSI cases with porosity transfer only the fluid quantities for 2D Temperature, 3D Temperature, Heat Transfer Coefficient, and Heat Flux.

Note:

Transfer of data between Fluent and Mechanical through CFD-Post is based on nodal data. Values that are visualized in CFD-Post may differ from values that are exported. For example in cases where Wall Heat Flux values are zero on the mapped surface, you would see zero total flux on the surface, as this calculation is based on face data. However, nodal data may have nonzero values on surface edge nodes, as it is interpolated from all faces that touch those nodes, including the ones from the neighboring surface (which may have nonzero flux). This is the data that will be mapped to Mechanical.

8.8.3.1. Options Tab

8.8.3.1.1. File

The **File** setting specifies the filepath and filename of the file to be exported. You may click the **Browse**  icon to select the name and location of the file to be exported. The default name is `export.sfe` or `export.xml` (depending on which File Format is chosen) and the default filepath is your current working directory.

8.8.3.1.2. Location

The **Location** setting selects the Mechanical surface object to export, which is generated by importing a `.cdb` file. For details, see [Import Mechanical CDB Surface \(p. 149\)](#).

Note:

The Mechanical load file does not contain mesh coordinate data, and must be interpreted along with the `.cdb` file originally imported into CFD-Post.

8.8.3.1.3. Unit System

The **Unit System** setting specifies the units for the exported data. By default, this uses the global units system selected in **Edit > Options**. For details, see [Setting the Display Units \(p. 210\)](#).

8.8.3.1.4. Boundary Data

The **Boundary Data** setting specifies Hybrid or Conservative boundary values. If **Boundary Data** is set to Current, this setting is picked up from each variable. For details, see [Variables Details View \(p. 91\)](#).

8.8.3.1.5. Export Data

The **Export Data** setting has the following options:

Option	Description
Normal Stress Vector	Exports Normal Stress variable data onto imported surf154 surfaces. Normal Stress is a vector variable calculated from the normal component of Force.
Tangential Stress Vector	Exports Shear Stress variable data onto imported surf154 surfaces. Shear stress data is calculated from the tangential component of Force.
Stress Vector	Exports Stress variable data onto imported surf154 surfaces. Stress data is calculated by vector summing the normal stress and shear data.
Heat Transfer Coefficient	Exports convection variable data onto imported surf152 surfaces. When selected, the Specify Reference Temperature check box will appear. See Specify Reference Temperature (p. 162) , below.

Option	Description
Heat Flux	Exports the Heat Flux variable data onto the surf152 surfaces.
Temperature	Exports the Temperature variable data on the nodes of the imported surface.

8.8.3.1.6. Fluids

Fluids is available only if either the Tangential Stress Vector or Stress Vector options are selected. The **Fluids** setting specifies which fluids, or All Fluids, that will affect the elements shear or stress values.

8.8.3.1.7. Specify Reference Temperature

Specify Reference Temperature is available only if the **Heat Transfer Coefficient** option is selected. Select this check box to enable you to specify a fixed reference temperature value or expression.

1. If you specify a reference temperature, then the exported heat transfer coefficient is calculated based on Heat Flux and Temperature data. If the case models radiation, then the heat transfer coefficient also includes the contribution to the heat flux from radiation.

Note that the variable Surface Heat Transfer Coef is not recognized by CFD-Post for one-way FSI.

2. If you do not specify a reference temperature, the exported data is based on the Wall Heat Transfer Coefficient and Wall Adjacent Temperature. This includes only the convective contribution to the heat flux, and does not include any radiative contribution.

For Fluent Cases: To transfer HTC from Fluent cases without specifying a reference temperature (method 2 above), the following variables have to be exported to the DAT/CDAT file:

- Wall Func. Heat Tran. Coef (which will be converted to the CFX variable Wall Heat Transfer Coefficient)
- Temperature. In Fluent, the wall adjacent temperature is calculated by averaging the adjacent cell temperatures to the wall nodes.

Note:

Enabling (or disabling) **Specify reference (bulk) temperature** affects all further calculations of the associated Ansys variables on the relevant locator (for example, Wall Adjacent Temperature on <surface name>), and so potentially changes any plots of these associated Ansys variables.

8.8.3.2. Formatting Tab

The **Formatting** tab enables you to specify only a precision value. This setting is the same for the **Export** command. For details, see [Export \(p. 151\)](#).

8.9. Mechanical Import/Export Commands

The Mechanical import/export facility enables a mapping of boundary data stored in a CFX results file to a surface stored in an Ansys mesh (.cdb) file.

8.9.1. Mechanical Import/Export Example: One-Way FSI Data Transfer

You can perform one-way FSI operations manually (by exporting CDB files from Mechanical APDL, importing the surface in CFD-Post, and exporting the SFE commands).

To create an Ansys load file using CFD-Post to transfer FSI data:

1. Load the fluids results file, from which you want to transfer results, into CFD-Post
2. Select **File > Import > Import Mechanical CDB Surface**. The **Import Ansys CDB Surface** dialog box appears.
3. In the **Import Mechanical CDB Surface** dialog box, either:
 - Select the CDB file that specifies the surface mesh of the solid object to which to transfer data. Also select the **Associated Boundary** for the surface to map onto, and make other selections as appropriate.
 - Select the XML document that provides all transfer information. Click **OK**, and the surface data is loaded.
4. Select **File > Export > Export Mechanical Load File**. The **Export Mechanical Load File** dialog box appears.
5. In the **Export Mechanical Load File** dialog box, select a filename to which to save the data. For the **Location** parameter value, select the imported Ansys mesh object. Under **File Format** select **Ansys Load Commands (FSE or D)**. (Alternatively, you can select **WB Simulation Input (XML)** to get XML output.) Also select the appropriate data to export: Normal Stress Vector, Tangential Stress Vector, Stress Vector, Heat Transfer Coefficient, Heat Flux, or Temperature. Click **Save**, and the data file is created.

The one-way FSI data transfer described above is performed automatically when using the **FSI: Fluid Flow (CFX) > Static Structural** custom system in Ansys Workbench. For details, see the [FSI: Fluid Flow \(Ansys CFX\) > Static Structural in the Workbench User's Guide](#) section in the Ansys documentation.

8.10. FSI with Mechanical APDL and CFX: Manual One-way Mapping

You can use CFD-Post to manually generate a load file for Mechanical APDL:

1. Write out a .cdb file that contains the surface or volume mesh by using the CDWRITE command in Mechanical APDL.

For surface load mapping, create a layer of SURF154 elements (pressure) or SURF152 elements (thermal) on the boundary of interest and write out only these elements to the .cdb file.

Note:

If you write out a .cdb file for Temperature surface mapping, there is no need to create surface-effect elements because Temperature will be the Degrees Of Freedom and is set directly. Instead, select the nodes on the surface of interest and write these to the .cdb file.

2. Load the .cdb file into CFD-Post using File > Import > Import Mechanical CDB Surface.

Select the associated CFD boundary (disable for body import).

Note:

Pick the solid side of a CHT interface.

Upon importing the .cdb file, a User Surface object is created in CFD-Post.

- For surface mapping, you can enable the visibility and view the mesh using the usual Render options.
- For volume mapping, enabling the visibility will not display the Mechanical volume mesh.

To see the node locations, create a Point Cloud object scoped to the User Surface with Sampling = Vertex and a Reduction Factor of 1.

3. Export the load file from CFD-Post using **File > Export > Export Mechanical Load File**.

- For Stress, Heat Flux, and HTC, the load file will contain SFE commands to apply loads via the SURF152/SURF154 elements.
- For Temperature:
 - A surface load file will contain D commands to set the Degrees of Freedom.
 - A body load file will contain BF commands.
- You must export in Celsius.
- A structural analysis reads BF loads in Celsius regardless of the units selection in Workbench.

You can open the exported file in a text editor to make sure the values look reasonable.

4. Read the load file into Mechanical APDL using the /input command.

Make sure your solution units are consistent with the values exported from CFD-Post. In particular, note that Mechanical is Celsius by default, while CFX is Kelvin.

5. After solving, check the Solution Information and the .err file to make sure the /input command was successful. If the file was not read, the solution will still proceed without the load applied.

8.11. Report Command

The **File > Report** menu item has the following options:

Report Templates

Invokes the **Report Templates** dialog box, where you can browse the list of existing templates or add (register) a new template. The existing templates are for turbomachinery simulations.

To learn how to use templates, see [Report Templates \(p. 76\)](#).

Load 'Generic Report' Template

Reloads the default template.

Refresh Preview

Updates the report that is displayed in the **Report Viewer**. You need to do this after making changes to your report.

This command is equivalent to clicking on the **Refresh** icon at the top of the **Report Viewer**.

Publish

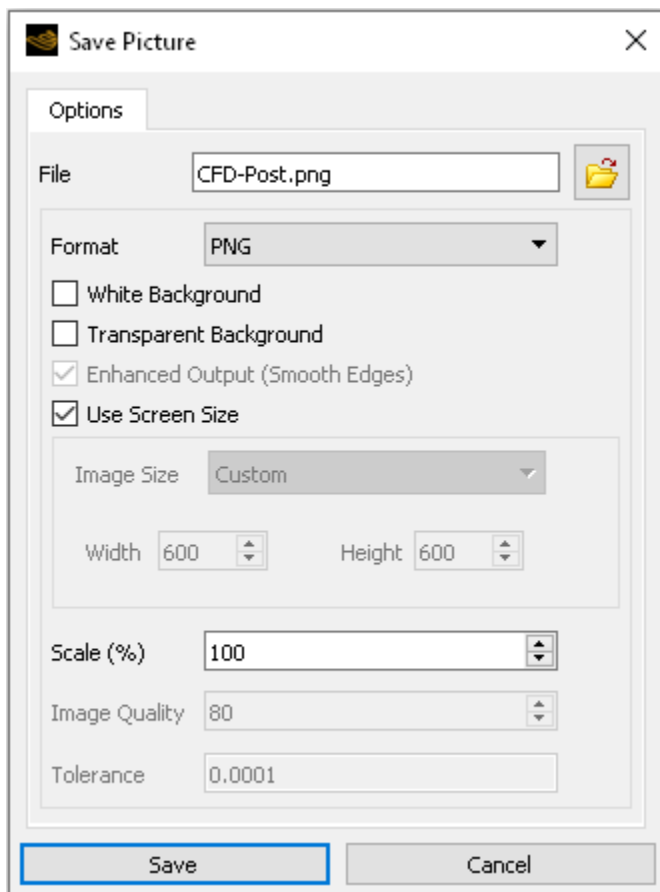
Displays the **Publish Report** dialog box, where you can configure the format and name of your report. See [Publishing the Report \(p. 87\)](#) for details.

This command is equivalent to clicking on the **Publish** icon at the top of the **Report Viewer**.

To learn more about publishing a report, see [Report \(p. 72\)](#).

8.12. Save Picture Command

To save the current contents of the viewer window to a file, select **File > Save Picture** from the main menu or click *Save Picture* . The **Save Picture** dialog box is displayed.



Note:

When you save a picture of the **Chart Viewer**, note that the font sizing and line thickness in the chart image output are adjusted to approximately reproduce the appearance of the chart at the initial screen size (700x700 pixels), and are independent of the output image resolution.

Options Tab

The **Options** tab has the following settings:

File

Enables you to specify the filename of the file. You may enter the filename and path into the **File** field, or click the *Browse*  icon and search for the directory in which the file is to be saved.

Files are always saved with the file extension corresponding to the selected graphics format.

Format

To choose a file format, click . When creating a new image file, the file format you choose affects the quality of the image:

PNG

Portable Network Graphics is a raster file format (*.png) that supports lossless image compression.

AVZ (3D)

Ansys Viewer Format is a file format (*.avz) used to present interactive three-dimensional views. It can be displayed using the Ansys Viewer.

JPEG

A compressed file format (*.jpg) developed for compressing raw digital information. File sizes are relatively small. Due to compression artifacts, this format is not recommended for line drawings.

PPM

Portable Pixel Map is a file format (*.ppm) with uncompressed data and is not recommended for large images.

PostScript and Encapsulated PS

PostScript (*.ps) and Encapsulated PS (*.eps) are generally recommended for output to a printer or line drawings. However, some graphics objects and features can cause the PS/EPS to output as a very large bitmap file, in which case a PNG file would be a more efficient alternative. Note that the Ansys logo and the axis do not cause the PS/EPS output to become a bitmap.

Note:

Previously generated CFD-Viewer State (*.cvf) files can be converted by using an application supplied as part of the CFD-Post installation:

```
<CFDPOSTROOT>/bin/cfx5cvfconvert <cvf-file>
```

White Background Check Box

You can save the current image with a white background by selecting **White Background**.

When **White Background** is selected, certain white objects may be colored black and certain black objects may be colored white in the image file. Objects that are not affected can usually be manually colored by editing them.

Note:

You can adjust the viewer background colors in the preferences (**Edit > Options > CFD-Post > Viewer**).

Transparent Background Check Box

Transparent Background specifies a transparent background for a PNG image.

Note:

GPU rendering is required to support transparent backgrounds. In the user interface, ensure that the **Use GPU Rendering for Printing** option is selected (restart the application after changing that option). For details, see [Viewer Setup \(p. 211\)](#). If playing a session file in batch mode to save PNG images, run CFD-Post with GPU rendering enabled. For details, see [Optional Command Line Arguments \(p. 37\)](#).

Enhanced Output (Smooth Edges) Check Box

Note:

This setting is available only when using software rendering. Note that software rendering can be slower than GPU rendering and does not correctly render some graphics objects (that is, OpenGL shader graphics). It is recommended that you use GPU rendering whenever possible. You can turn GPU rendering on and off from the preferences (**Edit > Options > Common > Viewer Setup > Use GPU Rendering for Printing**).

When **Enhanced Output (Smooth Edges)** is selected, the image is processed by antialiasing.

Use Screen Size Check Box

When **Use Screen Size** is selected, the output has the same width and height, measured in pixels, as shown in the viewer. You can clear the check box to specify the width and height using other options.

Image Size

With **Use Screen Size** not selected, you can set **Image Size** to one of the following options:

- HD: 1280 pixels wide and 720 pixels high
- FHD: 1920 pixels wide and 1080 pixels high
- QHD: 2560 pixels wide and 1440 pixels high
- 4k: 3840 pixels wide and 2160 pixels high
- 8k: 7680 pixels wide and 4320 pixels high
- Custom: Set **Width** and **Height** in pixels.

Width/Height

With **Use Screen Size** not selected, and with **Image Size** set to Custom, you can specify the width and height of the image in pixels by entering values for **Width** and **Height**.

Scale (%)

Scale (%) is used to scale the size of bitmap images to a fraction (in percent) of the current viewer window size. This option is available when **Use Screen Size** is selected.

Image Quality

Image Quality is available only for the JPEG format. A value of 100 specifies the highest image quality; a value of 1 specifies the lowest image quality.

Tolerance

Note:

This setting is available only when using software rendering. Note that software rendering can be slower than GPU rendering and does not correctly render some graphics objects (that is, OpenGL shader graphics). It is recommended that you use GPU rendering whenever possible. You can turn GPU rendering on and off from the preferences (**Edit > Options > Common > Viewer Setup > Use GPU Rendering for Printing**).

Tolerance is a non-dimensional value used in face sorting when generating pictures. Larger values result in faster generation times, but may cause defects in the resulting output.

Click **Save** to save the current viewer contents to an image file.

8.13. Loading Recently Accessed Files

CFD-Post saves the filepaths of the last six results files, state files, and session files. To re-open a recently used file, select it from the **Recent Results Files**, **Recent State Files**, or **Recent Session Files**, as appropriate.

8.14. Quit Command

To exit from CFD-Post, select **File > Quit** from main menu. Objects created during your CFD-Post session are not automatically saved. If there is no state file in memory, the state was changed since the file was opened, or since the last state save, a dialog box asks whether you want to save the state before closing. For details, see [Save State Command and Save State As Command \(p. 147\)](#).

8.15. File Types Used and Produced by CFD-Post

This section describes the file types used by CFD-Post and the software that outputs those file types:

8.15.1. [Ansys CFX Files](#)

8.15.2. [Ansys Meshing Files](#)

8.15.3. [Ansys Mechanical Results Files](#)

8.15.4. [Ansys Icepak Files](#)

8.15.5. [CGNS Files](#)

8.15.6. [Common Fluids Format \(CFF\) Files](#)

[8.15.7. Ansys Fluent Files](#)

[8.15.8. Ansys Forte Files](#)

[8.15.9. Ansys FENSAP-ICE Files](#)

[8.15.10. CFX-4 Dump Files](#)

[8.15.11. CFX-TASCflow Results Files](#)

Note:

Of the files that CFD-Post can load, only CFX-Solver Results files with extension `.res` contain both hybrid and conservative values. For details, see [Hybrid and Conservative Variable Values in the CFX Reference Guide](#).

8.15.1. Ansys CFX Files

Case Files (.cfx)

A case file is generated when you save a simulation in CFX-Pre. The case file contains the physics data, region definitions, and mesh information for the simulation and is used by CFX-Pre as the 'database' for the simulation setup.

A case file is a binary file and cannot be directly edited.

CFX-Mesh Files (.gtm)

GTM files (`.gtm`) contain mesh regions that can be used to set up a simulation in CFX-Pre or viewed in CFD-Post.

Note:

GTM files that contain mesh data should not be confused with GT-SUITE `.gtm` files, which are used by GT-SUITE, for example when running a CFX simulation coupled with GT-SUITE.

Limitations with a GTM File in Multiple Results Files

When loading a state file that loads multiple results files including a GTM file, load the mesh file first, and then apply the state.

CFX-Solver Input Files (.def, .mdef)

A CFX-Solver input file is created by CFX-Pre. The input file for a single configuration simulation (`.def`) contains all physics and mesh data; the input file for multi-configuration simulations (`.mdef`) contains global physics data only (that is, Library and Simulation Control CFX Command Language specifications). An `.mdef` input file is supplemented by Configuration Definition (`.cfg`) files that:

- Are located in a subdirectory that is named according to the base name of the input file

- Contain local physics and mesh data.

Note:

Use the `-norun` command line option (described in [Command-Line Options and Keywords for cfx5solve in the CFX-Solver Manager User's Guide](#)) to merge global information into the configuration definition files, and produce a CFX-Solver input (`.def`) file that can be run by the CFX-Solver.

CFX-Solver Results Files (.res, .mres, .trn, .bak)

Intermediate and final results files are created by the CFX-Solver:

- Intermediate results files, which include transient and backup files (`.trn` and `.bak`, respectively) are created while running an analysis.
- Final results files for single and multi-configuration simulations (`.res` and `.mres`, respectively) are written at the end of the simulation's execution. For multi-configuration simulations, a configuration result file (`.res`) is also created at the end of each configuration's execution.

Each results file contains the following information as of the iteration or time step at which it is written:

- The physics data (that is, the CFX Command Language specifications)
- All or a subset of the mesh and solution data.

An alternate file format, Common Fluids Format (CFF), is also available. For details, see [Common Fluids Format \(CFF\) Files \(p. 176\)](#).

CFX-Solver Backup Results Files (.bak)

A backup file (`.bak`) is created at your request, either by configuring the settings on the **Backup** tab in **Output Control** in CFX-Pre, or by choosing to write a backup file while the run is in progress in the CFX-Solver Manager.

CFX-Solver Transient Results Files (.trn)

A transient results file (`.trn`) is created at your request, by configuring the settings on the **Output Control > Trn Results** tab in CFX-Pre.

CFX-Solver Error Results Files (.err)

An error results file (`.err`) is created when the CFX-Solver detects a failure and stops executing an analysis. The `.err` file can be loaded into CFD-Post and treated the same way as a `.bak` file, but if the CFX-Solver encounters another failure while writing the `.err` file, it may become corrupted and accurate solutions cannot be guaranteed.

Session Files (.cse)

Session files are produced by CFD-Post and contain CCL commands. You can record the commands executed during a session to a file and then play back the file at a later date. For details, see [New Session Command \(p. 215\)](#).

You can also modify session files in a text editor.

State Files (.cst)

State files are produced by CFD-Post and contain CCL commands. They differ from session files in that only a snapshot of the current state is saved to a file. You can also write your own state files using any text editor. For details, see [Save State Command](#) and [Save State As Command](#) (p. 147) and [Load State Command](#) (p. 146).

8.15.1.1. Transient Blade Row Postprocessing

Transient Blade Row models available in Ansys CFX (described in [Transient Blade Row Modeling in the CFX-Solver Modeling Guide](#)) make it possible to obtain temporally accurate solutions of transient blade row interaction, with vastly reduced memory requirements and computational time. There are two main types of case where these transient blade row models can be applied:

- Single-domain modeling, such as frozen gust (inlet disturbance (for one domain only, either rotor or stator)).
- Single-stage, multi-domain modeling, such as transient rotor-stator, or any multistage model.

When a transient blade row case is loaded, CFD-Post automatically creates valid solution variables for postprocessing (see [Variables Tree View](#) (p. 89)).

Locator Object Limitations:

- For transient blade row cases, transient statistics for boundary-only variables (such as Force, Mass Flow, Heat Flux) are not available on the following postprocessing locators: points, lines, cut-planes, isosurfaces, and iso clips.
- CFD-Post displays the global range for plots in transient blade row cases differently than in other cases. In a transient blade row case, the global range is computed and extended dynamically based on the selected domain(s) and timestep. For example if you change the definition of a plot to include another domain, or if you change the timestep, the global range in the plot's legend will be extended automatically to cover both the original and the new plot definitions.

Transient blade row cases also interact with the timestep selector; see [Using the Timestep Selector with Transient Blade Row Cases](#) (p. 331) for details.

8.15.1.2. Limitations with Ansys CFX Files

- For transient rotor stator simulations with rotating frames of reference, velocity gradients in the rotating domain will be read incorrectly into CFD-Post. To avoid this problem, you can define user-defined variables for the velocity gradients.

8.15.2. Ansys Meshing Files

Both .cldb files (created in the Meshing application) and .dsdb files (created in Simulation) behave the same in CFD-Post as .gdm (and .def) files. For details, see [File Types Used and Produced by CFD-Post](#) (p. 169).

Note:

- You must have Ansys Workbench installed in order to be able to load Ansys Meshing files (cldb and dsdb) into CFX-Pre or CFD-Post.
 - CFD-Post does not support .cldb files generated by the Meshing application prior to Release 11.0.
-

8.15.3. Ansys Mechanical Results Files

Note:

CFD-Post does not support reading results files from Mechanical application Release 2021 R1 or later.

Ansys solver files are created from the Ansys solver. CFD-Post is able to read results for temperature, velocity, acceleration, magnetic forces, stress, strain, and mesh deformation. The Mechanical solver files may have load-step variables and time steps; CFD-Post will represent both as time steps. The valid file types are *.rst (deprecated), *.rth (deprecated), *.rmg (deprecated), *.inn, *.inp, *.cdb.

When Ansys solver files are read together with CFX-Solver files, fluid dynamics and solid mechanics results can be analyzed simultaneously. For details on how to load multiple files, see [Load Results Command](#) (p. 141).

The deformations due to change in temperature and stress/strain of the mesh can be amplified by using the **Deformation** option available by right-clicking the viewer background. For details, see [CFD-Post 3D Viewer Shortcut Menus](#) (p. 121).

8.15.3.1. Limitations with Ansys Mechanical Results Files

There are some important limitations with Mechanical results files:

- If you are postprocessing Mechanical harmonic-analyzed files and all of the variable values are incorrectly displaying as zeroes, you can set the environment variable CFD-POST_RST_SKIP_LAST_DATASET=1 to plot the variables with nonzero values from the second-last dataset of the results file (which does not contain frequency values of zero).
- CFD-Post does not fully support undefined and user-defined as values for RST units. In those cases, CFD-Post assumes SI units.

If you want to use CFD-Post to postprocess a Mechanical RST file outside of Ansys Workbench and you do not want to have CFD-Post assume SI units, you can set an environment variable

(CFDPOST_RST_SHOW_UNITS_DIALOG=1) to cause the **Solution Units** dialog box (described in [Solution Units Dialog Box \(p. 145\)](#)) to appear. Note that this will cause the **Solution Units** dialog box to appear even when CFD-Post can read the units in the results file.

- The Ansys Mechanical solver does not output minimum/maximum ranges for each of the calculated variables; these ranges are calculated when the results file is loaded by CFD-Post. Calculating the range for a very large problem would, however, require prohibitively large amounts of CPU time. As a result, range values are calculated for the loaded time step only. This means that values that appear as global range, are in fact ranges that exist for that time step only, at first. As more time steps are added, the global range is extended accordingly. If you want to enable the calculation of true global ranges (and incur the potentially large CPU time each time you load a non-Ansys CFX file), you can do this by selecting **Edit > Options** and selecting **Pre-calculate global variable ranges**, under **Files**. For details, see [Files \(p. 205\)](#).
- CFD-Post plots only Mechanical variables that exist in RST files; unlike Mechanical, it will not calculate other variables automatically. Therefore, some variables that you would expect to be able to plot (as in Mechanical) either will be missing or will have all zero values in CFD-Post.
- RST files do not store data for principal stresses. There is a slight difference for principal stress calculated in CFD-Post and Mechanical. CFD-Post calculates principal stresses on a node by averaging the stresses on each element touching the node. However, Mechanical calculates principal stresses based on node-averaged stresses. This difference can be minimized by the use of a finer mesh.

You can also configure Mechanical to calculate principal stresses the same way as CFD-Post by issuing the command `avprin, 1`, and replotting the values.

- By default, a Mechanical results file does not contain the definitions of any components that you may have created in the simulation set up and so these will not be available as regions for plotting in CFD-Post. However, it is possible to produce an additional "components" file that does contain these definitions. If CFD-Post finds a file with the name `<filename>.cm` in the same directory and with the same filename (excluding the file extension) as the Mechanical results file, then it will read component definitions from this file. For instance, if you are postprocessing the Mechanical results file `OscillatingPlate.rst`, CFD-Post will look for the file `Oscillating-Plate.cm` in the same directory to find component definitions. You can make Mechanical write the components file by including a command `CMWRITE, <jobname>, cm` in your Mechanical input file, before the `SOLVE` command. If `.cm` files are to be loaded into CFD-Post, job names need to be consistent across restarts, input file processing, and regular runs.
- Components files (CM files) must have been output in blocked format (which is the default output format). Refer to the Ansys Mechanical documentation to learn how to control the Mechanical output format.

All regions from components files are read as surfaces. If a region is volumetric, CFD-Post will read the outer surface only.

CFD-Post will read only nodal components. Components that consist of elements will be ignored.

- CFD-Post can read a limited number of Mechanical results files that contain shell elements only. It depends on the problem setup details as to whether a file can be successfully read or not. CFD-Post cannot read any Mechanical results files that contain no 3D or shell elements.

- When reading RST files, CFD-Post ignores mid-side nodes and duplicate nodes. The latter situation occurs when a case has multiple bodies with matching meshes on the interfaces. The simulation picks up duplicate nodes and plots accordingly, giving a discontinuous plot. However CFD-Post picks up only one of the nodes, causing one domain to appear to spill into the next.
- In Mechanical, simulation characteristics such as maximum values are derived from actual local node values. In CFD-Post values need to be presented on global nodes, therefore CFD-Post takes a simple average from all shared elements' local values. When compared the two calculations will be similar, but not exactly the same.
- Mechanical results files that contain mid-edge nodes will cause mesh reports to be inaccurate (the Connectivity range may be reported as zero).
- When Mechanical electrical data is read in CFD-Post, units are not displayed for quantities such as Electric Potential or for Electric Flux^[1].
- Mechanical results files contain shell elements with data on both the top and bottom sides. CFD-Post currently does not support separate data on the two sides. Only data from the top side will be read. If you want to load results from the bottom side, you can set the environment variable `CFDPOST_RST_READ_BOTTOM_SHELL_DATA`.
- When exporting Heat Flux to Mechanical, you should always select the fluid side of the interface.

8.15.4. Ansys Icepak Files

When using Ansys Icepak with CFD-Post, a custom session file (`IcepakCustomVariables.cse`) loads into CFD-Post when you load results files (`.cas` and `.dat`) or refresh the Results cell in Ansys Workbench. This session file creates variables required to use the Ansys Icepak files fully within CFD-Post.

Icepak exports the solution variables: Pressure, X/Y/Z velocities, and temperatures. In addition, for some models you can select additional variables in Icepak to export to CFD-Post:

User defined memory 0/1/2

The X/Y/Z direction thermal conductivities at every node in the domain.

Heat Flux Vector

Akin to velocity vectors, you can use this variable to plot heat flux vectors at any location in the domain.

Thermal Cross

The cross product of the Heat Flux and the gradient of Temperature. You can use this to locate regions of large heat flux vectors that are not aligned with high temperature gradients, therefore implying possible regions for shorting heat flow.

[1] Electric Flux is the electric current density.

Thermal Chokepoint

The dot product of Heat Flux and the gradient of Temperature. You can use this to locate regions of high heat flux coupled with high thermal resistances.

8.15.5. CGNS Files

CFD-Post has limited support for reading meshes and solutions from CGNS (CFD General Notation System) files of version 2.4 or 4.X. Extensions for such files are typically .cgns or .cgs. The following documentation describes the supported and unsupported features of CGNS.

Note:

- By default, variable names are adjusted to conform to the CFX naming style, which may not always be desirable. To preserve the original variable names in the file, open **Edit > Options > CFD-Post > Files** and clear **Translate variable names to CFX-Solver style names**.
- Only one CGNS file can be postprocessed at a time; multiple simultaneous CGNS files are not supported.

Supported	Not supported
3D problems	1D and 2D problems
Elements of the following types: TRI_3, QUAD_4, TETRA_4, PENTA_6, PYRA_5, HEXA_8 and MIXED	Elements of any type not listed in the "Supported" column
Base #1	Base selection
Steady-state solutions (solution #1)	Transient solution
	Regions
	Zone connections
	Periodic crossings

8.15.6. Common Fluids Format (CFF) Files

CFD-Post has limited support for reading Common Fluids Format (CFF) files, which are written by some Ansys solvers, for example those of Ansys Fluent and Ansys CFX.

CFF data is supplied within a number of files that include:

- At least one case file (extension ".cas.XXX", where "XXX" is solver-specific).

A case file includes the mesh and physics settings such as the definitions of domains and boundary conditions. It may also contain additional data depending on the solver that wrote the file.

A case file contains no reference to the associated results data file(s). However, if there is exactly one results data file in the same directory as a case file, and if it has the same file name (excluding the extension), CFD-Post will assume that the results file is associated with the case file.

- At least one results data file (extension ".dat.XXX", where "XXX" is solver-specific).

A results data file contains the solution data and other output data. It also contains a reference to its associated case file.

- (Typically) a fluids project file (extension ".flprj").

The project file contains information that is used for transient postprocessing and file associations when sequences of files are supplied. Note that some solvers provide the files referenced by the ".flprj" file in a subdirectory named the same as the ".flprj" file without the file type extension.

The solver-specific part of the file extension is generally ".h5" for Fluent and ".cfx" for CFX.

The recommended way to load CFF files into CFD-Post is to load the fluids project file. This is especially important for solutions that involve multiple case and data files, which must be loaded in the correct sequence. If you do not have the fluids project file, then you can load either the results data file or the case file:

- If you load a results data file, the corresponding case file will also be loaded. This is especially useful when there is more than one results data file for a given case file.
- If you load a case file, then CFD-Post will check to see if there is exactly one corresponding results data file; if there is, it will also be loaded.

The following topics are discussed:

[8.15.6.1. Supported Functionality](#)

[8.15.6.2. Limitations with Ansys CFF Files](#)

8.15.6.1. Supported Functionality

The following topics are discussed:

[8.15.6.1.1. Physical Objects \(Domains, Boundary Conditions and Subdomains\)](#)

[8.15.6.1.2. Topological Regions](#)

[8.15.6.1.3. Meshes](#)

[8.15.6.1.4. Mesh Zones](#)

[8.15.6.1.5. Steady State and Transient simulations](#)

[8.15.6.1.6. Multi-configuration \(Modified Mesh/Modified Physics\)](#)

[8.15.6.1.7. Single-phase and Multi-phase Solutions](#)

[8.15.6.1.8. Naming of Variables](#)

8.15.6.1.1. Physical Objects (Domains, Boundary Conditions and Subdomains)

- When supplied, physical objects, such as those listed above, are shown in the CFD-Post tree as objects and their geometries are shown in the CFD-Post viewer. Normal operations can be carried out on these objects.
- Physical objects are user-defined.

- Physical objects are defined in terms of Topological Regions that are, in turn, defined in terms of the mesh (see [Meshes \(p. 178\)](#)).
- Names of physical objects are preserved as supplied except when they don't conform with naming requirements, in which case they are automatically translated to conform (see [Name Limitations in CFD-Post \(p. 181\)](#)).

8.15.6.1.2. Topological Regions

- Topological regions are groups of one or more Mesh Zones (see below), normally of the same dimensionality
- Topological regions are read from the file but are normally not displayed because physical objects are preferred. However, if physical objects such as domains and boundary conditions are not supplied, the topological regions are displayed.
- In the case where neither physical objects nor topological regions are supplied by the file, the underlying mesh zones are displayed (see [Mesh Zones \(p. 178\)](#)).

8.15.6.1.3. Meshes

- Meshes can be defined in 2D (Axisymmetric and Planar) and 3D. Both can be read by CFD-Post.
- Meshes might be defined in two forms: one that explicitly defines the cells and faces in terms of vertices; one where only the faces are defined in terms of vertices and the cell definition must be derived from the faces.
- In CFD-Post, 2D meshes are extruded to 3D meshes. The extruded distance might not be identical to that seen when previous file formats are read.

8.15.6.1.4. Mesh Zones

- Meshes are defined in terms of collections of cells, faces and nodes known as zones. **Note:** These are not be confused with solver zones.
- Unless neither physical objects nor topological regions are supplied, mesh zones will not be presented in the user interface because it is unlikely that they have any physical significance.

8.15.6.1.5. Steady State and Transient simulations

- Both steady state and transient simulations can be read by CFD-Post.
- In the case of transient simulations, CFD-Post can only read simulations using a fluids project file that defines the order of the time steps.
- Solver Timestep, Time, Crank Angle and other information might be read from the fluids project file depending on what is supplied by the solver.

Note:

Timesteps are not displayed if they cannot be successfully loaded and validated.

8.15.6.1.6. Multi-configuration (Modified Mesh/Modified Physics)

- Limited support is available for multi-configuration runs obtained from CFX-Solver cases.
- Multi-configuration runs from Fluent are supported.

Note:

Some transient cases loaded from Fluent or CFX may be discovered to be multi-configuration on read when they are not truly multi-configuration. This does not have a major impact on postprocessing.

8.15.6.1.7. Single-phase and Multi-phase Solutions

- Supported by CFD-Post
- Where possible, CFD-Post uses the phase name supplied by the solver (see [Name Limitations in CFD-Post \(p. 181\)](#)).

8.15.6.1.8. Naming of Variables

- Most solvers supply results variables using their native naming convention. Some of these variables have been translated to follow CFD-Post naming conventions for backwards compatibility.
 - It is possible to alter the naming convention used by CFD-Post by using the option in **Edit > Options > CFD-Post > Files > Variables > Common File Format**. By default, the convention is that of CFD-Post.
 - When the Native naming convention is used, CFD-Post uses the variable name supplied by the solver.
- Species names are preserved where possible (see [Name Limitations in CFD-Post \(p. 181\)](#)). This may mean that the names differ from those seen in previous file formats.

8.15.6.2. Limitations with Ansys CFF Files

Note that not all of the data available in other file formats is necessarily available in CFF files. The documentation of a solver that writes CFF files might mention limitations on which data it writes to CFF files.

Note:

The HDF5 version used in 2025 R1 will result in failure when the underlying file system does not support file locking, or where locks have been disabled. To disable all file locking operations, create an environment variable named `HDF5_USE_FILE_LOCKING` and set the value to the five-character string **FALSE**.

CFD-Post has limitations for reading and using the data contained in CFF files:

- CFD-Post does not support reading CFF files that were written by Ansys solvers older than Release 2020 R2.
- CFD-Post does not support reading CFF files that were written by Ansys solvers other than CFX-Solver or Fluent.
- CFD-Post offers limited or no functionality in these general areas:
 - Derived variables (limited support)

Results loaded into CFD-Post can show differences in variable values at specific locations depending on the data source. For example, there can be differences in derived variable values between CFF files and legacy files.
 - Axis of rotation
 - Rotation speed
 - Solution residual and monitor data
- CFD-Post offers limited or no functionality in these areas when reading CFF files from CFX:
 - Mesh coordinate transformations
- CFD-Post offers limited or no functionality in these areas when reading CFF files from Fluent:
 - Mesh (msh . h5) files (limited support)
 - Hanging nodes
 - Overset mesh
 - Particle tracks

Note that existing XML particle track files can be written by Fluent and read into CFD-Post.
 - Volume mesh data
 - Lattice Boltzmann: mesh or data
 - The same limitations that CFD-Post has with reading legacy files from Fluent, as described in [Limitations with Ansys Fluent Files \(p. 185\)](#)

The following topics are discussed:

- [8.15.6.2.1. Name Limitations in CFD-Post](#)
- [8.15.6.2.2. Solver Model Limitations in CFD-Post](#)
- [8.15.6.2.3. Transient limitations](#)
- [8.15.6.2.4. Solution Limitations in CFD-Post](#)
- [8.15.6.2.5. Solver Specific Limitations in CFD-Post](#)
- [8.15.6.2.6. CFF-Post File Limitations](#)

8.15.6.2.1. Name Limitations in CFD-Post

Names read from CFF files are preserved as much as possible unless a non-native naming convention is used for variables. However, CFD-Post might change some characters in any name to meet the requirements of the expression language and CCL.

There is a user preference, **Common Fluids Format > Name Convention**, that controls the optional renaming of variables loaded from CFF files. For details, see [Variables \(p. 207\)](#).

Fluent species names may not be consistent with those read from legacy cas/dat files. They are however, consistent with those seen in Fluent.

When reading Fluent files that lack the names of some periodic and symmetry boundaries, the names generated might differ from those generated when reading corresponding legacy files.

8.15.6.2.2. Solver Model Limitations in CFD-Post

There is limited support for automatic Turbomachinery functionality.

8.15.6.2.3. Transient limitations

You should always read transient data using the supplied .flprj files. CFD-Post does not generate transient sequences from a directory of CFF files.

8.15.6.2.4. Solution Limitations in CFD-Post

- CFD-Post reads and displays data as supplied by the solver, except when cell or face data is interpolated onto the vertices.
- CFD-Post does not derive all the variables that are supported with legacy files. Some derived variables might show different values than those calculated when reading legacy files.
- Some variable names might not be translated to CFD-Post names. There is limited support for variable name translation from some solvers.

Note:

The following limitation applies to Solution files from Fluent:

- Variable Wall Adjacent Temperature might be incorrect.
-

8.15.6.2.5. Solver Specific Limitations in CFD-Post

Some functionality might be limited when compared to support for legacy results files. For details, refer to the documentation of the solver that writes the CFF files.

8.15.6.2.6. CFF-Post File Limitations

Support for Common Fluids Format - Post files is limited.

Note:

The following limitation(s) apply to CFF-Post files from Fluent:

- Turbo initialization is not supported.
-

8.15.7. Ansys Fluent Files

CFD-Post can load Fluent version 6 and version 12.0 or later (preferred) result files (which include case, data, and fluids project files) and mesh files for postprocessing.

Fluent writes files that contain data that enable Fluent to restart smoothly. The data in these files is not designed for postprocessing outside of Fluent and is limited when doing so. Fluent can also export mesh and solution data to a number of other formats; CFD-Post provides support for several of these formats.

8.15.7.1. Ansys Fluent Mesh Files

It is also possible to load mesh files. Mesh files have extensions:

- .msh.h5 (Common Fluids Format)
- .msh (legacy)
- .msh.gz (legacy, compressed version of .msh)

Important:

CFD-Post can read files written from Fluent, but the reading of mesh files written from TGrid or GAMBIT is not supported and may cause CFD-Post to terminate abnormally.

Important:

CFD-Post cannot import Boundary Mesh files, even though Boundary Mesh files have a .msh.h5, .msh, or .msh.gz file extension.

8.15.7.2. Fluent Restart Files

Fluent Restart files are comprised by one or more case files and one or more data files and may be written in a number of formats.

Case files have extensions:

- .cas.h5 (Common Fluids Format)

- .cas (legacy)
- .cas.gz (legacy; compressed version of .cas)

Data files have extensions:

- .dat.h5 (Common Fluids Format)
- .dat (legacy)
- .dat.gz (legacy; compressed)

Restart files do not contain all variables you might want to postprocess. CFD-Post does not calculate derived variables, therefore only variables available in the file can be used. However, you can export any variable to the data files from Fluent 12.0 or later.

Additional Variables

Additional variables selected via the **Data File Quantities** panel in Fluent for output to a Fluent Restart file are written to the user-specified section of a data file. Any variable of same name from the basic section of the data file will not be read in CFD-Post; precedence goes to the variable in the user-specified section.

8.15.7.3. Fluent Postprocessing Files

Fluent Restart files are comprised of one or more case files and one or more data files and may be written in a number of proprietary and 3rd-party formats.

The CDAT format is designed for use with CFD-Post and EnSight, and requires at least one legacy case file to be provided by Fluent. It is not possible to load a CDAT file with a case file in CFF.

CDAT files have the following extensions:

- .cdat (legacy; variables interpolated to nodes specifically for CFD-Post)
- .cdat.gz (legacy; compressed version of .cdat)

8.15.7.4. Ansys Fluent Project Files

Fluent might also write a Fluids Project file. Typically this is needed when loading a sequence of files in the CFF format.

Fluids project files have extension .flprj.

To load CFF files from Fluent, you should load the fluids project (.flprj) file if it exists. For details on CFD-Post's handling of CFF files, see [Common Fluids Format \(CFF\) Files \(p. 176\)](#).

Note:

To load legacy (non CFF) files from Fluent, you should select only one file from a sequence of related Fluent files in the **Load** dialog box (normally the final timestep's data file); other related files are loaded automatically.

8.15.7.5. Ansys Fluent Particle Track Files

CFD-Post can load Fluent particle track files. Particle track files have a `.xml` file extension and are loaded separately to the Fluent Restart files.

To load a particle track file, you must use the **File > Import > Import Fluent Particle Track File** option after loading the case and data file.

See "Limitations in the Import and Display of Fluent Particle Tracks" in [Limitations in CFD-Post using Fluent Files \(p. 192\)](#).

8.15.7.6. Summary of Ansys Fluent File Extensions

File Type	Mesh File	Case File	Data File	Project File
CFF Mesh	<code>.msh.h5</code>			
CFF Restart		<code>.cas.h5</code>	<code>.dat.h5</code>	<code>.flprj</code>
CFF Post		<code>.cas.post</code>	<code>.dat.post</code>	<code>.flprj</code>
Legacy Mesh	<code>.msh/.msh.gz</code>			
Legacy Restart		<code>.cas/.cas.gz</code>	<code>.dat/.dat.gz</code>	
Legacy CDAT		<code>.cas/.cas.gz</code>	<code>.cdat/.cdat.gz</code>	

Important:

Fluent files of extension `.dat.h5`, `.dat`, and `.dat.gz` can contain variables that show differences when displayed in CFD-Post when compared to Fluent. To avoid these differences, use Fluent files of extension `.cdat` or `.cdat.gz`.

You should explicitly specify the quantities that you want to postprocess when exporting information from Fluent to CFD-Post via `.cdat` or `.cdat.gz` files. See [Exporting to Ansys CFD-Post in the *Fluent User's Guide*](#) for details on exporting such files.

Alternatively, if you do not want to export additional files, you can select additional postprocessing quantities to write to a Fluent `.dat` or `.dat.gz` file. See [Setting Data File Quantities in the *Fluent User's Guide*](#) for details on how to select additional postprocessing quantities. Using regular `.dat/.dat.gz` files can result in quantitative differences from Fluent while postprocessing in CFD-Post.

8.15.7.7. Considerations When Naming Geometries in Fluent

- You need to be careful when choosing geometry names in Fluent when the file will be read in CFD-Post. The geometry names must not contain special characters such as '-', '|', and ':'. All such characters will be replaced by a space (which is allowed in names in CFD-Post).

- There might be problems postprocessing transient Fluent cases that involve changes of geometry name or changes of boundary type at intermediate time steps.

8.15.7.8. Limitations with Ansys Fluent Files

Fluent files are supported with the following limitations:

File-based Limitations

- The restrictions on loading CFF files also apply to Fluent CFF Case and Mesh (.h5) files. For details, see [Limitations with Ansys CFF Files \(p. 179\)](#).
- CFD-Post can read Fluent case files that have imprinted surfaces defined in them. However, the imprinted surfaces are ignored in CFD-Post.

Limited support of files from Previous Versions

- Grid interfaces from Fluent versions 6.3 and older are not supported by CFD-Post. If your .cas file has old grid interfaces, read the .cas and .dat file into Fluent Release 12.0 (or later), run at least one iteration, and save the file to change to the new grid interfaces. This converts grid interfaces to use the virtual polygon method and makes the file readable in CFD-Post. Attempting to read old grid interfaces may cause CFD-Post to exit.
- To postprocess forces or fluxes using the DBNS solver of Fluent for cases from versions prior to Release 12.0, you must read the case into Fluent Release 12.0 (or later), iterate at least once, and then write out the .cas and .dat files.
- Holes may appear in Planes/iso-surfaces created using old Fluent mesh/case files (from earlier than Release 14.5) for hex-core/cut-cell mesh cases.

Limitations When Loading Fluent Files

- There is no support for loading of a subset of domains. All domains are always loaded by CFD-Post.
- CFD-Post cannot read Fluent cases that have CAS and DAT files output in different directories.
- For transient Fluent cases, there is no support for adding or removing time steps in the timestep selector.

Mesh Limitations

- The number of nodes, faces and cells that can be read from files written by Fluent may exceed the number supported by CFD-Post.
- In Ansys Fluent, any cell can have any number of faces. The maximum number of nodes in a polyhedron cell is 256 and in a polygon face is 128 (however the contour-creation algorithm has a limit of 64 nodes per face).
- For some cases (for example, shell conduction model), the number of cells/elements reported by Fluent is more than that of CFD-Post. This difference is due to the additional cells Fluent creates internally for solving some physics; these are never written into the case file. Fluent reports include these cells in addition to those available to CFD-Post.

Limitations With Fluent Cases Defining Rigid Body Motion

- CFD-Post can read zone-motion variables (origin, axis, omega, grid-velocity) on profile boundaries from Fluent files only if the variables are constants. For cases where any of these values are specified as non-constants, CFD-Post ignores the variable. In such conditions, CFD-Post cannot transform the velocity from absolute to the relative frame or the reverse. Other dependent variables (such as Mach Number and Vorticity) will not be available unless explicitly exported from Fluent to a Fluent Postprocessing file or written as an additional variable (see [Additional Variables \(p. 183\)](#)). There will be a warning message issued when these files are read.

Limitations for 2D Cases

- Ansys Fluent can solve 2D problems. The mesh and data within the Ansys Fluent files for these cases is defined using 2D cells and faces.

CFD-Post will extrude the 2D case on read so that it is displayed as a quasi-3D case.

If the 2D case is axisymmetric, the extrusion will be performed about the Y-axis such that it forms a 3D wedge.

Note:

For axisymmetric cases, the point values in CFD-Post may differ from values reported in Fluent due to the extrusion of the 2D domain in the theta direction.

If the case is not axisymmetric, the extrusion will be performed in the direction of the Z axis such that it uses the *reference-depth*.

- There are very small differences between Fluent and CFD-Post in the way that area is calculated for axisymmetric cases; this area is used in quantitative functions. Fluent calculates the area for any axisymmetric case as $2\pi r$, where r is centroid-y of the facet. CFD-Post extrudes the 2D geometry to create a 3D wedge (of wedge angle 7.5°) then calculates the area by repeating the wedge to create a 360° cylinder. This is similar to approximating the perimeter of a circle by measuring the perimeter of an inscribed uniform polygon.
- The results of calculations by CFD-Post for Fluent 2D cases are for a *reference-depth* from the Fluent case file. For axisymmetric cases, the reference-depth is 2π ; that is, the results calculations are for the complete cylindrical region, and not what is shown in the viewer (which is a sector from the complete cylinder). This matches the behavior of Fluent.

Results will be consistent for all quantitative calculations on all locations. For example, a slice plane will be assumed to be cutting the full cylinder.

8.15.7.9. Quantitative Differences Between CFD-Post and Fluent

Quantities may be handled in CFD-Post differently than in Fluent. There may be a number of reasons for this, some of which are detailed below.

Solution Quantities

- Fluent Restart files provide solution data at Cell and/or Face centers. Nodal values are not available. The data provided by Fluent has to be averaged to Node locations for use within CFD-Post.

Faces in different surfaces that share one or more nodes may provide different face values or their parent cells provide different cell values. Boundaries are therefore given different priorities such that a value from a boundary with a higher priority will be used in the interpolation process over one of a lower priority.

In a case with two or more domains, in Fluent the nodes on the boundary will get their values from the domain that has the higher priority; CFD-Post uses the average of the boundary values from both domains.

Fluent Postprocessing files provide results data that may be available at nodal locations. If this is the case, these values will be read from the file and used by CFD-Post. In the absence of nodal data, face and cell data may be available and the use of this data will be governed by the same algorithms (with the same limitations) as those used for Fluent Restart files.

- Only nodal values are used for all qualitative purposes (except for contour plots) and for all quantitative purposes. It is expected that there may be differences in quantitative results when compared to Fluent.

Vector Quantities

- Averaging of vector quantities to nodes differs between CFD-Post and Fluent. In Fluent, vector magnitudes are averaged to nodes explicitly; in CFD-Post, only vector components are averaged to nodes, while the magnitude is calculated from the components at the nodes. The two magnitudes will differ in cases with sharp vector gradients or high face angles (usually due to a coarse mesh).

For example, if a node has four faces attached that have shear stresses in directions radially away from the node, in CFD-Post the shear stress values at the node will be much smaller in magnitude compared with the face stresses because the stresses in opposite directions cancel out. In Fluent, the direction is ignored and only magnitude is taken into account while calculating the stress magnitude at the node.

- Due to differences between Fluent and CFD-Post in the handling of vector quantities, velocity values can be different on 'interior' zones (conformal domain interfaces).

Domains

- In a case with two domains, the nodes on the boundary in Fluent will get their values from the domain that has the higher priority; CFD-Post uses the average of the boundary values from both domains.

Boundaries

- On boundaries, CFD-Post produces more accurate quantitative results involving geometric variables (such as X, Y, Z) than Fluent reports. This is because Fluent uses geometric variable data from adjacent cell centers instead of the boundary face centers. However, you can get CFD-Post results to match Fluent results exactly by setting the environment variable `CFDPOST_MATCH_FLU-ENT_RESULTS` to 1 before running CFD-Post.
- In CFD-Post, on boundaries that have zero velocity, Total Temperature and Total Pressure will have the same values as Temperature and Pressure, respectively (as expected). In Fluent, Total Temperature is different from Temperature for boundaries that have zero velocity; similar differences apply between Total Pressure and Pressure. This is a limitation in Fluent.

If you export 'Total Pressure' from Fluent, the CFD-Post results will be closer to what Fluent shows.

Periodic Boundaries

- When plotting velocity components on periodic boundaries, there may be differences in CFD-Post, when using Fluent Restart files, compared to Fluent. You should use Fluent Postprocessing files, as outlined in [CDAT for CFD-Post and EnSight](#), to get the correct values.
- A periodic surface in Fluent is actually a pair of surfaces. In CFD-Post this pair appears as a Periodic object and a corresponding Periodic Shadow. When looking at quantitative results in CFD-Post, you need to look at a surface group that contains the "periodic/periodic-shadow" to see output that is in agreement with Fluent's results.

Hanging Nodes

- CFD-Post and Fluent display contours differently in the vicinity of a hanging node. Fluent takes values from cells only on one side, causing a discontinuity of contours. In CFD-Post, the hanging node is made to be a conformal node and takes values from cells on both sides, making a smoother contour.

Interfaces

- For cases with 1:1 interfaces, due to a difference in the handling on nodes at these interfaces, the number of nodes reported by CFD-Post will be different than the number reported by Fluent. However, the number of cells should match.
- Field variables may be discontinuous across an interface that joins domains of differing porosity. Variable values on the interface boundaries are not averaged across the interface. You can inspect variable values on each side of the interface separately.

Specific Variable Considerations when Postprocessing Fluent Results in CFD-Post

- For Fluent results loaded into CFD-Post, all variables, including X, Y, and Z, are stored at either cell centers (for 3D regions) or face centers (for boundary regions) — not at nodes. Fluent can export node based values to Fluent Postprocessing files as outlined in [CDAT for CFD-Post and EnSight in the *Fluent User's Guide*](#), to get the correct values.

As a result, when nodal data doesn't exist, the `minVal()` and `maxVal()` functions return minimum and maximum cell-centered values of X,Y, or Z on 3D locations, and minimum and maximum face-centered values of X, Y, or Z on boundary regions.

- The nodal based results in CFD-Post are similar but not always identical to those shown in Fluent because Fluent can apply some model-specific transformations not available in CFD-Post when cell and face values are mapped to node values.
- In CFD-Post, a node value on a wall boundary takes precedence over other values for the same node for other boundary types, so all wall nodes will have wall values irrespective of whether they are also used in any other boundary.
- CFD-Post does not smooth out values across non-conformal interface boundaries; that is, there must be a 1-1 mapping of nodes across the interface. As a result, contour and color plots as well as iso-surfaces are discontinuous across these interfaces. Therefore, the node values displayed by CFD-Post at non-conformal interfaces may differ from those shown by Fluent.

- Plots and calculations that are based on node values (rather than cell/face values) can have undesired results where nodes are shared by multiple boundaries at an adjoining edge.
- Also see [Quantitative Differences Between CFD-Post and Fluent \(p. 186\)](#).

Gradients

- There may be substantial differences between gradients calculated in the Fluent solver and gradients calculated in CFD-Post. The Fluent solver uses options such as boundary treatments and limiters to calculate gradients; CFD-Post calculates gradients independently of the Fluent solver, and does not have access to all of the same data.

The gradients computed by the Fluent solver can be transferred into CFD-Post by exporting them as variables and then loading them into CFD-Post.

- For Fluent files, the gradients computed by CFD-Post are discontinuous across domains.

Heat Flux

- The variable `Boundary Heat Flux Sensible` is available only for boundary types velocity-inlet, mass-flow-inlet, pressure-inlet, pressure-outlet, pressure-far-field, and outflow.
- For Fluent files using the energy model, `Heat Flux` is available for all boundaries and `Wall Heat Flux` is available only for walls. The values of these two variables will be the same on walls.

Also see "Wall Heat Flux".

Mass Flow

- CFD-Post may report an incorrect sign for `Mass Flow` through a conformal internal boundary (known as an "interior" type in Fluent) that is located at the boundary between two domains. To determine the direction of mass flow, use a CEL expression based on the normalized dot product of the local velocity and the face normal. The equations for computing the corrected mass flow through a conformal internal boundary, based on the local mass flow summed over the boundary, are:

Corrected Mass Flow = $\text{sum}(\text{abs}(\text{Mass Flow}) * \text{massFlowCorrectionNorm}) @ <\text{region}>$

$\text{massFlowCorrection} = \text{Normal X} * u + \text{Normal Y} * v + \text{Normal Z} * w$

$\text{massFlowCorrectionNorm} = -\text{massFlowCorrection} / \text{abs}(\text{massFlowCorrection})$

- For some cases, the fluxes (`Mass Flow( ) @ <surface>` or `AreaInt(Boundary Heat Flux) @ <surface>`) from CFD-Post are different from the values reported by the **Flux Reports** panel from Fluent.

This is due to some additional physics model-based calculations done by Fluent that are not available in CFD-Post. However, you can use the Fluent **Surface Integral** or **Volume Integral** panel results for comparison with CFD-Post.

Mesh Variables

- You cannot use X, Y, or Z variables in expressions or plots in moving mesh transient cases. For example, instead of an Isosurface of X, use a YZ Plane.

Molar Weight

- Molar Weight will always have units of kg/mol in CFD-Post, but units of kg/kmol in Fluent. This will be true for all quantities involving 'mol' or 'kmol' in units.

To learn how to control the units displayed by CFD-Post, see [Setting the Display Units \(p. 210\)](#).

Pressure

- In the cavitation model in Fluent, the minimum value for Pressure is limited by the cavitation pressure; this is not done in CFD-Post.

Shear Stress

- CFD-Post will not display any Shear Stress values on coupled non-conformal interfaces as shear stresses are undefined on such interfaces.
- A DBNS solver with laminar flow will have no shear stress on any walls. Force calculations will not include any viscous component in such cases.

Species, Reaction, and VOF

- For Species, Reaction and VOF cases, values in CFD-Post results match with Fluent "Vertex" values instead of "Face" values.

To have results from CFD-Post match results from Fluent, set the environment variable `CFD_POST_BOUNDARY_DATA_FROM_ELEMENTS` to 1.

Total Pressure

- CFD-Post reads Total Pressure from the Fluent results file if such data exists. However, if the data is not supplied in the file, Total Pressure is not calculated by CFD-Post.

Unsteady Statistics

- Unsteady statistic variables (such as Mean X Velocity and RMS Static Pressure) are not consistent when read from the standard Fluent Restart file. If needed, you can export these variables explicitly in a Fluent Postprocessing file or append the variables to the Fluent Restart file (see [Additional Variables \(p. 183\)](#)).

User-Defined Memory

- CFD-Post does not support User-Defined Node Memory (UDNM) variables from Fluent.
- CFD-Post reads User-Defined Memory (UDM) and User-Defined Scalars (UDS) as follows:
 - When Fluent Restart files are read into CFD-Post, UDM/UDS variables will appear with names like "User Defined Memory 0"/"Scalar 0".
 - When Fluent Postprocessing files are read into CFD-Post, CFD-Post will show all UDM/UDS variables that were exported to a Fluent Postprocessing file.

- Fluent files do not contain the units for user-defined scalars, user-defined memory, or custom field functions, so these will be dimensionless in CFD-Post.

Velocity

- For Eulerian multiphase cases, the velocity-related variables for secondary phases are incorrect in CFD-Post compared to Fluent when using Fluent Restart files. You should use Fluent Postprocessing files, as outlined in [CDAT for CFD-Post and EnSight](#), to get the correct values.
- Plots of velocity vectors on wall boundaries do not match between CFD-Post and Fluent. Fluent always uses adjacent cell velocity for plotting vectors whereas CFD-Post uses node velocity (interpolated from cell/face values).
- Velocity magnitude values for Fluent in CFD-Post are not in good agreement with Fluent results for cases with multiple-frame-of-reference or sliding-mesh models.

For cases solved with relative velocity:

- The `Velocity in Stn Frame` plotted in CFD-Post is equivalent to `Velocity Magnitude` in Fluent. Similarly, other quantities dependent on `Velocity`, such as `Total Pressure` or `Total Temperature`, will have the suffix "in Stn Frame" (for example, `Total Temperature in Stn Frame`, corresponding to `Total Temperature`) for stationary frame variables in Fluent. Variables in the relative frame of motion will be without this suffix.
- There is no Fluent equivalent for the CFD-Post variable `Velocity` because this represents a relative velocity in the local reference frame of the domain (which is not available for postprocessing in Fluent).
- There is no CFD-Post equivalent for Fluent's `Relative Velocity`. In Fluent, `Relative Velocity` is always relative to a global frame of reference (which you can select in Fluent's **Reference Values** panel; if no reference frame is selected, an absolute velocity is used).
- When loading Fluent results, CFD-Post does not calculate global ranges by default because this would be too time-consuming (there is a warning to this effect when you load a Fluent case). However, when the variable is used for the first time (for example, when it is plotted), and as timesteps are loaded, the global range will be continually updated.

Volume Fraction

- If `Volume Fraction` is not available in the list of variables for multiphase Fluent cases, the phase forces reported by CFD-Post will be the same as the total force.
- For the cavitation model, there are differences in the volume fraction values between Fluent and CFD-Post.

Wall Adjacent Temperature

- Nodal values for `Wall Adjacent Temperature` are only available on faces if they are supplied by Fluent.

This might result in undefined plots on non-wall locations when using `Wall Adjacent Temperature`.

Exporting Wall Adjacent Temperature on non-wall locations from CFD-Post might also show "null" values where data does not exist.

Wall Heat Flux

- The values of Wall Heat Flux reported by CFD-Post for moving and deforming mesh cases will not match those for Fluent. This is because Fluent adds pressure work to get the energy balance.

Wall Heat Transfer Coefficient

- The Global variable range for a Wall Heat Transfer Coefficient is incorrectly reported as zero; use the Local variable range instead.

8.15.7.10. Limitations in CFD-Post using Fluent Files

Processing Fluent Data in CFD-Post

- Plot objects (other than Contour Plots) cannot display cell or face data directly; they can display only nodal averages. However, cell and face data will be used in quantitative reports on volumes, boundaries, planes, iso clips, and isosurfaces (averages, mass flows, integrals). Lines, polyline, and points will use only nodal averages for quantitative calculations.
- Global variable ranges shown in plots are nodal (averaged) ranges.
- CFD-Post does not account for surface tension forces.
- Certain real gas properties are not available for use in CFD-Post: gas constant, molecular viscosity, specific heat, and sound speed.

Case Comparison

- Case comparison is not available for face-based contours of Fluent results.

Contour Plots

- CFD-Post and Fluent might use different sources of data when generating contour plots. For details, see [Variable Location: Vertex and Face Options \(p. 265\)](#).
- The value ranges shown in a contour plot might differ from the value ranges reported by the function calculator.
- Function calculator results are based on either cell/face values or nodal values, in accordance with how the variables are stored.

Function Calculator

- The Function Calculator might give variable averages on slice planes, isosurfaces, and interiors that are different from those given by Fluent.

These differences might occur when the surface is cutting through a mesh face that joins two mesh elements. In this situation, CFD-Post might use the element-center data from a different element than Fluent uses.

Note that as both elements are equally valid choices, both calculations are correct.

- Computing the sum of any variable on any surface returns a value in CFD-Post that is the Fluent value divided by 2π .

Lines

- A line in CFD-Post of type **Sample** gives results that match with Fluent's 'line' if the environment variable `CFDPOST_BOUNDARY_DATA_FROM_ELEMENTS` is set to 1. However, a line in CFD-Post of type **Cut** gives results that do not match with Fluent's 'rake' because the former is infinite and the latter is clipped.

Streamlines

- Surface streamlines cannot be created on wall boundaries because wall velocities are zero.

It is recommended that you create the streamline based on the `Wall Shear` vector. Ensure that `Wall Shear` is in the file; if it is not, return to Fluent and export that variable.

Alternatively, you can use the near wall velocity for streamlines (and other plots) by setting `CFDPOST_BOUNDARY_DATA_FROM_ELEMENTS` before running CFD-Post.

- In the regions where the mesh is coarse and vector variable gradients are steep, streamlines in CFD-Post might hit walls earlier than similar streamlines in Fluent. This is due to a difference in computational methods, and can be avoided by refining the mesh.
- When creating Streamlines for Dual Cell heat exchanged Fluent cases, exclude Auxiliary Fluid domains from the Domains list because these domains can cause the streamlines to be terminated too early.

Turbo Workspace

- CFD-Post can initialize the turbo space only for domains that are enclosed with inlet, outlet, hub, and shroud regions. For more complex geometries, you must set up the problem such that the region of interest is isolated into a separate domain that can be initialized in CFD-Post.
- When choosing a report template for a Fluent turbo report, choose Release 12 templates (which do not have the word "Rotor" in the template name).
- Report templates that have "Rotor" in the name are from Release 11 and require variables that are not available from Fluent turbo files.
- For rotating machinery applications, identification of components and ordering, regions, rotation axis, number of passages, and interfaces cannot be done automatically; you must supply this information on the Turbo initialization panel. When generating turbo reports, select variables, instance transforms, and expressions will require manual updates; for details see [Procedures for Using Turbo Reports when Turbomachinery Data is Missing](#) (p. 80).
- Also see "Velocity" in [Quantitative Differences Between CFD-Post and Fluent](#) (p. 186).

Limitations in the Import and Display of Fluent Particle Tracks

- If particle track files from Fluent were written with `rpvar dpm/io/cfd-post/export-int64?` set to `#t` (default is `#f`), then, in order for CFD-Post to read the particle track files, you must have environment variable `CFXPOST_READ_64BIT_FLUENT_PARTICLE_IDS` set to 1.
- It is not possible to group or color transient particles by stream.
- The size of exported files and the intermediate history file is limited to 2 GB on architectures that have `sizeof(long)==4` (for example: win64, ntx86, and lnx86).
- When Fluent particle tracks cross periodic boundaries, there will be a gap between the point on one side of the periodic boundary and the point on the other side. This is most visible if instancing is enabled, but appears only in transient cases.
- CFD-Post displays particle tracks as segments, whereas Ansys Fluent displays particle tracks as points. This is particularly shown by transient cases when viewing tracks for a particular timestep; CFD-Post displays tracks as segments of the track from the previous timestep to the current timestep, while Ansys Fluent shows points at the current timestep. Because of this, the range of the color variable will include data that is not included by Ansys Fluent when displaying particle tracks.
- Also see [Exporting Steady-State Particle History Data in the Fluent User's Guide](#).

Polyflow Classic and FIDAP Limitations

- Some Polyflow Classic and FIDAP cases may have interior surfaces that are read into CFD-Post as boundaries. Unlike other boundaries, these "interior boundaries" cannot be used to create Polyline objects by intersecting them with a slice plane.
- CFX Results files generated by Polyflow Classic do not contain all the necessary information required for some automatic calculations in CFD-Post, including force and torque functions.

Limitations of Load Transfer to Mechanical

- In cases where forces on surfaces are transferred between Fluent and Mechanical, the forces displayed on the surface are read directly from the Fluent results file, while the forces on the corresponding Mechanical surface are averaged to the nodes. When viewed in CFD-Post, the values of the surface and Mechanical forces will not match.

8.15.8. Ansys Forte Files

CFD-Post can load Forte results files (`.ftind`) for postprocessing.

8.15.9. Ansys FENSAP-ICE Files

CFD-Post can load grid (`*.grid`) and solution (`*.soln`, `*.droplet`, `*.crystal`, `*.swimsol`) files from FENSAP-ICE. A [View Set-up File \(p. 195\)](#) (`*.fsp`) can also be used to specify a list of domains, grids, and solutions over a single time step or multiple iterations.

When opened through a FENSAP-ICE Workbench component, or from the FENSAP-ICE graphical environment, CFD-Post is launched through a [View Set-up File \(p. 195\)](#) (`*.fsp`):

- Upon launching a computation, the FENSAP-ICE project manager will set-up a `cfdpst.fsp` file in the run directory. This file contains references to the most common output files in this folder.
- Using the **View** action, with CFD-Post selected as post-processor, a `.cmd_post.fsp` file is created, and will contain the specific files selected for the view action.

8.15.9.1. Grid Files

A grid file can be loaded in CFD-Post by selecting a `*.grid` file, or via a [View Set-up File \(p. 195\)](#) (`*.fsp`). Both formats (Airflow grid and C3D Solid grid) are supported.

Note:

If the file name does not comply with the file extension, the file can also be loaded using the **All files (*)** option, and the type of file will be auto-detected.

All volumes and boundary zones from the grid file will be loaded. The FENSAP-ICE boundary condition numeric identifiers will show up as zone labels such as BC_1000, BC_2000, BC_2001.

8.15.9.2. Solution Files

The recognized solution file extensions are: `*.soln` (FENSAP Airflow), `*.droplet`, `*.crystal` (DROP3D droplet impingement). A solution file must be read alongside a grid file. For example, alongside a `*.soln` file, a `*.grid` file is needed.

A [View Set-up File \(p. 195\)](#) enables you to specify a pair of grid and solution files, of any name and disk location.

8.15.9.3. Solution Files - Icing

The recognized icing solution file extension is `*.swimsol`. The icing surface grid (`*.map.grid`) must be alongside the `*.swimsol` file.

A [View Set-up File \(p. 195\)](#) enables you to specify a pair of grid and solution files, of any name and disk location.

Note:

The icing solution is mapped on a surface grid.

8.15.9.4. View Set-up File

A view set-up file has the extension `.fsp`, and enables you to define a list of domains. For each domain, you can specify a single, or a list of, grid and solution files.

Domain definition syntax is as follows:

```
domain=DomainName
grid=GridFileList
soln=SolutionFileList
```

A .fsp file can contain multiple domains, with each having a different name. The grid and solution file list can be a single filename, a list of filenames separated by a comma, or a filename with a wildcard (for example, grid.*.disp, soln.0000??). A single grid can be used with multiple solutions, or multiple grids can be used with multiple solutions, however, the two lists must be of the same length, the grid/solution of each index will be loaded as a pair.

Note:

All filenames are relative to the view set-up file directory.

8.15.9.4.1. Examples

Single grid with one solution:

```
domain=Air
grid=../naca0012
soln=soln
```

Two domains, each with a grid and solution pair:

```
domain=row01
grid=../grid.row01
soln=soln.row01
domain=row02
grid=../grid.row02
soln=soln.row02
```

Multiple domains, grid only:

```
domain=row01
grid=TurboMeshing/row1/grid
domain=row02
grid=TurboMeshing/row2/grid
domain=row03
grid=TurboMeshing/row3/grid
```

Ice solution on map.grid and ice.grid:

```
domain=map-swimsol
grid=map.grid
soln=swimsol
domain=ice-swimsol
grid=ice.grid
soln=swimsol
```

Multiple numbered solutions:

```
domain=Air
grid=../naca0012
soln=soln.000005, soln.000010, soln.000015, soln.000020
```

Multiple grids and solution


```
domain=Remeshing
grid=../naca0012,grid.disp.000002,grid.disp.000003
soln=soln.000001,soln.000002,soln.000003
```

Note:

soln.000001 will be loaded with ../naca0012, soln.000002 with grid.disp.000002, and so on.

8.15.9.5. Limitations

- Grids with element groups defining materials or subdomains are loaded as a single domain. Material identifiers cannot be viewed with CFD-Post.
- Facet-based solution files (hflux.dat and surface.dat) are not supported. The equivalent nodal solution is available through the airflow solution file (*.soln)
- Boundary profiles (timebc.dat) are not supported. They can be viewed with the Viewmerical tool, included with FENSAP-ICE. For more details, see [Post-Processing in the Ansys FENSAP-ICE User Manual](#).

8.15.10. CFX-4 Dump Files

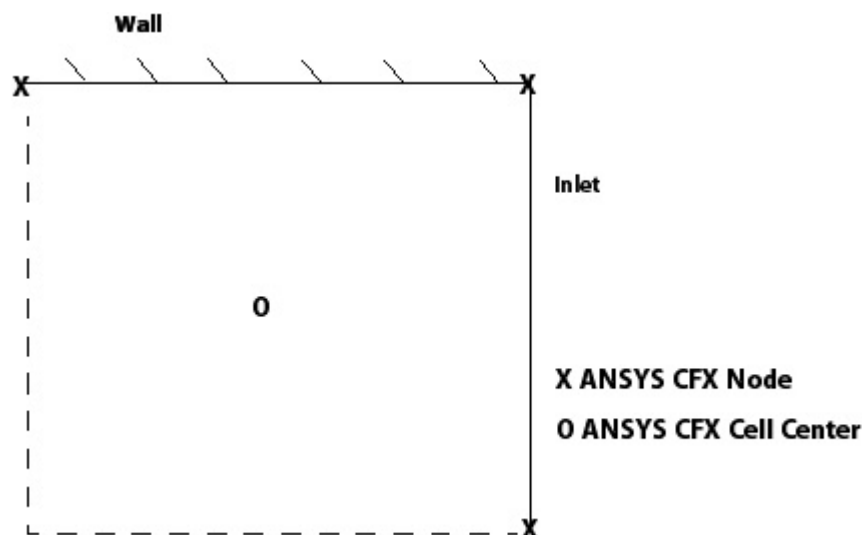
CFD-Post can load dump files (*.d*mp*) created by CFX-4. When you load the results, you may be prompted to provide the solution units that were used in the simulation. For details, see [CFD-Post Solution Units](#) (p. 207).

8.15.10.1. Limitation with CFX-4 Files

There is an important limitation with CFX-4 results files that should be noted: the CFX-4 Solver does not output minimum/maximum ranges for each of the calculated variables. These ranges are calculated when the results file is loaded by CFD-Post. Calculating the range for a very large problem would, however, require prohibitively large amounts of CPU time. As a result, range values are calculated for the loaded time step only. This means that values that appear as global range, are in fact ranges that exist for that time step only.

8.15.10.2. Interpolation of Results

The CFX-4 Solver uses a cell-based solution method, whereas CFX-Solver uses a node-based solution method. Possible problems can be encountered at the intersection of patches, such as in the following diagram:



When interpolating from cell-centered to node-centered data, the data at a given node is affected by all surrounding cells. In order to get the correct behavior at boundary patches, a priority number is assigned to each patch by CFD-Post. This means that, for example in the above diagram, if the wall has a higher priority number than the inlet, the value of the node is interpolated from the wall value of the CFX solution. When considering a situation in 3D, the priority of all faces is read and interpolation occurs from the face(s) with the highest priority. CFD-Post uses the same default values for every problem, so there are cases in which accuracy can be compromised. These errors can be minimized by refining the grid density in the region around problem areas.

Quantitative calculations can suffer a loss of accuracy due to the limitation described above. The results of mass flow calculations should, therefore, be assumed to be approximations for the purposes of quantitative analysis.

8.15.11. CFX-TASCflow Results Files

CFD-Post can import CFX-TASCflow results files for postprocessing. When you load the results, you may be prompted to provide the solution units that were used in the simulation. For details, see [CFD-Post Solution Units](#) (p. 207).

If TBPOST_COMP_X parameters (where X is the component number) are defined in the GCI file, either on their own or within the TBPOST_COMP_LIST macro, they are used to obtain the list of turbo components to load. Each defined component is treated as a separate domain inside CFD-Post, allowing for their individual turbo initialization.

Note:

If this list exists but is incomplete, only the defined components are loaded. If you cannot load a turbo file, it may be due to an incompatibility in the component definition. As a workaround, remove TBPOST related parameter and macro definitions from the GCI file.

8.15.11.1. Limitations with CFX-TASCflow Files

- When loading rso or grd files, bcf and prn files are required.

- bcf files must be complete (must contain all domain and boundary condition definitions).
- When using the Turbo Post functionality, separate region names are required for the following 2D location types:
 - Hub
 - Shroud
 - Blade
 - Inlet
 - Outlet
 - Periodic1
 - Periodic2

If these regions have not been specified separately (that is, the hub and blade are both part of one region), you will either need to recreate them in the CFX-TASCflow preprocessor or specify the turbo regions from line locators. For details, see [Initialize All Components \(p. 383\)](#).

- Mass flow and torque are not written to rso files by CFX-TASCflow. These values are approximated in CFD-Post and may not be suitable for use in a formal quantitative analysis.

8.15.11.2. Variable Translation

By default, CFD-Post does not modify the variable names in the rso file. If you want to use all of the embedded CFD-Post macros and calculation options, you need to convert variable names to CFX variable names. You can convert the variable names to CFX variable names upon reading the file by selecting **Edit > Options**, then, in the **Options** dialog box, selecting **CFD-Post > Files > Variables > Non CFX Files > Translate variable names to CFX-Solver style names**.

Translation is carried out according to the following:

CFX-TASCflow	Translated to CFX Variable
T	Temperature
TKE	Turbulent Kinetic Energy
EPSILON	Turbulence Eddy Dissipation
VISC_TURBULENT	Eddy Viscosity
VISC_MOLECULAR	Molecular Viscosity
CONDUCTIVITY	Thermal Conductivity
SPECIFIC_HEAT_P	Specific Heat Capacity at Constant Pressure
SPECIFIC_HEAT_V	Specific Heat Capacity at Constant Volume
PTOTAL	Total Pressure
PTOTAL_REL	Total Pressure in Rel Frame
PTOTAL_ABS	Total Pressure in Stn Frame
POFF	Pressure Offset

CFX-TASCflow	Translated to CFX Variable
P_CORRECTED	Pressure Corrected
TTOTAL	Total Temperature
TTOTAL_REL	Total Temperature in Rel Frame
TTOTAL_ABS	Total Temperature in Stn Frame
TOFF	Temperature Offset
T_CORRECTED	Temperature Corrected
TAU_WALL	Wall Shear
YPLUS	Solver Yplus
Q_WALL	Wall Heat Flux
P	Pressure
PRESSURE_STATIC	Static Pressure
PRESSURE_REL	Relative Pressure
MACH	Mach Number
MACH_ABS	Mach Number in Stn Frame
MACH_REL	Mach Number in Rel Frame
HTOTAL	Total Enthalpy
HTOTAL_REL	Total Enthalpy in Rel Frame
HTOTAL_ABS	Total Enthalpy in Stn Frame
ENTHALPY	Static Enthalpy
ENTROPY	Static Entropy
FE_VOLUME	FE Volume
CONTROL_VOLUME	Volume of Finite Volumes
DIST_TURB_WALL	Wall Distance

Chapter 9: CFD-Post Edit Menu and Options (Preferences)

Undo and **Redo** commands are available in the **Edit** menu. Additionally, there are a variety of options that can be set to customize the software.

This chapter describes:

- [Undo and Redo Commands \(p. 201\)](#)
- [Options Command \(p. 202\)](#)

9.1. Undo and Redo Commands

The undo and redo capability is limited by the amount of available memory.

In stand-alone mode, the undo stack is cleared whenever a **New**, **Open**, or **Close** action occurs. Similarly, when using CFX-Pre/CFD-Post from within Ansys Workbench, the undo stack is cleared in CFX-Pre/CFD-Post after the application receives commands from Ansys Workbench.

Issue the **Undo** command by doing any of the following:

- Select **Edit > Undo**.
- Click *Undo*  on the toolbar.
- Press **Ctrl + Z**

Note:

- You can repeatedly issue the **Undo** command.
 - Some viewer manipulations cannot be reverted using the **Undo** command.
 - Some commands that you issue have multiple components. For example, when you create some objects the software creates the object and sets the visibility of the object on (in two separate operations). Thus, when you perform an undo operation in such a situation, you are setting the visibility of the object off; you must choose undo a second time to "uncreate" the object.
 - **Undo** cannot be used when recording session files.
-

The redo feature is used to do an action that you have just undone using the **Undo** command. Issue the **Redo** command by doing any of the following:

- Select **Edit > Redo**.
- Click *Redo*  on the toolbar.
- Press **Ctrl + Y**

9.2. Options Command

The **Options** dialog box enables you to set various general preferences. Settings are retained per user.

1. Select **Edit > Options**.

The **Options** dialog box appears.

2. Set options as required. For descriptions of the available options, see:

- [CFD-Post Options \(p. 203\)](#)
- [Common Options \(p. 209\)](#)

If desired, you can use the **CFX Defaults** or the **Workbench Defaults** buttons at the bottom of the dialog box to quickly set CFD-Post to have the standard appearance and operation of CFX or Workbench respectively. The CFD-Post settings that are affected by these buttons are:

- **CFD-Post > Viewer > Background**
- **CFD-Post > Viewer > ANSYS Logo/Text Color/Edge Color**
- **Common > Viewer Setup > Mouse Mapping**

3. Click **OK**.

Your changes are generally implemented immediately. However, when changing the following user preferences, it will be necessary to restart CFD-Post for the setting to take effect:

- **CFD-Post > General > Enable Beta Features**
- **CFD-Post > Viewer > Object Highlighting**
- **CFD-Post > Viewer > Hide Ansys Logo**
- **CFD-Post > Viewer > Stereo**
- **Common > Temporary Directory**
- **Common > Viewer Setup > Double Buffering**

9.2.1. CFD-Post Options

When the **Options** dialog box appears, CFD-Post options can be configured under **CFD-Post**.

Interpolation Tolerance

The **Interpolation Tolerance** sets the amount of the area outside the domain that will be treated as a part of the domain when interpolating variables. For example, a point that is within this tolerance distance will be given a value that is interpolated from the nearest domain boundary face.

By default the tolerance "layer" is 0.2% of the domain. You can set the value to 0 to turn the Interpolation Tolerance off.

Note that this value should be set to a value less than half the size of the smallest openings or features of the domain geometry. This prevents a point from being detected in two overlapping Interpolation Tolerance regions.

If the **Interpolation Tolerance** setting is too high, there could be undefined nodes and malformed elements in the virtual layer, especially around sharply-contoured regions of the domain surface.

Changes to the interpolation tolerance can affect how data is interpolated onto objects such as Sample Planes, Polylines, User surfaces, Surfaces of Rotation, and so on. Each of these objects, depending on how close the defined points are to the domain, may have slightly changed behavior, depending on how close to the domain the points are defined.

9.2.1.1. General

Angular Shift for Rotating Locations

This option takes effect when a new case is loaded, and affects rotating domains and Monitor Surfaces defined with a rotating coordinate frame. The option can be set to:

Automatic

This is the default option. CFD-Post displays the rotated position of the rotating locations considering the currently-loaded timestep. CFD-Post uses the rotated position for a domain (and associated locations such as domain boundaries) only if the domain is connected with a transient rotor-stator interface, either directly or through one or more connected domains. The rotated position for a Monitor Surface is always used.

Always rotate

CFD-Post displays the rotated position of the rotating locations applicable to the currently-loaded timestep. Domains and Monitor Surfaces are always displayed in their rotated position regardless of any interfaces in the setup.

Never rotate

CFD-Post always uses the initial position of the rotating domains and Monitor Surfaces.

Because the rotating domains may be in a rotated position when they are loaded into CFD-Post for postprocessing, force calculations made on the rotating domains can have different results depending on the setting of the option above.

9.2.1.1.1. Beta Options

- **Enable Beta Features**

Some Beta features are hidden in the user interface. You can select this option to "unhide" those features. When selected, such features are identified by "(Beta)" in the user interface. Note that Beta features are unofficial and not well tested.

9.2.1.1.2. Load Options

Changes made in this section will take effect the **next** time you load a file.

- **Warn prior to loading the same results file more than once** controls whether you are warned when you attempt to load a file that is already loaded. The warning appears when both the filename and the file contents are the same as a currently loaded file.
- Select **Multi-Domain > Show domain selector before load** to enable you to choose which domains to load when more than one domain exists in the results file. If this option is turned off, then all domains will be loaded next time you load a results file.

9.2.1.1.3. Advanced

- Under **Keyframe Animation > Command Timeout**, specify the minimum number of milliseconds that must pass after clicking to change the number of keyframes in the **Animation** dialog box (in the **# of Frames** field with the **Keyframe Animation** option selected) before a subsequent mouse click can be registered. This time enables CFD-Post to finish generating the currently specified number of keyframes without the specified number changing during the generation process. The allowed range for **Command Timeout** is 400 to 1000.
- If **Enable GPU Shader Rendering** is selected (default), then:
 - The graphics card GPU (if eligible, and with sufficiently recent drivers) is used to compute objects (for example, Turbo Surfaces) that display Transient Blade Row (TBR) results.
 - GPU Accelerated Animation becomes available for objects except those that are based on a domain for which more than one data instance is set in the domain details view, on the **Data Instancing** tab. In other words, data instancing as set in the domain details view is incompatible with GPU Accelerated Animation.
 - For objects that display Transient Blade Row (TBR) results, graphical instancing, as set in the domain details view on the **Instancing** tab, acts like data instancing would without GPU acceleration, in the sense that variables are not simply copied to new instances but are instead calculated for each instance using a Fourier series.
 - In the details view for boundaries and other surfaces, on the **Color** tab, the **Contour** setting "Banded" is functional, enabling you to color a surface with contour bands without having to create a contour object. For details, see [Contour \(p. 54\)](#).

Note:

Due to limitations of the graphics card and/or graphics driver, the graphics card might not be able to render an image using all of the sine-cosine pairs of Fourier

coefficients (per vertex) available to be included in the graphics data. For multi-disturbance (see [Multiple Disturbances in the CFX-Solver Modeling Guide](#)) and mesh deformation (see [Mesh Deformation in the CFX-Solver Modeling Guide](#)) cases in particular, you might see the following error message:

CreateShader ERROR: The number of vertex attributes used may have exceeded the hardware limit. Please consult the documentation for ways to resolve the issue.

If this error occurs, try setting the following environment variable to a value less than 7, for example 3:

`CFDPOST_GPU_ANIMATION_TBR_MAXIMUM_FOURIER_COEFFICIENT_PAIRS`

This reduces the maximum number of Fourier coefficient pairs in the Fourier series sent to the GPU. For example, a setting of 3 would use the first 6 Fourier coefficients.

By restricting the number of Fourier coefficient pairs, the graphical fidelity of plotted variables may be diminished. Consequently, you should set the environment variable temporarily, only for cases that require it. For most cases not involving multi-disturbance or mesh deformation, you can leave the environment variable undefined; doing so enables graphical rendering using the full set of Fourier coefficient pairs.

9.2.1.2. Files

Changes made in this section will take effect the **next** time you load a file.

9.2.1.2.1. CFX

- Under **Mass Flow Expression Method Used for Physical Surfaces**, select an option to control how mass flow data is calculated, for example by the `massFlow`, `massFlowAve`, and `massFlowAveAbs` functions:
 - **Use surface shape approximated by the nearest control volume faces (not recommended for cases with GGI interfaces)**

This method can provide more accurate massFlow rates. However, it may provide inaccurate results if the surface over which the mass flow is evaluated crosses a GGI interface.

- **Use surface mass flows computed from interpolated nodal values (recommended for general cases)**

This method is recommended because the results do not suffer if the surface over which the mass flow is evaluated crosses a GGI interface.

- When **Transient > Load missing variables from nearest FULL time step** is cleared, it makes all variables that are not written to the partial results file undefined for the current timestep. When selected, CFD-Post loads the missing variables from the nearest full results file. This option is used when partial transient results files do not contain all of the variables calculated by the

CFX-Solver. By default, these variables will be undefined (but still visible in the variables list) for the current timestep.

Important:

Take care when using this option because values that are plotted may not apply to the current timestep.

- Select **Regions > Don't load mesh regions** if you do not want to have region definitions loaded when you load a file that contains them.
- Select **Turbo > Don't prompt to auto-load reports** to prevent CFD-Post from automatically asking you if you want to load a report upon loading results files.

9.2.1.2.2. FLUENT

- **Load interior face zones** controls whether or not interior face zones are displayed.

Fluent cases contain cell zones and face zones. (Cell zones are similar to element sets in CFD-Post, and face zones are similar to face sets.) All cell zones are read into CFD-Post and are listed as domains. Of the face zones, by default only those that bound a cell-zone/domain are shown in CFD-Post^[1]. However, Fluent meshes can also contain 'interior' face zones that are useful for postprocessing. Interior face zones are inside a domain and do not form a boundary of the domain. To see interior face zones, enable **Load interior face zones**.

Note:

The names of 'interior' zones in CFD-Post are kept same as that in Fluent, except that characters that are not allowed in CFD-Post (such as '-', ':' and so on) are replaced by space characters.

- **Show warning for incompatible variables in old files** controls whether a message box will appear if the Fluent .dat file you are loading contains incompatible variables. These variables, when written by versions of Fluent earlier than 18.0 may show differences when displayed in CFD-Post when compared to Fluent. For details, see [Exporting to Ansys CFD-Post](#).

9.2.1.2.3. CGNS

- **Define vector variables** controls whether or not vector variables are created if Cartesian vector components are present in a CGNS file. It should not be necessary to turn this option off unless this is required to allow compatibility with state or session files from previous releases.

^[1] CFD-Post will never list:

- Default interior zones.
- Walls created during creation of non-conformal interfaces.
- Sliding interface zones.
- Any other zones that cannot be displayed from the **Mesh Display** panel in Fluent.

9.2.1.2.4. Variables

- **Non CFX Files > Translate variable names to CFX-Solver style names** converts variable names into CFX variable names for results files that are both:
 - Not from the CFX-Solver, and
 - Not written using the Common Fluids Format (CFF).

For example, the variable P in a CFX-TASCflow file will be converted to Pressure.

Note:

Variables from CFF files are not affected by the **Non CFX Files > Translate variable names to CFX-Solver style names** setting. Instead, they are affected by the **Common Fluids Format > Name Convention** setting (described below).

Important:

- By default, CFD-Post will not modify the variable names in the .rso file. If you want to use all of the embedded CFD-Post macros and calculation options, you will need to convert variable names to CFX types.
 - In order to use the **Turbo Charts** feature with Fluent files, you must have **Translate variable names to CFX-Solver style names** selected.
-

The complete list of translated variables is given in [Variable Translation \(p. 199\)](#).

- Clear **Non CFX Files > Pre-calculate global variable ranges** to turn off the calculation of all variable ranges.
- Set **Common Fluids Format > Name Convention** to one of the following options:
 - CFD-Post

Variables read from a CFF file are, where applicable, renamed with standard CFX variable names, which are used by various features in CFD-Post, such as functions, reports, and macros. If this option is not used, such features might not work.

- Native

Variables read from a CFF file are not renamed.

- Common Fluids Format

Variables read from a CFF file are, where applicable, renamed with standard Ansys Fluids variable names, which are used by various Ansys Fluids products. Not all variables supplied in the CFF file are renamed; some variables retain their supplied name.

9.2.1.2.5. CFD-Post Solution Units

CFD-Post has a **Solution Units** option that is available from the **Options** tab.

The solution units assumed, which are read when the file was loaded, are displayed on the right. When files that do not store solution units (such as CFX-4 dump files, CFX-TASC files, Fluent files, or Ansys results files) are loaded, you will be prompted to specify the solution units. You can enable the **Don't prompt for Solution Units before loading results** toggle to suppress this prompt, in which case the default units of kilograms, meters, seconds, Kelvin, and radians will be used.

The units shown on this dialog box are not necessarily those used by CFD-Post, but are the solution units used in the currently loaded file. The units used by CFD-Post are set elsewhere; for details, see [Setting the Display Units \(p. 210\)](#). CFD-Post needs to know the solution units used in the file so that it can convert them to the units specified. When CFX files are loaded into CFD-Post, the solution units that were used by the CFX-Solver are automatically read from the file. For this reason, **Don't prompt for Solution Units before loading results** is ignored when loading CFX files and selected by default for other file types.

When postprocessing a results file in CFD-Post, the units used are not necessarily those used in the results file. CFD-Post will convert to your preferred units.

Note:

In CFD-Post, the temperature solution units must be an absolute scale (for example, Kelvin [K] or Rankine [R]); you cannot use Celsius and Fahrenheit. Temperature quantities elsewhere in Ansys CFX can be set in Celsius and Fahrenheit.

9.2.1.3. Turbo

These settings are related to turbomachinery simulation results loaded into CFD-Post.

9.2.1.4. Viewer

To configure the viewer, right-click the viewer and select **Viewer Options**.

9.2.1.4.1. Object Highlighting

Controls how an object that is generated after a change to the setting of this option is highlighted in the viewer. Such highlighting occurs when in picking mode, when selecting a region in a list, or when selecting items in the tree view.

Under **Type**, select one of the following:

- **Surface Mesh**: Displays the surface mesh for selected regions using lines.
- **Wireframe**: Traces objects that contain surfaces with green lines.
- **Bounding Box**: Highlights the selected objects with a green box.

Note:

When you load a state file, the highlighting is dictated by the setting that is stored in the case, rather than by the current preferences setting.

9.2.1.4.2. Background

Set **Mode** to Color or Image.

9.2.1.4.2.1. Color

Use **Color Type** to set either a solid color or a gradient of colors; use **Color** to set the color (and **Color 2** for gradients).

9.2.1.4.2.2. Image

Select one of a list of predefined images or a custom image.

If selecting a custom image, choose an image file and a type of mapping. Image types that are supported include *.jpg, *.png, and *.ppm. Mapping options are Flat and Spherical. Flat maps are stationary while spherical maps surround the virtual environment and rotate with the objects in the viewer.

Custom images have some restrictions: all background images and textures sent to the viewer must be square and must have dimensions that are powers of 2 (for example, 512 x 512 or 1024 x 1024).

If the dimensions of your background image is not a power of 2, the viewer sizes the image to be a power of 2 by doing bicubic resampling.

To make the background image square, transparent pixels are added to the smaller dimension to make it the same as the larger dimension. The transparent pixels enable you to see the regular viewer background, which gives you control over what fill color your background has.

9.2.1.4.3. Ansys Logo

Contains options for displaying or hiding the Ansys logo in the **3D Viewer**.

9.2.1.4.4. Text/Edge Color

Select a color by clicking in the box, or clicking the *Ellipsis*  icon.

9.2.1.4.5. Axis/Ruler Visibility

Select or clear **Axis Visibility** or **Ruler Visibility** to show or hide the axis indicator or ruler in the viewer.

9.2.1.4.6. Stereo

See [Stereo Viewer](#) (p. 130).

9.2.2. Common Options

Auto Save

Select the time between automatic saves.

To turn off automatic saves, set **Auto Save** to Never.

Note:

This option affects more than one CFX product.

Temporary directory

To set a temporary directory, click *Browse*  to find a convenient directory where the autosave feature will save state files.

9.2.2.1. Appearance

The appearance of the user interface can be controlled from the **Appearance** options. The default user interface style will be set to that of your machine. For example, on Windows, the user interface has a Windows look to it.

1. Under **GUI Style**, select the user interface style to use.
2. For **Font** and **Formatted Font**, specify the fonts to use in the application.

Note:

It is important not to set the font size too high (over 24 pt. is not recommended) or the dialog boxes might become difficult to read. Setting the font size too small might cause some portions of the text to not be visible on monitors set at low resolutions. It is also important not to set the font to a family such as Webdings, Wingdings, Symbols, or similar type faces, or the dialog boxes become illegible.

Formatted Font has no function in CFD-Post.

9.2.2.2. Threading

Select **Enable Threading** and set **Hardware Capacity Fraction** to a value greater than 0 to make use of multiple CPU cores to improve the general performance of applications that share this common setting, at the possible expense of CPU availability for other applications. The maximum allowed value of 1 (default) uses all available CPU cores.

Note:

The use of multithreading in CFD-Post can alter face IDs and node numbering for some surfaces, such as isosurfaces and turbo surfaces, affecting CSV data exported from those surfaces. However, the resulting nodal data remains consistent. If face ID and node numbering consistency is important then you should use only one thread.

9.2.2.3. Setting the Display Units

These settings control the *preferred units* of the CFX application. Preferred units are:

- The units of the data that CFD-Post uses when information is displayed.

For example, if your preferred units are SI and you load a results file that contains data in British Technical units, the values you see in CFD-Post will be in SI.

- The default units when you enter information.

Note that preferred units are not necessarily the same as the units stored in results files.

To set the preferred units:

1. Under **System**, select the unit system to use. Unit systems are sets of quantity types for mass, length, time, and so on.

The options under **System** include SI, CGS, English Engineering, British Technical, US Customary, US Engineering, or Custom. Only Custom enables you to redefine a quantity type (for example, to use inches for the dimensions in a file that otherwise used SI units).

The most common quantity types appear in the main **Options** dialog box; to see *all* quantity types, click **More Units**.

2. Select or clear **Always convert units to Preferred Units**.

If **Always convert units to Preferred Units** is selected, the units of entered quantities are immediately converted to those set in this dialog box.

For example, if you have set **Velocity** to $[m \ s^{-1}]$ in this dialog box to make that the preferred velocity unit, and elsewhere you enter 20 $[mile \ hr^{-1}]$ for a velocity quantity, the entered value is immediately converted and displayed as 8.94078 $[m \ s^{-1}]$.

The two sets of units are:

- The units presented on this dialog box, which control the default units presented in the user interface, as well as the units used for mesh transformation.
- The solution units; for details, see [CFD-Post Solution Units \(p. 207\)](#).

9.2.2.4. Viewer Setup

1. Select **Double Buffering** to use two color buffers for improved visualization.

Double Buffering is a feature supported by most OpenGL implementations. It provides two complete color buffers that swap between each other to animate graphics smoothly. If your implementation of OpenGL does not support double buffering, you can clear this check box.

2. Select or clear **Unlimited Zoom**.

By default, zoom is restricted to prevent graphics problems related to depth sorting. Selecting **Unlimited Zoom** allows an unrestricted zoom.

3. Select or clear **Use GPU Rendering for Printing**.

If your graphics hardware is compatible with GPU rendering, selecting this preference is strongly recommended.

When selected, this preference causes saved pictures in CFD-Post to use GPU rendering and saved pictures in CFX-Pre and TurboGrid to (by controlling the default value of a setting in the **Save Picture** dialog box) use GPU rendering by default.

When you save a picture with GPU rendering, your graphics hardware renders an image that closely matches what is shown in the viewer.

If GPU rendering is not set or cannot be used (for example, if compatible GPU hardware is not found), software rendering is used instead. Software rendering is relatively slow and does not always render as shown in the viewer. One benefit of software rendering is that it has no graphics hardware requirements.

Note:

Batch mode printing in CFX-Pre and TurboGrid does not support GPU rendering.

9.2.2.4.1. Mouse Mapping

The mouse-mapping options enable you to assign viewer actions to mouse clicks and keyboard/mouse combinations. These options are available when running in stand-alone mode. To adjust or view the mouse mapping options, select **Edit > Options**, then **Common > Viewer Setup > Mouse Mapping**. For details, see [Mouse Button Mapping \(p. 124\)](#).

Chapter 10: CFD-Post Monitor Menu

The Monitor menu enables you to view solution data while the solver is executing. To use this feature, you must have predefined a User Location in CFX-Pre and set up CFX-Pre to output variable data onto a Monitor Surface. For more information, see [Monitor Surfaces in the CFX-Pre User's Guide](#).

This chapter describes:

- [10.1. Monitor Run in Progress](#)
- [10.2. Start Auto Update](#)
- [10.3. Stop Auto Update](#)
- [10.4. Update Once](#)

10.1. Monitor Run in Progress

To enable monitoring in CFD-Post, you must use the **Monitor Run in Progress** option and select the run directory you want to monitor:

Note:

Monitoring of multi-configuration or operating point runs is not supported. It is only possible to monitor run directories of individual configuration runs or operating point jobs.

1. Select **Monitor** > **Monitor Run in Progress**.

The **Select a Run Directory (.dir)** dialog box is displayed.

2. Browse to the directory containing the run.
3. Select the run directory.
4. Click **Choose**.

You can now monitor the run in CFD-Post on your predefined Monitor Surface using the following options.

10.2. Start Auto Update

The **Start Auto Update** option continually reads the most recent solution data from the current run in progress. You can see the solution update over time by creating a post-processing object (such as a contour) on your predefined monitor surface.

After selecting **Start Auto Update**, all user actions related to object creation and editing are disabled. When the solver run is finished, the last iteration is read into CFD-Post, and all user actions are made available to you.

10.3. Stop Auto Update

The **Stop Auto Update** option stops CFD-Post from reading further solution data generated by the CFX-Solver and the last read iteration remains loaded. This enables the previously disabled user actions in CFD-Post related to object creation and editing.

10.4. Update Once

The **Update Once** option reads solution data up to the last solved iteration, with the most recent being displayed in CFD-Post. Afterwards, the CFD-Post user interface is enabled so you can perform regular actions related to object creation and editing.

Chapter 11: CFD-Post Session Menu

Session files contain a record of the commands issued during a CFD-Post session. The actions that cause commands to be written to a session file include:

- Viewer manipulation performed using the commands available by right-clicking in the viewer window.
- All actions available from the **File** and **Edit** menus.
- Creation of expressions.
- Creation of new objects and changes to an object committed by clicking **OK** or **Apply** on any of the panels available from the **Tools** and **Insert** menus/toolbars.
- Commands issued in the **Tools > Command Editor** dialog box.

This chapter describes:

[11.1. New Session Command](#)

[11.2. Start Recording and Stop Recording Commands](#)

[11.3. Play Session Command](#)

11.1. New Session Command

When a session file is not currently being recorded, you can select **Session > New Session**. This opens the **Set Session File** dialog box where you can enter a filename for your session file. Once you have saved the file, it becomes the current session file. Commands are not written to the file until you select **Session > Start Recording**.

1. Browse to the directory in which you want to create the session file, and then enter a name for the file ending with a **.cse** (CFD-Post) extension.
2. Click **Save** to create the file.

This will not start recording to the session file. To start recording, you must select **Session > Start Recording**.

If you create more than one session file during a CFD-Post session, the most recently created file is the current session file by default. You can set a different file to be the current session file by selecting an existing file from the **New Session > Set Session File** window and then clicking **Save**. Because the file exists, a warning dialog box appears:

- If you select **Overwrite**, the existing session file is deleted and a new file is created in its place.
- If you select **Append**, commands will be added to the end of the existing session file when recording begins.

11.2. Start Recording and Stop Recording Commands

The **Start Recording** action writes into the current session file the CCL commands you issue. A session file must first be set before you can start recording (see [New Session Command \(p. 215\)](#)). **Stop Recording** terminates writing of CCL commands to the current session file. You can start and stop recording to a session file as many times as necessary.

Important:

- A session file cannot be played if it contains an **Undo** command. To run a session file that contains an **Undo** command, first edit the session file to remove the command.
 - Some Case Comparison difference plots and objects will only be updated at the end of the session file playback. To force them to be updated during the session playback (for example, because the session file saves images or figures that include these plots), disable and then re-enable Case Comparison when recording the session file.
-

11.3. Play Session Command

Selecting **Session > Play Session** opens the **Play Session File** dialog box in which you can select the session file to play. The commands listed in the selected session file are then executed.

Important:

Existing objects with the same name as objects defined in the session file are replaced by those in the session file (for example, if **P lane 1** exists in this CFD-Post session file, playing the session file will overwrite any existing object with the name **P lane 1**).

To play a session file:

1. From the menu bar, select **Session > Play Session**.
 2. In the **Play Session File** dialog box, browse to the directory containing the session file and select the file you want to play.
 3. Click **Open** to play the session file. The commands listed in the selected session file are executed. Existing objects with the same name as objects defined in the session file are replaced by those in the session file.
-

Note:

You can play session files in stand-alone CFD-Post, but not in CFD-Post in Ansys Workbench.

Note:

While recording a session file, any preference changes made using the **Options** dialog box (**Edit > Options**) are recorded. However, after playing back the session file, the preference

changes generally affect the current session only and are not reflected in the **Options** dialog box.

Chapter 12: CFD-Post Insert Menu

The **Insert** menu in CFD-Post is used to create new objects (such as locators, tables, charts, and so on), variables, and expressions.

A *locator* is a place or object that another object uses to plot or calculate values. For example, if you were to select a plane from which to start a streamline, the plane would be a locator.

Interpolation in CFD-Post:

For both plots and quantitative evaluation, iso clip and user surface locators interpolate values using general (tri-linear) interpolation from mesh nodes to surface nodes. Planes and isosurfaces use more accurate "edge interpolation" for plots and, in Fluent cases, element-to-face interpolation for quantitative evaluation (such as area averages). Similarly, boundaries directly map to mesh nodes or faces. Consequently, quantitative operations on a user surface or on an iso clip that is based on a mesh density location (slice plane, isosurface, or boundary), are not going to evaluate to precisely the same number as the underlying mesh density location.

Locator Object Limitation:

For transient blade row cases, transient statistics for boundary-only variables (such as Force, Mass Flow, Heat Flux) are not available on the following postprocessing locators: points, lines, cut-planes, isosurfaces, and iso clips.

This chapter describes:

- 12.1. Location Submenu
- 12.2. Vector Command
- 12.3. Contour Command
- 12.4. Streamline Command
- 12.5. Particle Track Command
- 12.6. Volume Rendering Command
- 12.7. Text Command
- 12.8. Coordinate Frame Command
- 12.9. Legend Command
- 12.10. Instance Transform Command
- 12.11. Clip Plane Command
- 12.12. Color Map Command
- 12.13. Variable Command
- 12.14. Expression Command

[12.15. Table Command](#)

[12.16. Chart Command](#)

[12.17. Comment Command](#)

[12.18. Figure Command](#)

12.1. Location Submenu

When you select any of the objects from the **Insert > Location** submenu, an **Insert <Object>** dialog box appears in which you can either accept the default name for the new object or enter a new one. CFD-Post will not enable you to create objects that have duplicate names.

Click **OK** on the dialog box to open the relevant details view in the **Outline** workspace. A new object will be created in the database when you click **Apply** in the details view of the location object.

Tip:

You can also access locator objects from the *Location* icon  **Location** on the toolbar.

The following topics are discussed:

[12.1.1. Point Command](#)

[12.1.2. Point Cloud Command](#)

[12.1.3. Line Command](#)

[12.1.4. Plane Command](#)

[12.1.5. Volume Command](#)

[12.1.6. Isosurface Command](#)

[12.1.7. Iso Clip Command](#)

[12.1.8. Vortex Core Region](#)

[12.1.9. Surface of Revolution Command](#)

[12.1.10. Polyline Command](#)

[12.1.11. User Surface Command](#)

[12.1.12. Surface Group Command](#)

[12.1.13. Turbo Surface Command](#)

[12.1.14. Turbo Line Command](#)

12.1.1. Point Command

A *point* is an object in 3D space that has a set of coordinates. You can use a point to locate the position of a variable minimum or maximum or as an object with which other objects can interact.

The following characteristics of points will be discussed:

- [Point: Geometry Tab \(p. 221\)](#)
- [Point: Color Tab \(p. 223\)](#)

- [Point: Symbol Tab \(p. 223\)](#)
- [Point: Render Tab \(p. 223\)](#)
- [Point: View Tab \(p. 223\)](#)

Note:

There are several ways to insert a point:

- From the menu bar, select **Insert > Location > Point**.
 - From the toolbar, select **Location > Point**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the **3D Viewer**.
-

12.1.1.1. Point: Geometry Tab

12.1.1.1.1. Domains

The **Domains** setting selects the domains in which the point will exist.

Note:

For a case with immersed solids, the setting **All Domains** refers to all domains *except* the immersed solids. To display all of the domains in a case that contains immersed

solids, click the *Location Editor*  icon and hold down the **Ctrl** key while selecting **All Domains** and **All Immersed Solids**.

Variables used for plots or calculations on immersed solid domain boundaries are not taken from the immersed solid domain; instead, they are interpolated from the fluid/porous domain in which the solid is immersed. The accuracy of such interpolation is dependent on the mesh densities of both the fluid/porous domain and the surface of the immersed solid domain. To visualize, or perform computations with, variables that are associated with the immersed solid domain, use slice planes, user surfaces, or other locators that are offset into the immersed solid domain, and set the applicable **Domains** setting to refer to the immersed solid domain.

12.1.1.1.2. Definition

12.1.1.1.2.1. Method

The **Method** setting has the following options:

Option	Description
XYZ	Enables you to set a coordinate in 3D space for the Point.

Option	Description
Node Number	Enables you to select a node to which to attach the Point.
Variable Minimum	Places the Point at the selected variable's lowest value. Select whether the object you want to plot will be based on hybrid or conservative values. For details, see Hybrid and Conservative Variable Values .
Variable Maximum	Places the Point at the selected variable's greatest value. Select whether the object you want to plot will be based on hybrid or conservative values. For details, see Hybrid and Conservative Variable Values .

Note:

You can move only points that have been specified with the XYZ option.

When using **Variable Minimum** or **Variable Maximum** option on a point in multi-file or comparison mode, the point is placed at the location of the overall minimum/maximum. If you want to place the point at the minimum/maximum value for the individual cases, select the appropriate case in the point's **Domain List** selector.

12.1.1.1.2.2. Point

Point is available only if the XYZ option is selected. The **Point** setting specifies the Cartesian coordinates for the Point object. Once the point is created, you can use the mouse pointer to drag the point around in the domain. For details, see [Picking Mode \(p. 126\)](#).

12.1.1.1.2.3. Node Number

Node Number is available only if the Node Number **Method** is selected. The **Node Number** setting specifies at which node to place the Point object. When more than one domain is selected, a point is created for the specified node number in each domain (if it exists). If the node number does not exist in one domain but exists in another, you should select only the domain in which the node exists or an error message will be displayed.

12.1.1.1.2.4. Location

Location is available only if the Variable Minimum or the Variable Maximum options are selected. The **Location** setting specifies an object for the Point to be located in. When more than one domain is selected, a point is created for the minimum or maximum value of the variable within each domain.

12.1.1.1.2.5. Variable

Variable is available only if the Variable Minimum or the Variable Maximum options are selected. The **Variable** setting selects the variable to be used to find the maximum or minimum point.

12.1.1.1.3. Nearest Node Value

Nearest Node appears when any option except the **Node Number** option is selected. The **Nearest Node** text displays the numerical value of the nearest node to the point's current position.

12.1.1.2. Point: Color Tab

The **Color** tab controls the color settings. For details, see [Color Tab \(p. 52\)](#).

12.1.1.3. Point: Symbol Tab

12.1.1.3.1. Symbol

The **Symbol** setting has the following options:

Symbol	Description
Crosshair	A 3D "+" sign.
Octahedron	A 3D diamond that has eight faces.
Cube	A box.
Ball	A sphere.

12.1.1.3.2. Symbol Size

The **Symbol Size** setting specifies the size of the Point symbol. Each **Symbol Size** unit represents 5% of the domain span. The domain span, which is dependent on the geometry, is equal to the largest difference from the X, Y, and Z ranges.

12.1.1.4. Point: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.1.1.5. Point: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.2. Point Cloud Command

To create multiple points, select **Insert > Location > Point Cloud**. You can create uniform vector plots independent of the mesh by using the point cloud object. You can also create streamlines that use a point cloud as the locator.

The following characteristics of point clouds will be discussed:

- [Point Cloud: Geometry Tab \(p. 224\)](#)
- [Point Cloud: Color Tab \(p. 226\)](#)

- [Point Cloud: Symbol Tab \(p. 226\)](#)
 - [Point Cloud: Render Tab \(p. 226\)](#)
 - [Point Cloud: View Tab \(p. 227\)](#)
-

Note:

There are two ways to insert a point cloud:

- From the menu bar, select **Insert > Location > Point Cloud**.
 - From the toolbar, select **Location > Point Cloud**.
-

12.1.2.1. Point Cloud: Geometry Tab

12.1.2.1.1. Domains

For details, see [Domains \(p. 221\)](#).

12.1.2.1.2. Method

The options are:

- From Locations (default)

The **Definition** settings apply. For details, see [Definition \(p. 224\)](#).

- From File

Set **File** to the name of a CSV file of the profile or table format. For details on the file format, see [Profile Data Format in the CFX-Pre User's Guide](#) and [Table Data Format in the CFX-Pre User's Guide](#). When you click **Apply**, the point cloud is defined according to the data set, within the CSV file, that is specified by the **Section Name** setting.

12.1.2.1.3. Definition

The **Definition** settings apply only when **Method** is set to From Locations.

12.1.2.1.3.1. Locations

The **Locations** setting selects the location or locations in which the point cloud is created.

Tip:

Click *Location Editor*  to open the **Location Selector** dialog box, which displays the complete list of available locations.

12.1.2.1.3.2. Sampling

The **Sampling** setting has the following options:

Option	Description
Equally Spaced	Generates points with roughly the same distance between them.
Rectangular Grid	Generates a rectangular grid of points on the surface. This option should be used only on flat surfaces.
Vertex	Generates the points on the vertices of the mesh. The maximum number of points is the total number of vertices in the mesh.
Face Center	Generates the points at the center of the mesh faces. The maximum number of points is the total number of faces in the mesh.
Free Edge	Generates the points on the outer edge at the center of the edge segments.
Random	Generates the points randomly. If the seed is positive, the point distribution can be reproduced.

12.1.2.1.3.3. # of Points

of Points is available only when either the **Equally Spaced** or **Random** option is selected. The **# of Points** setting specifies the number of equally spaced points you want generated on the surface of the mesh.

For the **Equally Spaced** option, the actual number of points generated is guided by the provided **# of Points** but may not be exactly equal to this.

12.1.2.1.3.4. Spacing

Spacing is available only when the **Rectangular Grid** option is selected. The **Spacing** setting specifies a value that represents a fraction of the maximum domain extent. For example, if your domain has a maximum extent of 1 [m] and a **Spacing** of 0.1 was used, a rectangular grid with 0.1 [m] spacing would be created.

12.1.2.1.3.5. Aspect Ratio

Aspect Ratio is available only when the **Rectangular Grid** option is selected. The **Aspect Ratio** setting stretches the rectangle in a direction parallel to the grid axes. If a value less than one is entered, the grid will be stretched in one direction. If a value greater than one is entered, the grid will be stretched in the direction perpendicular to the previous direction.

12.1.2.1.3.6. Grid Angle

Grid Angle is available only when the **Rectangular Grid** option is selected. The **Grid Angle** setting specifies the magnitude and direction of grid rotation.

12.1.2.1.3.7. Reduction

Reduction is available only when the **Vertex**, **Face Center**, or **Free Edge** options are selected. The **Reduction** setting has the following options:

Option	Description
Max Number of Points	Enables the option to specify the maximum number of points allowed to be plotted.
Reduction Factor	Enables the option to specify a reduction factor from the full number of points.

12.1.2.1.3.8. Max Points

Max Points is available only if the **Max Number of Points** option is selected. The **Max Points** setting specifies a value for the maximum number of points allowed. If the maximum number of vertices is greater than that of the specified value, then the points taken will be randomly selected.

12.1.2.1.3.9. Factor

Factor is available only if the **Reduction Factor** option is selected. The **Factor** setting specifies a value by which to decrease the total number of points in the Point Cloud object. The final number of vectors is $total/n$, where *total* is the total number of seeds, and *n* is the reduction value entered into the box.

12.1.2.1.3.10. Seed

Seed is available only if the **Random** option is selected. The **Seed** setting generates a different set of random points for each value entered. The distribution cannot be replicated or reproduced for negative seed values. For negative seed values, the random series is based on the system time. Different compilers may generate different distributions for the same positive seed value.

Note:

Similar sampling options are also available directly on **Vector** and **Streamline** objects.

12.1.2.2. Point Cloud: Color Tab

The color settings can be changed on the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.1.2.3. Point Cloud: Symbol Tab

For details, see [Point: Symbol Tab \(p. 223\)](#).

12.1.2.4. Point Cloud: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.1.2.5. Point Cloud: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.3. Line Command

A line locator can exist between two points anywhere inside or outside the domain.

The following characteristics of lines will be discussed:

- [Line: Geometry Tab \(p. 227\)](#)
- [Line: Color Tab \(p. 228\)](#)
- [Line: Render Tab \(p. 228\)](#)
- [Line: View Tab \(p. 228\)](#)

Note:

There are several ways to insert a line:

- From the menu bar, select **Insert > Location > Line**.
 - From the toolbar, select **Location > Line**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
-

12.1.3.1. Line: Geometry Tab

12.1.3.1.1. Domains

For details, see [Domains \(p. 221\)](#).

12.1.3.1.2. Definition

12.1.3.1.2.1. Method

The only available option is the **Two Points** option.

12.1.3.1.2.2. Point 1

The **Point 1** fields specify the start point of the line.

12.1.3.1.2.3. Point 2

The **Point 2** fields specify the end point of the line.

12.1.3.1.3. Line Type

12.1.3.1.3.1. Cut/Sample Options

Selecting **Cut** will extend the line in both directions until it reaches the edge of the domain. Points on this line exist where the line intersects with a mesh element face.

Tip:

In cases with very thin elements near a boundary, a cut line normal to the boundary may stop too early and not quite reach the boundary. To correct this, you can set the environment variable CFX_CUT_LINE_TOLERANCE before starting CFD-Post. For example:

```
CFDPOST_CUT_LINE_TOLERANCE='1.0e-8 [m]'
```

The smaller the number, the more likely it is that the line will reach the boundary. However, if it is too small, the line could end up with a number of coincident (repeated) points.

Selecting **Sample** creates a line existing between the two points entered. It is mesh-independent, and the number of points along the line corresponds to the value you enter in the **Samples** box.

12.1.3.1.3.2. Samples

Samples is available only if the **Sample** option is selected. The **Samples** setting specifies a value for the number of evenly-spaced sampling points along the line.

12.1.3.1.4. Line Translation Using Picking Mode

You can use picking mode to select or translate a line in the viewer. To move a line, select picking mode by clicking *Single Select*  in the **Selection Tools** toolbar and drag the line to a new location. The line properties will automatically update in the details view. For details, see [Picking Mode \(p. 126\)](#).

12.1.3.2. Line: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.1.3.3. Line: Render Tab

You can change the **Line Width** by entering a value corresponding to the pixel width of the line. You can specify the value between 1 and 11 by using the graduated arrows, the embedded slider, or by typing in the value.

12.1.3.4. Line: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.4. Plane Command

A plane is a two-dimensional area that exists only within the boundaries of the computational domain.

The following characteristics of planes will be discussed:

- [Plane: Geometry Tab \(p. 229\)](#)
- [Plane: Color Tab \(p. 232\)](#)
- [Plane: Render Tab \(p. 232\)](#)
- [Plane: View Tab \(p. 232\)](#)

Note:

There are several ways to insert a plane:

- From the menu bar, select **Insert > Location > Plane**.
 - From the toolbar, select **Location > Plane**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the **3D Viewer**.
-

12.1.4.1. Plane: Geometry Tab

12.1.4.1.1. Domains

For details, see [Domains \(p. 221\)](#).

12.1.4.1.2. Definition

12.1.4.1.2.1. Method

The **Method** setting has the following options:

Option	Description
YZ Plane	Defines a plane normal to the X axis.
ZX Plane	Defines a plane normal to the Y axis.
XY Plane	Defines a plane normal to the Z axis.
Point and Normal	Enables you to specify a point on the plane and a normal vector to the plane.
Three Points	Enables you to define a plane by providing three points that lie in the plane.

12.1.4.1.2.2. X

X is available only if the **YZ Plane** option is selected. The **X** setting specifies an offset value from the X axis.

12.1.4.1.2.3. Y

Y is available only if the **ZX Plane** option is selected. The **Y** setting specifies an offset value from the Y axis.

12.1.4.1.2.4. Z

Z is available only if the **XY Plane** option is selected. The **Z** setting specifies an offset value from the Z axis.

12.1.4.1.2.5. Point

Point is available only if the **Point** and **Normal** option is selected. The **Point** setting specifies the 3D coordinates of the point that lies on the plane.

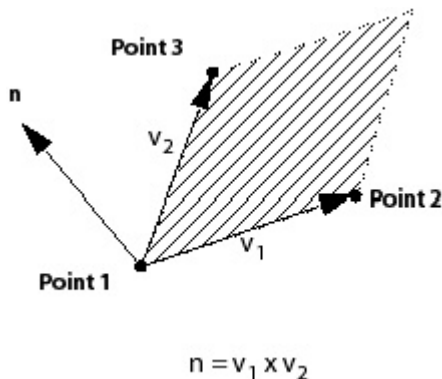
12.1.4.1.2.6. Normal

Normal is available only if the **Point** and **Normal** option is selected. The plane normal is calculated as a vector from the origin to the specified coordinates.

12.1.4.1.2.7. Point 1, Point 2, and Point 3

These options are only available if the **Three Points** option is selected. The **Point 1**, **Point 2**, and **Point 3** settings specify three points that lie on the plane.

The normal vector to the plane is calculated using the right-hand rule. The first vector is from **Point 1** to **Point 2**, and the second is from **Point 1** to **Point 3**, as shown in the following diagram. For example, the direction of this vector might be important if you are using the plane to define a Clip Plane.



12.1.4.1.3. Plane Bounds

12.1.4.1.3.1. Type

The **Type** setting has the following options:

Option	Description
None	Cuts through a complete cross-section of each domain specified in the Domains list. A slice plane is bounded only by the limits of the domain. The Plane Type must be set to Slice for this option (default).
Circular	Causes the boundary of the plane to be in the shape of a circle. The circle is centered at the origin for the YZ, ZX, and XY Planes. For the other two methods, the circle is centered at the first point entered in the Definition frame.
Rectangular	Causes the boundary of the plane to be a rectangular shape. The rectangle is centered at the origin for the YZ, ZX, and XY Planes. For the other two methods, the rectangle is centered at the first point entered in the Definition frame.

12.1.4.1.3.2. Radius

Radius is available only if the **Circular** option is selected. The **Radius** setting specifies a radius for the circular boundary. You can enter a value or select the *Expression*  icon to the right of the **Radius** setting to specify the radius as an expression.

12.1.4.1.3.3. X/Y/Z Size

These settings are available only if the **Rectangular** option is selected. Two of these options will be displayed because a plane is a 2D object. These settings will specify a width and height for the rectangular boundary. The size of the rectangle is determined with reference to the plane's origin (that is, the plane is resized around its center).

12.1.4.1.3.4. X/Y/Z Angle

This setting is available only if the **Rectangular** option is selected. Only one of these settings is displayed at once. This setting specifies an angle to rotate the plane counterclockwise about its normal vector by the specified number of degrees.

12.1.4.1.3.5. Invert Plane Bound Check Box

Invert Plane Bound is available only if the **Circular** or the **Rectangular** option is selected. If this check box is selected, the area defined by the rectangle or circle is used as a cut-out area from a slice plane that is bounded only by the domains. The area inside the bounds of the rectangle or circle do not form part of the plane, but everything on the slice plane outside of these bounds is included.

12.1.4.1.4. Plane Type

12.1.4.1.4.1. Slice Option

Select the **Slice** option to cut the plane so that it lies only inside the domain.

A slice plane differs from a sampling plane. A sampling plane is a set of evenly-spaced sampling points that are independent of the mesh. When you create a slice plane, the sampling points are placed at locations where the slice plane intersects an edge of the mesh, causing an uneven distribution of the sampling points. The density of these sampling points in a slice plane is related to the length scale of the mesh.

When you use the slice plane for **Vector** plots, the seeds are the points where the plane intersects a point on the edge of three mesh elements. You can view the seeds by turning on the **Show Mesh Lines** option on the **Render** tab for the plane.

12.1.4.1.4.2. Sample Option

Select the **Sample** option to specify the amount of seeds in the plane.

When creating a sampling plane, the **Plane Bounds** must be either **Circular** or **Rectangular**. For the **Circular** option, the density of sampling points is determined by the radius of the plane specified in the **Plane Bounds** tab and the number of radial and circumferential sampling points. For **Rectangular** bounds, you must specify the size of the bounds for your plane in each of the plane directions. The density of sampling points depends on the size of the plane and the number of samples in each of the two coordinate directions that describe the plane.

Certain types of plots will show small differences across GGI interfaces. This is to be expected when the nodes of the computational grids on each side of a GGI connection do not match. For example, contour lines or fringe lines may not match exactly across a GGI interface. This is a very minor effect and is not an indicator of any problem.

12.1.4.1.5. Plane Translation using Picking Mode

For details, see [Line Translation Using Picking Mode \(p. 228\)](#).

12.1.4.2. Plane: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.1.4.3. Plane: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.1.4.4. Plane: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.5. Volume Command

A **Volume** is a collection of mesh elements that can be used as a locator for graphic objects or calculations. Volumes will not be displayed as perfect shapes (for example, a perfect sphere) because mesh elements are either included in or excluded from the **Volume** object.

The following characteristics of volumes will be discussed:

- [Volume: Geometry Tab \(p. 233\)](#)
- [Volume: Color Tab \(p. 237\)](#)
- [Volume: Render Tab \(p. 237\)](#)
- [Volume: View Tab \(p. 237\)](#)

Note:

There are several ways to insert a volume:

- From the menu bar, select **Insert > Location > Volume**.
 - From the toolbar, select **Location > Volume**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the **3D Viewer**.
-

12.1.5.1. Volume: Geometry Tab

12.1.5.1.1. Domains

For details, see [Domains \(p. 221\)](#).

12.1.5.1.2. Element Types

The **Element Types** setting has the following options:

Option	Description
Tet	Displays volume that is connected to a tetrahedral mesh.
Pyramid	Displays volume that is connected to a pyramid-shaped mesh.
Wedge	Displays volume that is connected to a wedge-shaped mesh.
Hex	Displays volume that is connected to a hexagonal mesh.

12.1.5.1.3. Definition

12.1.5.1.3.1. Method

The **Method** setting has the following options:

Option	Description
Sphere	Creates a sphere-shaped volume. Enables you to specify a center point and radius for the sphere volume.

Option	Description
From Surface	Creates a volume on a surface. Enables you to select a surface from the Location setting. Some surface types may not be available.
Isovolume	Creates a volume at a specified variables value. Enables you to specify a variable and one or two values (depending on the Mode) to create one or two isosurfaces that bound the isovolume.
Surrounding Node	Creates a volume at a node. Enables you to specify a node by number.

12.1.5.1.3.2. Point

Point is available only if the Sphere option is selected. The **Point** setting specifies a center point for the sphere volume. The point can be anywhere in 3D space.

12.1.5.1.3.3. Radius

Radius is available only if the Sphere option is selected. The **Sphere** setting specifies a radius for the sphere volume.

12.1.5.1.3.4. Location

Location is available only if the From Surface option is selected. The **Location** setting selects from a list of valid locations for the volume to exist on.

12.1.5.1.3.5. Variable

Variable is available only if the Isovolume option is selected. The **Variable** setting selects a variable to plot the volume on. A **Value** for the variable must be selected before the volume can be defined.

12.1.5.1.3.6. Hybrid/Conservative Options

These options are available only if the Isovolume option is selected. For help on which field to select, see [Hybrid and Conservative Variable Values](#).

12.1.5.1.3.7. Mode (for the Sphere and From Surface options)

The **Mode** setting has the following options:

Option	Description
Intersection	Creates a volume at the specified radius for the Sphere option. For the From Surface option, the volume is created on the surface of the object.
Below Intersection	Creates a volume for all of the radii less than the specified radius for the Sphere option. For the From Surface option, the volume is plotted for all values less than the given value on the location object.

Option	Description
Above Intersection	Opposite to the Below Intersection option.

12.1.5.1.3.8. Mode (for the Isovolum option)

The **Mode** setting has the following options:

Option	Description
At Value	Creates a volume for all the mesh elements in the domain equal to the entered value.
Below Value	Creates a volume for all the mesh elements in the domain above the entered value.
Above Value	Creates a volume for all the mesh elements in the domain less than the entered value.
Between Value	Creates a volume for all the mesh elements in the domain in between the two entered values.

12.1.5.1.3.9. Value Fields

The **Value** fields are available only if the Isovolum option is selected. The **Value** fields specify values to compare to using the **Mode** options. For example, if **Value** is set to 2 and **Mode** is set to At Value, the Volume will plot where the variable is equal to 2.

12.1.5.1.3.10. Inclusive Check Box

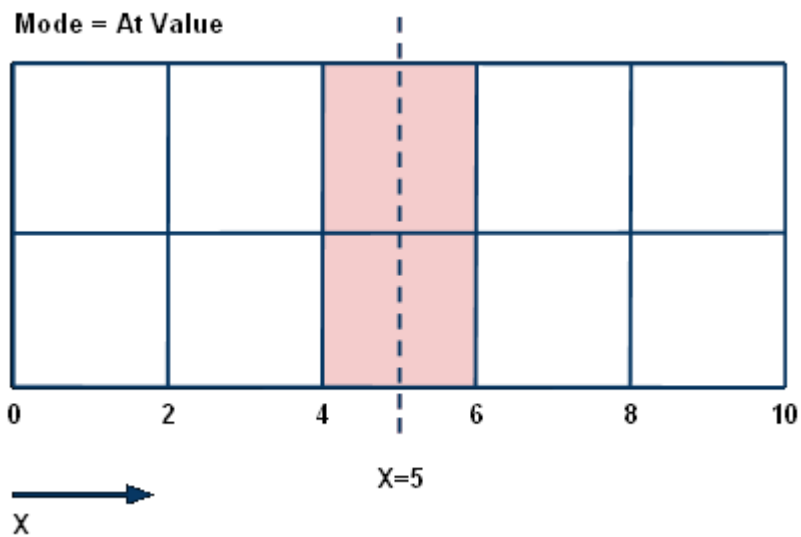
Select the **Inclusive** check box to add the entered values to an above or below comparison **Mode**. For example, if the **Inclusive** check box is selected with the Below Intersection option, the volume will include the radius entered or surface selected.

12.1.5.1.3.11. How CFD-Post Calculates Isovolumes

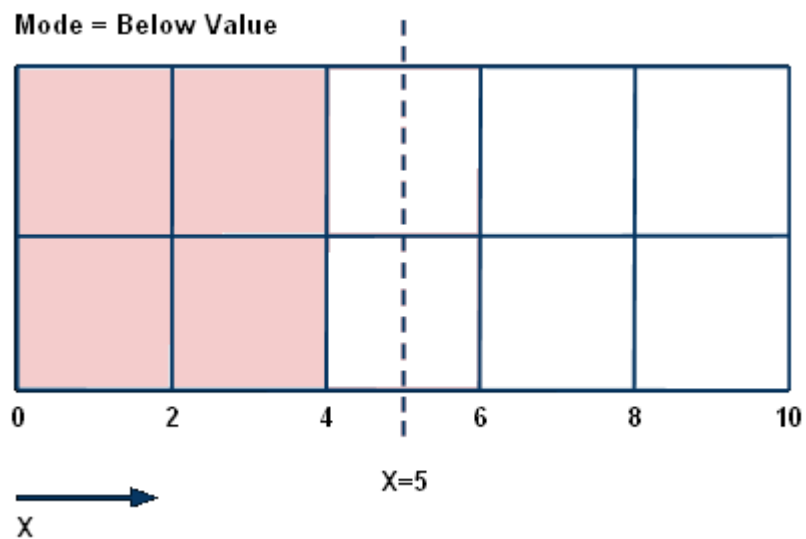
In order to see the effects of the **Inclusive** check box on the **Minimum Face Angle** variable set to the **Below Value** mode, do the following:

1. Open a results file.
2. Create a volume, accepting the default name.
3. Set **Variable** to **Minimum Face Angle**, **Mode** to **Below Value**, and then click **Apply**. Few volumes appear.
4. Now enable **Inclusive** and click **Apply** again. Many more volumes appear.

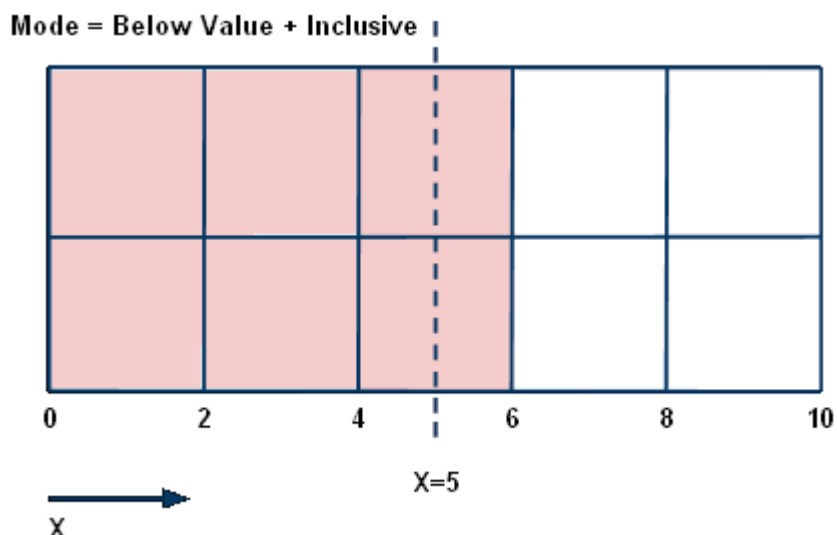
The differences you see are caused by how CFD-Post calculates values for a given point on a mesh (**Mode** is set to **At Value**):



...as compared to **Mode** being set to **Below Value**



...and as compared to **Mode** being set to **Below Value** with **Inclusive** being selected:



12.1.5.1.4. Remove Internal Faces

Remove Internal Faces is on by default, causing CFD-Post to remove all internal mesh element faces from a volume object. Turning off this setting can save significant rendering time by skipping the removal process. As a side effect of turning off this setting, internal faces can appear in a volume object that is transparent or clipped.

Note:

It is a known issue that some mesh faces might be missing in the **3D Viewer** when **Remove Internal Faces** is off. This issue seems more likely to occur if the object being displayed contains many elements and involves more than one element type.

12.1.5.2. Volume: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.1.5.3. Volume: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.1.5.4. Volume: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.6. Isosurface Command

An *isosurface* is a surface upon which a particular variable has a constant value, called the *level*. For instance, an Isosurface of pressure would be a surface consisting of all the points in the geometry

where the pressure took a value of $1.32\text{e}+05$ Pa. In CFD-Post, isosurfaces can be defined using any variable. You can also color the isosurface using any variable or choosing a constant color.

The following characteristics of isosurfaces will be discussed:

- [Isosurface: Geometry Tab \(p. 239\)](#)
- [Isosurface: Color Tab \(p. 239\)](#)
- [Isosurface: Render Tab \(p. 239\)](#)
- [Isosurface: View Tab \(p. 239\)](#)

Note:

There are several ways to insert an isosurface:

- From the menu bar, select **Insert > Location > Isosurface**.
- From the toolbar, select **Location > Isosurface**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Note:

When you are creating an isosurface using a variable that is fundamentally discontinuous within a domain, some unexpected portions of the isosurface may appear near the discontinuity. A common example is a variable Theta that is discontinuous at 0° in a full 360° domain. In such a case, creating an isosurface for any value of Theta will result in two basic parts: the expected part of the isosurface that was defined by the Theta value that was selected; a spurious part that appears along the border between nodes with Theta values near 0° and nodes with Theta values near 360° . The spurious part should be ignored.

Note:

When you are creating an isosurface (or another object such as a contour plot or a chart), and the range of the specified variable is discontinuous at any point in its domain, the resulting range may be different from what you expect it to be. For example, consider a case with a cylindrical domain in which you would expect the Theta value to have a minimum value of 0° and a maximum value of 360° . In reality, the range will depend on the vertices of the mesh and the Theta values of the vertices. For example, an extremely coarse mesh with only 10 equidistant nodes on the circumference with Theta values of 0° , 36° , 72° , 108° , 144° , 180° , 216° , 252° , 288° , 324° , and 0° , will result in a range of 0° to 324° for Theta. If the nodes appear on Theta values of, for example, 10° , 46° , 82° , 118° , 154° , 190° , 226° , 298° , 334° , and 10° , then the range for Theta will be from 10° to 334° . Therefore, the

exact range of Theta will be dependent on the mesh. Refining the mesh will cause the actual range to be more similar to the expected range.

Note:

Isosurface locator face IDs and node numbering are typically altered by the use of multithreading (see [Threading](#) (p. 210)). As a result, sampled vector plots on isosurface locators might look different when generated with multithreading compared to when generated without multithreading. Multithreading has no effect on vector plots that place a vector at each and every node.

12.1.6.1. Isosurface: Geometry Tab

12.1.6.1.1. Domains

For details, see [Domains](#) (p. 221).

12.1.6.1.2. Definition

12.1.6.1.2.1. Variable

The **Variable** setting specifies the variable that you want to plot.

Tip:

Click the *Location Editor*  to open the **Variable Selector** dialog box, which displays the complete list of available options.

12.1.6.1.2.2. Hybrid/Conservative Option

For help on which field to select, see [Hybrid and Conservative Variable Values](#).

12.1.6.1.2.3. Value

The **Value** setting specifies a numerical value or expression to plot for the given variable.

12.1.6.2. Isosurface: Color Tab

You can change the color settings by clicking the **Color** tab; for details, see [Color Tab](#) (p. 52).

12.1.6.3. Isosurface: Render Tab

To change the rendering settings, click the **Render** tab. For details, see [Render Tab](#) (p. 55).

12.1.6.4. Isosurface: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.7. Iso Clip Command

An *iso clip* hides the portion of one or more locators subject to one or more constraints (visibility parameters) that you specify.

There are several ways to insert an iso clip:

- From the menu bar, select **Insert > Location > Iso Clip**.
- From the toolbar, select **Location > Iso Clip**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Iso clips have the following restrictions:

- Iso clip locators interpolate values using a method that is slightly less accurate than that used for slice planes and isosurfaces. For details, see [Interpolation in CFD-Post \(p. 219\)](#).
- Iso clips cannot clip volumes.
- Iso clips cannot have mix of faces and lines.
- Setting **Visible when [value]** to = always results in lines, never faces.
- Setting **Visible when [value]** to both $\geq$ and $\leq$ produces only faces, no lines (due to restriction 3), yet clipped lines may mix with mesh lines if the latter are shown.

The following characteristics of iso clips will be discussed:

- [Iso Clip: Geometry Tab \(p. 240\)](#)
- [Iso Clip: Color Tab \(p. 241\)](#)
- [Iso Clip: Render Tab \(p. 241\)](#)
- [Iso Clip: View Tab \(p. 241\)](#)

12.1.7.1. Iso Clip: Geometry Tab

12.1.7.1.1. Domains

For details, see [Domains \(p. 221\)](#).

12.1.7.1.2. Location

Click the *Location Editor*  icon to open the **Location Editor** dialog box, which displays the complete list of available options. If you specify multiple locators, they must all have the same dimensionality (for example, all must be planes, rather than a combination of lines and planes).

12.1.7.1.3. Visibility Parameters

The **Visibility parameters** area is where you set the variables that hide the values that fail to meet a specified condition on a locator specified in the **Locations** field. For example, if the locator is an X-Y plane and the visibility is restricted to $Y \geq 0$, $Y \leq .1$, and $X \geq .15$, only areas that have values within those bounds will be displayed.

You create a new clip setting by clicking the New  icon or by right-clicking in the **Visibility parameters** area and selecting **New**. These actions cause the **Visibility Parameter Properties** settings to appear:

Variable

Sets the variable that controls where the iso clip regions are placed. Typically you would specify geometric variables.

Visible when [value]

Sets the display of regions ($\geq$, $\leq$) or a line ($=$).

Boundary Data

Enables you to set the boundary data to use of hybrid or conservative variable values. For details, see [Hybrid and Conservative Variable Values](#).

12.1.7.2. Iso Clip: Color Tab

You can change the color of the locator or the variable that is colored on the locator by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.1.7.3. Iso Clip: Render Tab

To change the rendering settings, click the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.1.7.4. Iso Clip: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.8. Vortex Core Region

A *vortex* is a circular or spiral set of streamlines; a *vortex core* is a special type of isosurface that displays a vortex. The CFD-Post vortex core visualization tools are designed to help you identify and understand vortex regions.

The following characteristics of vortex cores will be discussed:

- [Vortex Core Region: Geometry Tab \(p. 242\)](#)

- [Vortex Core Region: Color Tab \(p. 248\)](#)
- [Vortex Core Region: Render Tab \(p. 248\)](#)
- [Vortex Core Region: View Tab \(p. 248\)](#)

Note:

There are several ways to insert a vortex core region:

- From the menu bar, select **Insert > Location > Vortex Core Region**.
 - From the toolbar, select **Location > Vortex Core Region**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
-

12.1.8.1. Vortex Core Region: Geometry Tab

12.1.8.1.1. Domains

The **Domains** setting specifies the domains where the vortex core should be found. Selected domains do not need to be contiguous.

For details, see [Domains \(p. 221\)](#).

12.1.8.1.2. Definition Area

The **Definition** area is where you define the type and the strength of the vortex core.

12.1.8.1.2.1. Method

The **Method** setting specifies sets of equations that detect vortices as spatial regions. Click the drop-down arrow to choose a method:

Absolute Helicity	Absolute value of the dot product of velocity vector and vorticity vector.
Eigen Helicity	Dot product of vorticity and the normal of swirling plane (that is, the plane spanned by the real and imaginary parts of complex eigenvectors of velocity gradient tensor).
Lambda 2-Criterion	The negative values of the second eigenvalue of the symmetry square of velocity gradient tensor. Derived through the hessian of pressure.
Q-Criterion	The second invariant of the velocity gradient tensor. For a region with positive values, it could include regions with negative discriminants and exclude region with positive discriminants.
Real Eigen Helicity	Dot product of vorticity and swirling vector that is the real eigenvector of velocity gradient tensor.
Swirling Discriminant	The discriminant of velocity gradient tensor for complex eigenvalues. The positive values indicate existence of swirling local flow pattern.

Swirling Strength	The imaginary part of complex eigenvalues of velocity gradient tensor. It is positive if and only if the discriminant is positive and its value represents the strength of swirling motion around local centers.
Vorticity	Curl of velocity vector.

Note:

There is no recommended vortex core method; the appropriate choice of vortex core is always case-dependent.

12.1.8.1.2.1.1. Vortex Core Mathematics

A number of methods are based on eigen analysis in local velocity gradient tensor. The following are the related notations and equations.

For the velocity gradient tensor

$$D = [d_{ij}] = \begin{bmatrix} d_{11} & d_{12} & d_{13} \\ d_{21} & d_{22} & d_{23} \\ d_{31} & d_{32} & d_{33} \end{bmatrix} = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial u}{\partial z} \\ \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y} & \frac{\partial v}{\partial z} \\ \frac{\partial w}{\partial x} & \frac{\partial w}{\partial y} & \frac{\partial w}{\partial z} \end{bmatrix} \quad (12.1)$$

The eigenvalues of the gradient tensor satisfies

$$\lambda^3 + P\lambda^2 + Q\lambda + R = 0 \quad (12.2)$$

where

$$P \equiv -\text{tr}(\underline{D}) = -\nabla \cdot \underline{u} = -(d_{11} + d_{22} + d_{33}) \quad (12.3)$$

$$Q \equiv \frac{1}{2} [P^2 - \text{tr}(\underline{D}\underline{D})] \quad (12.4)$$

$$= (d_{22}d_{33} - d_{23}d_{32}) + (d_{11}d_{22} - d_{12}d_{21}) + (d_{33}d_{11} - d_{13}d_{31})$$

$$R \equiv \frac{1}{3} [-P^3 + 3PQ - \text{tr}(\underline{D}\underline{D}\underline{D})] = \quad (12.5)$$

$$d_{11}(d_{23}d_{32} - d_{22}d_{33}) + d_{12}(d_{21}d_{33} - d_{31}d_{23}) + d_{13}(d_{31}d_{22} - d_{21}d_{32})$$

Now let

$$q \equiv Q - \frac{1}{3}P^2 \quad (12.6)$$

$$r \equiv R + \frac{2}{27}P^3 - \frac{1}{3}PQ \quad (12.7)$$

Then, if the discriminant is

$$\Delta \equiv \left(\frac{1}{2}r\right)^2 + \left(\frac{1}{3}q\right)^3 > 0 \quad (12.8)$$

then the tensor has one real eigenvalue λ_r and a pair of conjugated complex eigenvalues $\lambda_{cr} \pm i\lambda_{ci}$

That is, the tensor can be decomposed as

$$[d_{ij}] = [v_r v_{cr} v_{ci}] \begin{bmatrix} \lambda_r & 0 & 0 \\ 0 & \lambda_{cr} & \lambda_{ci} \\ 0 & -\lambda_{ci} & \lambda_{cr} \end{bmatrix} [v_r v_{cr} v_{ci}]^{-1} \quad (12.9)$$

We denote

$$\xi_2 = \sqrt[3]{\sqrt{\Delta} - \frac{r}{2}} \quad (12.10)$$

and

$$\xi_3 = \sqrt[3]{\sqrt{\Delta} + \frac{r}{2}} \quad (12.11)$$

Then

$$\lambda_r = \tilde{\lambda}_r - \frac{P}{3} = \xi_2 - \xi_3 - \frac{P}{3} \quad (12.12)$$

$$\lambda_{cr} = -\frac{\xi_2 - \xi_3}{2} - \frac{P}{3} \quad (12.13)$$

$$\lambda_{ci} = \frac{\xi_2 + \xi_3}{2} \sqrt{3} \quad (12.14)$$

The last one is called *Swirling Strength*, and represents the strength of the local swirling motion. In CFD-Post, the magnitude of both Swirling Vector and Swirling Normal is the Swirling Strength. The direction of the Swirling Vector is that of the real eigenvector (v_r in Equation 12.19 (p. 244)) and the direction of the Swirling Normal is that of v_n defined in Equation 12.26 (p. 245).

The following relationships are useful:

$$\xi_2 \xi_3 = \frac{q}{3} \quad (12.15)$$

$$\lambda_{ci}^2 = q + \frac{3}{4} \tilde{\lambda}_r^2 = Q + \frac{3}{4} \left(\lambda_r + P \right) \left(\lambda_r - \frac{P}{3} \right) \quad (12.16)$$

$$\Delta = \frac{1}{3} \lambda_{ci}^2 \left(\frac{\lambda_{ci}^2}{3} + \frac{3 \tilde{\lambda}_r^2}{4} \right) \quad (12.17)$$

$$Q = \frac{1}{4} \|\nabla \times \bar{U}\| + 2(tr^2(S) - tr(SS)) \quad (12.18)$$

Now the real eigenvector meets:

$$[D - \lambda_r I] v_r = 0 \quad (12.19)$$

We can calculate the real eigenvector using one of the non-zero vectors:

$$\begin{bmatrix} d_{12}d_{23} - d_{13}(d_{22} - \lambda_r) \\ d_{13}d_{21} - d_{23}(d_{11} - \lambda_r) \\ (d_{11} - \lambda_r)(d_{22} - \lambda_r) - d_{12}d_{21} \end{bmatrix} \quad (12.20)$$

$$\begin{bmatrix} d_{12}(d_{33} - \lambda_r) - d_{32}d_{13} \\ d_{13}d_{31} - (d_{11} - \lambda_r)(d_{33} - \lambda_r) \\ d_{32}(d_{11} - \lambda_r) - d_{31}d_{12} \end{bmatrix} \quad (12.21)$$

$$\begin{bmatrix} (d_{22}-\lambda_r)(d_{33}-\lambda_r)-d_{32}d_{23} \\ d_{23}d_{31}-d_{21}(d_{33}-\lambda_r) \\ d_{21}d_{32}-d_{31}(d_{22}-\lambda_r) \end{bmatrix} \quad (12.22)$$

The complex eigenvectors' real and imaginary parts meet:

$$[D-\lambda_{cr}I]v_{cr}=-\lambda_{ci}v_{ci} \quad (12.23)$$

$$[D-\lambda_{cr}I]v_{ci}=-\lambda_{ci}v_{cr} \quad (12.24)$$

Therefore, if

$$A \equiv DD-2\lambda_{cr}D+(\lambda_{cr}^2+\lambda_{ci}^2)I \quad (12.25)$$

then, $Av_{cr}=0$ and $Av_{ci}=0$. That is, all rows of matrix A are normal to both v_{cr} and v_{ci} , therefore they are all proportional to

$$v_n = \frac{v_{cr} \times v_{ci}}{\|v_{cr} \times v_{ci}\|} \quad (12.26)$$

So any non-zero row vector of matrix A can be used to calculate v_n .

This is useful to get the eigen-helicity $H_e = v_n \cdot \omega$, where ω is the vorticity vector.

On S and $S^2+\Omega^2$ let $S \equiv \frac{(D+D^T)}{2}$ and $\Omega \equiv \frac{(D-D^T)}{2}$

Then $D=S+\Omega$ and $S^2+\Omega^2 = \text{Sym}(D^2)$ have all real eigen-values ($\lambda_1 \leq \lambda_2 \leq \lambda_3$).

The region with negative of λ_2 is used in the method proposed by F. Hussain. By using the eigen-values and eigenvectors of velocity gradient tensor D , we have

$$[d_{ij}]^2 = [v_r v_{cr} v_{ci}] \begin{bmatrix} \lambda_r & 0 & 0 \\ 0 & \lambda_{cr}^2 - \lambda_{ci}^2 & 2\lambda_{cr}\lambda_{ci} \\ 0 & -2\lambda_{cr}\lambda_{ci} & \lambda_{cr}^2 - \lambda_{ci}^2 \end{bmatrix} [v_r v_{cr} v_{ci}]^{-1} \quad (12.27)$$

So, in the case the second eigenvalue is $\lambda_{2(S^2+\Omega^2)} = \lambda_{cr}^2 - \lambda_{ci}^2$

Also, we can express the tensor DD as

$$DD = 2\lambda_{cr}D - (\lambda_{cr}^2 + \lambda_{ci}^2)I + [(\lambda_r - \lambda_{cr})^2 + \lambda_{ci}^2]v_r v_n^T \quad (12.28)$$

Now when we look into the eigenvalues and vectors of S , the same should apply to $S^2+\Omega^2$.

Let

$$S = \begin{bmatrix} S_0 & S_3 & S_3 \\ S_3 & S_1 & S_4 \\ S_5 & S_4 & S_2 \end{bmatrix} \quad (12.29)$$

Its eigenvalues meet

$$\lambda^3 - A\lambda^2 - B\lambda - C = 0 \quad (12.30)$$

where

$$A \equiv \frac{s_0 + s_1 + s_2}{3} \quad (12.31)$$

$$B \equiv (s_0 s_1 + s_1 s_2 + s_2 s_0) - (s_3^2 + s_4^2 + s_5^2) \quad (12.32)$$

$$C \equiv s_0 s_1 s_2 + 2s_3 s_4 s_5 - s_0 s_4^2 - s_1 s_5^2 - s_2 s_3^2 \quad (12.33)$$

Then the three eigenvalues are:

$$\sigma_1 = A + \rho \cos(\theta) \quad (12.34)$$

$$\sigma_2 = A + \rho \cos\left(\theta + \frac{2\pi}{3}\right) \quad (12.35)$$

$$\sigma_3 = A + \rho \cos\left(\theta + \frac{4\pi}{3}\right) \quad (12.36)$$

where

$$\eta = \frac{(s_0 - s_1)^2 + (s_1 - s_2)^2 + (s_2 - s_0)^2}{2} + 3(s_3^2 + s_4^2 + s_5^2) \quad (12.37)$$

$$\rho = \frac{2}{3} \sqrt{\eta} \quad (12.38)$$

$$\theta = \frac{1}{3} \cos^{-1} \left(\frac{4}{\rho^3} \left(C + A \left(\frac{\eta}{3} - A^2 \right) \right) \right) \quad (12.39)$$

Because θ is in the range of $(0, \frac{\pi}{3})$, we have $\sigma_2 \leq \sigma_3 \leq \sigma_1$. Therefore, the second eigenvalue for a 3x3 symmetry tensor is $A + \rho \cos\left(\theta + \frac{4\pi}{3}\right)$.

The eigenvector corresponding to an eigenvalue λ can be one of the non-zero vectors

$$\begin{bmatrix} s_3 s_4 - s_5 (s_2 - \lambda) \\ s_5 s_3 - s_4 (s_0 - \lambda) \\ (s_0 - \lambda)(s_1 - \lambda) - s_3 s_3 \end{bmatrix} \quad (12.40)$$

$$\begin{bmatrix} s_3 (s_2 - \lambda) - s_4 s_5 \\ s_5 s_5 - (s_0 - \lambda)(s_2 - \lambda) \\ s_4 (s_0 - \lambda)(s_1 - \lambda) - s_3 s_5 \end{bmatrix} \quad (12.41)$$

$$\begin{bmatrix} (s_1 - \lambda)(s_2 - \lambda) - s_4 s_4 \\ s_4 s_5 - s_3 (s_2 - \lambda) \\ s_3 s_4 - s_5 (s_1 - \lambda) \end{bmatrix} \quad (12.42)$$

12.1.8.1.2.1.2. Vortex Core References

Bibliography

- [1] M. S. Chong, A. E. Perry, and B. J. Cantwell. Copyright © 1990. Phys. Fluid. A General Classification of Three Dimensional Flow Fields. 765-777. A 2.
- [2] U. Dallman, A. Hilgenstock, B. Schulte-Werning, S. Riedelbauch, and H. Vollmers. Copyright © 1991. AGARD Conf. Proc. CP-494. On the Footprints of Three-Dimensional Separated Vortex Flows Around Blunt Bodies.

- [3] R. Haimes and D. Sujudi. Copyright © 1995. Dept. of Aeronautics and Astronautics, MIT, Cambridge, MA. Identification of Swirling Flow in 3D Vector Fields. Tech. Report.
- [4] J. C. R. Hunt, A. A. Wary, and P. Moin. Copyright © 1988. NASA Ames / Stanford University in Oroc. 1988 Summer Program of the Center for Turbulent Research. Eddies, Streams, and Convergence Zones in Turbulent Flows. 193-207.
- [5] J. Jeong and F. Hussain. Copyright © 1995. Journal of Fluid Mechanics. On the Identification of a Vortex. 69-94. 285.
- [6] M. Jiang, R. Machiraju, and D. Thompson. Copyright © 2002. Eurographics – IEEE VGTC Symposium on Visualization. A Novel Approach to Vortex Core Region Detection.
- [7] S. K. Robinson, S. J. Kline, and P. R. Spalart. Copyright © 1988. In Proc. Zoran P. Zaric Memorial International Seminar on Near Wall Turbulence. Statistical Analysis of Near-wall Structures in Turbulent Channel Flow.
- [8] M. Roth and R. Peikert. Copyright © 1998. A Higher-order Method for Finding Vortex Core Lines.
- [9] J. Sahner, T. Weinkauff, and H.-C. Hege. Copyright © 2005. Eurographics – IEEE VGTC Symposium on Visualization. Galilean Invariant Extraction and Iconic Representation of Vortex Core Lines.
- [10] S. Zhang and D. Choudhury. Copyright © 2006. Phys. Fluids 18. Eigen Helicity Density: A New Vortex Identification Scheme and its Application in Accelerated Inhomogeneous Flows.
- [11] J. Zhou, R. J. Adrian, and S. Balachander. Copyright © 1996. Phys. Fluids 8. Autogeneration of Near Wall Vertical Structure in Channel Flow. 288-291.
- [12] J. Zhou. Copyright © 1997. Ph.D. thesis, Department of Theoretical and Applied Mechanics, University of Illinois at Urbana-Champaign, Urbana, Illinois. Self-sustaining Formation of Packets of Hairpin Vortices in a Turbulent Wall Layer.
- [13] J. Zhou, R. J. Adrian, S. Balachander, and T. M. Kendall. Copyright © 1999. Journal of Fluid Mechanics. Mechanisms for Generating Coherent Packets of Hairpin Vortices in Channel Flow. 353-396. 387.

12.1.8.1.2.2. Level

The **Level** setting controls the strength of the vortex core that is displayed. The **Level** setting is normalized between **Method** types so that it is easy for you to compare the output of the different methods.

12.1.8.1.2.3. Actual Value

The **Actual Value** setting displays the isosurface value. This read-only value varies between methods.

12.1.8.2. Vortex Core Region: Color Tab

To learn how to use color to show how a variable changes through a region or just to change the color of the vortex core regions, see [Color Tab \(p. 52\)](#).

Note:

The ranges of vortex core variables are calculated by CFD-Post and will be local to the timestep (that is, the range will not be calculated across all timesteps).

12.1.8.3. Vortex Core Region: Render Tab

To learn how to control the display of mesh lines, textures, and vortex core faces, see [Render Tab \(p. 55\)](#).

12.1.8.4. Vortex Core Region: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.9. Surface of Revolution Command

A **Surface of Revolution** is a surface created by revolving a polyline about an axis. The polyline may be as simple as a single line segment or as complicated as a general curve.

The following characteristics of surfaces of revolution will be discussed:

- [Surface of Revolution: Geometry Tab \(p. 249\)](#)
- [Surface of Revolution: Color Tab \(p. 251\)](#)
- [Surface of Revolution: Render Tab \(p. 251\)](#)
- [Surface of Revolution: View Tab \(p. 251\)](#)

Note:

There are several ways to insert a surface of revolution:

- From the menu bar, select **Insert > Location > Surface of Revolution**.
 - From the toolbar, select **Location > Surface of Revolution**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
-

12.1.9.1. Surface of Revolution: Geometry Tab

12.1.9.1.1. Domains

For details, see [Domains](#) (p. 221).

12.1.9.1.2. Definition

12.1.9.1.2.1. Method

The **Method** setting has the following options:

Option	Description
Cylinder	Creates a cylinder using two axial and one radial coordinate points.
Cone	Creates a cone using two axial and radial coordinate points.
Disc	Creates a disc using one axial and two radial coordinate points.
Sphere	Creates a cylinder using one axial and radial coordinate points.
From Line	Enables you to specify a line or polyline to revolve about the axis (to be specified later).

12.1.9.1.2.2. Point 1 (a,r) and Point 2 (a,r)

These fields are not available for the **From Line** option. These fields specify axial and radial coordinates to define the surface of revolution.

Only one set of coordinates are available for the **Sphere** option. The axial value offsets the sphere in the direction of the rotational axis, and the radial value is used as the radius of the sphere.

12.1.9.1.2.3. Line

Line is available only if the **From Line** option is selected. The **Line** setting selects a valid line or polyline to use for rotation around the axis.

Tip:

Click the *Location Editor*  icon to open the **Location Selector** dialog box, which displays the complete list of available lines.

Note:

Calculations of quantities (such as area) performed on a surface of revolution created using the **From Line** option may be incorrect in the following situations:

- Multiple input lines are selected with overlapping segments.

- An input line passes through a region where multiple domains overlap. Domains may overlap in a single case or between multiple cases if more than one case is loaded.

To ensure that calculations are correct, these situations should be avoided.

12.1.9.1.2.4. # of Samples

of Samples is not available if the **From Line** option is selected. The **# of Samples** setting sets the amount of sample points in the direction of the rotational axis.

12.1.9.1.2.5. Theta Samples

The **Theta Samples** setting specifies the amount of sample points evenly rotated around the rotational axis. For example, increasing this setting would make a cylinder's curve around its origin more accurate (more like a circle).

12.1.9.1.2.6. Project to AR Plane Check Box

The **Project to AR Plane** check box is available only if the **From Line** option is selected. If **Project to AR Plane** is selected (default), then the **Theta** values will be projected to the plane of constant **Theta**. This produces a more refined mesh.

12.1.9.1.3. Rotation Axis

12.1.9.1.3.1. Method

The **Method** setting has the following options:

Option	Description
Principal Axis	Enables you to specify a principal axis to rotate around.
Rotation Axis	Enables you to specify a custom axis to rotate around using a line.

12.1.9.1.3.2. Axis

Axis is available only if the **Principal Axis** option is selected. The **Axis** setting enables you to select from a list the X, Y, or Z axis to rotate around.

12.1.9.1.3.3. From/To Text Boxes

The **From** and **To** text boxes are available only if the **Rotation Axis** option is selected. These fields create a line representing the axis about which the Solid of Revolution is created.

12.1.9.1.4. Angle Range Check Box

Select the **Angle Range** check box if you want to specify a minimum or maximum angle to rotate to.

12.1.9.1.4.1. Min./Max. Angle

These settings specify a minimum and/or maximum angle to rotate to.

12.1.9.1.5. Axial/Radial Offset

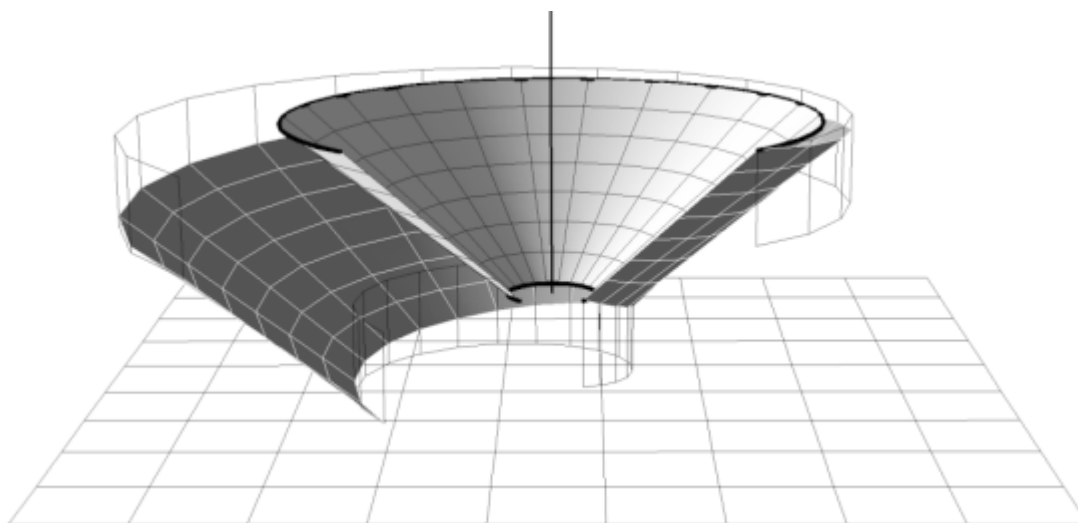
12.1.9.1.5.1. Start/End A

These settings specify a start and end offset along the axis of rotation.

12.1.9.1.5.2. Start/End R

These settings specify a start and end offset for the radius.

The following image shows two partial cones with the same profile and theta limits. For the end profile of one of the cones, the radial offset is positive and the axial offset is negative, causing the radius to increase and the axial coordinate to decrease with increasing theta (as determined by the right hand rule with reference to the axis shown). Two other surfaces of revolution were included in the figure to help illustrate axial displacements.



12.1.9.2. Surface of Revolution: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.1.9.3. Surface of Revolution: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.1.9.4. Surface of Revolution: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.10. Polyline Command

A polyline is a line connecting a series of points. The points may have local (path) variables associated with them. The polyline can interact with CFD data and can be colored using path variables or domain variables.

The following characteristics of polylines will be discussed:

- [Polyline: Geometry Tab \(p. 252\)](#)
- [Polyline: Color Tab \(p. 254\)](#)
- [Polyline: Render Tab \(p. 254\)](#)
- [Polyline: View Tab \(p. 254\)](#)

Note:

There are several ways to insert a polyline:

- From the menu bar, select **Insert > Location > Polyline**.
 - From the toolbar, select **Location > Polyline**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
-

12.1.10.1. Polyline: Geometry Tab

12.1.10.1.1. Method

The Method setting has the following options:

Option	Description
From File	Enables you to specify a file that has the point data contained within it. Set Section Name to the name of the applicable data set within the file. The data file format is described in POLYLINE Data Format (p. 156) .
Boundary Intersection	Enables you to select a boundary and an object to intersect it with. The line will then plot on the intersection.
From Contour	Enables you to plot using contour data (for example, a velocity of 5 m/s).

12.1.10.1.2. File

File is available only if the **From File** option is selected. The **File** setting specifies the filename of a file to insert. You can type in the filename or click *Browse*  to open the **Import** dialog box and search for the file. The only valid file types to import are *.txt and *.csv.

Tip:

This method enables you to read polylines or lines from another case (if that case has the required geometry). First export a polyline or a line from another case, make sure to select **Export Geometry Information**, then use the **From File** method in the other case to import the lines along with any local data. You can also create your own file containing your data, such as experimental data, by using the same format. For a description of the polyline file format, see [POLYLINE Data Format \(p. 156\)](#).

12.1.10.1.3. Domains

Domains is available only if the **Boundary Intersection** option is selected. The **Domains** setting selects a domain for the polyline to exist in. For details, see [Domains \(p. 221\)](#).

12.1.10.1.4. Boundary List

Boundary List is available only if the **Boundary Intersection** option is selected. The

Boundary List setting specifies a boundary. Click the *Location Editor*  icon to open the **Location Selector** dialog box, which displays the complete list of available boundaries.

Note:

When intersecting with a thin surface boundary, the resulting polyline will include both sides of the boundary. To intersect only one side, pick the primitive region that defines one side of the thin surface instead of the entire boundary.

Note:

If a mesh region is selected for the **Boundary List**, the polyline may not be successfully generated if the mesh region is not part of an external boundary.

12.1.10.1.5. Intersect With

Intersect With is available only if the **Boundary Intersection** option is selected. The **Intersect With** setting specifies a graphic object that intersects the boundary.

12.1.10.1.6. Contour Name

Contour Name is available only when the **From Contour** option is selected. The **Contour Name** setting selects a predefined contour plot. If you have not created a contour, see [Contour Command \(p. 264\)](#).

12.1.10.1.7. Contour Level

Contour Level is available only when the From Contour option is selected. The **Contour Level** setting specifies a contour level. The amount of contour levels is predefined by the [Contour Command](#) (p. 264).

12.1.10.2. Polyline: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab](#) (p. 52).

12.1.10.3. Polyline: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab](#) (p. 55).

12.1.10.4. Polyline: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab](#) (p. 60).

12.1.11. User Surface Command

A user surface can be defined in a number of different ways:

- From a file containing data points.
- From the intersection of a boundary and an existing locator.
- From a contour fringe number.
- By transforming an existing surface.
- Offset from an existing surface. The offset can be uniform or described by a variable.

Note:

User Surface locators interpolate values using a method that is slightly less accurate than that used for slice planes and isosurfaces. For details, see [Interpolation in CFD-Post](#) (p. 219).

The following characteristics of user surfaces will be discussed:

- [User Surface: Geometry Tab](#) (p. 255)
- [User Surface: Color Tab](#) (p. 259)
- [User Surface: Render Tab](#) (p. 259)

- [User Surface: View Tab \(p. 259\)](#)

Note:

There are several ways to insert a user surface:

- From the menu bar, select **Insert > Location > User Surface**.
 - From the toolbar, select **Location > User Surface**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
-

12.1.11.1. User Surface: Geometry Tab

12.1.11.1.1. Method

The **Method** setting has the following options:

Option	Description
From File	Same as for the polyline object. For details, see Polyline: Geometry: Method (p. 252) . The data file format is described in USER SURFACE Data Format (p. 156) .
Boundary Intersection	Same as for the polyline object. For details, see Polyline: Geometry: Method (p. 252) .
From Contour	Same as for the polyline object. For details, see Polyline: Geometry: Method (p. 252) .
Transformed Surface	Create a user surface by transforming a preexisting surface. You may specify a rotation, translation, and uniform scale for the user surface.
Offset From Surface	Create a user surface by offsetting it from a preexisting surface. You may specify different methods of offset for the user surface.
ANSYS	Similar to the From File option, except that this option uses Ansys files to load into the instance. You may also specify an associated boundary for the file to be loaded onto. For details, see Specify Associated Boundary Check Box (p. 258) .
From STL File	Similar to the From File option, except that this option uses .stl files to load into the instance. Only ASCII .stl files are supported.

Option	Description
	You can also load STL files from the File menu in CFD-Post. For details, see Importing .stl Files (p. 149) .

Note:

When multiple cases are loaded, user surfaces defined using either the Transformed Surface or Offset From Surface methods exhibit different behavior depending on where the preexisting surface is:

- The preexisting surface exists within domains from only one case.
 - Examples: a slice plane that intersects, or otherwise has been restricted to, the domains of only one case; a boundary that exists (by name) in only one case.
 - Result: The user surface exists in each loaded case, and has the same geometry for all cases.
- The preexisting surface exists within domains of multiple loaded cases.
 - Examples: a slice plane that intersects the domains of multiple cases; a boundary that exists (by name) in multiple cases, and is selected in the list of preexisting surfaces specified by **Surface Name**.
 - Result: The user surface corresponds to the preexisting surface. That is, for every part of the preexisting surface that exists in each case, there is a corresponding part of the user surface that exists in the same case.

12.1.11.1.2. File

File is the same for the polyline object. For details, see [Polyline: Geometry: File \(p. 253\)](#).

Tip:

This method enables you to read surfaces from another case. First export a surface (such as a plane or a boundary) from another case and make sure to select **Export Geometry Information** and **Export Line and Face Data**. Then use the From File method in the other case to import the surface along with any local data. You can also create a file containing your own data, such as experimental data, by using the same format. For a description of the surface file format, see [USER SURFACE Data Format \(p. 156\)](#).

12.1.11.1.3. Domains/Boundary List/Intersect With

These settings are the same as for a polyline, except that instead of outlining the intersection, a line of intersection is formed between the boundaries and the location. Each mesh element that the line passes through forms part of the User Surface. For details, see [Domains \(p. 253\)](#).

12.1.11.1.4. Contour Name/Contour Level

These settings are the same as for a polyline, except that instead of outlining the contour, the User Surface fills in all of the area above the contour level entered and below the contour level above. Also, when applicable, **Contour Level 1** creates a surface below the first contour line. For details, see [Contour Name \(p. 253\)](#).

12.1.11.1.5. Surface Name

Surface Name is available only if either the Transformed Surface or Offset From Surface options are selected. The **Surface Name** setting selects a surface on which to plot the User Surface.

12.1.11.1.6. Rotation Check Box

The **Rotation** check box is available only if the Transformed Surface option is selected. Select the **Rotation** check box to specify a rotation for the User Surface. For details, see [Apply Rotation Check Box \(p. 293\)](#).

12.1.11.1.7. Translation Check Box

The **Translation** check box is available only if the Transformed Surface option is selected. Select the **Translation** check box to specify a translation for the User Surface. For details, see [Apply Translation Check Box \(p. 294\)](#).

12.1.11.1.8. Scale Check Box

The **Scale** check box is available only if the Transformed Surface option is selected. Select the **Scale** check box to specify a scale for the User Surface. Use the **Scale** field to specify a uniform scale factor.

12.1.11.1.9. Type

Type is available only if the Offset From Surface option is selected. The **Type** setting has the following options:

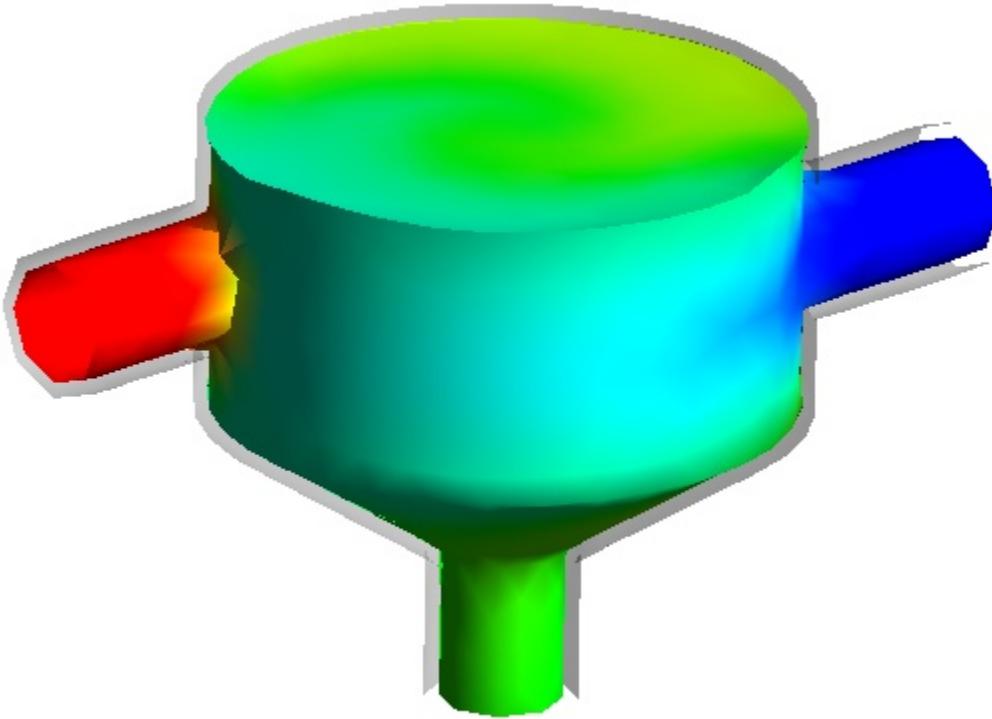
Option	Description
Normal	Enables you to offset the User Surface normal to selected surface.
Translational	Enables you to offset the User Surface from the selected surface by moving the User Surface.

12.1.11.1.10. Mode

Mode is available only if the Offset From Surface option is selected. The **Mode** setting has the following options:

Option	Description
Uniform	Enables you to specify a uniform offset.
Variable	Enables you to select a variable to plot from the surface.

An example of a uniform normal offset of -0.1 [m] to the Default surface of the static mixer, colored by Temperature, is shown in the diagram.



12.1.11.1.11. Distance

Distance is available only if the Uniform option is selected. The **Distance** setting specifies an offset distance, whether it is translational or normal.

12.1.11.1.12. Variable

Variable is available only if the Variable option is selected. The **Variable** setting specifies a variable to plot.

When the distance is described by a variable, you can also incorporate the variable into an expression. For example, after you have chosen a variable you can click in the **Distance** box and amend it with valid CFX Expression Language (CEL) (for example, $0.5 * \text{Temperature}$).

12.1.11.1.13. Direction

Direction is available only if the Translational option is selected. The **Direction** setting selects a direction to offset the User Surface. Increased values do not increase the translational offset, they merely change the ratio that the offset X, Y, and Z directions are placed at. For example, $[2, 3, 1]$ and $[4, 6, 2]$ would identically offset the User Surface.

12.1.11.1.14. Specify Associated Boundary Check Box

The **Specify Associated Boundary** check box is available only if the ANSYS option is selected. This setting is also available in an import menu. For details, see [Import Mechanical CDB Surface](#) (p. 149).

12.1.11.1.15. Length Units

The **Length Units** field is available only if the **From STL File** or **Ansys (.cdb files only)** options are selected. Assigns the chosen length unit to the geometric information in the file. This will overwrite any previous length units specified in the file.

12.1.11.2. User Surface: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.1.11.3. User Surface: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.1.11.4. User Surface: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.1.12. Surface Group Command

A surface group enables you to create a locator consisting of multiple surface locators.

The following characteristics of user surface groups will be discussed:

- [Surface Group: Geometry Tab \(p. 260\)](#)
- [Surface Group: Color Tab \(p. 260\)](#)
- [Surface Group: Render Tab \(p. 260\)](#)
- [Surface Group: View Tab \(p. 260\)](#)

Note:

There are several ways to insert a surface group:

- From the menu bar, select **Insert > Location > Surface Group**.
 - From the toolbar, select **Location > Surface Group**.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
-

12.1.12.1. Surface Group: Geometry Tab

12.1.12.1.1. Domains

The Domains setting selects the domains in which the surface group will exist. For details, see [Domains](#) (p. 221).

12.1.12.1.2. Locations

The **Locations** setting specifies a location or locations on which to plot the Surface Group. For details, see [Locations](#) (p. 224).

12.1.12.2. Surface Group: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab](#) (p. 52).

12.1.12.3. Surface Group: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab](#) (p. 55).

12.1.12.4. Surface Group: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab](#) (p. 60).

12.1.13. Turbo Surface Command

Turbo surfaces are graphic objects that can be viewed and used as locators, just like other graphic objects.

Note:

There are two ways to insert a turbo surface:

- From the menu bar, select **Insert > Location > Turbo Surface**.
- From the toolbar, select **Location > Turbo Surface**.

For details on working with turbo surfaces, see [Turbo Surface](#) (p. 388).

12.1.14. Turbo Line Command

Turbo lines are graphic objects that can be viewed and used as locators, just like other graphic objects.

Note:

There are two ways to insert a turbo line:

- From the menu bar, select **Insert > Location > Turbo Line**.
 - From the toolbar, select **Location > Turbo Line**.
-

For details on working with turbo lines, see [Turbo Line \(p. 391\)](#).

12.2. Vector Command

A **Vector Plot** is a collection of vectors drawn to show the direction and magnitude (optional) of a vector variable on a collection of points. These points, known as seeds, are defined by a location.

When post-processing a GGI simulation, the velocity vectors can be plotted in the local frame of reference for each domain (**Velocity Field Selection**) or in the absolute frame of reference for each domain (**Velocity in a Stationary Frame**). These two choices produce the same plot in all stationary frame domains, but plot either the rotating frame or absolute frame velocity vectors in domains that are in the rotating frame of reference.

The following characteristics of vectors will be discussed:

- [Vector: Geometry Tab \(p. 262\)](#)
 - [Vector: Color Tab \(p. 263\)](#)
 - [Vector: Symbol Tab \(p. 263\)](#)
 - [Vector: Render Tab \(p. 264\)](#)
 - [Vector: View Tab \(p. 264\)](#)
-

Note:

There are several ways to insert a vector plot:

- From the menu bar, select **Insert > Vector**.
 - From the toolbar, click the *Vector*  icon.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the **3D Viewer**.
-

12.2.1. Vector: Geometry Tab

12.2.1.1. Domains

For details, see [Domains](#) (p. 221).

12.2.1.2. Definition

12.2.1.2.1. Locations

Locations is the same for the Point Cloud object. For details, see [Locations](#) (p. 224).

12.2.1.2.2. Sampling

Sampling and all of the settings that correspond to it are the same for the Point Cloud object. For details, see [Sampling](#) (p. 225).

12.2.1.2.3. Variable

The **Variable** setting selects a variable from the list to plot at the selected location.

Tip:

Click the *Location Editor*  icon to open the **Variable Selector** dialog box, which displays the complete list of available variables.

12.2.1.2.4. Hybrid/Conservative Options

For details, see [Hybrid and Conservative Variable Values](#).

12.2.1.2.5. Projection

The **Projection** setting has the following options:

Option	Description
None	Original vectors are plotted without any projection.
Coord Frame	Plots vector components aligned with a principal axis or an axis of a custom coordinate frame.
Normal	Plots vector components normal to the location. Applicable only for surface locations.
Tangential	Plots vector components tangential to the location. Applicable only for surface locations.

When a rotation axis is defined (set in the Turbo tab, or by reading a turbo case), the **Projection** setting has the following additional options:

Option	Description
Axial	Plots vector components along the rotation axis. Available when a rotation axis is defined.
Radial	Plots vector components radially to the rotation axis. Available when a rotation axis is defined.
Circumferential	Plots vector components along the theta direction about the rotation axis. Available when a rotation axis is defined.

12.2.1.2.6. Direction

There are two drop-down list boxes for this setting. The first list represents the options for the range of the vector. The second list box represents the available directions to plot the vector in.

12.2.2. Vector: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.2.3. Vector: Symbol Tab

12.2.3.1. Symbol

The **Symbol** setting has the following options to select a shape for the vector:

Option	Description
Line Arrow	Displays the vector as a line arrow. This option takes the least amount of memory and is suggested for large vector field plots.
Arrow2D	Displays a filled line arrow.
Arrow3D	Displays a 3D filled line arrow.
Arrowhead	Displays the tip of the Arrow2D option.
Arrowhead3D	Displays a 3D version of the Arrowhead option.
Fish3D	Displays a 3D fish.
Ball	Displays a sphere at every vector point. This option does not specify a direction, only a scalar value.
Crosshair	Displays a 3D "+" sign. This option, through its natural shape, displays the normal and the tangential vector to the surface automatically. However, the crosshair does not point to the actual direction (does not have an arrow pointing the direction of the actual vector).
Octahedron	Displays a filled Crosshair option.
Cube	Displays a 3D box. One face of the cube lies tangent to the surface and one of the corners points in the direction of the vector.

12.2.3.2. Symbol Size

The **Symbol Size** setting specifies the scale for the vectors symbol.

12.2.3.3. Normalize Symbols Check Box

Select the **Normalize Symbols** check box to make all of the vectors the same size.

12.2.4. Vector: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.2.5. Vector: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.3. Contour Command

A contour plot is a series of lines linking points with equal values of a given variable. For example, contours of height exist on geographical maps and give an impression of gradient and land shape.

Note:

An alternative to creating a contour plot is coloring an object with contour bands. For details, see [Contour \(p. 54\)](#).

The following characteristics of contours will be discussed:

- [Contour: Geometry Tab \(p. 265\)](#)
- [Contour: Labels Tab \(p. 267\)](#)
- [Contour: Render Tab \(p. 268\)](#)
- [Contour: View Tab \(p. 268\)](#)

Note:

There are several ways to insert a contour plot:

- From the menu bar, select **Insert > Contour**.
 - From the toolbar, click the *Contour*  icon.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the **3D Viewer**.
-

12.3.1. Contour: Geometry Tab

12.3.1.1. Domains

For details, see [Domains](#) (p. 221).

12.3.1.2. Locations

For details, see [Locations](#) (p. 224).

12.3.1.3. Variable

For details, see [Mode: Variable and Use Plot Variable](#) (p. 52).

12.3.1.4. Range

For details, see [Range](#) (p. 53). In addition to the options specified in the link, there is the following option. **Value List** is a comma-separated list that enables you to specify the actual values at which contours should be plotted. For example, if plotting temperature in a combustor, you might try a value list of 300, 500, 700, 900, and 1100K. It should be noted that entering a value list overrides the number specified in the [# of Contours](#) (p. 267) field.

Note:

When dealing with the face values of variables X, Y, and Z, CFD-Post computes the global and local ranges using vertex values whereas Fluent computes the ranges using cell-center values. As a result, the computed ranges may differ between CFD-Post and Fluent.

12.3.1.5. Variable Location: Vertex and Face Options

The **Variable Location** setting controls details about the source of the variable data on which the contour is based. This setting is enabled only when Fluent results are loaded.

The two options are:

- **Vertex**

The variable data is interpolated from mesh vertex values. This is the default setting for all types of results. Note that this option is always in effect for non-Fluent results.

- **Face** (only for postprocessing Fluent results)

The variable data is taken (without interpolation) from one of two sources, depending on availability:

- Wherever face-center values are available on a locator, those values are used for contour creation.

Face-center values are typically available on boundaries and interfaces for Fluent files.

- If face-center values are not available on a locator, then cell-center values of the mesh elements are used for contour creation, provided that such cell-center values are available. For example, for locators that "cut" through mesh elements (for example, planes and isosurfaces), the variable data is taken from the (cell-center) values for the mesh elements that are "cut" by the locator.
- If neither face-center values nor cell-center values are available, CFD-Post issues a warning message and does not generate a contour.

Note:

- Contours created in CFD-Post with the **Face** option are similar to contours created in Fluent with the **Node values** option switched off.

A notable difference between such contours in CFD-Post and Fluent is that Fluent contours always use cell-center values even when face-center values are available. It follows that such contours will be different between CFD-Post and Fluent for locators that have face-center values, typically boundaries and interfaces. On such locators, the contour minimum and maximum values in CFD-Post should match the minimum and maximum face set values in Fluent.

- When dealing with the face values of variables X, Y, and Z, CFD-Post computes the global and local ranges using vertex values whereas Fluent computes the ranges using cell-center values. As a result, the computed ranges may differ between CFD-Post and Fluent.

12.3.1.6. Boundary Data: Hybrid and Conservative Options

The **Boundary Data** setting controls details about the source of the variable data on which the contour is based.

The two options are:

- **Hybrid**

Hybrid variable values are used. Such values exist only in CFX results.

- **Conservative**

Conservative variable values are used. Such values exist only in CFX results.

For details, see [Hybrid and Conservative Variable Values](#).

Note:

In the following cases, CFD-Post makes the **Boundary Data** setting available, even if not all the domains in the **Domains** specification for the contour contain CFX results:

- A domain having CFX results is part of the **Domains** specification.
- In the current session of CFD-Post, the variable has been plotted using hybrid values in a domain that is not selected in **Domains**.

In these cases, if you make a contour with the **Hybrid** option, undefined values are plotted over domains that do not contain hybrid variables.

12.3.1.7. Color Scale

For details, see [Color Scale \(p. 53\)](#).

12.3.1.8. Color Map

For details, see [Color Map \(p. 53\)](#).

12.3.1.9. # of Contours

The **# of Contours** setting specifies the number of contours in the plot. This will not increase the range, it will increase only the number of contours within the range.

12.3.1.10. Clip to Range Check Box

Select the **Clip to Range** check box to plot values only within the specified **Range**. If selected, you should use this setting in conjunction with the **User Specified range**.

12.3.2. Contour: Labels Tab

12.3.2.1. Show Numbers Check Box

Select the **Show Numbers** check box to display numbers for the contour lines and edit their appearance. The contour numbers will appear next to the contour values in the legend.

12.3.2.1.1. Text Height

The **Text Height** setting specifies a value for the text height. The value corresponds to a ratio of the height of the **3D Viewer**. For example, a value of 1 would display the contour numbers to be the full height of the **3D Viewer**.

12.3.2.1.2. Text Font

The **Text Font** setting specifies a font from the list.

12.3.2.1.3. Color Mode

The **Color Mode** setting has the following options:

Option	Description
Default	Displays the text as gray.
User Specified	Enables you to specify a custom color.

12.3.2.1.4. Text Color

The **Text Color** setting selects a custom color. You can select a predefined color by clicking the color bar.

Tip:

Click the *Location Editor*  icon to open the **Select color** dialog box, which displays the complete range of available colors.

12.3.3. Contour: Render Tab

The **Render** tab for a contour does not contain an **Apply Texture** section, but does contain the other sections described in [Render Tab \(p. 55\)](#).

12.3.4. Contour: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.4. Streamline Command

A *streamline* is the path that a particle of zero mass would take through the fluid domain. The path is calculated using a Runge-Kutta method of vector variable integration with variable timestep control. Streamlines start at each node on a given locator.

The assumption of steady-state flow is assumed when a streamline is created, even with a transient simulation. Although the CFD-Post streamline algorithm is efficient, the calculation of large numbers of streamlines in a large domain can still take a long period of time. Therefore, when calculating streamlines for a solution for the first time, start by plotting a small number of streamlines and then increase the number of streamlines until the best generation time vs. detail ratio is found.

Note:

In multi-domain turbo cases, streamlines may not always cross from one domain to next. This can happen when there is no overlap between the two domains, or when the domain interface is not modeled as an interface, but rather as outlet/inlet pair (for example, in Fluent). If you want to view streamlines in both domains in such cases, you can start the streamlines from both "inlets", or from the inlet and the outlet, setting the Direction to be Forward and Backward.

The following characteristics of streamlines will be discussed:

- [Streamline: Geometry Tab \(p. 269\)](#)
- [Streamline: Color Tab \(p. 271\)](#)

- [Streamline: Symbol Tab \(p. 272\)](#)
- [Streamline: Limits Tab \(p. 273\)](#)
- [Streamline: Render Tab \(p. 274\)](#)
- [Streamline: View Tab \(p. 274\)](#)

Note:

There are several ways to insert a streamline:

- From the menu bar, select **Insert > Streamline**.
 - From the toolbar, click the *Streamline*  icon.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the **3D Viewer**.
-

12.4.1. Streamline: Geometry Tab

12.4.1.1. Type

The **Type** setting has the following options:

Option	Description
3D Streamline	Plots the streamline inside a specified domain from a location.
Surface Streamline	Plots the streamline on a surface from a location. A Surface Streamline is defined as a line that is, at a given instant of time, everywhere tangent to the surface projections of the surface values of a vector variable. Note that, in the current version of CFD-Post, a surface streamline cannot cross from one domain to another.

12.4.1.2. Definition

12.4.1.2.1. Domains

Domains is available only if the 3D Streamline option is selected. For details, see [Domains \(p. 221\)](#).

12.4.1.2.2. Start From (3D Streamline)

Start From is available only if the 3D Streamline option is selected. The **Start From** setting selects a location or locations to start from. For details, see [Locations \(p. 224\)](#).

If you are starting your streamlines from an inlet, outlet, or slice plane, you are advised to use the **Factor** field to reduce the number of streamlines. If your solution is likely to contain recirculation areas, or regions of high vorticity, you are advised to reduce the **Max Segments** number

to a few hundred streamlines. If the streamlines stop part of the way through the domain, increase the **Max Segments** value until you receive good results.

12.4.1.2.3. Surfaces

Surfaces is available only if the Surface Streamline option is selected. The **Surfaces** setting selects a location or locations to plot on. For details, see [Locations \(p. 224\)](#).

12.4.1.2.4. Start From (Surface Streamline)

Start From is available only if the Surface Streamline option is selected. This setting is similar to the **Sampling** setting for a Point Cloud object. For details, see [Sampling \(p. 225\)](#). The differences between the settings are that you cannot select the Random option for this setting and that the **Start From** setting also has the Locations option.

Note that, unlike 3D streamlines, surface streamlines cannot pass through multiple domains.

12.4.1.2.5. Locations

Locations is available only if the Locations option is selected. For details, see [Locations \(p. 224\)](#).

12.4.1.2.6. Sampling

Sampling is available only if the 3D Streamline option is selected. The **Sampling** setting is identical to the **Sampling** setting for a Point Cloud object, except that you cannot select the Random option for this setting. For details, see [Sampling \(p. 225\)](#).

12.4.1.2.7. Preview Seeds Button

Click the **Preview Seeds** button to display in the **Viewer** where the streamlines will originate from.

12.4.1.2.8. Variable

Use **Variable** to select a variable to plot. Using the Velocity variable is recommended. For details, see [Variable \(p. 262\)](#).

12.4.1.2.9. Hybrid/Conservative Options

For help on which field to select, see [Hybrid and Conservative Variable Values](#).

12.4.1.2.10. Direction

The **Direction** setting has the following options:

Option	Description
Forward	Specifies that the streamline goes only in the positive direction from the start point.
Backward	Specifies that the streamline goes only in the negative direction from the start point.

Option	Description
Forward and Backward	Specifies that the streamline goes in both the positive and negative directions from the start point.

12.4.1.3. Cross Periodics Check Box

Cross Periodics is available only if the 3D Streamline option is selected. Select the **Cross Periodics** check box to have the streamline cross from one periodic interface to the opposite boundary. A periodic interface can be defined by selecting a periodic option for a domain interface.

Domain interfaces are used for multiple purposes:

- Connecting domains or assemblies

Domain Interfaces are required to connect multiple unmatched meshes within a domain (for example, when there is a hexahedral mesh volume and a tetrahedral mesh volume within a single domain) and to connect separate domains.

- Modeling changes in reference frame between domains

This occurs when you have a stationary and a rotating domain or domains rotating at different rates.

- Creating periodic interfaces between regions

This occurs when you are reducing the size of the computational domain by assuming periodicity in the simulation.

12.4.1.4. Simplify Streamline Geometry Check Box

Simplify Streamline Geometry is available only if the Surface Streamline option is selected. Select the **Simplify Streamline Geometry** check box to interpolate a linear line in between points if the streamline is almost linear. This will have negative effects if you plot a variable on the streamline because the linearly interpolated line will omit the points in between the points that create the line.

12.4.2. Streamline: Color Tab

There are additional **Mode** options for coloring streamlines that are not available for other objects:

- Time

The Time option colors the streamline by the amount of time a massless particle would take to get to each point of the streamline, starting at the location.

- Animated Time

The Animated Time option is intended for use with the **Sweep** animation type of GPU Accelerated Animation (see [GPU Accelerated Animation \(p. 337\)](#)). Using this option, streamlines are colored according to a periodic time variable using special color maps. The resulting animation shows movement along streamlines.

The periodic time variable and other variables computed for the streamline are listed in the **Variables** workspace in the tree under **User Locations and Plots > [name of streamline]**. The details view (accessible by right-clicking the variable and selecting **Edit**, or by double-clicking the variable) shows the variable range.

The initial value of **Time Interval** is -1 [s], which upon creating the streamline is automatically changed to the maximum value of variable **Time on Streamline 1** (for a streamline named **Streamline 1**). Changing the value of **Time Interval** changes how quickly the periodic time cycles (from 0 to 1) along streamlines.

You can change the color map to affect how quickly segments of the streamlines appear to fade during an animation. To improve the smoothness of animations, you might want to reduce **Limits** tab > **Step Tolerance**. Note that, to maintain the overall length of streamlines, you might then also have to increase **Limits** tab > **Upper Limits > Max Segments**.

- Unique

The Unique option gives each streamline a different color along its whole length, and can be used to track individual streamlines through a domain.

For details on the rest of the **Color** tab, see [Color Tab \(p. 52\)](#).

12.4.3. Streamline: Symbol Tab

The **Symbol** tab adds markers to each streamline at given time intervals.

12.4.3.1. Show Symbols Check Box

Select the **Show Symbols** check box to draw symbols at a user-specified time interval along the streamline.

12.4.3.1.1. Min Time

The **Min Time** setting specifies a minimum time to start plotting the symbols. The time value

can also be an expression. To create an expression, click the *Expression*  icon and enter the expression.

12.4.3.1.2. Max Time

The **Max Time** setting specifies a maximum time to stop plotting the symbols. The time value

may also be an expression. To create an expression, click the *Expression*  icon and enter the expression.

12.4.3.1.3. Interval

The **Interval** setting specifies the time interval at which you want to plot the symbols.

12.4.3.1.4. Symbol

The same options are available for the **Symbol** setting for the vector object. For details, see [Symbol](#) (p. 263). The symbols are drawn along the vector for the streamline at the given point.

12.4.3.1.5. Symbol Size

This setting is identical to the **Symbol Size** setting for the vector object. For details, see [Symbol Size](#) (p. 264).

12.4.3.2. Show Streams Check Box

Select the **Show Streams** check box to display the streamline or streamlines.

12.4.3.2.1. Stream Type

The **Stream Type** setting has the following options:

Option	Description
Line	Plots the streamline as a line.
Tube	Plots the streamline in tube shape.
Ribbon	Plots the streamline in a flat tube shape. Ribbons also displays axial rotation of the fluid as it passes through the domain.

12.4.3.2.2. Line Width/Tube Width/Ribbon Width

These settings control the width of the streamline.

12.4.3.2.3. # of Sides

of Sides is available only if the Tube option is selected. The **# of Sides** setting specifies the number of sides to the tube. The minimum number of sides is 3 and the maximum is 20.

12.4.3.2.4. Initial Direction

Initial Direction is available only if the Ribbon option is selected. The **Initial Direction** setting specifies the initial direction of the ribbon streamline.

12.4.4. Streamline: Limits Tab

The **Limits** tab enables modification of the tolerance, segments, and maximum time settings.

12.4.4.1. Step Tolerance

The values for the streamline (location and direction) are calculated at points determined by the step tolerance mode. You can choose to have streamline elements calculated relative to the mesh (grid) or at absolute increments as shown in the table that follows:

12.4.4.1.1. Mode

The **Mode** setting has the following options:

Option	Description
Grid Relative	Specifies that the streamline must lie within the specified fraction of the local grid cell size. Selecting Grid Relative means that the Tolerance is directly proportional to the mesh spacing. In areas where the mesh has been refined (such as areas where the flow pattern changes quickly), the Tolerance setting reduces the distance between streamline points proportionately. This in turn produces more accurate streamlines in these areas.
Absolute	Specifies that the calculation points for streamline elements must lie within the Tolerance distance specified.

12.4.4.1.2. Tolerance

The **Tolerance** setting specifies the accuracy of the path. As the **Tolerance** setting becomes finer, the accuracy increases but the calculation time increases.

12.4.4.2. Upper Limits

12.4.4.2.1. Max Segments

The **Max Segments** setting specifies the maximum number of segments allowed for a streamline before it ends.

12.4.4.2.2. Max Time

The **Max Time** setting specifies the maximum time allowed to pass before the streamline ends. A time of zero, in this case, represents infinite time (because zero would actually plot nothing).

12.4.4.2.3. Max Periods

The **Max Periods** setting is available only if the **Cross Periodics** check box is selected in the **Geometry** tab. This setting sets the number of times a streamline is able to pass through a periodic boundary.

12.4.5. Streamline: Render Tab

The render settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.4.6. Streamline: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.5. Particle Track Command

In complex flows, it is often useful to track the flow of discrete particles through the flow field. These particles interact with the fluid, following a path that is determined by the particle properties, as well as by the mean and turbulent flow behavior. The tracking is useful in two ways:

- Particle tracking can trace the mean flow behavior in and around complex geometries.
- The injection of several particles from a point can help to display the turbulence properties of the flow.

In Ansys Fluent, particle tracking information is contained in a Fluent Particle Track XML file. For details on creating a particle track file in Ansys Fluent, see [Exporting Steady-State Particle History Data in the Fluent User's Guide](#). To learn how to import such a file into a case loaded in CFD-Post, see [Import Fluent Particle Track File \(p. 149\)](#). Once the Fluent Particle Track file has been loaded, a **Reread** button appears in the Particle Track details view. Clicking that button causes the particle track file to be reread and automatically updates any object that has a dependency on that file.

In Ansys CFX, particle tracking information is written to the results file. The parameters are set in the pre-processor. CFD-Post also provides support for track files created in CFX by enabling the import of particle tracking data from a separate file. If a CFX results file contains particle tracking data, an object will exist in the tree view of type `Res Particle Track`.

The following characteristics of particle tracks will be discussed:

- [Particle Track: Geometry Tab \(p. 276\)](#)
- [Particle Track: Color Tab \(p. 278\)](#)
- [Particle Track: Symbol Tab \(p. 278\)](#)
- [Particle Track: Render Tab \(p. 279\)](#)
- [Particle Track: View Tab \(p. 279\)](#)
- [Particle Track: Info Tab \(p. 280\)](#)

Note:

There are several ways to insert a particle track:

- To insert a Fluent particle track, select **File > Import > Import Fluent Particle Track File**.
 - From the menu bar, select **Insert > Particle Track**.
 - From the toolbar, click the *Particle Track*  icon.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
-

12.5.1. Particle Track: Geometry Tab

12.5.1.1. Method

For Ansys CFX cases, the **Method** setting has the following options:

Option	Description
From Res	Creates the particle track from the current .res file. This option is available only with a valid results file.
CFX-4 Tracks From File	Creates the particle track data from the selected file. Note: You cannot load Fluent Particle Track Files using this option; such files can be loaded only via File > Import > Import Fluent Particle Track File.

12.5.1.1.1. Domains

Domains is available only if the From Res option is selected. For details, see [Domains \(p. 221\)](#).

12.5.1.1.2. Material

Material is available only if the From Res option is selected. The **Material** setting selects a material to emulate with the particle track.

12.5.1.2. File

File is available only for Ansys CFX cases. The **File** setting specifies the filename of a file to load.

You may type in the filename or click *Browse*  to open the **Select CFX-4 Particle Track File** dialog box, and search for the file.

12.5.1.3. Injections

For Ansys Fluent cases, the **Injections** setting enables you to filter by injection region.

12.5.1.4. Reduction Type

The **Reduction Type** setting has the following options:

Option	Description
Maximum Number of Tracks	Enables you to set the maximum number of tracks to be plotted.
Reduction Factor	Enables you to specify a reduction factor to decrease the number of tracks to be plotted.

12.5.1.4.1. Reduction

Reduction is available only if the **Reduction Factor** option is selected. This setting is the same as **Factor** for the Point Cloud object. For details, see [Factor \(p. 226\)](#).

12.5.1.4.2. Max Tracks

Max Tracks is available only if the **Maximum Number of Tracks** option is selected. The **Max Tracks** setting specifies the maximum number of tracks to be plotted.

12.5.1.5. Limits Option

The **Limits Option** setting has the following options:

Option	Description
Up to Current Timestep	Plots the track values up to the current timestep only.
Since Last Timestep	Plots the track values from the previous timestep to the current timestep.
User Specified.	Enables you to specify a beginning and ending time/distance.

Note:

For transient cases, CFD-Post uses transient timesteps (as shown in the timestep selector) to determine **Since Last Timestep** values; CFD-Post does *not* limit timesteps by solver timesteps, or by what is written in the particle tracking XML file as separate times. Therefore, if **Since Last Timestep** is selected, the tracks will be limited to the values between the previous timestep (from that selected in the Timestep Selector) and the currently selected timestep. If this is a steady-state case, this option will not limit the tracks in any way.

12.5.1.5.1. Limit Type and Start/End <variable>

Limit Type and **Start/End <variable>** are available only if the **User Specified** option is selected. The **Limit Type** setting specifies the limiting variable for the plot, and **Start/End <limiter>** specify a start and end value for the selected limiter.

- For CFX cases, **Limit Type** can be either **Time** or **Distance**.
- For Fluent cases, **Limit Type** can be either **Time** or **Diameter**.

12.5.1.6. Filter Check Box

Select the **Filter** check box to specify filters. The settings included are **Start Region**, **End Region**, **Diameter**, **Track**, and the **Match ALL/Match ANY** options.

12.5.1.6.1. Start/End Region Check Boxes

These settings are available only if the **From Res** option is selected. Select the **Start/End Region** check boxes to filter out the tracks that do not start or end in the selected region.

12.5.1.6.2. Diameter Check Box

Select the **Diameter** check box and set the corresponding text and list boxes to place restrictions on particles at the injection location.

12.5.1.6.3. Track Check Box

Select the **Track** check box and enter numbers corresponding to tracks to display indicated tracks by entering a comma-delimited list of track numbers. You may also enter a range of track numbers. For example:

- -5 specifies tracks 1 to 5
- 40- specifies track numbers above 40
- 10-100 specifies tracks 10 to 100.

You may view the **Info** tab to view the **Total Tracks** and **Tracks Shown**. For details, see [Particle Track: Info Tab \(p. 280\)](#).

12.5.1.6.4. Match ALL/Match ANY Options

Select **Match All** to display only the tracks that meet all of the specified **Filter** conditions. Selecting **Match Any** draws all tracks that meet one or more of the selected conditions.

12.5.2. Particle Track: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

12.5.3. Particle Track: Symbol Tab

12.5.3.1. Show Symbols Check Box

This setting and its options are similar to those for the **Show Symbols** check box for the streamline object, as described in [Show Symbols Check Box \(p. 272\)](#). The differences are that a particle track has a **Max Time** setting and different symbol size options.

The **Size Option** choices are **Constant** and **Particle Diameter**:

- When **Size Option** is **Constant**, the symbol size is constant for all particles. The particle size displayed is a mean particle diameter size multiplied by the value you set with the **Scale** setting.
- When **Size Option** is **Particle Diameter**, the **Scale Type** can be **Absolute** or **Relative**.
 - When **Scale Type** is **Absolute**, the particle size displayed is a mean particle diameter size multiplied by the value you set with the **Scale** setting.

- When **Scale Type** is **Relative**, symbol sizes are scaled by the domain.

Note:

Enabling symbols for single-point tracks will cause CFD-Post to show symbols for each point that exists between the **Min. Time** and **Max. Time**. The interval setting has no effect because there is no interval to show. This behavior is different than for tracks that have segments, because tracks without segments are not visible unless a symbol is drawn on them.

For particle tracks that have no lines for the given set of tracks, CFD-Post evaluates each point for the tracks against the **Min. Time** and **Max. Time** and, if the time on that vertex falls between those times, a symbol is drawn. Therefore, track symbols will exist on track vertices that have differing times (that is, it will not just be a symbol at 1.0 s for all tracks; rather, it will be a symbol for all track points with a time between 0.5 s and 1.0 s).

12.5.3.1.1. Max Time is

The **Max Time is** setting has the following options:

Option	Description
User Specified	Enables you to enter a custom value for the maximum time value. This is the default for steady-state simulations.
Current Time	Uses the current timestep as the maximum time value. This is the default for transient simulations.

12.5.3.2. Show Tracks Check Box

The settings for this check box are the same as for the **Show Streams** check box for the streamline object. For details, see [Show Streams Check Box \(p. 273\)](#).

12.5.3.3. Show Track Numbers Check Box

Select the **Show Track Numbers** check box to display and edit the appearance of track numbers. The track numbers will be displayed at the beginning of each numbered track. This setting and its options are similar to the **Show Numbers** check box for the contour object. For details, see [Show Numbers Check Box \(p. 267\)](#).

12.5.4. Particle Track: Render Tab

The render settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.5.5. Particle Track: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.5.6. Particle Track: Info Tab

The **Info** tab displays information about the current state of the particle tracks. Note that the **File** field is read-only for Fluent particle track cases.

Note that CFD-Post shows the tracks displayed after reduction and filtering. **Track Limiting** is just a way to show parts of a track that are displayed (after reduction and filtering); if a track happens not to be visible during the **Track Limiting** range, then nothing will be shown for that track, but the number of tracks and the track index range is always reported for all tracks.

Tip:

If you want to see the track numbers for that particle tracks that are visible within the limits, you can turn on **Show Track Numbers** in the **Symbol** tab.

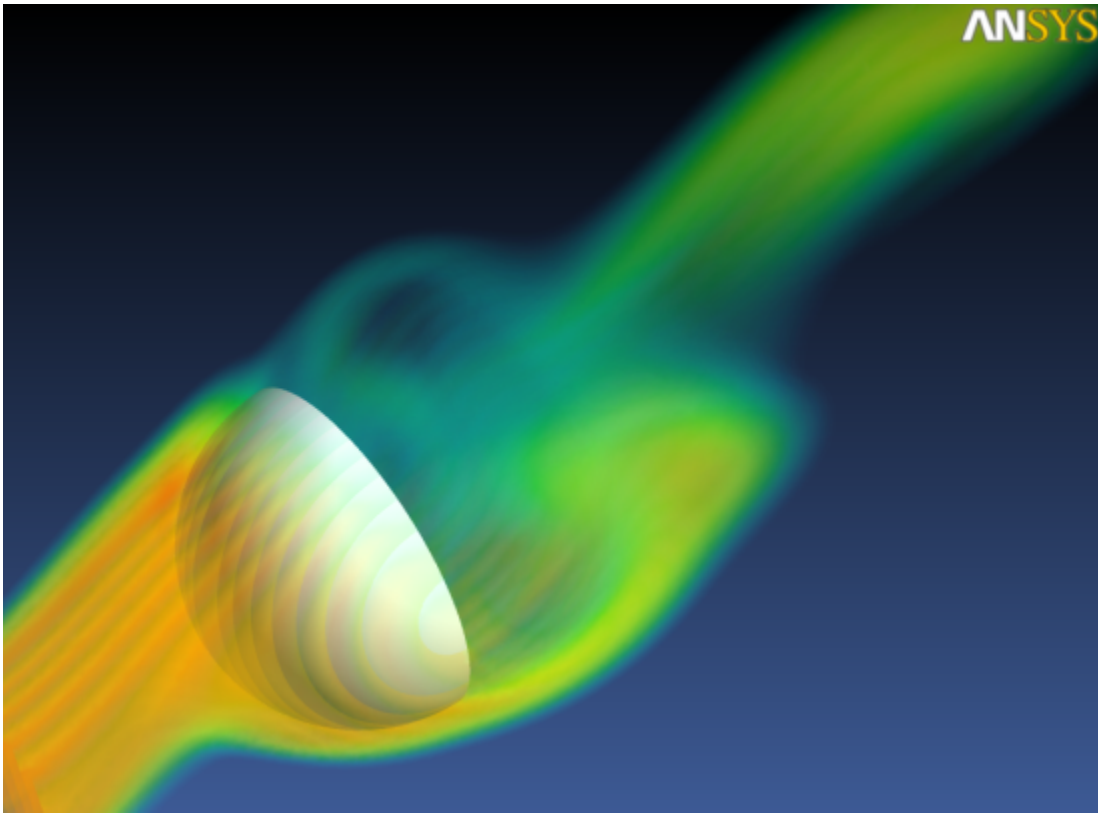
If you have changed settings on another tab menu, you must click **Apply** before the information is updated.

12.6. Volume Rendering Command

The Volume Rendering feature enables you to visualize field variables throughout the entire domain by varying the transparency and color of the plot as a function of the variable value. For example, you can make realistic images of smoke and analyze how it spreads and how the smoke affects visibility.

Tip:

A gradient background in the viewer can interfere with the interpretation of volume rendering results. You may find it useful to set the viewer background to solid white so that the gradients of the solution itself are easier to see. You can set the viewer background with the **Tools > Options > CFD-Post > Viewer > Background: Color Type** option.



The following characteristics of volume rendering will be discussed:

- [Volume Rendering: Geometry Tab \(p. 281\)](#)
- [Volume Rendering: Color Tab \(p. 282\)](#)
- [Volume Rendering: Render Tab \(p. 282\)](#)
- [Volume Rendering: View Tab \(p. 283\)](#)

12.6.1. Volume Rendering: Geometry Tab

The **Geometry** tab has the following settings:

Domains

Where the Volume Rendering will be calculated.

Variable

The name of the variable to be used in determining the transparency of the object.

Range

Range enables you to plot using the global, local, or a user specified range of a variable. The range affects the variation of transparency used when plotting the object in the viewer.

- The **Global** range option uses the variable values from the results in all domains (regardless of the domains selected on the **Geometry** tab) and all time steps (when applicable) to determine the transparent and opaque values.
- The **Local** range option uses only the variable values on the current object at the current time step to set the transparent and opaque range values. This option is useful to use the full transparency range on an object.
- The **User Specified** range option enables you to specify your own transparent and opaque values. You can use this to concentrate the full transparency range into a specific variable range.

Boundary Data

You can specify the use of **Hybrid** or **Conservative** values. For help on the use of Hybrid or Conservative values, see [Hybrid and Conservative Variable Values in the CFX Reference Guide](#).

Transparency Scale

Sets calculation of the object transparency with either a Linear or Logarithmic scale. By default, the scale is Linear.

Transparency Map

The name of the color map to use when rendering the transparency of the object.

Note:

You can create your own transparency map by clicking the map editor icon . It is possible to invert the transparency gradient (making larger data values more transparent), but using an inverted transparency gradient with a user-specified range of values may cause "holes" to appear in the plot.

Resolution

The number of plane cuts per axis. The larger the number of plane cuts, the finer the resolution in the volume rendering.

Transparency Factor

The factor applied to the overall transparency of the plot. Values can range from 0 (fully opaque) to 1 (fully transparent).

12.6.2. Volume Rendering: Color Tab

The color settings can be changed by clicking the **Color** tab. For details, see [Color Tab \(p. 52\)](#). Note that the transparency of the selected color map is disregarded; only the colors of the selected color map are used.

12.6.3. Volume Rendering: Render Tab

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

12.6.4. Volume Rendering: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

Important:

Volume rendering data displays properly for translational and reflective instances, but not for rotational instances.

For details on changing the view settings, see [View Tab \(p. 60\)](#).

12.7. Text Command

Text can be added to the viewer for titles, annotations, or comments in CFD-Post.

The following characteristics of text will be discussed:

- [Text: Definition Tab \(p. 283\)](#)
- [Text: Location Tab \(p. 285\)](#)
- [Text: Appearance Tab \(p. 285\)](#)

Note:

There are several ways to insert a text object:

- From the menu bar, select **Insert > Object > Text**.
- From the toolbar, click the **Text**  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

12.7.1. Text: Definition Tab

12.7.1.1. Text String

The **Text String** setting enters text for the object. When <aa> appears, auto-annotation will be embedded there.

12.7.1.2. Embed Auto Annotation Check Box

Select the **Embed Auto Annotation** check box to insert auto-annotation into the text string.

Note:

In GPU Accelerated Animations, text objects are not updated, including those with embedded auto-annotation.

12.7.1.2.1. Type

The **Type** setting has the following options:

Option	Description
Expression	Adds an expression, selected from a list, to the text string.
Timestep	Adds the current timestep to the text string.
Time Value	Adds the current time value to the text string.
Filename	Adds the filename or the entire pathname to the text string.
File Date	Adds the date that the file was created to the text string.
File Time	Adds the time that the file was created to the text string.
Crank Angle	Adds the crank angle associated with the current timestep. This is only applicable to internal combustion engine cases.

12.7.1.2.2. Expression

Expression is available only if the Expression option is selected. The **Expression** setting specifies an expression to enter into the text string.

12.7.1.2.3. Format (for Filename option)

Format is available only if the Filename option is selected. The **Format** setting selects either Entire Path or Filename Only to insert into the text string.

12.7.1.2.4. Format (for the File Date and File Time options)

Format is available if either the File Date or File Time options are selected. The **Format** setting selects a time format to enter into the text string.

12.7.1.2.5. Determine the number formatting automatically Check Box

You can allow CFD-Post to define an appropriate format for the display of floating point numbers in certain annotations (Crank Angle, Expression, and Time Value) or choose the format yourself:

- If you select **Determine the number formatting automatically**, CFD-Post will change the formatting to the one that best suits the data being plotted.
- If you clear **Determine the number formatting automatically**, you can choose between scientific notation and fixed notation, and set the amount of precision.

12.7.1.3. More/Fewer Buttons

Click the **More** button to create another line of text. Click the **Fewer** button to remove these added lines of text.

12.7.2. Text: Location Tab

12.7.2.1. Location

12.7.2.1.1. Position Mode

The **Position Mode** setting has the following options:

Option	Description
Two Coords	Specifies the text to sit in the viewer in the 2D plane.
Three Coords	Specifies the text to be fixed to one point in the viewer and rotate with that point when the view is rotated.

12.7.2.1.2. X Justification

This setting is available only if the Two Coords option is selected and is the same for the Legend object. For details, see [X Justification](#) (p. 290).

12.7.2.1.3. Y Justification

This setting is available only if the Two Coords option is selected and is the same for the Legend object. For details, see [Y Justification](#) (p. 290).

12.7.2.1.4. Position (for Two Coords option)

Position is available only if the Two Coords option is selected. The **Position** setting specifies a fixed 2D point at which the text will be displayed.

12.7.2.1.5. Position (for Three Coords option)

Position is available only if the Three Coords option is selected. The **Position** setting specifies a 3D point at which the text will be displayed.

12.7.2.1.6. Rotation

The **Rotation** setting specifies a rotation for the text about the bottom-left corner of text in a counterclockwise direction. When the **X/Y Justification** is set to Center, the object rotates about the center point of the text.

12.7.3. Text: Appearance Tab

12.7.3.1. Height

The **Height** setting specifies a text height. The value is equivalent to a fraction of the viewer size.

12.7.3.2. Color Mode

This setting and its corresponding settings are the same for the Contour object. For details, see [Color Mode \(p. 267\)](#).

12.7.3.3. Font

The **Font** setting specifies a font type for the text from a list.

12.8. Coordinate Frame Command

In CFD-Post it may be necessary to define a new coordinate frame for certain quantitative operations, which are described in [Function Calculator \(p. 347\)](#).

Note:

- There are several ways to insert a coordinate frame:
 - From the menu bar, select **Insert > Coordinate Frame**.
 - From the toolbar, click the *Coordinate Frame*  icon.
 - Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.
- You cannot use a user-defined coordinate frame as part of a general CEL expression. For example, `radius = sqrt(x_myAxis^2 + y_myAxis^2)` is not valid.

For information on how to define a coordinate frame, see [Coordinate Frame Details \(p. 287\)](#).

12.8.1. Coordinate Frame: Definition Tab

12.8.1.1. Type

The **Type** setting is always set to Cartesian.

12.8.1.2. Origin

The **Origin** setting specifies 3D coordinates corresponding to the location of the new Coordinate Frame.

12.8.1.3. Z Axis Point

The **Z Axis Point** setting specifies a point on the Z axis from the origin.

12.8.1.4. X-Z Plane Pt

The **X-Z Plane Pt** setting specifies a point in the XZ plane used to define the positive X axis direction.

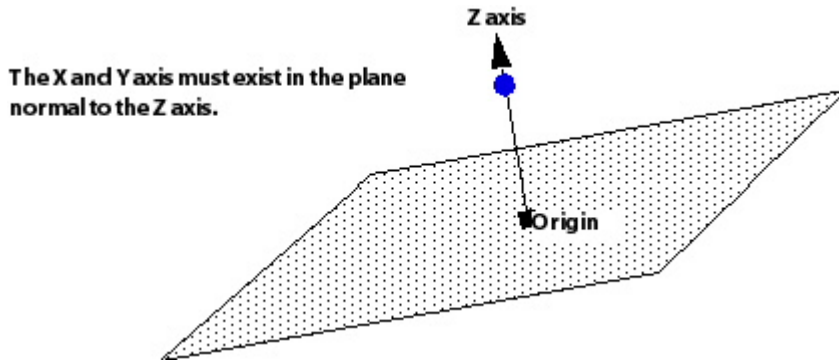
12.8.1.5. Symbol Size

The **Symbol Size** setting scales the size of the coordinate frame being edited.

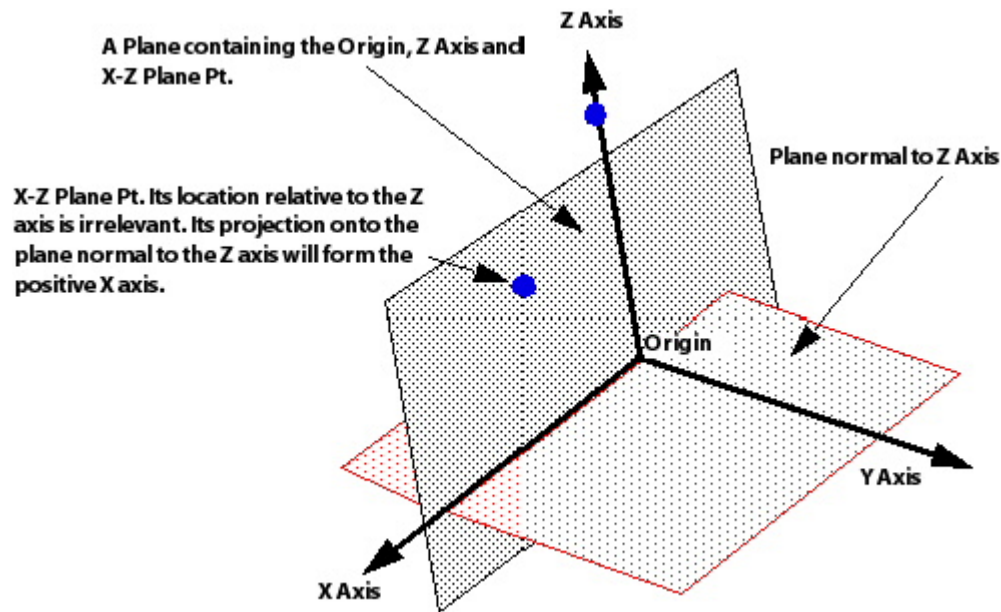
12.8.1.6. Coordinate Frame Details

A coordinate frame is created by specifying three points. It is important to understand how these three points are used to create a coordinate frame.

The first point is the origin for the new coordinate frame (labelled **Origin** in the **Definition** tab). The second point is used to create a Z axis in the new frame. A vector is calculated from the **Origin** to the point defined in the **Z Axis Point** box and used as the third axis of the new coordinate frame. The plane normal to the Z axis is now set and contains both the X and Y axes.



A third point entered into the **X-Z Plane Pt** box is needed to define the location of the X and Y axis in the plane normal to the Z axis. The **X-Z Plane Pt** point, along with the two points already specified, define a plane that lies in the X-Z plane (see diagram below). Because the X axis must now lie in both the X-Z plane and the plane normal to the Z axis, its location must be the line of intersection between the two planes. The positive direction for the X axis is the same side as the **X-Z Plane Pt** point lies with respect to the Z axis.



Finally, because the Y axis must be perpendicular to both the X Axis and the Z Axis, its positive direction is determined by the right-hand rule.

If **X-Z Plane Pt** is specified such that it lies on Axis 3, an error is displayed. The projection of the **X-Z Plane Pt** onto the plane normal to the Z axis would be on the origin and does not give enough information to define the X axis.

12.9. Legend Command

Default and user-defined legends can be plotted in the viewer to show the mapping between colors and quantities for plots that are colored by variable values.

The following characteristics of legends will be discussed:

- [Default Legends \(p. 289\)](#)
- [User-defined Legends \(p. 289\)](#)
- [Legend: Definition Tab \(p. 289\)](#)
- [Legend: Appearance Tab \(p. 291\)](#)

Note:

There are several ways to insert a legend:

- From the menu bar, select **Insert > Legend**.
- From the toolbar, click the *Legend*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

12.9.1. Default Legends

Each view/figure has a **Default Legend** object that appears whenever an eligible plot is created or updated. As further objects are added to, or updated in, a viewport, the **Default Legend** updates to show the variable values for the latest plot.

Only the default legend for the selected view/figure is shown in the tree view. The other default legends continue to exist, even when not displayed in the tree view.

12.9.2. User-defined Legends

To create a user-defined legend, select **Insert > Legend**.

12.9.3. Legend: Definition Tab

12.9.3.1. Plot

The **Plot** setting is available only when creating or modifying a user-defined legend (not the default legends). Select from a list of objects for the legend to act on.

12.9.3.2. Title Mode

The **Title Mode** setting has the following options:

Option	Description
No Title	Omits the title.
Variable	Sets the title to the name of the variable mapped by the legend.
Variable and Location	Variable and Location is the same as Variable except that the name of the locator is appended to the title.
User Specified	Enables you to specify a custom title.

12.9.3.3. Title

The **Title** field is available only after the **User Specified** option has been selected. This setting enters a custom title.

12.9.3.4. Show Legend Units Check Box

Clearing the **Show Legend Units** check box will hide the legend units. By default, the check box is selected, and so units are displayed.

Note:

The legend will always display Temperature in absolute units: if C or K are selected as temperature units, the legend's data will be displayed in K; if F or R are selected, the legend's data will be displayed in R. For details, see [Function Calculator \(p. 347\)](#).

12.9.3.5. Vertical / Horizontal Options

Selecting **Vertical** or **Horizontal** will display the legend vertically or horizontally in the viewer.

12.9.3.6. Location

12.9.3.6.1. X Justification

The **X Justification** setting has the following options:

Option	Description
None	Enables you to specify a custom X location using the Position fields.
Left	Places the legend on the left side of the viewer.
Center	Places the legend in the center of the viewer.
Right	Places the legend on the right side of the viewer.

12.9.3.6.2. Y Justification

The **Y Justification** setting has the following options:

Option	Description
None	Enables you to specify a custom Y location using the Position fields.
Top	Places the legend at the top of the viewer.
Center	Places the legend in the center of the viewer.
Bottom	Places the legend at the bottom of the viewer.

12.9.3.6.3. Position

The **Position** fields specify a custom point at which to position the legend. This setting is available after the **None** option is selected for the **X** and/or **Y Justification** settings. The values entered are fractions of the screen width/height for x and y respectively. For example, 0.2 for the X value would place the legend 1/5 across the screen from the left. A value of 0.2 for the Y direction would place the legend 1/5 up from the bottom of the viewer. The placement uses the bottom left corner of the legend as a reference.

12.9.4. Legend: Appearance Tab

12.9.4.1. Sizing Parameters

12.9.4.1.1. Size

The **Size** setting scales the legend height to a fraction of the viewer height.

12.9.4.1.2. Aspect

The **Aspect** setting specifies the width of the color range bar.

12.9.4.2. Text Parameters

12.9.4.2.1. Precision

The **Precision** setting specifies the number of significant digits after the decimal place that the legend can hold. You may also choose to display the numbers in a Fixed or Scientific format.

12.9.4.2.2. Value Ticks

The **Value Ticks** setting holds the number of intervals that you want shown by the legend.

Note:

The **Value Ticks** setting is not applicable for a legend that relates to a contour plot.

12.9.4.2.3. Font

The **Font** setting specifies a font for the interval labels.

12.9.4.2.4. Color Mode

The **Color Mode** setting specifies whether to use a User Specified color or the Default color for the title and interval labels.

12.9.4.2.5. Color

When **Color Mode** is set to User Specified, the **Color** setting specifies a color for the title and interval labels. You can click the color bar to browse through predefined colors, or click the

Color Selector  icon and select a color from the **Select color** dialog box.

12.9.4.2.6. Text Rotation

The **Text Rotation** setting specifies a value in degrees to rotate the text at (in a counterclockwise direction from horizontal).

12.9.4.2.7. Text Height

The **Text Height** setting specifies a value corresponding to the text height of the legend relative to the viewer size. You may enter a value between 0.005 and 0.1.

Note:

When using a legend as the basis for quantitative analysis, you should ensure that lighting is turned off for any objects colored by a variable. This will give you exact matches between object colors and legend colors.

12.10. Instance Transform Command

Instance Transforms are used to specify how an object should be drawn multiple times. CFD-Post can create Instance Transforms using rotation, translation, and reflection. For example, if you have a mesh that contains one blade from a blade row that contains 51 blades, you would set **Number of Passages** to 51. You could then choose to display any number of blades by setting **Number of Graphical Instances**.

To apply an Instance Transform to an object, select the **Apply Instancing Transform** check box on the **View** tab for the object and select the transform from a list.

The following characteristics of instance transforms will be discussed:

- [Default Transform Object \(p. 292\)](#)
- [Instance Transform: Definition Tab \(p. 293\)](#)
- [Instance Transform: Example \(p. 295\)](#)

Note:

There are several ways to insert an instance transform:

- From the menu bar, select **Insert > Instance Transform**.
- From the toolbar, click the *Instance Transform*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the **3D Viewer**.

12.10.1. Default Transform Object

By default, an Instance Transform called **Default Transform** (which is set to apply no instancing by default) is applied to all objects where Instance Transforms are possible. As a result, editing the definition of **Default Transform** will cause all plots and objects to be transformed (unless you modify the View properties for a particular object). An example is available for applying Instance Transforms. For details, see [Instance Transform: Example \(p. 295\)](#). Note that instancing is purely geo-

metric (in the **Viewer**). This means that quantitative calculations are carried out for the original geometry.

12.10.2. Instance Transform: Definition Tab

The **Definition** tab for an Instance Transform object is similar to the **Instancing** tab for a domain (see [Instancing Tab \(p. 62\)](#)) and the **Instancing** tab for a turbo component (see [Instancing Tab \(p. 387\)](#)). (The **Definition** tab for an Instance Transform object is different in that its **Axis Definition** settings and **Instance Definition** settings cannot be set from a results file.)

12.10.2.1. Instancing Info From Domain Check Box

Clear the **Instancing Info From Domain** check box to enable creating a custom Instance Transform object. Selecting **Instancing Info From Domain** will ignore the application of the Instance Transform inside the domain.

12.10.2.2. Number of Graphical Instances

The **Number of Graphical Instances** setting specifies the number of copies to be made of the object when it is transformed.

If the Instance Transform object is using more than one of the following check boxes, (**Apply Rotation**, **Apply Translation**, and **Apply Reflection/Mirroring**) the order in which each segments are applied are rotation, translation, then reflection.

12.10.2.3. Apply Rotation Check Box

Select the **Apply Rotation** check box if you want to apply a rotation.

12.10.2.3.1. Method

The **Method** setting has the following options:

Option	Description
Principal Axis	Rotates about a principal axis.
Rotation Axis	Rotates about a user-specified axis.

12.10.2.3.2. Axis

Axis is available only if the **Principal Axis** option is selected. The **Axis** setting specifies a principal axis to rotate about.

12.10.2.3.3. From/To Fields

These settings are available only if the **Rotation Axis** option is selected. These settings create an axis of rotation.

12.10.2.3.4. Full Circle Check Box

Select the **Full Circle** check box to uniformly distribute the copies around 360 degrees of rotation.

12.10.2.3.5. Determine Angle From

The **Determine Angle From** setting has the following options:

Option	Description
Instances in 360	Splits 360 degrees into the amount of passages entered and places a copy at each passage, if possible.
Value	Evenly distributes copies from zero to the specified angle.

12.10.2.3.6. Number of Passages

Number of Passages is available only if the Instances in 360 option is selected. The **Number of Passages** setting specifies a value for the number of passages in 360 degrees.

12.10.2.3.7. Passages per Component

Passages per Component is available only if the Instances in 360 option is selected. The **Passages per Component** setting specifies a value for the number of passages per component.

12.10.2.3.8. Angle

Angle is available only if the Value option is selected. The **Angle** setting specifies the rotational angle.

12.10.2.4. Apply Translation Check Box

Select the **Apply Translation** check box if you want to specify a translation.

12.10.2.4.1. Translation

The **Translation** setting specifies a 3D translation.

12.10.2.5. Apply Reflection Check Box

Select the **Apply Reflection** check box to select a reflection method and direction.

Tip:

A quick way to define a reflection for your case is to right-click the Wireframe near the reflection plane and select **Reflect/Mirror**.

12.10.2.5.1. Method

The **Method** setting has the following options:

Option	Description
YZ Plane	Specifies a reflection about the YZ plane.
ZX Plane	Specifies a reflection about the ZX plane.
XY Plane	Specifies a reflection about the XY plane.
From Plane	Specifies a reflection about a user-specified plane.

12.10.2.5.2. X/Y/Z

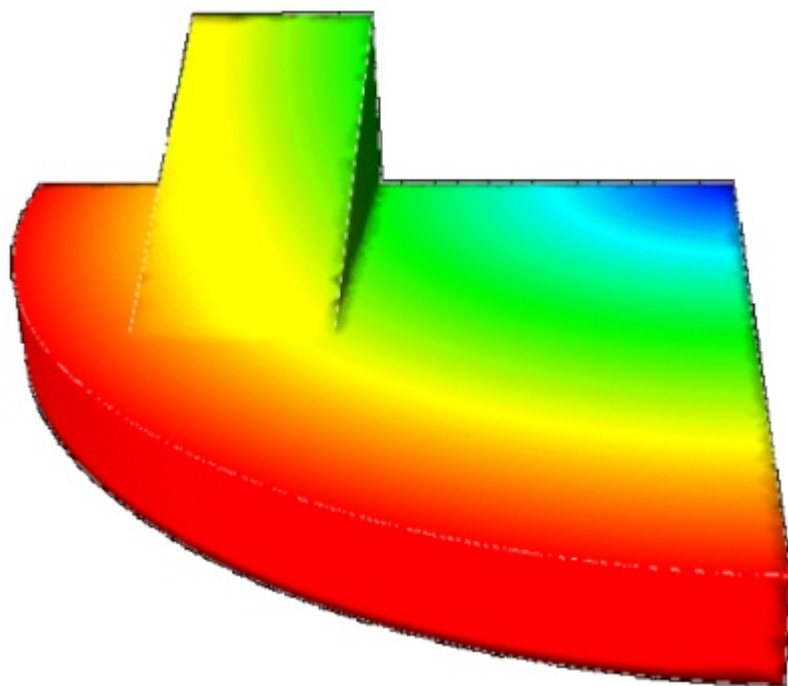
These settings are available only if one of the principal plane options are selected. These settings specify the distance along the normal axis to the plane to reflect by.

12.10.2.5.3. Plane

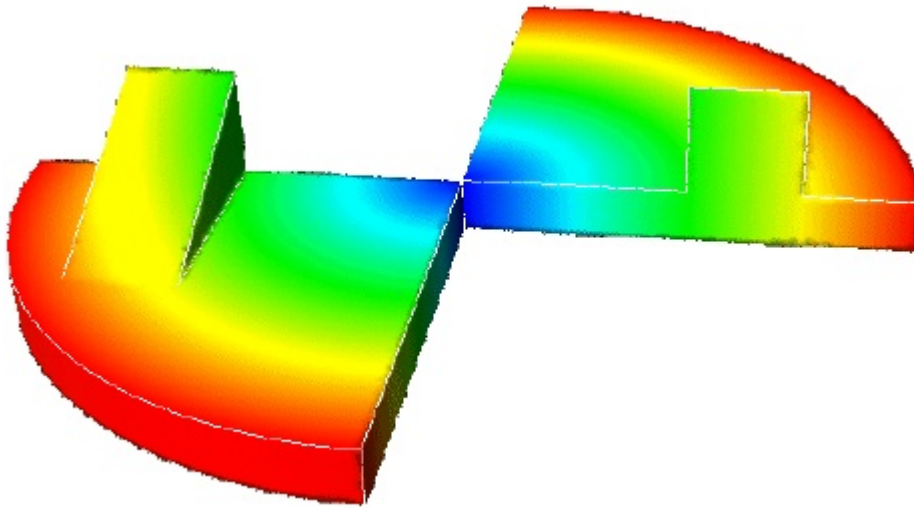
Plane is available only if the From Plane option is selected. The **Plane** setting specifies a plane from the list.

12.10.3. Instance Transform: Example

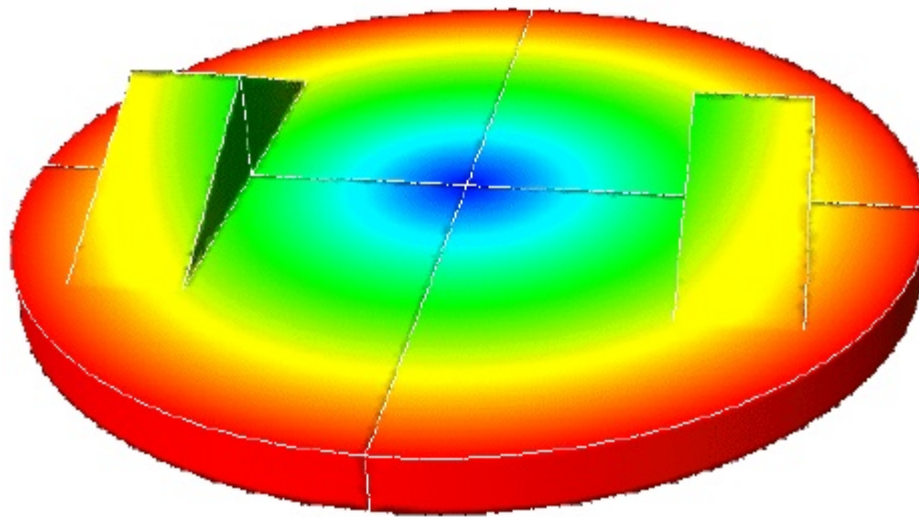
The following example shows how coupling of rotation and reflection instancing can be used to simulate reflection in two planes using a random geometry.



The axis of rotation is defined using the **Rotation Axis** feature on the **Rotation** section of the **Instance Transform** tab. An axis parallel to the z-axis was set. Rotation only was applied initially. An angle of 180 degrees was implemented.



The next step involves creating an XY plane (called PPlane 1) at $X = -1$ and $Z = 1$. For details, see [Plane Command](#) (p. 229). After clicking to expand the Reflection/Mirroring submenu, reflection is applied on PPlane 1.



12.11. Clip Plane Command

A clip plane enables you to define a plane that hides all objects displayed in the viewer that lie to one side of the plane. For example, you could use an XY plane and clip it at $Z = 1$ so that objects are visible only where Z is less than or equal to 1 (or greater than or equal to 1 if the **Flip Normal** check box is selected).

A clip plane will act on all objects in the viewer, including the Wireframe, but will not affect other functions such as calculations (that is, a calculation will still use the entire location, whether visible or not).

Note:

There are several ways to insert a clip plane:

- From the menu bar, select **Insert > Clip Plane**.
- From the toolbar, click the *Clip Plane*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

12.11.1. Clip Plane: Geometry Tab

12.11.1.1. Definition

12.11.1.1.1. Method

Method has the same options and settings as for the **Plane** object, except for the **From Slice Plane** option. For details, see [Method \(p. 229\)](#). **From Slice Plane** enables you to select a predefined slice plane.

12.11.1.1.2. Slice Plane

Slice Plane is available only if the **From Slice Plane** option is selected. The **Slice Plane** setting selects a plane to clip by.

12.11.1.2. Flip Normal Check Box

Select the **Flip Normal** check box to cut all objects in the negative normal direction. If the check box is cleared, the Clip Plane cuts all objects in the positive normal direction.

Note:

To enable/disable Clip Planes, you must use a Viewer shortcut menu command. For details, see [CFD-Post 3D Viewer Shortcut Menus \(p. 121\)](#).

12.12. Color Map Command

To access the Color Map editor, from the menu bar select **Insert > Color Map**.

Note:

The Transparency editor is similar to the Color Map editor, except that it controls the density of a color, rather than the color itself. It is currently used only in conjunction with Volume Rendering objects.

You can apply a color to the opaque color point to better visualize the transparency gradient in the Preview area of the Transparency editor, but the color will not be used in the Viewer.

To access the Transparency editor, click the icon on the **Volume Rendering** editor.

The color map editor has the following controls:

Color Map Style

The **Color Map Style** controls whether the Color Map is a **Gradient**, which forms continuous bands of colors between any number of "color points" that you set, or **Zebra**, which forms bands between only two color points, using a number of divisions that you set with the **divisions** counter.

In gradient mode, all Color Map controls other than **divisions** are enabled; in zebra mode, **Insert**, **Delete**, and **Distribute** are all disabled and the **Position** indicator is read-only (and reflects the setting in the **divisions** indicator).

Preview

The **Preview** both shows the results of your edits and enables you to modify your color points. One color point will always be longer than the others; this indicates the color point that you can drag with the mouse or modify with the controls in the **Color Point Properties** area: the **Color** definition bar, the **Transparency** slider, and (in gradient mode) the **Position** indicator.

You can navigate from one color point to the next by:

- Clicking a color point
- Clicking **Next** or **Previous**.

In gradient mode, you can insert a new color point by:

- Clicking **Insert** to add a color point mid way between the current color point and its neighbor.
- Clicking on the **Preview** bar to insert the color point and, if necessary, adjusting its location by typing a value in the **Position** field.

Color

The **Color** control enables you to change the color of the active color point. When you click the color field, it cycles through ten preset colors. To define any color, click the *Color selector*  icon to the right of the **Color** option and select one of the available colors.

Transparency

The **Transparency** slider enables you to control how opaque each color is.

Insert, Delete, Next, Previous, Distribute

The color point buttons control the number of color points, which color point is active, and the distribution of color points. In Zebra mode, only the buttons that control the active color point are enabled.

Symbol	Option	Description
	Insert	Add a color point mid way between the current color point and its neighbor
	Delete	Delete the selected color point
	Previous	Select the previous color point
	Next	Select the next color point
	Distribute	Evenly distribute the defined color points

Make available in other cases, Set as default

These settings control where the color map is stored; unless you specify otherwise, the color map you define will be available only with the current file.

If you select **Make available in other cases**, the color map will be stored in your preferences file when you click **Apply**.

If you select **Set as default**, when you click **Apply** the color map will be stored in your preferences file and will be the default color map for all future objects in all future files. For this reason **Make available in other cases** will also be selected automatically.

Note:

The default CFD-Post color map is not the same as the default Fluent color map. To use the default Fluent color map for a particular locator (such as a contour):

1. Select **File > Load Results** and double-click the desired file.
 2. Select the locator from the **Insert** menu.
 - a. On the **General** tab for the locator, set **Color Map** to **Fluent Rainbow**.
 - b. On the **Render** tab, clear the **Lighting** check box.
 - c. Make any other changes desired and click **OK**.
-

12.13. Variable Command

There are several ways to insert a variable:

- From the menu bar, select **Insert > Variable**.
- From the toolbar, click the *Variable*  icon.
- In the CFD-Post workspace, click the **Variables** tab.

Each of these methods inserts a new variable and opens the **Variables** workspace. For details, see [Variables Workspace \(p. 89\)](#).

12.14. Expression Command

There are several ways to insert an expression:

- From the menu bar, select **Insert > Expression**.
- From the toolbar, click the *Expression*  icon.
- In the CFD-Post workspace, click the **Expressions** tab.

Each of these methods inserts a new expression and opens the **Expression** workspace. For details, see [Expressions Workspace \(p. 96\)](#).

12.15. Table Command

The **Insert > Table** command opens a table for editing in the **Table Viewer**. In addition to that method, you can also create a table as follows:

- From the toolbar, click the *Create Table*  icon.
- In the **Table Viewer**, click the *New Table*  icon.

Each of these methods inserts a new table under the Report object. To see the table in the report, you must generate the report. For details, see [Report \(p. 72\)](#).

To learn how to work with tables, see [Editing in the Table Viewer \(p. 300\)](#).

12.15.1. Editing in the Table Viewer

Note:

When multiple cases are loaded, the **Default Case** field enables you to specify which case the table values apply to. If the cases are in case-comparison mode, you have the option of creating a table that uses values from the differences in values between cases 1 and 2.

Changing the **Default Case** field removes all unsaved values and definitions from the table.

To enter data into a cell, select a cell and type in the information you want. To edit the current contents of a cell in the cell itself (rather than in the cell definition field), double-click the cell.

The cell contents can be formatted with bold, italic, and underline fonts; left, center, and right justification; word wrapping; font sizes; and text and background colors. Multiple cells can be merged into a single larger cell to enable large items (for example, titles) to span multiple cells. For details, see [Table 12.2: Table Viewer Tools Toolbar \(p. 303\)](#).

To perform a formatting operation on multiple cells, click in the upper-left cell of the group and, while pressing **Shift**, click in the lower-right cell of the group. While the group is highlighted, toolbar operations are applied to all cells in the group.

Numeric data, (that is, numbers alone, numbers with units, and expression results), can be formatted to display in scientific or fixed notation with a specified number of significant digits.

Table contents can be cut (**Ctrl+C**) and pasted (**Ctrl+V**) into Microsoft Excel documents and vice versa.

12.15.1.1. Shortcut Menu

To access the shortcut menu for a table, type = into a cell and right-click the cell, or right-click the field for the selected cell above the table. The shortcut menu has all of the commands listed in [Table 12.1: Shortcut Menus Toolbar \(p. 301\)](#), plus an **Edit** submenu that has the standard editing commands.

For faster expression entry, there is also a **Shortcut Menus** toolbar above the table with the following items. Type = into the cell and click the given menu to display a variety of items that can be inserted automatically at the current cursor location. All, except **Annotation**, are also available in the details view for expressions.

Table 12.1: Shortcut Menus Toolbar

Type of Item to Insert	Description
 Function ▾	Select from the following submenus: • CFD-Post Select from a list of predefined and user-defined functions from CFD-Post to insert into the cell. For details, see CFX Command Language (CCL) in CFD-Post (p. 417). • CEL Select from a list of predefined CEL functions. For details, see CEL Mathematical Functions in the CFX Reference Guide.
 Expression ▾	Enables you to specify CFD-Post expressions or expressions that you have created with the Expressions workspace. For an

Type of Item to Insert	Description
	example of using the Expressions workspace, see Expressions Workspace: Example (p. 99) .
 Variable ▾	Select from a list of existing variables to insert into the cell.
 Location ▾	Select from a list of existing locations to insert into the cell.
 Constant ▾	Select from a list of mathematical constants to insert into the cell.
 Annotation ▾	Select from the following menu items/submenus: • Time Step Inserts the value of the current timestep. • Time Value Inserts the value of the current time value. • File Name submenu – Name Inserts the name of the current results file, including the extension. – Path Inserts the file path of the current results file. • File Date submenu Select from a list of different date formats to insert into the cell. The inserted value represents the date that the file was created. • File Time submenu Select from a list of different time formats to insert into the cell. The inserted value represents the time of day that the file was created.

12.15.1.2. Expressions

Tables in CFD-Post have the ability to evaluate and display expression results and update those results when variables and/or locations they depend on change.

To enter an expression, edit a cell and prefix a valid CFD-Post expression with an equals sign (=). For example, you may enter the following into a cell:

```
=2*areaAve(Pressure)@inlet
```

When the focus leaves the cell, the table displays the evaluated result of that equation in the cell. When selecting a cell containing an expression, the expression is displayed in the cell editor box immediately above the table. You can edit the expression in the cell editor box. Alternatively, you can double-click the cell and edit the equation from the cell itself. For details on how to enter common expressions and functions quickly, see [Shortcut Menu \(p. 301\)](#).

If there is an error in evaluating the expression contained in a table cell, the cell will be colored red.

Units for expression evaluations that return a temperature will always be displayed in K or R. For details, see [Function Calculator \(p. 347\)](#).

The toolbar above the **Table Viewer** contains the following icons:

Table 12.2: Table Viewer Tools Toolbar

Icon	Description
	Creates a new table.
	Opens the Load Table from file dialog box. Tables can be loaded from files in two different formats: • CFD-Post State Files (*.cst) - Loads the table CCL from the given state file. If the file contains tables with names that already exist, numbers will be added to the end of the names of the imported tables to differentiate them from existing tables. • Comma Separated Values Files (*.csv) - Loads the values in the CSV file into a new table. You can specify the table name in the Load Table dialog box.
	Opens the Save Table to file dialog box. Tables can be saved to several formats: • CFD-Post State (*.cst) - Saves the current table to a state file. Tables saved in state files will maintain expressions and formatting and, when reloaded, will exactly reproduce the original table. • HTML (*.htm, *.html) - Saves the current table to an HTML file. Note that the saved HTML table will contain expression results, and not the expressions. All formatting will be converted to the HTML equivalent. Word-wrapping is always on. The Save Table dialog box contains additional formatting options including table title, caption, borders, margins, spacing, and gridline visibility. • Comma Separated Values (*.csv) - Saves the current table to a CSV file. Note that the saved table will contain expression

Icon	Description
	results, not the expressions. No formatting information is saved to the file. The Save Table dialog box provides the option to clear the output of trailing separators for table rows that have fewer columns than other rows. If this option is on, extra commas will appear on some lines so that all rows in the CSV file will contain the same number of columns. This format can be directly imported to Microsoft Excel. Text (* .txt) - Behaves identically to the CSV option, except that you can specify the separator.
	Edit operations for contents of cells: <i>Cut</i>, <i>Copy</i>, and <i>Paste</i>. To select a rectangle of cells for an operation, click in the cell in the upper-left corner, then Shift-click the cell in the lower-right corner. The cells become highlighted and can be operated upon as a unit.
	Font operations for text in cells: <i>Bold</i> , <i>Italic</i> , and <i>Underline</i> .
	Text-alignment operations: <i>Left</i> , <i>Center</i> , and <i>Right</i> .
	Makes all cells in the table wrap text.
	Launches the Cell Formatting dialog box, where you can specify scientific or fixed notation, the precision, and whether to show the value or the units (at least one of the value or units must appear).
	Changes the size of the font used in the cell.
	Opens the Select color dialog box for setting the background color.
	Opens the Select color dialog box for setting the text color.
	Causes a cell to span rows or columns (<i>Merge Cells</i>) or reverses that operation (<i>Unmerge Cells</i>).

Here is an example of formatting applied to a table:

Figure 12.1: Sample Table Formatting

	A	B	C	D
1	Proximity to Outlet	Min Temperature	Max Temperature	Difference
2	-6.04 [in]	293.15 [K]	313.15 [K]	20.00 [K]
3	-0.55 [in]	293.15 [K]	305.40 [K]	12.25 [K]
4	2.18 [in]	293.15 [K]	304.35 [K]	11.20 [K]
5	5.04 [in]	293.15 [K]	303.02 [K]	9.87 [K]
6	8.00 [in]	293.15 [K]	301.81 [K]	8.66 [K]

To format the table shown above:

1. Cells A1-D1: Applied bold font, background color, and text centering. Manually resized cell widths individually.   
2. Cell A1: Applied text wrapping and resized cell height manually. 
3. Cells A2-D6: Right-justified text. 
4. Cells A2-A3: Manually changed the font color. 

Note:

To perform a formatting operation on multiple cells, click in the upper-left cell of the group and, while pressing **Shift**, click in the lower-right cell of the group. While the group is highlighted, toolbar operations are applied to all cells in the group.

12.16. Chart Command

Charts are graphs that use lines and/or symbols to display data. You can create charts that can be used on their own or in reports.

The following characteristics of charts will be discussed:

- [Creating a Chart Object \(p. 305\)](#)
- [Viewing a Chart \(p. 324\)](#)
- [Example: Charting a Velocity Profile \(p. 325\)](#)

Note:

When using the **Turbo** workspace to post-process a turbo-machinery case, several "Turbo Charts" are created by default. For details, refer to [Turbo Charts \(p. 397\)](#).

12.16.1. Creating a Chart Object

To create and view a chart object:

1. Click *Create Chart*  or select **Insert** > **Chart**.
The **Insert Chart** dialog box appears.
2. Enter a name for the new chart object.
3. Click **OK**.

The chart object appears under the **Report** heading in the tree view. A details view appears for the new chart object and the **Chart Viewer** takes focus.

4. Edit the chart settings as appropriate for each tab:
 - [Chart: General Tab \(p. 306\)](#)
 - [Chart: Data Series Tab \(p. 310\)](#)
 - [Chart: X Axis Tab \(p. 314\)](#)
 - [Chart: Y Axis Tab \(p. 316\)](#)
 - [Chart: Line Display Tab \(p. 319\)](#)
 - [Chart: Chart Display Tab \(p. 323\)](#)
5. Click **Apply** to see the results of your changes displayed in the **Chart Viewer**.
6. Optionally, on the **Data Series** tab click the *Get Information on the Item* icon  to view summary data for the current series.
7. Optionally, click **Export** to save the chart data in a Comma Separated Values (CSV) file. You can load the values in the CSV file into external programs such as Microsoft Excel.

To see the chart in the report, you must generate the report as described in [Report \(p. 72\)](#).

12.16.1.1. Chart: General Tab

The **General** tab is used to define the chart type, the main title, and the report caption.

12.16.1.1.1. Type

The **Type** setting has the following options:

Option	Description
XY - Line	Plots X axis variable vs. the Y axis variable. XY - Line charts use polyline or line locators to plot values that vary in space.
XY - Scatter	Plots X axis variable vs. the Y axis variable. XY - Scatter charts use symbols to represent the X-axis and Y-axis variable values for each point in the selected location.
Transient or Sequence	Plots an Expression (typically time) on the X axis and enables you to specify a variable to plot on the Y axis. Transient or Sequence charts use expressions or a point locator to plot the variation of a scalar value vs. time.
Histogram	Plots the number of values or the proportion of values that fall into each specified category.

12.16.1.1.2. Display Title: Title

The **Title** setting specifies a title for the Chart object. Clear the **Display Title** check box if you don't require a chart title.

12.16.1.1.3. Report: Caption

The **Caption** is the description of the Chart object that appears in the report.

12.16.1.1.4. Fast Fourier Transform

The **Fast Fourier Transform** check box can be selected only for Transient or Sequence charts. When the **Fast Fourier Transform** check box is selected, the following options are available:

Modify Input Signal Filter

Enables you to select the signal filter to be Hanning (default), Barlett, Blackman, Hamming, or None. For details, see [Fast Fourier Transform \(FFT\) Theory \(p. 307\)](#).

Subtract mean

Causes the mean to be subtracted from each value to better show the amplitude of the noise.

Note:

This feature applies to *all* loaded files.

Full range of input data vs. Setting Min/Max Limits

You can choose to analyze the **Full range of input data** or to set **Min** and **Max** values. To get the range for the **Min** and **Max** values, click *Get range from FFT output* .

Reference Values

Click **Reference Values** to display the **Reference Values** dialog box. There you can set the following values (which will apply to all Fast Fourier Transform charts): **Reference Acoustic Pressure**, **Length**, **Velocity**, and whether to **Save as default**.

12.16.1.1.4.1. Fast Fourier Transform (FFT) Theory

When interpreting time-sequence data from a transient solution, it is often useful to look at the data's frequency attributes. For instance, you may want to determine the major vortex-shedding frequency from the time-history of the drag force on a body recorded during a simulation, or you may want to compute the frequency distribution of static pressure data recorded at a particular location on a body surface. Similarly, you may need to compute the frequency distribution of turbulent kinetic energy using data for fluctuating velocity components.

To interpret some of these time-dependent data, you need to perform Fourier transform analysis. The Fourier transform enables you to take any time-dependent data and resolve it into an equivalent summation of sinusoids.

The CFD-Post FFT module assumes that the input data have been sampled at equal intervals and are consecutive (in the order of increasing time).

The lowest non-zero frequency that the FFT module can pick up, $f_{\min}$, is given by $1/(N\Delta t)$, where N is the total number of samples, and Δt is the sampling interval (or timestep size). If

the sampled sequence contains frequencies lower than this, these frequencies will be aliased into higher frequencies.

The highest frequency that the FFT module can pick up, $f_{\max}$, is given by $\lfloor N/2 \rfloor / (N\Delta t)$. Note that the floor function has been applied to $N/2$ to round it down to the nearest integer. Note that, when N is even, $f_{\max} = 1 / (2\Delta t)$.

The value at zero frequency represents the mean value. The mean value can be taken out of the plot by selecting **Subtract mean** check box.

12.16.1.1.4.1.1. Windowing in Fast Fourier Transforms

The discrete FFT algorithm is based on the assumption that the time-sequence data passed to the FFT corresponds to a single period of a periodically repeating signal. Because in most situations the first and the last data points will not coincide, the repeating signal implied in the assumption can have a large discontinuity. The large discontinuity produces high-frequency components in the resulting Fourier modes, causing an aliasing error. You can condition the input signal before the transform by "windowing" it, in order to avoid this problem.

Suppose there are N consecutive discrete (time-sequence) data that are sampled with a constant interval Δt :

$$\varphi_k \equiv \varphi(t_k), \quad t_k \equiv k\Delta t, \quad k=0, 1, 2, \dots, (N-1) \quad (12.43)$$

Windowing is done by multiplying the original input data (φ_j) by a window function W_j :

$$\tilde{\varphi}_j = \varphi_j W_j \quad j=0, 1, 2, \dots, (N-1) \quad (12.44)$$

There are four different window functions:

Hamming's window:

$$W_j = \begin{cases} 0.54 - 0.46 \cos\left(\frac{8\pi j}{N}\right) & j \leq \frac{N}{8}, j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (12.45)$$

Hanning's window:

$$W_j = \begin{cases} 0.5 \left[1 - \cos\left(\frac{8\pi j}{N}\right) \right] & j \leq \frac{N}{8}, j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (12.46)$$

Barlett's window:

$$W_j = \begin{cases} \frac{8j}{N} & j \leq \frac{N}{8} \\ 8 \left(1 - \frac{j}{N} \right) & j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (12.47)$$

Blackman's window:

$$W_j = \begin{cases} 0.42 - 0.5 \cos\left(\frac{8\pi j}{N}\right) + 0.08 \cos\left(\frac{16\pi j}{N}\right) & j \leq \frac{N}{8}, j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (12.48)$$

These window functions preserve 3/4 of the original data, affecting only 1/4 of the data the ends.

12.16.1.1.4.1.2. Using Fast Fourier Transforms

The Fourier transform utility enables you to compute the Fourier transform of a signal, $\varphi(t)$, a real-valued function, from a finite number of its sampled points.

For a periodic set of N sampled points, ϕ_k , the discrete Fourier transform expresses the signal as a finite trigonometric series:

$$\varphi_k = \sum_{n=0}^{N-1} \hat{\varphi}_n e^{2\pi i k n / N} \quad k=0, 1, 2, \dots, (N-1) \quad (12.49)$$

where the series coefficients $\hat{\varphi}_n$ are computed as

$$\hat{\varphi}_n = \frac{1}{N} \sum_{k=0}^{N-1} \varphi_k e^{-2\pi i k n / N} \quad n=0, 1, 2, \dots, (N-1) \quad (12.50)$$

The previous two equations form a Fourier transform pair that enables you to determine one from the other.

Note that when you vary n from 0 to $N-1$ in [Equation 12.49 \(p. 309\)](#) or [Equation 12.50 \(p. 309\)](#), the following is true:

- n and $N-n$ form a complex conjugate pair.
- The value at $n=0$ is unique and corresponds to zero frequency.
- When N is even, the range of index $1 \leq n \leq N/2-1$ corresponds to positive frequencies, and the range of index $N/2+1 \leq n \leq N-1$ corresponds to negative frequencies. The frequency at $N/2$, called the Nyquist frequency, is a unique value.
- When N is odd, the range of index $1 \leq n \leq (N-1)/2$ corresponds to positive frequencies, and the range of index $(N+1)/2 \leq n \leq N-1$ corresponds to negative frequencies. In this case, the Nyquist frequency does not exist.

For the actual calculation of the transforms, the CFD-Post adopts the Fast Fourier transform (FFT) algorithm, which significantly reduces operation counts in comparison to the direct transform. Specifically, CFD-Post uses the FFT algorithm from the Intel Math Kernel Library. This FFT algorithm allows for the processing of an N -point FFT where N can be even or odd.

12.16.1.1.5. Refresh Settings

Performing a refresh means re-reading files; therefore refreshing data from a series of files (such as transient files for a transient case) is potentially a time-consuming operation. If necessary, CFD-Post automatically performs a refresh of all charts when printing a chart and when generating or refreshing a report. At other times, you have control over when to perform a "refresh" of the data. When the **General** tab is set to create an XY transient or sequence chart, settings appear that control how charts are refreshed:

Refresh chart on Apply

Causes only the currently displayed chart to be refreshed when you click **Apply**.

Refresh all charts on Apply

Causes all loaded charts to be refreshed when you click **Apply**. This is provided as a convenience; it enables you to work on a series of charts, refreshing all of them when you are done your editing (rather than clicking **Apply** on each chart individually).

12.16.1.2. Chart: Data Series Tab

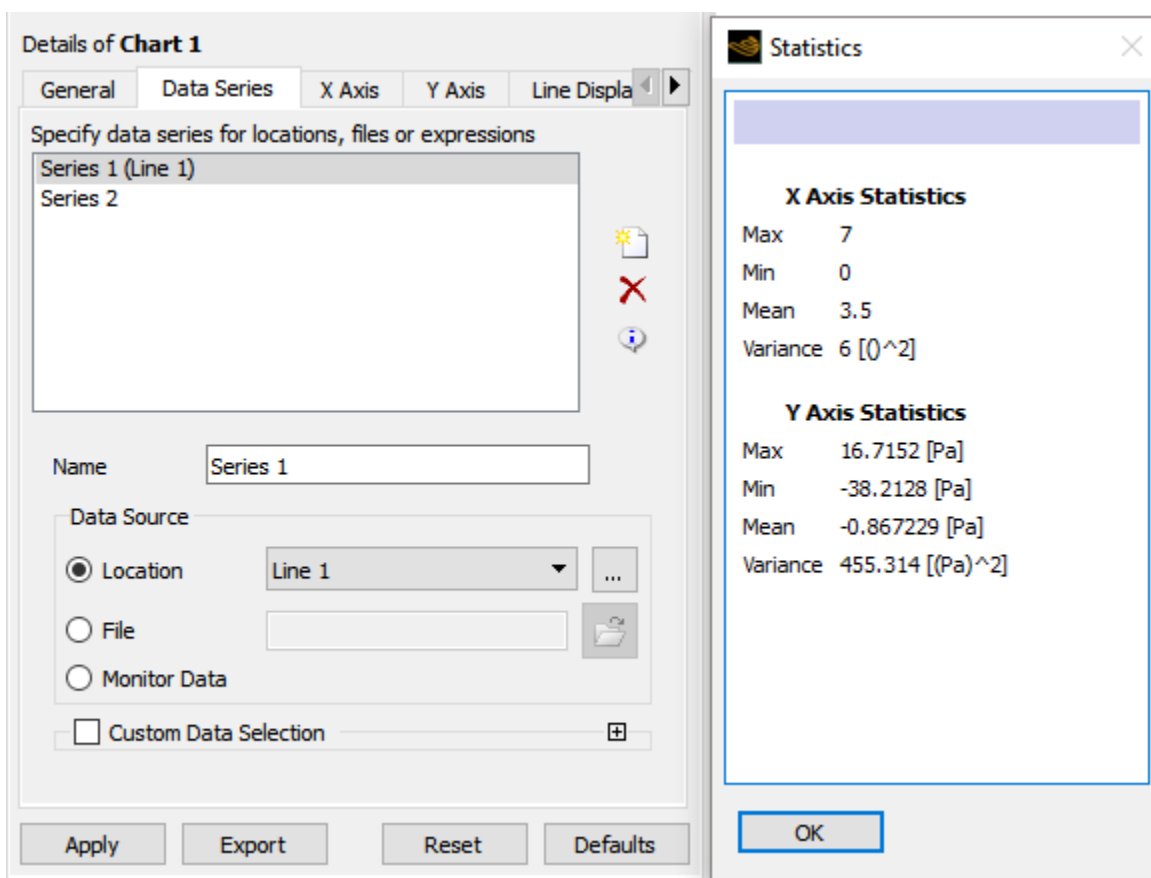
The **Data Series** tab is used to specify the data series to be plotted on the chart.

12.16.1.2.1. Name Controls

At the top of the **Data Series** tab is a list of the data series for the chart. The icons beside the list enable you to:

- Add a data series (*New* )
- Delete a data series (*Delete* )
- See statistics for a data series in a dialog box (*Statistics* )

Corresponding functions are available when you right-click a data series name.



12.16.1.2.2. Data Source

The fields in the **Data Source** area become enabled after you create an initial series.

- For a **General** > **XY** chart or a **General** > **Histogram** chart, choose a **Location** or **File** or **Monitor Data** to define the source of the data for the series you are creating. A typical location would be a line or a streamline; a typical data file would be a CSV file.
- For a **General** > **XY - Transient or Sequence** chart, a typical **Location** would be a point. However, such a chart will also accept a **File** or **Expression** or **Monitor Data** as the data source. For example, you could use an expression to plot `areaAve(Temperature)@Outlet` as a function of time.

12.16.1.2.2.1. Data Source File Format

The data delimiter in a data source file is usually a comma.^[1]

Files that use commas as delimiters have at least one section of numerical data. If there is only one section of data, then the name specification (consisting of the [Name] header and the line that follows it) and the [Data] header are optional, but not independently; they must either be both included or both omitted. If there is more than one section of data, the name specification and [Data] header are both required. The line that follows the [Data] header

^[1] There are two exceptions to this: if the extension of the file is .xy, then the delimiter is a tab character; if the extension of the file is .out, then the delimiter is a space character.

is always optional; omitting it causes default variable names and default units to be assumed. An example section follows:

```
[Name]
Velocity Profile

[Data]
Z [ m ],Velocity [ m s^-1 ]
-0.10000000,4.53693390
-0.09797980,4.54303789
-0.09595960,4.54667473
-0.09393939,4.54347515
-0.09191919,4.54762697
...
```

The line following the keyword [Data] may specify the variable names and units for each column of numerical data that follows. If variable names are not provided, default names are applied.

This is the same format as the export file in CFD-Post. It is also the same format that CFX-Solver Manager uses when exporting lines from a chart. Data exported from these sources can be imported directly into CFD-Post to produce a chart line.

For Histograms, the second column is used as the variable data.

Note:

When exporting histogram data, the Y Axis data (that is, the counts) will be stored in the CSV file with the variable name "Count(Histogram Data)". The "(Histogram Data)" suffix is required if this information is imported into CFD-Post because it indicates to CFD-Post that this variable must be interpreted as histogram data and not as regular chart line data.

Note:

If **File** is selected as the data source for a Transient or Sequence chart, and a Fast Fourier Transform (FFT) is being calculated, then the data in the file must be in particular units:

- If the FFT **X Function** setting has a value of Strouhal Number, Octave Band Full or One Third Full, the X Axis data in the file must have units of [s] (seconds).
 - If the FFT **Y Function** setting has a value of Sound Amplitude, Sound Pressure Level, A Weighted, B Weighted or C Weighted, then the Y Axis data in the file must have units of [Pa] (Pascals).
-

12.16.1.2.3. File Variable Selection

The **File Variable Selection** settings are available when you are defining a data series from a file. They indicate which data, within the selected file, is to be used to define the data series.

Section Name is used to select a section of data from the file. This setting does not appear if there is only one section of data in the data source file.

X Axis Variable and **Y Axis Variable** specify which two columns of data are to be used to generate a chart line.

12.16.1.2.4. Custom Data Selection Controls

The **Custom Data Selection Controls** settings are available when you are defining a data series from a location or expression, with **General** > **Fast Fourier Transform** not selected.

When selected, the **Custom Data Selection** controls enable you to override the settings on the X Axis and Y Axis tabs for each series individually. See [Chart: X Axis Tab \(p. 314\)](#) and [Chart: Y Axis Tab \(p. 316\)](#) for more details on what each setting means.

You can also specify the use of **Hybrid** or **Conservative** values or to use the absolute value of data points. For help on the use of Hybrid or Conservative values, see [Hybrid and Conservative Variable Values](#).

Other available settings depend on the chart type; see [X Axis Data Selection \(p. 314\)](#) for details.

12.16.1.2.5. Monitor Variable Selection

The **Monitor Variable Selection** settings are available when you are defining a data series from monitor data or solution residuals.

Set **Source** to one of:

- **Current Cases**

Monitor data from all cases are included in the series, as separate lines. For example, if results files StaticMixer_001.res and StaticMixerRef_001.res are loaded and the residual monitor MAX P-Mass is selected for the series, then multiple lines will be generated, one for each results file.

- **File**

Monitor data is taken from the file you specify. The allowed file types are .res, .mres, .bak, and .mon.

X Axis Variable and **Y Axis Variable** specify which variables are to be used to generate a chart line.

Take absolute value of data points controls whether the values of data points are always positive.

Note:

If a complete multi-configuration (.mres) run is loaded as a single case, only the monitor data that is present in the last configuration will be added to the series.

12.16.1.3. Chart: X Axis Tab

The **X Axis** tab is used to set properties for all data series that do not have custom data selection (which is set on the **Series** tab). The options available on the X Axis tab vary according to the **General** tab's **Type** setting.

12.16.1.3.1. X Axis Data Selection

The **Data Selection** settings control which variable is used as the data source and how the data is processed:

When **General** > **Type** is XY or Histogram, the **Variable** field can be set to any variable. **Hybrid** vs. **Conservative** sets how conservation equations for the boundary control volumes are solved. See [Hybrid and Conservative Variable Values](#) for details. **Take absolute value of data points** controls whether the values of data points are always positive.

When **General** > **Type** is XY – Transient or Sequence, the **Expression** field can be set to expressions provided in CFD-Post or to expressions that you have defined. For example, you could use this option to plot areaAve(Temperature)@Outlet as a function of time.

When **General** > **Type** is XY – Transient or Sequence and **Fast Fourier Transform** is selected, you can define an **X Function**.

12.16.1.3.1.1. Specifying an X Function

The options for the X Function are related to the discrete frequencies at which the Fourier coefficients are computed. You can apply the following specific analytic functions. Because the corresponding definitions for the y-axis functions include contributions from both elements of a complex conjugate pair (i.e. n and $N-n$), definitions are provided for $n \leq N/2$ when N is even, and $n \leq (N-1)/2$ when N is odd.

Frequency is defined as:

$$f_n = \frac{1}{N\Delta t} n \quad \begin{cases} n=0,1,2,\dots,N/2 & \text{when } N \text{ is even} \\ n=0,1,2,\dots,(N-1)/2 & \text{when } N \text{ is odd} \end{cases} \quad (12.51)$$

where N is the number of data points used in the FFT.

Strouhal Number is the nondimensionalized version of the frequency defined in the equation for **Frequency**:

$$St_n \equiv \frac{f_n L_{\text{ref}}}{U_{\text{ref}}} \quad (12.52)$$

where L_{ref} and U_{ref} are the reference length and velocity scales. Note that you can set L_{ref} and U_{ref} by clicking **Reference Values** on the **General** tab.

Fourier Mode is the index in

$$\varphi_k = \sum_{n=0}^{N-1} \hat{\varphi}_k e^{2\pi i k n / N} \quad k=0,1,2,\dots,N-1 \quad (12.53)$$

and/or

$$\hat{\varphi}_n = \frac{1}{N} \sum_{k=0}^{N-1} \hat{\varphi}_k e^{-2\pi i k n / N} \quad n=0,1,2,\dots,N-1 \quad (12.54)$$

which represents the n th or k th term in the Fourier transform of the signal.

Octave Band Full is a range of discrete frequency bands for different octaves within the threshold of hearing. The range of each octave band is double to that of the previous band (see [Table 12.3: Octave Band Frequencies and Weightings \(p. 315\)](#)).

One Third Full is a range of discrete frequency bands within the threshold of hearing. Here the range of each band is one-third of an octave, meaning that there are three times as many bands for the same frequency range.

Table 12.3: Octave Band Frequencies and Weightings

Lower Freq. (Hz)	Center Freq. (Hz)	Upper Freq. (Hz)	dB A	dB B	dB C
11	16	22	-56.7	-28.5	-8.5
22	31.5	45	-39.4	-17.1	-3.0
45	63	90	-26.2	-9.3	-0.8
90	125	180	-16.1	-4.2	-0.2
180	250	355	-8.6	-1.3	0.0
355	500	710	-3.2	-0.3	0.0
710	1000	1400	0.0	0.0	0.0
1400	2000	2800	1.2	-0.1	-0.2
2800	4000	5600	1.0	-0.7	-0.8
5600	8000	11200	-1.1	-2.9	-3.0
11200	16000	22400	-6.6	-8.4	-8.5

12.16.1.3.2. Category Divisions

These controls are enabled when the chart type is **Histogram**.

If **Category Divisions** are set to **Automatic**, you are able to specify the **Number of Divisions**. If **Category Divisions** are set to **User Defined**, you are able to specify the **Division Values**.

The **Division Values** field enables you to type points where you want to create histogram boundaries. You can either enter user-defined category divisions by typing a comma-separated

ordered list directly into the **Division Values** field, or click **More**  to open up an editor for the division values (which includes the ability to set the values in a particular unit). If you use the editor, then the values do not need to be entered in order as you will be offered the chance to sort the values when you close the editor.

12.16.1.3.3. Axis Range

You can choose to **Determine ranges automatically** or to set **Min** and **Max** values. To get the range for the **Min** and **Max** values, click *Get range from existing chart* .

The default X-axis scale is linear but can be set to be a **Logarithmic scale**. Select **Invert Axis** to reverse the direction of the scale.

12.16.1.3.4. Axis Number Formatting

You can have axis numbers set automatically or choose the format yourself:

- If you select **Determine the number formatting automatically**, CFD-Post will change the formatting to the one that best suits the data being plotted.
- If you clear **Determine the number formatting automatically**, you can choose between scientific notation and fixed notation, and set the amount of precision.

12.16.1.3.5. Axis Labels

You can choose to **Use data for axis labels**, or to use a **Custom Label**.

12.16.1.4. Chart: Y Axis Tab

The **Y Axis** tab is used to define the characteristics of the Y axis of the chart you are going to produce. For descriptions of many of the fields on this tab, see [Chart: X Axis Tab \(p. 314\)](#). Fields unique to this tab are described below.

12.16.1.4.1. Y Axis: Data Selection

When the **General** tab is set to create an XY or an XY transient or sequence chart, the **Data Selection** includes a **Variable** that you can set; you can also control whether the boundary data is Hybrid or Conservative, and whether or not to take the absolute value of data points.

When the **General** tab is set to create a histogram, the Y axis **Value** can be:

- Count

When Count is selected, the display shows the number of values lying within each category, or, if the **Weighting** is not set to NONE, the total weight within the category.

- Percentage

When Percentage is selected, the display is scaled to show the percentage of values or weights lying within each category. The Percentage is calculated using all of the data on the selected location, even if some of the data is not displayed because it lies outside of the selected category boundaries. Hence the total Percentage shown in the selected categories may add up to less than 100%.

The **Weighting** can be:

- None
- Geometrical
- Mass Flow

The weighting setting changes the shape of the histogram by removing mesh dependencies. For example, if mesh density varies along a line, counts are biased towards areas of higher density; the `Geometrical` setting removes that bias.

When the **General** tab is set to `Fast Fourier Transform`, a **Y Function** field appears where you can choose one of the following settings. The definitions are provided for $n \leq N/2$ when N is even, and $n \leq (N-1)/2$ when N is odd. The definitions include contributions from both elements of a complex conjugate pair: n and $N-n$.

A Weighted, Sound Pressure Level (dB A)

This is the calculated sound pressure level weighted by the A-scale function to more closely approximate the frequency response of the human ear. A-weighting is applied for loudness levels below 55 phons (55 dB at 1 kHz) and is the most commonly used weighting function. See [Figure 12.2: A-, B-, and C-weighting Functions \(p. 317\)](#) for a graphical representation. This option is available only when **X Function** on the **X-Axis** tab is set to `Octave Band Full` or `One Third Full`.

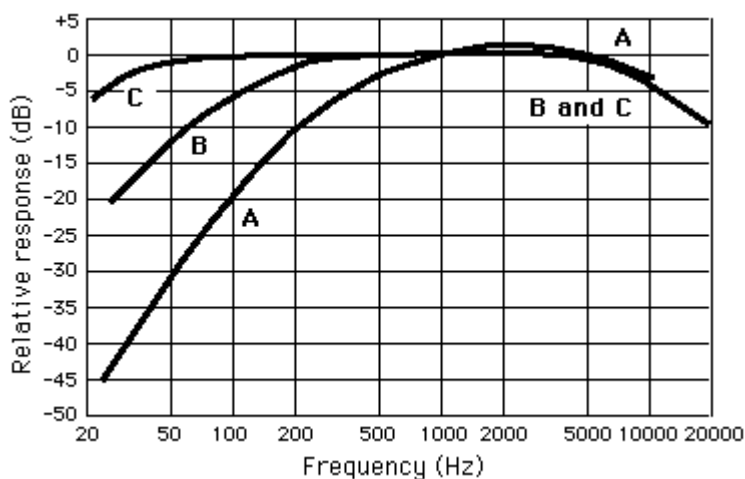
B Weighted, Sound Pressure Level (dB B)

This is the calculated sound pressure level weighted by the B-scale function. B-weighting is applied for loudness levels between 55 and 85 phons, though it is rarely used. See [Figure 12.2: A-, B-, and C-weighting Functions \(p. 317\)](#) for a graphical representation. This option is available only when **X Function** on the **X-Axis** tab is set to `Octave Band Full` or `One Third Full`.

C Weighted, Sound Pressure Level (dB C)

This is the calculated sound pressure level weighted by the C-scale function. C-weighting is applied for loudness levels above 85 phons and is commonly used for high-intensity sounds such as traffic studies. See [Figure 12.2: A-, B-, and C-weighting Functions \(p. 317\)](#) for a graphical representation. This option is available only when **X Function** on the **X-Axis** tab is set to `Octave Band Full` or `One Third Full`.

Figure 12.2: A-, B-, and C-weighting Functions



Power Spectral Density

This is the distribution of signal power in the frequency domain. Its value and units depend on the choice of **X Function**. For the detailed spectral representation with all resolved harmonics (that is, when the **X Function** option is set to either **Frequency**, **Strouhal Number**, or **Fourier Mode**), the power spectral density (*PSD*) has units of the signal magnitude squared over the frequency (for example, Pa²/Hz) and is defined for the frequency f_n as

$$PSD(f_n) = E(f_n) / \Delta f \quad \begin{cases} n=1,2,\dots,N/2 & \text{when } N \text{ is even} \\ n=1,2,\dots,(N-1)/2 & \text{when } N \text{ is odd} \end{cases} \quad (12.55)$$

where Δf is the frequency step in the discrete spectrum, and $E(f_n)$ is the Fourier mode power.

When N is even, the Fourier mode power is computed as:

$$E(f_n) = \begin{cases} 0.5(2|\hat{\phi}_n|)^2 & n=1,2,\dots,N/2-1 \\ |\hat{\phi}_n|^2 & n=N/2 \end{cases} \quad (12.56)$$

When N is odd, the Fourier mode power is computed as:

$$E(f_n) = 0.5(2|\hat{\phi}_n|)^2 \quad n=1,2,\dots,(N-1)/2 \quad (12.57)$$

For the octave analysis (that is, when the **X Function** option is set to either **Octave Band Full** or **One Third Full**), the power spectral density has units of the signal magnitude squared (for example, Pa²), and is defined for the frequency band f_{band} as

$$PSD(f_{\text{band}}) = \sum E(f_n) \quad (12.58)$$

where n includes all of the Fourier modes belonging to the band.

Sound Amplitude

This is similar to the **Sound Pressure Level (dB)** option, and is a logarithmic conversion of the pressure signal Magnitude into decibel units. The sound amplitude in dB, A_{sp} , is calculated for either a Fourier mode or a frequency band using

$$A_{\text{sp}} = 10 \left(\log \frac{A}{p_{\text{ref}}} \right) \quad (\text{dB}) \quad (12.59)$$

Sound Pressure Level

This is the decibel level. For either general or acoustic data, when the sampled data is pressure (for example, static pressure or sound pressure), the sound pressure level in dB, L_{sp} , is calculated in decibel units using

$$L_{sp} = 10 \left(\log \frac{PSD}{p_{ref}^2} \right) \quad (\text{dB}) \quad (12.60)$$

where PSD is the power spectral density for either a particular Fourier mode or a particular frequency band (see [Equation 12.55 \(p. 318\)](#) and [Equation 12.58 \(p. 318\)](#)). p_{ref} is the reference acoustic pressure, with a default value of 2×10^{-5} Pa.

Magnitude

This is the amplitude. For the detailed spectral representation with all resolved harmonics (that is, when the **X Function** option is set to either Frequency, Strouhal Number, or Fourier Mode), the magnitude (A) is defined for the frequency f_n in one of two ways.

When N is even:

$$A(f_n) = \begin{cases} |\hat{\phi}_n| & n=0, N/2 \\ 2|\hat{\phi}_n| & n=1, 2, \dots, N/2-1 \end{cases} \quad (12.61)$$

When N is odd:

$$A(f_n) = \begin{cases} |\hat{\phi}_n| & n=0 \\ 2|\hat{\phi}_n| & n=1, 2, \dots, (N-1)/2 \end{cases} \quad (12.62)$$

where, in both cases, $A(f_0)$ is the mean signal value.

For the octave analysis (that is, when the **X Axis Function** is either Octave Band or 1/3-Octave Band), the magnitude is defined for the frequency band f_{band} as

$$A(f_{band}) = \sqrt{2PSD(f_{band})} \quad (12.63)$$

where $PSD(f_{band})$ is calculated according to [Equation 12.58 \(p. 318\)](#).

12.16.1.5. Chart: Line Display Tab

On the **Line Display** tab you can set the **Line Style** to a variety of settings, including Automatic, Solid, Dash, Dot, and so on.

For scatter charts, Automatic is interpreted as None, although you can add lines to scatter charts if desired. Note that If lines are manually added to a scatter chart, the points are joined in the order that CFD-Post reads them and do not represent any underlying data structures.

You can select the **Use series name for legend name** check box to derive the name of the line (as it appears in the legend) from the name of the series (as defined on the **Data Series** tab and, if more than one case is loaded, from the case name). Alternatively, you can clear that check box and type in a new **Legend Name**.

You can have CFD-Post **Automatically generate Line Color** or you can:

1. Clear the **Automatically generate Line Color** check box.
2. Optionally click the bar beside **Line Color** to cycle through 10 basic colors. Click the right-mouse button to cycle backwards. Alternatively, you can choose any color by clicking *Color selector*  to the right of the **Line Color** setting.
3. Optionally change the selection for **Line Type** or **FFT Line Type**. For details, see [Line Type and FFT Line Type Options](#) (p. 320).

Use the **Symbols** drop-down menu to place a graphic at every data point of the series. **Automatic** is interpreted as **None** for line charts and, for scatter charts, chooses a different symbol for each data set.

Use the **Symbol Color** control to set a color for the graphic the same way you did for the **Line Color** or select **Automatically generate Symbol Color**.

Note:

Line width and symbol size can be set on the **Chart Display** tab for the chart as a whole, but cannot be set for each line individually.

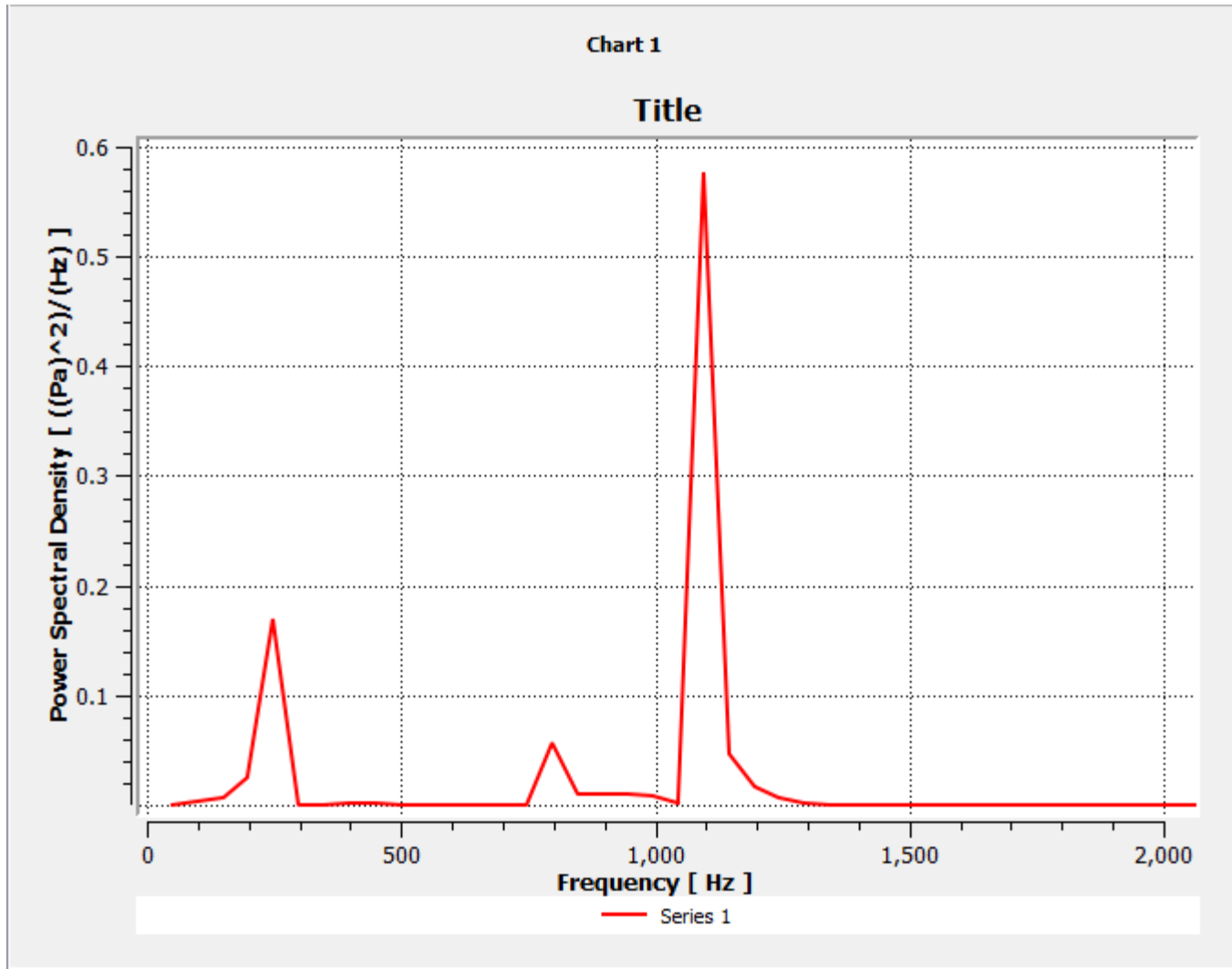
12.16.1.5.1. Line Type and FFT Line Type Options

On the **General** tab, if **Fast Fourier Transform** is selected, the line type is controlled by the **FFT Line Type** setting, otherwise, the line type is controlled by the **Line Type** setting; the applicable setting is shown.

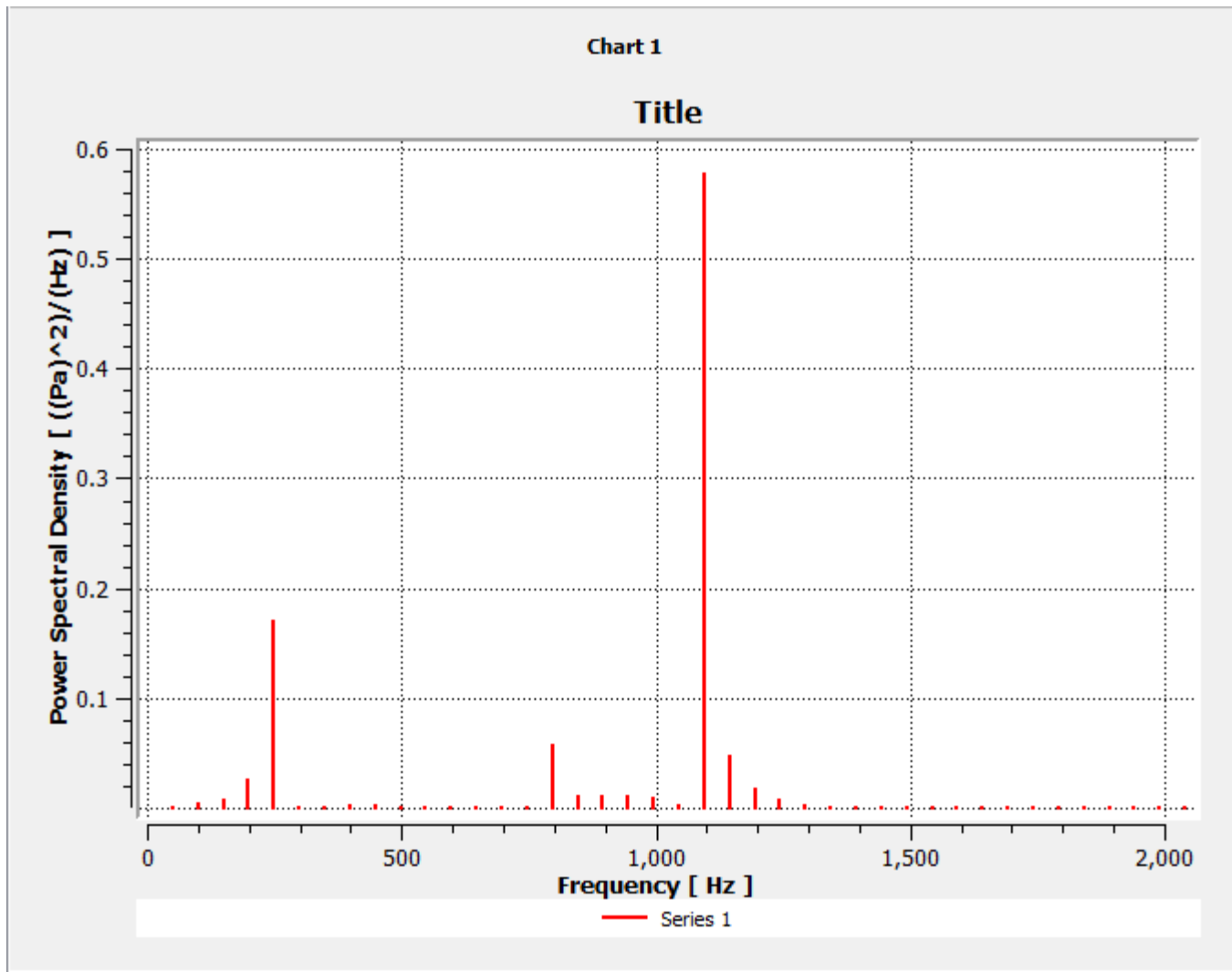
Both the **FFT Line Type** and **Line Type** settings have the following options:

- Lines (the default for non-FFT plots) - This line type is suitable for general plots.

Figure 12.3: Chart made with lines

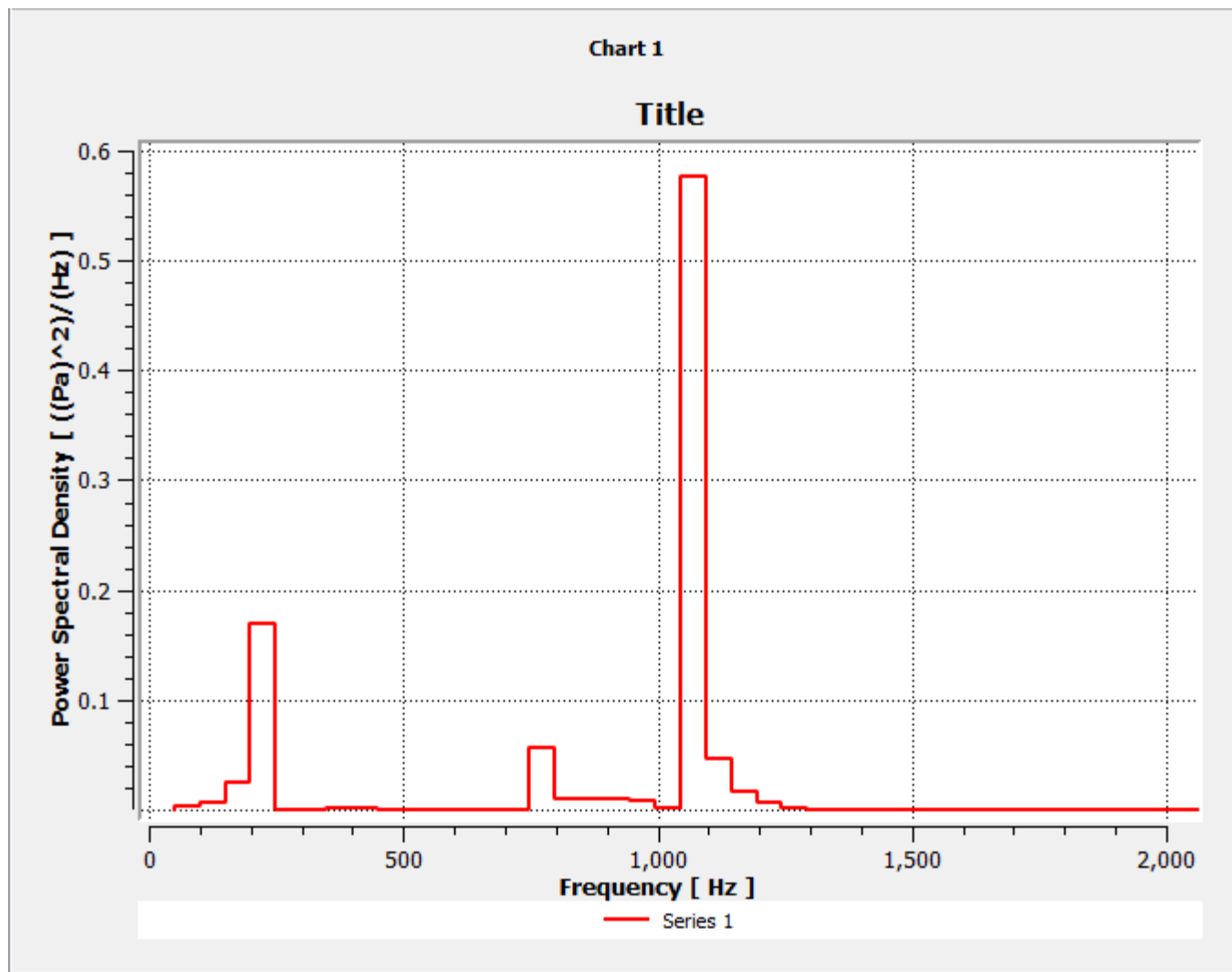


- Bars (the default for FFT plots) - This line type is suitable for FFT plots.

Figure 12.4: Chart made with bars

- Steps - This line type is suitable for FFT plots.

Figure 12.5: Chart made with steps



12.16.1.5.2. Fill Area Controls

When **Fill Area** is selected, you can choose to have a fill color generated automatically or at all times (**Always On**). The **Automatic** setting generates a fill when the chart's **General** tab's **Type** is set to Histogram.

Note:

Plotting fill areas for graphs that have multiple y values for a given x (such as streamlines) does not produce useful results.

12.16.1.6. Chart: Chart Display Tab

The following sections describe the **Chart Display** tab.

12.16.1.6.1. Display Legend Area

The legend is the text that you entered in the **Name** field on the **Data Series** tab displayed beside the line color that represents its data source.

When **Display Legend** is selected, the chart's legend appears, either **Outside Chart** or **Inside Chart**.

When **Outside Chart** is selected, set the **Location** and **Justification** of the legend:

- **Location** is Below or Above, **Justification** can be Left, Center, or Right.
- **Location** is Right or Left, **Justification** can be Top, Center, or Bottom.

When **Inside Chart** is selected, the legend is displayed in a box that appears on the chart. Use the **X Justification** and **Y Justification** controls to locate a fixed-size box at standard locations, optionally using **Width/Height** to change the size of the box. Alternatively, set **X Justification** and **Y Justification** to **None** so that you can use **Position** and **Width/Height** to control the size and position of the box exactly.

12.16.1.6.2. Sizes Area

Here you set the width of the line and the size of the symbol (if any) that you defined on the **Line Display** tab. These sizes apply to all lines and all symbols (you cannot set sizes for individual lines or symbols).

12.16.1.6.3. Fonts Area

Here is where you control the font type and size of the **Title** (which you defined on the **General** tab), the **Axes Titles**, the **Axes** values, and the **Legend**.

Note:

By default, the titles of the axes are derived from the variables used in the line definition (not necessarily from the **X Axis** and **Y Axis** tabs because a transient chart that uses an expression and any chart that uses custom data selection will set the variables used directly). You can override these default titles by going to the **X Axis** and **Y Axis** tabs, clearing the **Use data for axis labels** check box, and typing in a **Custom Label** name.

The legend text is defined by default as a combination of the series definitions on the **Series** tab and, when more than one case is loaded, the case names, but can be specified on a line-by-line basis directly on the **Line Display** tab by clearing the **Use series name for legend name** check box and typing in a **Legend Name**.

12.16.1.6.4. Grid Area

Here is where you configure the background grid (if any) and the thickness of its major and minor lines.

12.16.2. Viewing a Chart

After a chart object has been created, you can view it:

- In the **Chart Viewer** after selecting the chart in the **Outline** tree view
- In a separate window by right-clicking the chart in the **Outline** tree view and selecting **Show in Separate Window**
- In a report, after including the chart in a report. For details, see [Viewing the Report \(p. 87\)](#).

Time charts, which depict transient runs, have a **Refresh** button at the top of the page. When CFD-Post determines that the chart requires updates, a note appears beside the **Refresh** button. Refreshes are generally not automatic in order to ensure that you can make a series of changes without having to wait through the update required by each change. However, time charts are updated automatically when you print the chart, when a report is previewed, or when a report is generated (HTML/text).

Note:

As time charts are compute-intensive, they are generated only after user action. And because time chart data is not included in a state file, loading a state file will show an empty chart until you click **Apply** in the chart details view or **Refresh** in the **Chart Viewer**.

12.16.3. Example: Charting a Velocity Profile

This example demonstrates how a polyline locator can be used to create a chart of a velocity profile.

1. Load the following results file, which is provided with a tutorial: `StaticMixer_001.res`.
2. Insert a plane (**Insert** > **Location** > **Plane**) and define its location using the point and normal method.

Define the point to be (0,0,0) and the normal to be (0,1,0) so that the plane is normal to the Y axis; click **Apply** when you are done. For details, see [Plane: Geometry Tab \(p. 229\)](#).

3. Insert a polyline (**Insert** > **Location** > **Polyline**) and define its location using the Boundary Intersection method.

Set **Boundary List** to out and **Intersect With** to the plane you just created; click **Apply** when you are done. For details, see [Polyline: Geometry Tab \(p. 252\)](#).

4. Create a new chart by clicking *Create chart* .
5. In the **Insert Chart** dialog box, enter a name for the chart, and then click **OK**.

The details view for the chart appears.

6. On the **General** tab, set **Title** to Velocity Profile at Outlet.
7. Click the **Data Series** tab.
8. Set data source **Location** to the name of the polyline you just created.
9. Click the **X Axis** tab.
10. Set the x-axis Variable to X.

The x-coordinate direction is parallel to the polyline in this example so the plot shows a variable profile across the outlet.

11. Click the **Y Axis** tab.
12. Set the y-axis Variable to **Velocity**.
13. Click **Apply**.

A chart showing **Velocity** versus **X** is displayed in the **Chart Viewer**.

12.16.4. Example: Comparing Differences Between Two Files

You can use Case Comparison mode with the **Chart Viewer** to automatically see the differences in values between the two files:

1. Load the following results files, which are provided with a tutorial: `elbow1.cdat` and `elbow3.cdat`. (Press the **Ctrl** key while selecting the two files, then click **Open**.)

Two viewports open, one with **elbow1** and the other with **elbow2** loaded.

2. Insert a line (**Insert** > **Location** > **Line**). Accept the default values for **Geometry** > **Method**, but set **Line Type** to **Cut**. On the **Color** tab, set **Mode** to **Variable** and **Variable** to **Temperature**. Set the **Range** to **Local**.

Click **Apply** when you are done.

3. In the **Outline** tree view, double-click **Case Comparison**. The **Case Comparison** details view appears. Select **Case Comparison Active** and click **Apply**.

A third viewport opens that displays the temperature difference between the two cases.

4. Click *Create chart* .

5. In the **Insert Chart** dialog box, enter a name for the chart, and then click **OK**.

The details view for the chart appears.

6. On the **General** tab, set **Title** to **Comparison of Temperatures in the Elbow**.
7. Click the **Data Series** tab.
8. Set data source **Location** to the name of the line you just created.
9. Click the **X Axis** tab.
10. Set the x-axis **Variable** to **X**.

The x-coordinate direction is parallel to the polyline in this example so the plot shows a variable profile across the outlet.

11. Click the **Y Axis** tab.
12. Set the y-axis **Variable** to **Temperature**.

13. Click **Apply**.

A chart showing **Temperature** versus **X** is displayed in the **Chart Viewer**. Three lines are there: one for each of the sets of temperature values, and a third line that shows the difference between those values.

Note:

- You can change some of the properties of each line individually (including turning them on and off) by using the **Line Display** tab.
 - The **Difference** line plots only the variable difference on the y-axis. For example, if you defined a chart of Velocity (y-axis) against Pressure (x-axis), then the difference line will plot Velocity Difference against Pressure, not Velocity Difference against Pressure Difference.
-

12.17. Comment Command

You can create comment objects to include in the report. Comments are used to add text to a report in the form of titles and paragraphs.

To define a comment object:

1. From the toolbar, click the *Comment*  icon or select **Insert > Comment**.

The **Insert Comment** dialog box appears.

2. Enter a name for the comment object.
3. Click **OK**.

The **Comment Viewer** appears.

The comment object appears in the tree view, under the Report object.

4. Enter a heading and/or a paragraph of text.

A heading is entered into the **Heading** box. The **Level** setting controls the level of the heading text in the report.

Paragraph text is entered into the large text box below the **Heading** box. Some *Rich Text* features are supported using toolbar icons that appear at the top of the **Comment Viewer**. Pictures can be inserted in the paragraph text area. External hyperlinks can be included in the paragraph text, but will not work in the **Report Viewer** of CFD-Post. External hyperlinks will work when the report is viewed in a web browser.

To see the comment in the report, you must generate the report. For details, see [Report \(p. 72\)](#).

12.18. Figure Command

You can create a figure (an image of the objects in the **3D Viewer**) to include in the report. There are two ways to create a figure:

- From the menu bar, select **Insert > Figure**.
- From the toolbar, click the *Figure*  icon.

To see the new figure, you must open the **Report Viewer** and refresh or publish the report. For details, see [Report \(p. 72\)](#).

Chapter 13: CFD-Post Tools Menu

The **Tools** menu offers access to quantitative analysis utilities, the animation editor, and the timestep selector. The **Command Editor** dialog box is also available so that you can enter CFX Command Language (CCL) directly.

This chapter describes:

- 13.1. Timestep Selector
- 13.2. Animation
- 13.3. Quick Editor
- 13.4. Probe
- 13.5. Function Calculator
- 13.6. Macro Calculator
- 13.7. Mesh Calculator
- 13.8. Case Comparison
- 13.9. Command Editor

13.1. Timestep Selector

For a transient results file, the **Timestep Selector** dialog box enables you to load the results for different timesteps by selecting the timestep and clicking **Apply**.

When reading transient cases, CFD-Post re-reads and re-imports the mesh, if the transient file contains them. This feature enables CFD-Post to support transient rotor/stator problems, as well as moving-mesh cases.

When CFD-Post reads a transient blade row results file, the variables are reconstructed automatically and the flow solution time is taken to the last time step. By default, the **Simulation Timestep** option is used in the **Timestep Selector**, and includes the timesteps used in the CFX-Solver. It is possible to recreate this timestep list by selecting a different **Timestep Sampling** option, and the solution will be accordingly reconstructed for these new time location.

Note:

All variables will always appear in the variables list for all transient files, even if the transient file does not contain some of the variables. If you have the **Load missing variables from nearest FULL timestep** option selected (**Edit > Options > Files > Transient Cases**), then the missing variable data will be loaded from the nearest full timestep. Otherwise, the data will be colored with the undefined color in these cases.

The following list describes the column headings in the list box. Some of these columns also appear in the **Case Comparison** editor in CFD-Post.

- The **Configuration** column indicates the configuration name as set in CFX-Pre. This column appears when you have a multi-configuration (.mr.es) file loaded.
- The **#** column displays the index number for the timestep. These values always begin at 1 and increase by 1.
- The **Step** column displays the timestep number, which is used for synchronization by time step. These values always increase; because they are unique, they can be used in scripts.

For most cases, the values in the **Step** column are the same as those in the **Solver Step** column. However, if you have a multi-configuration case or a case with run history, loaded using the **Load complete history as: A single case** option (described in [Load Results Command \(p. 141\)](#)), then the **Step** is calculated to give a unique, increasing value through all the configurations. It differs from index because it can maintain a consistent value even though (for example) some transient files (.trn) that were present when the run completed are no longer available. (For example, suppose that a case has transient files at three timesteps and these appear in CFD-Post as steps 1, 2, and 3. If you delete the middle transient file, CFD-Post will show entries in the timestep selector at steps 1 and 3, but not 2. If a script was loading step 3, it will load the same results as previously.) Note, however, that if an entire results file (.res) that is referenced by the multi-configuration results file (or the run history) is no longer available, **Step** cannot maintain a consistent value for the remaining entries in the timestep selector. For example, if you load just Step 10, you will not necessarily get the same results loaded at the same timestep as you would have if you had loaded Step 10 before you deleted the .res file.

- The **Solver Step** column displays the solver timestep or outer iteration number. In multi-configuration cases, the solver step may not always increase across different cases and may not be unique.

Solver Step can be used in expressions. Timestep-related expressions such as Current Time Step and Accumulated Time Step refer to the **Solver Step**.

- The **Time [s]** column shows the real time duration corresponding to the timestep. The units are always seconds.
- The **Type** list displays the Partial or Full results file corresponding to that timestep.
- The **Phase** column appears for transient blade row cases. For details, see [Using the Timestep Selector with Transient Blade Row Cases \(p. 331\)](#).
- The **Crank Angle** column appears for internal combustion engine simulations. It displays the crank angle at each associated time step. The crank angle is calculated by multiplying the time by the rotation speed of the crank.

The following icons/commands appear on the right side of the dialog box and/or the shortcut menu accessible by right-clicking on a timestep in the list box.

Icon/Command	Description
Switch To	Selects the timestep. Same as double-clicking the timestep.
 Add timesteps	Opens the Add Timestep Files window, which enables you to select one or more results files and load them into the memory.

Icon/Command	Description
 Delete	Only applies when there are added timesteps. This command enables you to delete added timesteps from the timestep selector.
 Animate	Opens the Animation dialog box. For details, see Animation (p. 334) .

13.1.1. Adding Timesteps

After you load a results file, the **Timestep Selector** dialog box shows the set of timesteps from that file. You can add to the set by clicking *Add Timesteps*  and selecting a file of type res, bak, or trn, or a directory containing files of type trn.

Select **Ignore duplicate timesteps** to avoid loading duplicate timesteps when loading a new file or directory. If this option is not selected, duplicate timesteps will appear at the end of the list, and will be given a unique timestep number.

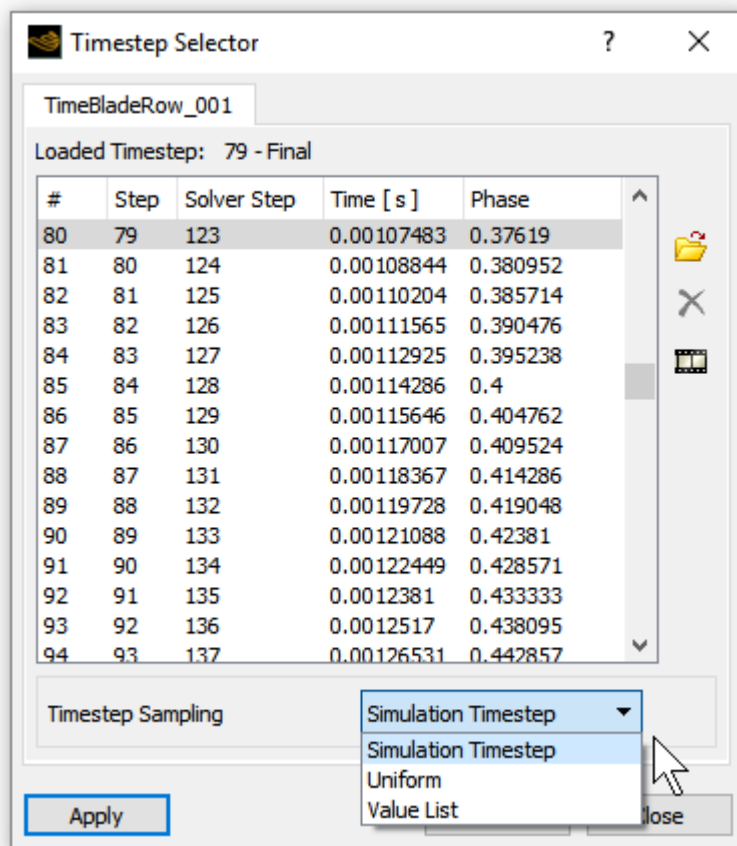
Note:

Adding timesteps to steady-state runs that contain particle tracks causes particles to be displayed up to the current time (which is zero for steady-state runs). To see the full particle track:

1. Open the particle track in question for editing.
 2. On the **Geometry** tab, set **Limits Option** to User Specified and **End Time** to the maximum time value for the simulation.
-

13.1.2. Using the Timestep Selector with Transient Blade Row Cases

The Timestep Selector shows discrete timesteps based on the **Timestep Sampling** option selected. Unlike traditional transient simulations, these discrete timesteps do not represent transient files but time locations where the compressed data is evaluated using Fourier Coefficients from the results file (For more information, see [CFX-Solver Output File \(Transient Blade Row Runs\) in the CFX-Solver Manager User's Guide](#)). These evaluated solutions do not attempt to reproduce the transient iterative process towards a steady periodic behavior. However, they are evaluated from the start of the transient run to facilitate comparisons to traditional transient calculations.



For transient blade row cases, the Timestep Selector has an extra column, **Phase**, and a new sampling control (**Timestep Sampling**):

- The **Phase** values are based on the common period.
- The **Timestep Sampling** options are:

Simulation Timestep (default)

The timestep list (and hence phase position list) is obtained from the results file.

Uniform

You can specify the number of timesteps per phase period (**# of Timesteps per Phase**). The timestep list shows the appropriate number of divisions for each phase period that is listed in the results file.

You can control the range of phase values in the list by selecting **Specify Phase Range** and then specifying minimum and maximum phase values. It is often sufficient to perform postprocessing on a single phase period.

Value List

You can create a custom timestep list based on a set of phase positions that you provide.

The listing of timesteps changes as the Timestep Sampling options are changed. When you click **Apply**, these changes are set appropriately. The selected timestep will be automatically updated to the closest phase position in the new timestep list.

Tip:

For an overview of working with transient blade row cases, see [Transient Blade Row Post-processing \(p. 172\)](#).

13.1.3. Multiple Files

When multiple files are loaded, they appear on separate tabs on the top of the **Timestep Selector** dialog box. The **Sync Cases** setting is available to synchronize the cases in the following ways:

- Off

The Off option causes each set of results to be independent in terms of the selected timestep.

- By Time Step

The By Time Step option causes each set of results to switch to "match" the timestep you select for any set of results. All sets of results are therefore synchronized by timestep. The **Match** setting controls the matching criterion. The Same Step option causes results with identical timesteps to be synchronized, and results without identical timesteps to remain at their current timestep. The Nearest Available option causes the closest timestep to be selected for each set of results if there is not an exact match.

- By Time Value

The By Time Value option causes each set of results to switch to "match" the time value you select for any set of results. All sets of results are therefore synchronized by time value. The **Match** setting controls the matching criterion. The Same Value option causes results with identical time values to be synchronized, and results without identical time values to remain at their current time value. The Nearest Available option causes the closest time value to be selected for each set of results if there is not an exact match. The remaining **Match** options enable different degrees of matching; they are: Within 1%, Within 5%, and Within 10%.

- By Index

The By Index option causes each set of results to switch to "match", as closely as possible, the index number you select for any set of results. All sets of results are therefore synchronized by index.

- By Crank Angle

The By Crank Angle option is only available when multiple internal combustion engine simulations are loaded. It causes each set of results to switch to "match" the crank angle you select for any set of results. All sets of results are therefore synchronized by crank angle. The **Match** setting controls the matching criterion. The Same Value option causes results with identical crank angles to be synchronized, and results without identical crank angles to remain at their current crank angle. The Nearest Available option causes the closest crank angle to be selected for each

set of results if there is not an exact match. The remaining **Match** options enable different degrees of matching; they are: Within 1%, Within 5%, and Within 10%.

Note:

For transient blade row cases, Timestep and Index are the same. Hence, timestep sync By Time Step and By Index are identical options.

13.2. Animation

CFD-Post can produce animations of the following types:

- [Sweep Animation \(p. 334\)](#), which is a means to automatically sweep objects across their defined range.
- [Timestep Animation \(p. 336\)](#), in which the animation is created by stepping through available timesteps.
- [GPU Accelerated Animation \(p. 337\)](#), in which the animation is automatically animated through time using graphics hardware.
- [Keyframe Animation \(p. 340\)](#), in which you define the start and end points of each section of animation using *keyframes*, then link these end points together by having CFD-Post create a number of intermediate frames.

Selecting **Tools > Animation** produces the **Animation** dialog box, where you can choose the type of animation you want.

The following topics are discussed:

[13.2.1. Animation Types](#)

[13.2.2. Animation Dialog Box](#)

[13.2.3. Saving an Animation](#)

[13.2.4. Saving the Animation State \(*.can file\)](#)

13.2.1. Animation Types

The following animation types are available:

[13.2.1.1. Sweep Animation](#)

[13.2.1.2. Timestep Animation](#)

[13.2.1.3. GPU Accelerated Animation](#)

[13.2.1.4. Keyframe Animation](#)

13.2.1.1. Sweep Animation

The Sweep Animation option provides a means to automatically sweep objects across their defined range to visualize the data throughout the domain. Planes, isosurfaces, turbo surfaces, streamlines, and particle tracks can all be animated with the Quick Animator.

To activate the Quick Animator, right-click an object in the **3D Viewer** and select **Animate**, or open **Tools > Animation** and select Sweep Animation.

Use the slider to select the number of frames per animation loop. The more frames you add, the more positions the animating object will go through. The number of frames increases logarithmically as you move the slider toward the **Slow** end.

You can animate as many objects, of any type, as you want. Just select the objects in the list and click *Play* ; all selected objects will animate.

To stop an animation in progress, click *Stop* . The objects will return to the state they were in before the animation began.

You can specify a number of repetitions (raise the *Repeat forever*  button to enable the **Repeat** field).

You can create an animation in any of a variety of formats by selecting the **Save Movie** option, specifying the **Format**, and providing a filename. Select the **Options** button to select video creation and quality options, just as for keyframe animations.

Note:

The Windows Media Video (WMV), AVI, and MPEG4 format options all use MPEG-4 encoding, so you will need a player that supports MPEG-4 to view animations in those formats.

13.2.1.1.1. Animating Planes

An animated plane will be shifted in a direction normal to its surface.

If the **Bounce** option is selected (default), the plane will move to the positive limit, and then in reverse to the negative limit, and then repeat, moving to the positive limit again. If the **Loop** option is selected, the plane will move to the positive limit, and then jump to the negative limit (in one frame), and then start moving to the positive limit again.

Depending on the shape of the domain relative to its bounding box and the plane orientation, the animating plane may disappear for a number of frames at the ends of its ranges.

13.2.1.1.2. Animating Isosurfaces

Isosurface value is modified to traverse through the entire variable range.

If the **Bounce** option is selected (default), the isosurface value is increased to its maximum value, and then decreased to its minimum value, and repeated. If the **Loop** option is selected, the isosurface value is increased to its maximum value, then set to its minimum value (in one frame), and then increased to its maximum value again.

13.2.1.1.3. Animating Turbo Surfaces

Depending on the surface type, Span, Streamwise Location or Theta value will be modified to sweep through its respective range.

Note:

Sweep animation will not work if a **Turbo Surface** is defined using the Cone option.

13.2.1.1.4. Animating Streamlines and Particle Tracks

Streamlines and Particle Tracks symbols are shifted along the lines.

By default, the symbol shape, interval, and size are overridden by the animation routines. If you want to change these settings, click the **Options...** button. To use symbol settings from the original object, clear the **Override Symbol Settings** check box.

The **Spacing** option specifies the interval at which to start a new batch of symbols. For example, with a spacing of 0.6, the symbols animating on the object will move 60% of the way along the lines, at which point another group of symbols will start at the beginning of the line. With a spacing of 1.0, all symbols will travel to the end of their lines before a new group of symbols starts at the beginning.

13.2.1.1.5. Animating Mesh Deformation Scaling

The Deformation scaling factor is animated from **Undeformed** to the current scaling factor. If the current scaling is set to **Undeformed**, the animation goes from **Undeformed** to **True Scale**.

13.2.1.2. Timestep Animation

The Timestep Animation option enables you to create an animation by stepping through timesteps, allowing you to easily animate results.

To put the **Animation** dialog box in Timestep Animation mode, set **Type** to Timestep Animation.

In addition to some of the standard buttons such as Play and Stop on the **Animation** dialog box, there are additional buttons:

- , go back one timestep.
- , go forward one timestep.

The **Control By** option determines which variable will be available to define the extents of the animation. There are 3 options:

- Timestep: Uses timesteps to define the animation. You can use the [Timestep Selector \(p. 329\)](#) () to select specific timesteps.
- Time: Uses time to define the animation, and is only available for transient simulations.

- **Crank Angle:** Uses crank angle to define the animation, and is only available for IC Engine simulations.

Select **Specify Range for Animation** to specify the start and end points of the animation (based on the variable chosen for **Control By**), otherwise CFD-Post will animate every timestep available.

You can expand the **Advanced Frame Selection Controls** panel if you want more control over which timesteps will be used to create the animation. The options are dependent on the case:

- **All Timesteps In Range:** Uses every timestep available to create the animation between the specified range (or every timestep in CFD-Post if no range has been specified).
- **Specified Number of Frames:** Enables you to specify the **# of Frames** and also their **Spacing**. You can space the frames by **Equal In Timestep**, **Equal In Time** (for transient cases), or **Equal In Crank Angle** (for IC Engine cases). When using this option, CFD-Post creates the animation using the start and end points of animation (either user specified or, by default, every timestep) and divides the animation according to the **# of Frames**. CFD-Post then creates the animation using the timesteps that are nearest to the division points (be they based on timestep, time, or crank angle depending on the **Spacing** option). The result may include duplicate timesteps if the specified frame interval is smaller than the interval between the available timesteps. These will not be seen when viewing the animation in CFD-Post but will exist in any generated video file.
- **Specified Time Interval:** Only available for transient simulations. Frames are included at the specified **Interval** between the start and end points of the animation. Similar to the **Specified Number of Frames** option, CFD-Post creates the animation using the timesteps nearest to the time values generated using the specified interval. The result may include duplicate timesteps if the specified frame interval is smaller than the interval between the available timesteps. These will not be seen when viewing the animation in CFD-Post but will exist in any generated video file.
- **Specified Crank Angle Interval:** only available for IC Engine simulations. Uses the same behavior as **Specified Time Interval** except that it uses the crank angle variable instead of time variable.
- **Specified Timestep Interval:** Uses the same behavior as **Specified Timestep Interval** except that it uses the timestep variable instead of the time variable.

You can specify a number of repetitions (raise the *Repeat forever*  button to enable the **Repeat** field).

You can create an animation in any of a variety of formats by selecting the **Save Movie** option, specifying the **Format**, and providing a filename. Select the **Options** button to select video creation and quality options, just as for keyframe animations.

13.2.1.3. GPU Accelerated Animation

The **GPU Accelerated Animation** option enables you to create animations using graphics card GPU acceleration. Such animations cycle with a relatively high frame rate.

To use GPU Accelerated Animation:

1. Ensure that preference **Enable GPU Shader Rendering** is selected.
For details, see [Advanced](#) (p. 204).
2. In the **Animation** dialog box, set **Type** to GPU Accelerated Animation.
3. Set **Animation Type** to **Sweep** or **Transient**.

Certain objects are eligible to be animated for each type. For example, streamlines that use the **Animated Time** coloring mode are eligible to be animated using the **Sweep** animation type, whereas objects colored using Fourier coefficients (from a transient blade row case) are eligible to be animated using the **Transient** animation type. Any objects not eligible to be animated, or not selected for animation, remain frozen during an animation.

4. In the list, select objects to animate, such as:
 - Turbo surfaces from a transient blade row case, colored by a variable
 - Surfaces colored by an expression
 - Streamlines that use the **Animated Time** option.

Note:

For any given object to be eligible for GPU Accelerated Animation, the details view for all associated domains must show, on the **Data Instancing** tab, that exactly 1 instance is set.

If you want to show multiple instances of an object, go to the details view of each associated domain and, on the **Instancing** tab, set the number of graphical instances. For objects that display TBR results, graphical instancing, as set in the domain details view on the **Instancing** tab, acts like data instancing would without GPU acceleration, in the sense that variables are not simply copied to new instances but are instead calculated for each instance using a Fourier series.

5. Click **Play** or **Stop** to start or stop the animation.
Note that the animation plays in a loop until you click **Stop**.
6. You can change the animation speed using the **Playback Time** slider.
7. You can change the **Phase Position** (that is, the phase position within the common period) of the selected objects.

When using GPU Accelerated Animation, the following limitations apply:

- Vectors are not animated or colored.
- Any object that is colored using an expression should use a user defined range for coloring because the local and global ranges defined by CFD-Post for such expressions might not reflect the true variable ranges.

- Numerical error can differ between CPU rendered images and GPU rendered images. Use best practices to reduce numerical error when formulating CEL expressions to color GPU rendered objects.
- For transient blade row results, accuracy of the graphics (in particular, accuracy of the variable values by which objects may be colored) depends on the number of Fourier coefficients used to calculate variable values. You can adjust this number by the environment variable `CFD_POST_GPU_ANIMATION_TBR_MAXIMUM_FOURIER_COEFFICIENT_PAIRS`. For details, see [Advanced](#) (p. 204).
- GPU Accelerated Animation does not support objects that have more than one data instance, as set in the domain details view on the **Data Instancing** tab. Instead, simply set the number of instances on the **Instances** tab; with preference **Enable GPU Shader Rendering** selected, the resulting instances are each calculated from a Fourier series, yielding instancing much like data instancing would without the preference selected.
- Your graphics card and its drivers must be sufficiently recent and capable of using OpenGL shaders.
- Text objects are not updated, including those with embedded auto-annotation.
- Coloring by a CEL expression has the following limitations:
 - For transient blade row results, the expression cannot contain more than one variable and cannot be used for a **Sweep** animation for any surface objects.
 - The following parts of a CEL expression are evaluated only once, at the current time step, and are then held constant throughout the entire animation:
 - Any user CEL function of type `User Function` (a user CEL function driven by an existing user CEL routine object)
 - Any location-dependent function, such as `areaAve`
 - The `Current Time Step` and `Current Phase Position` expressions are evaluated only once, at the current time step, and are then held constant throughout the entire animation.
 - User functions within the CEL expression must return a single value.
 - CEL functions `besselJ` and `besselY` are not supported for use in GPU rendering.

You can specify a number of repetitions (raise the *Repeat forever*  button to enable the **Repeat** field).

You can create an animation in any of a variety of formats by selecting the **Save Movie** option, specifying the **Format**, and providing a filename. Select the **Options** button to select video creation and quality options, just as for keyframe animations.

13.2.1.4. Keyframe Animation

Note:

The Windows Media Video (WMV), AVI, and MPEG4 format options all use MPEG-4 encoding, so you will need a player that supports MPEG-4 to view animations in those formats.

In CFD-Post, you can make animations based on keyframes. Keyframes define the start and end points of each section of animation. Keyframes are linked together by drawing a number of intermediate frames, the number of which is set by the **# of Frames** field in the **Animation** dialog box.

The basic approach to creating an animation sequence is to configure the problem in a particular state and then save this state as a keyframe. Next, change one or more aspects of the problem state. For example, change the viewer orientation by rotating the viewer object. You can then save this state as a second keyframe.

Animations are created by linearly interpolating the change in state of the viewer position between keyframe states. By default, 10 frames are created between keyframe states, but this is easily adjustable. If the camera position changes between keyframes, the view is interpolated between the two positions at each frame.

Every option and button that is accessible when **Animation** is active will increment by one linearly for each of the frames between the two states. For example, if one keyframe has 10 contour levels, and the next has 20 contour levels, then the number of contour levels will increment by one for each of the ten frames between the two states. Objects that are binary in state are toggled at the end of the keyframe sequence (for example, the visibility of an object). Animations can also be created using the different timesteps in a transient run.

Note:

If you have 2 keyframes with 10 frames between them, there are a total of 11 steps from one keyframe to the next.

13.2.1.4.1. Creating an Animation

The basic steps to creating an animation are as follows:

1. Once you have manipulated the user interface into a chosen start position, click **New**  to set the current state as **Keyframe 1**.
2. The keyframe becomes visible in the **Keyframe Creation and Editing** window.
3. Change the viewer and/or object parameters to obtain the second required state and click **New**  to create **Keyframe 2**.
4. When you click a keyframe to highlight it, the other options to the right of the keyframe list become active.

- To display the highlighted keyframe in the viewer, click *Edit Keyframe*  or double-click the keyframe itself. To apply changes in the viewer to the highlighted keyframe, click *Set Keyframe* . If more than 2 keyframes exist and you want to change their order, you can move a keyframe up and down by clicking on the blue arrows. To delete a keyframe, click *Delete* .
- To set the number of intermediate interpolated frames, click a keyframe and set the value in the **# of Frames** box.

After a second keyframe has been created, additional playback options are made available.

- The looping option enables you to specify whether you want the animation to play in one direction during each repeat or play forwards and backwards. For example, selecting **Repeat** of 3 on the **Loop** setting will play the animation three times, jumping from the last keyframe back to the first at the end of the first two cycles. Selecting **Bounce** for the same number of repeats will cause the animation to play forwards, and then backwards before playing forwards once more.

With the **Repeat** option, you specify how often the animation repeats before stopping. By default the *Repeat forever*  button is selected, so the animation will repeat continuously until you click *Stop* .

- The **Animate Camera** feature toggles whether the camera position is moved (interpolated) with the animation.

If it is switched off, all objects, except for the camera positions, are animated.

13.2.1.4.2. Animating Expressions

There is a limitation with respect to the animating of expressions. If the value of a parameter of an object is set to an expression, that expression is evaluated at the keyframes, but those values are not interpolated to obtain values at the frames between the keyframes. Thus, after the value of the parameter is determined for the first keyframe, that value does not change for the intermediate frames until it is recalculated at the next keyframe, which causes the animation to be unexpectedly discontinuous.

13.2.2. Animation Dialog Box

The following is a general explanation of the icons in the **Animation** dialog box:

Icon	Description	Icon	Description
	Create a new keyframe		Go to beginning
	Edit a keyframe		Go to previous keyframe
	Set the keyframe		Go to previous frame

Icon	Description	Icon	Description
	Move the keyframe up		Go to next frame
	Move the keyframe down		Go to next keyframe
	Delete the keyframe		Go to end
	Load animation state		Play forward
	Save animation state		Stop the animation
	More animation options		Repeat forever

13.2.2.1. Animation Options Dialog Box: Options Tab

The **Animation Options** dialog box is opened by expanding *More animation options*  at the bottom of the **Animation** dialog box, then clicking **Options**.

13.2.2.1.1. Animation Speed

The **Animation Speed** settings enable you to scale the animation to speed it up or slow it down without having to manually adjust the number of frames between keyframes in the animation.

The Approximate Animation Time is calculated with the following information: total number of frames in the animation, the number of repetitions, the frame rate (regardless of whether you are saving to a movie or not), and any animation speed adjustments.

Selecting an animation speed of **Normal** does not scale the animation by any factor.

Selecting an animation speed of **Slower** slows down the animation by adding sufficient additional frames to achieve the specified factor. Selecting **Generate more frames, spread evenly** automatically and transparently adds additional frames between keyframes. You will see the effect of this the next time you play the animation. This results in higher quality animations, but will take longer to compute because of the additional frames to interpolate. Selecting **Duplicate frames when saving movie** duplicates existing frames when generating the final movie output. The effect of this will be visible only when playing back the movie; you will see no effect when playing the animation in CFD-Post. This option is faster, but the quality of the movie may suffer: it may look a little jerky.

Selecting an animation speed of **Faster** speeds up the animation by removing sufficient frames to achieve the specified factor. Selecting **Generate fewer frames, spread evenly** automatically and transparently removes some of the frames between keyframes. You will see the effect of this next time you play the animation. The fewer frames between keyframes will be interpolated smoothly, as if you had reduced the number of frames manually. Selecting **Skip frames when saving movie** skips existing frames only when generating the final movie output. The effect of this will only be visible when playing back the movie file; you will see no effect when playing the animation in CFD-Post. This option is slower because all frames will be played in CFD-Post, but only some of the frames will be used to generate the movie.

13.2.2.1.2. Transient Case

The **Transient Case** setting is effective only for transient simulations and controls the way in which timesteps are selected. A particular frame is calculated. **Sequential Interpolation** evenly distributes frames over each transient output file. **Timestep Interpolation** evenly distributes frames based on the timestep number associated with each transient output file. **TimeValue Interpolation** evenly distributes frames based on the time value associated with each transient output file.

13.2.2.1.3. Print Options

13.2.2.1.3.1. Image Format

Select either a JPEG or PPM format for creation of the movie.

13.2.2.1.3.2. White Background

Toggles between a white/black background.

13.2.2.1.3.3. Enhanced Output (Smooth Edges)

Enables you to select higher quality output for the generated images.

13.2.2.1.3.4. Image Size

Enables you to specify the resolution of the resulting movie.

You can select any of the values in the drop-down list, including NTSC or PAL standard resolutions, or HD resolutions. Note that:

- HD Video 2k produces video 2560 pixels wide and 1440 pixels high
- HD Video 4k produces video 3840 pixels wide and 2160 pixels high.

You can select **Custom** to specify the pixel resolution in the **Width** and **Height** fields, or select **Use Screen Size** and specify a scale factor in the **Scale (%)** field.

13.2.2.1.3.5. Tolerance

Controls the amount of depth calculated for the creation of an image, where smaller values represent more accurate images. The benefit of relatively high values is that less processing is required. However, if the **Tolerance** value is too high (for instance, a value of 1), the back faces in an image may be displayed on top of near faces.

13.2.2.2. Animation Options Dialog Box: Advanced Tab

13.2.2.2.1. Save Frames As Image Files

If you have selected **Save Movie** (see [Sweep Animation \(p. 334\)](#)), selecting **Save Frames As Image Files** will prevent the deletion of the animation frame files from the temporary directory, where they are stored by default.

13.2.2.2.2. Output To User Directory

Selecting this option enables you to specify where you want the animation files to be saved by entering a path in the **Directory** field.

13.2.2.2.3. Frame Rate

The rate (in frames per second) at which the movie will be generated. The movie viewer may also dictate the playback rate.

13.2.2.2.4. Quality

Select a **Quality** from: Highest, High, Medium, Low, or Custom.

With the Custom setting, you may specify the **Variable Bit Rate** by clearing the **Variable Bit Rate** toggle and entering a bit rate. Reduce the **Bit Rate** value to lower the file size (and the file playback quality).

13.2.2.2.5. Don't Encode Last MPEG Frame

A single cycle of an animation loop starts and ends at the same frame. If you repeat a loop, that frame is encoded twice at the end of each cycle, leading to a brief pause at that point in the animation. Enable this setting to smooth the playback of repeated loop animations.

13.2.3. Saving an Animation

When **Save Movie** is selected and a filename is specified, the animation is saved to a file when the animation is played.

1. To select a file, click *Browse*  and browse to a convenient location.
2. Enter the name of the file; the extension is taken from the setting of the **Format** field.

Note:

The Windows Media Video (WMV), AVI, and MPEG4 format options all use MPEG-4 encoding, so you will need a player that supports MPEG-4 to view animations in those formats.

3. Click **Save** to save the file.

13.2.4. Saving the Animation State (*.can file)

You can load or save your animation state as a .can (CFX Animation) file. It saves the current status of all of the animation settings.

To open:

1. Browse to the correct directory to load the file.

2. Enter the name of the file or select it by using the mouse.
3. Click **Open**.

To save:

1. Browse to a convenient directory to save the file.
2. Enter or select the name of the file.
3. You should save the file as a .can (CFX Animation) type.

Note:

The animation state includes all of the keyframe settings for Keyframe Animation. Keyframe settings capture the complete state of CFD-Post as it was when the keyframe was created.

- **When running in stand-alone mode**

If an animation file containing keyframes is loaded into CFD-Post, then the whole of the current state will be overwritten, including which cases are currently loaded and which results files were used to load them.

- **When running under Ansys Workbench**

If an animation file containing keyframes is loaded into CFD-Post, then the whole of the current state will be overwritten, except that cases loaded automatically by Ansys Workbench (for example, from a Solution cell) will be retained with their current results. Files that are referred to in the keyframe state that have not been loaded automatically by Ansys Workbench (for example, files used for creating User Surfaces or Chart Series or results files loaded manually) may not be referenced correctly by the keyframe state if the project has been saved under another name or restored from an archive. Loading animation files created under another project or outside of Ansys Workbench is not supported.

13.3. Quick Editor

The Quick Editor in CFD-Post lets you easily perform repetitious modifications to certain objects. You can move planes along their normals to a specified location, set the value of isosurfaces, and set turbo surface locations.

To specify a value, you can enter a number in the value editor, move the slider, or click left/right mouse buttons to increment/decrement the value by a portion of the range. All changes are applied immediately; there is no need to click **Apply**.

13.4. Probe

Probe in CFD-Post enables you to determine exact variable values at specified points within a domain.

1. Select **Tools > Probe** or click *Probe* , or right-click an object in the viewer and select **Probe Variable**.

The **Probe** tool appears at the bottom of the viewer.

2. You can manually specify the probe coordinates in the **Probe At** fields or select a point in the viewer.

If *Probe only this variable*  is not selected, the probe variable will be automatically chosen. (For example, Temperature will be selected if you select a point on a plane that is colored by Temperature).

3. The probe variable can also be selected manually from the variable list.
4. If the desired variable does not appear in the list, select **Other...** and choose the variable from the **Variable Selector**.
5. If *Probe only this variable*  is selected, the probe variable will not change automatically when new coordinates are entered.
6. Select *Hide Probe tool*  to close the **Probe** tool.

The probed value appears in the box adjacent to the variable list and automatically updates every time a new coordinate or probe variable is selected.

If you probe on a **Point** object, the probe position will use the position coordinates of the **Point** object, not necessarily exactly where you chose.

Note:

- The **Probe** tool returns a conservative, not hybrid, variable value derived from volume, not surface, data.

For example, if you select a location on a surface that is colored according to a hybrid value, the conservative value returned by the **Probe** tool does not necessarily match the value indicated by the color.

For details, see [Hybrid and Conservative Variable Values](#).

- Probe locations will be selected more accurately when you zoom in tightly on the probe location when picking in the viewer. The smaller the object in the viewer is, the less accurate the picked location will be. A consequence is that you may get an undefined value on an outer boundary because the point location will be slightly outside the domain. This problem may disappear if you zoom in on the boundary and probe again. The **Edit > Options > CFD-Post > Interpolation Tolerance** setting controls the distance by which a point can fall outside a domain and still acquire data from that domain. For details, see [CFD-Post Options](#) (p. 203).

13.5. Function Calculator

The Function Calculator is used to provide quantitative information about the results. To use the Function Calculator:

1. Select the function to evaluate from this list.

2. Choose the location for the calculation.

Only locations valid for the selected function will be available.

3. If multiple cases are open, choose which cases the Function Calculator should act upon.

4. If applicable to the function, choose a variable from the list.

For most functions, you can click in the **Variable** box and enter an expression. The expression can include variables and any valid CEL (CFX Expression Language) function. For example, `abs(Velocity u)` could be entered so that the calculation is performed using the absolute values of the variable `Velocity u`. For details, see [CEL Operators, Constants, and Expressions](#) and [CFX Expression Language \(CEL\)](#).

User variables are also available. For details, see [Variables Workspace \(p. 89\)](#).

5. Select the direction if applicable to the function.

For some functions None is available.

For details, see [Quantitative Function List](#).

6. If applicable to the function, select the appropriate fluid.

For multiphase results, you can select which fluids to use in your calculation for selected functions. The **All Fluids** option can be selected to perform the calculation using all of the fluids in the results.

Note:

When calculating mass flow rate for a Fluent file, the option **Mixture** gives the same results as **All Fluids**. These two options appear because have different origins (Fluent and CFD-Post respectively); you may choose either for your calculations.

Click **Calculate** to calculate the result. Choose whether to base the calculation on hybrid or conservative values. Most quantitative calculations are best performed using conservative variable values. For details, see [Hybrid and Conservative Variable Values](#).

Note:

If the function result is a temperature, and if C or K are selected as temperature units, the result's units will be K. If F or R are selected, the temperature will be returned in R.

This has an implication for calculations of temperature differences measured in C or F. Expressions are always evaluated in absolute temperature units (K or R) and then, if necessary,

the result is converted to the user-selected units. For example, if you evaluate $1[\text{C}] - 1[\text{C}]$, internally it is evaluated as $274[\text{K}] - 274[\text{K}]$, which is $0[\text{K}]$ and is reported as such (with the units forced to be in an absolute scale). In plots (where CFD-Post cannot force the units to be K), the software cannot tell whether the result is a temperature difference or just the temperature, so the result is converted to user-selected units (in this case, $-273[\text{C}]$) and a value of -273 is reported in the plot legend. Thus when analyzing temperature differences, set the preferred temperature units to be in an absolute scale (K or R) in the **Edit > Options > Units** dialog box.

Important:

There are some important limitations concerning calculations performed on CFX-4 results files. For details, see [CFX-4 Dump Files \(p. 197\)](#).

13.5.1. Function Selection

The quantitative functions available from the Function Calculator in CFD-Post are integrated into CEL and can be used in any expression.

The available quantitative functions are outlined in the table below.

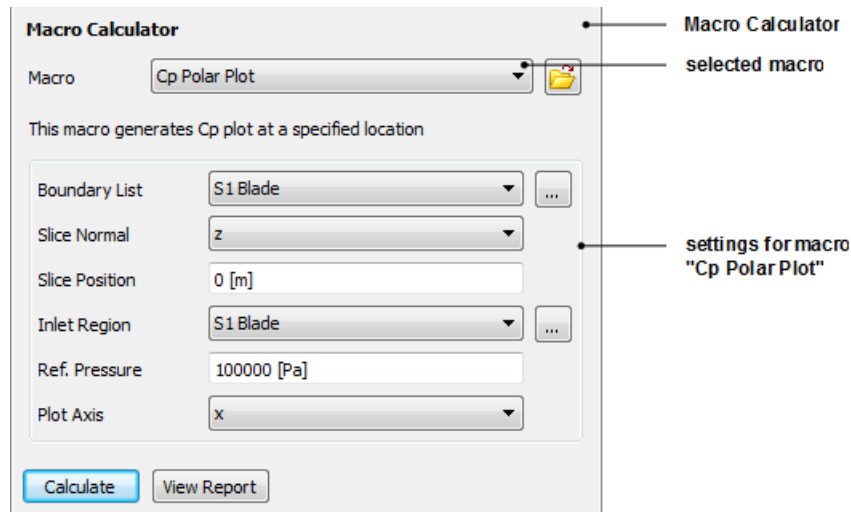
Function Name	Operation
area	Area of location
areaAve	Area-weighted average Note: Projected areaAve (for example, areaAve_x) works as expected only for surfaces that do not fold in the selected direction. In extreme case, if the surface is fully closed, the projected average will result in a randomly large number, as the projected area will be zero.
areaInt	Area-weighted integral (can be projected to a direction)
ave	Arithmetic average
count	Number of Nodes
countTrue	Number of nodes at which the logical expression evaluates to true.
force	Force on a surface in the specified direction
forceNorm	Length normalized force on a curve in the specified direction
length	Length of a line

Function Name	Operation
lengthAve	Length-weighted average
lengthInt	Length-weighted integration
massFlow	Total mass flow Note: Mass flow convention for CFD-Post is negative in the direction of the surface normal.
massFlowAve	Mass Flow-weighted average
massFlowAveAbs	Mass Flow-weighted average with absolute values of mass flow used in numerator and denominator of formula for averaging
massFlowInt	Mass Flow-weighted integral
maxVal	Maximum Value
minVal	Minimum Value
probe	Value at a point
sum	Sum over the calculation points
torque	Torque on a surface about the specified axis
volume	Volume of a 3D location
volumeAve	Volume-weighted average
volumeInt	Volume-weighted integral Note: The volumeInt function does not take into account the porosity of the location specified for porous domains. To include the porosity effect in your calculation, you need to manually multiply your argument by: • The Volume Porosity if you want to evaluate the integral on the fluid side • (1-Volume Porosity) if you want to evaluate the integral on the solid side

For details on each of the functions listed in the table above, see [Quantitative Function List in the CFX Reference Guide](#).

13.6. Macro Calculator

The Macro Calculator is a facility for running *macros*, which are small programs that perform calculations and generate various forms of output.



For example, there is a macro named `Fan Noise` that calculates the noise generated by a low-speed fan. For this macro, you need to specify inputs such as the number of blades, the number of harmonics, and the position of the observer.

Macro Calculator

Macro Fan Noise 

Calculates the tonal noise generated by a low speed fan (Mach Number less than 0.4)

Domain	S1  
Blade Selection	Automatic 
> Custom Blade	S1 Blade  
> Custom # Blades	0 
# of Harmonics	6 
Observer (r)	1 [m]
Observer (theta)	0 [degree]
Theta Sectors	36 
Directivity Harm. #	1 
Loading Coeff.	2.2
Acou. Ref. Pressure	2e-5 [Pa]
Acou. Ref. Power	1e-12 [W m^-3]
Sound Speed	340 [m/s]

The following topics are discussed:

- [13.6.1. Running Macros from the Macro Calculator](#)
- [13.6.2. Macro Availability](#)
- [13.6.3. Predefined Macros](#)
- [13.6.4. User-defined Macros](#)

13.6.1. Running Macros from the Macro Calculator

To run a macro (assuming that a case is already loaded):

1. From the menu bar, select **Tools > Macro Calculator**.
2. Select an appropriate macro from the drop-down list or open a CFD-Post session file that contains a user-defined macro definition. (In the latter case, opening the file both loads the macro into the Macro Calculator and adds that macro to the drop-down list.)
3. Fill in the settings that appear in the Macro Calculator.

These settings serve to provide input data to the macro. Which settings are required depends on which macro is selected.

4. Click **Calculate** to run the macro.

The macro might create objects, user variables, expressions, and other forms of output.

Some macros produce an HTML-formatted report that can be viewed by clicking the **View Report** button or by opening the **Report Viewer**.

13.6.2. Macro Availability

In the Macro Calculator, the **Macro** setting has a drop-down list that lists the available macros.

Some macros are predefined and are already in the list of available macros. Other macros (user-defined macros) can be added to the list of available macros in any of the following ways:

- In the Macro Calculator, click *Browse*  and select the macro file (a CFD-Post session file with extension .cse). This will make the macro available for only the current session of CFD-Post.
- Add the pathname of your macro file (which has the file extension .cse) to environment variable CFXPOST_USER_MACROS, which is a comma-separated list of the pathnames of your macro files.

13.6.3. Predefined Macros

There are predefined macros that are readily available in the Macro Calculator. These predefined macros are provided in the form of session files (file extension .cse) that are located in `<CFDPOST-ROOT>/etc/` and that are loaded via `<CFDPOSTROOT>/etc/CFXPostInit.ccl` upon starting CFD-Post.

Some macros produce an HTML-formatted report that can be viewed by clicking the **View Report** button or by opening the **Report Viewer**.

The following sections describe the predefined macros:

[13.6.3.1. Comfort Factors Macro](#)

[13.6.3.2. Cp Polar Plot Macro](#)

[13.6.3.3. Gas Compressor Performance Macro](#)

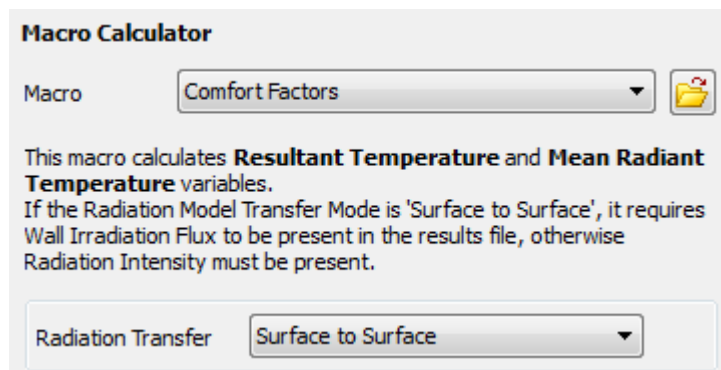
[13.6.3.4. Gas Turbine Performance Macro](#)

[13.6.3.5. Liquid Pump Performance Macro](#)

[13.6.3.6. Liquid Turbine Performance Macro](#)

[13.6.3.7. Fan Noise Macro](#)

13.6.3.1. Comfort Factors Macro



The **Comfort Factors** macro can be used to calculate values for Resultant Temperature and Mean Radiant Temperature in HVAC simulations.

In order to use the macro, the velocity and temperature variables are required. In addition:

- If **Radiation Transfer** is set to Surface to Surface then variable Wall Irradiation Flux is required.
- If **Radiation Transfer** is set to Participating Media then variable Radiation Intensity is required.

For CFX cases, the Radiation Transfer selection should match the radiation transfer mode used in the CFX-Solver. For non-CFX cases, the selection should depend on which radiation variables are available.

Variables Resultant Temperature and Mean Radiant Temperature are created using the values of expressions resultTemp and meanTemp, respectively. These expressions are visible on the **Expressions** tab.

Note:

As an alternative to calculating comfort factors in CFD-Post, the comfort factors may be calculated during the solution process; this would be required, for example, when the model simulates a ventilation system in which the control system depends dynamically on derived comfort factors.

Note:

For more information regarding variables Wall Irradiation Flux and Radiation Intensity, which may be used by the macro, see [Variables Relevant for Radiation Calculations in the CFX Reference Guide](#).

13.6.3.2. Cp Polar Plot Macro

Macro Calculator

Macro: **Cp Polar Plot**

This macro generates Cp plot at a specified location

Boundary List	S1 Blade	...
Slice Normal	z	
Slice Position	0 [m]	
Inlet Region	S1 Blade	...
Ref. Pressure	100000 [Pa]	
Plot Axis	x	

The **Cp Polar** macro produces a polar plot of the pressure coefficient (Cp) along a polyline. The macro creates the polyline using the Boundary Intersection method. For details, see [Polyline Command](#) (p. 252). The boundary and intersecting slice plane are defined in the Macro Calculator and passed to the subroutine as arguments. The boundaries selected for **Boundary List** in the Macro Calculator make up one surface for the intersection. The second surface is a slice plane created using the X, Y, or Z normal axis to the plane (**Slice Normal**) and a point on that axis (**Slice Position**).

The cp user variable is created by the macro from the cp expression. The cp expression can be defined as:

```
(Pressure - $pref [Pa])/dynHead
```

where \$pref is the **Ref. Pressure** set in the Macro Calculator and dynHead is a reference dynamic head (evaluated at the inlet) that can be defined as:

```
0.5 * areaAve(Density)@inlet * areaAve(Velocity)@inlet^2
```

The **Inlet Region** selected in the Macro Calculator is used as the `inlet` location in the calculation of dynHead.

Next, a Chart line of the cp variable versus the **Plot X Axis** value is created. The generated report contains the chart and the settings from the Macro Calculator.

The following information must be specified:

- Boundary List: A list of boundaries used in the simulation.
- Slice Normal: The axis that will be normal to the slice plane.
- Slice Position: The offset of the slice plane in the direction specified by the normal axis.
- Inlet Region: The locator used to calculate inlet quantities.
- Ref. Pressure: The reference pressure for the simulation.

- Plot Axis: The axis on which the results will be plotted.

13.6.3.3. Gas Compressor Performance Macro

Macro Calculator

Macro: Gas Compressor Performance

Inlet Region	S1 Blade	
Outlet Region	S1 Blade	
Rotor Blade(s)	S1 Blade	
Machine Axis	z	
Rot. Speed	1000 [rad/sec]	
Num. Main Blades	1	
Ref. Pressure	100000 [Pa]	
Ref Radius	0.05 [m]	
Fluid Gamma	1.4	

The **Compressor Performance** macro performs a series of calculations using the data set in the Macro Calculator. The following information must be specified:

- Inlet Region: The locator used to calculate inlet quantities.
- Outlet Region: The locator used to calculate outlet quantities.
- Rotor Blade(s): The locator used to calculate torque (one blade row) about the machine axis.
- Machine Axis: The axis of rotation of the compressor.
- Rot. Speed: The rotational speed of the compressor.
- Num. Main Blades: Some quantities calculated for a single blade set (main blade and any splitter blades) are multiplied by the number of blade sets in the full 360° wheel in order to produce the total value for the wheel.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: A reference radius between the hub and tip.
- Fluid Gamma: The ratio of specific heat capacity at constant pressure to specific heat capacity at constant volume (C_p / C_v).

13.6.3.4. Gas Turbine Performance Macro

The screenshot shows the 'Macro Calculator' dialog box with the 'Gas Turbine Performance' macro selected. The dialog contains the following fields and values:

Parameter	Value
Macro	Gas Turbine Performance
Inlet Region	S1 Blade
Outlet Region	S1 Blade
Rotor Blade(s)	S1 Blade
Machine Axis	z
Rot. Speed	1000 [rad/sec]
Num. Main Blades	1
Ref. Pressure	100000 [Pa]
Ref Radius	0.05 [m]
Fluid Gamma	1.4

The following information must be specified:

- Inlet Region: The locator used to calculate inlet quantities.
- Outlet Region: The locator used to calculate outlet quantities.
- Rotor Blade(s): The locator used to calculate torque (one blade row) about the machine axis.
- Machine Axis: The axis or rotation of the turbine.
- Rot. Speed: The rotational speed of the turbine.
- Num. Main Blades: Some quantities calculated for a single blade set (main blade and any splitter blades) are multiplied by the number of blade sets in the full 360° wheel in order to produce the total value for the wheel.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: Reference radius between the hub and tip.
- Fluid Gamma: The ratio of specific heat capacity at constant pressure to specific heat capacity at constant volume (C_p / C_v).

13.6.3.5. Liquid Pump Performance Macro

Macro Calculator

Macro: Liquid Pump Performance

Inlet	S1 Blade	
Outlet	S1 Blade	
Rotor Blade(s)	S1 Blade	
Machine Axis	z	
Rot. Speed	1000 [rad/sec]	
Num. Main Blades	1	
Ref. Pressure	100000 [Pa]	
Ref Radius	0.05 [m]	
Ref Height	0.05 [m]	
Ref. Density	1000 [kg/m^3]	
Gravity Accel.	9.81 [m/s^2]	

The following information must be specified:

- Inlet: The locator used to calculate inlet quantities.
- Outlet: The locator used to calculate outlet quantities.
- Rotor Blade(s): The locator(s) used to calculate torque (one blade row) about the machine axis.
- Machine Axis: The axis or rotation of the pump.
- Rot. Speed: The rotational speed of the pump.
- Num. Main Blades: Some quantities calculated for a single blade set (main blade and any splitter blades) are multiplied by the number of blade sets in the full 360° wheel in order to produce the total value for the wheel.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: Reference radius between the hub and tip.
- Ref Height: Cross-section height (that is, the height of the outlet region, or the height of the blade at the trailing edge).
- Ref Density: The reference density for the simulation.
- Gravity Accel.: The acceleration due to gravity.

13.6.3.6. Liquid Turbine Performance Macro

Macro Calculator

Macro Liquid Turbine Performance 

Inlet	S1 Blade	
Outlet	S1 Blade	
Rotor Blade(s)	S1 Blade	
Machine Axis	z	
Rot. Speed	1000 [rad/sec]	
Num. Main Blades	1  	
Ref. Pressure	100000 [Pa]	
Ref Radius	0.05 [m]	
Ref Height	0.05 [m]	
Flow Rate	0.1 [m^3/s]	
Head Rise	1 [m]	
Ref. Density	1000 [kg/m^3]	
Gravity Accel.	9.81 [m/s^2]	

The following information must be specified:

- Inlet: the locator used to calculate inlet quantities.
- Outlet: the locator used to calculate outlet quantities.
- Rotor Blade(s): the locators used to calculate torque (one blade row) about the machine axis.
- Machine Axis: the axis or rotation of the turbine.
- Rot. Speed: the rotational speed of the turbine.
- Num. Main Blades: Some quantities calculated for a single blade set (main blade and any splitter blades) are multiplied by the number of blade sets in the full 360° wheel in order to produce the total value for the wheel.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: reference radius between the hub and tip.
- Ref Height: Cross-section height (that is, the height of the outlet region, or the height of the blade at the trailing edge).
- Flow Rate: The volume flow rate.

- Head Rise: The pressure head at the inlet.
- Ref Density: The reference density for the simulation.
- Gravity Accel.: The acceleration due to gravity.

13.6.3.7. Fan Noise Macro

Macro Calculator

Macro Fan Noise 

Calculates the tonal noise generated by a low speed fan (Mach Number less than 0.4)

Domain	S1 	
Blade Selection	Automatic 	
> Custom Blade	S1 Blade 	
> Custom # Blades	0 	
# of Harmonics	6 	
Observer (r)	1 [m]	
Observer (theta)	0 [degree]	
Theta Sectors	36 	
Directivity Harm. #	1 	
Loading Coeff.	2.2	
Acou. Ref. Pressure	2e-5 [Pa]	
Acou. Ref. Power	1e-12 [W m⁻³]	
Sound Speed	340 [m/s]	

This macro calculates the noise levels of the turbomachinery as observed at a specific location. The following information must be specified:

- Domain: The domain in which the blade is located.
- Blade Selection: Set to Automatic for a single blade passage or Custom for a multiple blade passage. If this is set to Custom, you will need to specify the 2D region for the blade (**Custom Blade**) as well as the number of blades (**Custom # Blades**).
- # of Harmonics: The number of harmonics used in the calculation.
- Observer (r) and Observer (theta): The distance and location of the observer, relative to the blade.

- **Theta Sectors:** The number of sampling points (sectors) equally spaced over 360° at a given radius around the fan, used to calculate the noise values. A higher number leads to a more accurate solution, but takes more time to calculate.
- **Directivity Harm. #:** The harmonic level at which the sound pressure levels will be calculated.
- **Loading Coeff.:** A coefficient between 2 and 2.5.

The loading coefficient parameter defines the decay (or decrease) of the sound-pressure level vs the frequency. In general, the sound-pressure level decreases when the frequency increases. In his experiments, Lowson replaced the unsteady loading by a steady one multiplied by a decay function. Based on these experiments, this decay follows an exponential law with a negative slope. Lowson found that a loading coefficient between 2 and 2.5 gives a sound-pressure level close to the experimental data; that is, the loading coefficient defines the slope of the exponential law.

In general and for highly loaded blades, the decay of the sound-pressure level is very quick (one or two peaks in the sound-pressure level spectrum) and therefore a higher value of the loading coefficient will be appropriate.

- **Acou. Ref. Pressure:** Acoustic reference pressure (P_{ref}) is the international standard for the minimum audible sound of $2 \cdot 10^{-5}$ [Pa].

The acoustic reference pressure is used to convert the acoustic pressure into Sound Pressure in dB using the following equation:

$$SPL_m(dB) = 20 \log_{10} \left(\frac{P'_m}{P_{\text{ref}}} \right) \quad (13.1)$$

where P_{ref} is the acoustic reference pressure. The reference pressure depends on the fluid.

- **Acou. Ref. Power:** Acoustic reference power (W_{ref}) is used to convert the sound power SW_m from units of $[W m^{-3}]$ to units of dB.

The equation used is:

$$LW_m(dB) = 10 \log_{10} \left(\frac{SW_m}{W_{\text{ref}}} \right) \quad (13.2)$$

where:

- W_{ref} is the value of the acoustic reference power
- SW_m is the sound power and is defined by:

$$SW_m = \frac{\pi r_1^2}{\rho c_0} \int_0^\pi (P'_m)^2 \sin \phi \, d\phi \quad (13.3)$$

The acoustic reference power is: $1 \cdot 10^{-11} [W m^{-3}]$

- **Sound Speed:** The speed of the sound in the fluid at rest.

For details on completing this dialog box, see [Using the Fan Noise Macro \(p. 361\)](#).

13.6.3.7.1. Using the Fan Noise Macro

The Fan Noise macro calculates the tonal noise levels generated by a low-speed fan (primarily axial-flow fans). Tonal noise, or discrete-frequency noise, is due mainly to periodic forces exerted on fluid passing a fan. The Fan Noise macro can be applied to low speed fans having a tip Mach number less than 0.45. For a higher tip Mach number, the accuracy of the results is questionable. The fan must radiate in the free field where the observer can see the fan blades (the Fan Noise macro does not take into account the reflection effect). Thus, the Fan Noise macro cannot be applied to ducted fans.

The following topics are discussed:

[13.6.3.7.1.1. Fan Noise Theory in Brief](#)

[13.6.3.7.1.2. Fan Noise Macro Input](#)

[13.6.3.7.1.3. Fan Noise Output \(Reports\)](#)

[13.6.3.7.1.4. Fan Noise Examples](#)

13.6.3.7.1.1. Fan Noise Theory in Brief

Several methods have been developed to predict tonal noise; the Lowson Model is described here.

In the low-speed regime, the main noise component is a dipolar source. Lowson [Lowson, M. V., 1970, "Theoretical analysis of compressor noise", The Journal of Acoustics So. Am., Vol. 47 (1), 1970, pp. 371-385.] showed that the noise generated by a fan is directly related to the aerodynamic forces exerted on the fixed and rotating blades. First, in a semi-empirical way, he calculated these forces; then he took into account the distance between the source and the observer. In this case, the fan is considered as a noise source for which the frequency depends on the rotational speed and other parameters. In 1962, Lighthill established the acoustic pressure expression produced by a punctual force, F_i , in rectilinear motion.

$$p'(x_i, y_i) = \frac{x_i - y_i}{4 \pi c_0 r^2 (1 - M_r)^2} \frac{\partial F_i}{\partial t} \quad (13.4)$$

where:

$$x_i = (x, y, 0)$$

$$y_i = (0, R \cos \theta, R \sin \theta)$$

$$M_r = \frac{1}{c_0} \frac{dr}{d\tau}$$

$$\tau = t - \frac{r}{c_0}$$

$$F_i = (-F, F_y \sin \theta, F_y \cos \theta)$$

As shown in [Figure 13.1: Relative position of the source and the observer \(p. 362\)](#), x_i and y_i are the coordinates of the Observer O (r, ϕ, τ) and of the Source S (R, θ, t), respectively. M_r is the convective component of the rotational Mach number in the r direction. F_x and F_y are respect-

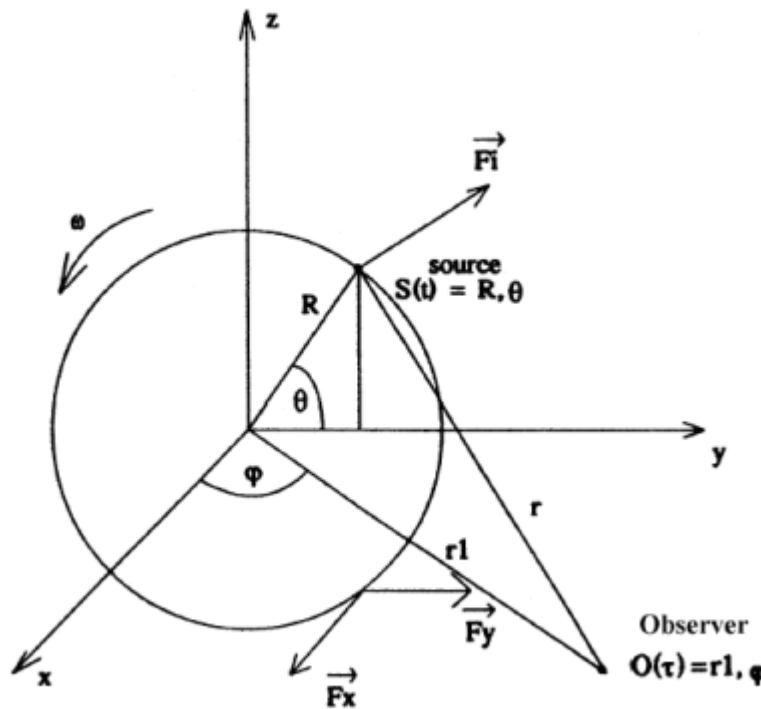
ively the thrust and the drag (torque) forces exerted on the blade. According to [Equation 13.4 \(p. 361\)](#), when the force F_i is constant, the acoustic pressure is equal to zero.

Lowson extended [Equation 13.4 \(p. 361\)](#) to create a more general equation:

$$p'(x_i, y_i) = \frac{x_i - y_i}{4\pi c_0 r^2 (1 - M_r)^2} \left[\frac{\partial F_i}{\partial t} + \frac{F_i}{1 - M_r} \frac{\partial M_r}{\partial t} \right] \quad (13.5)$$

This relation describes the contribution of the convective phenomenon due to the term $\partial M_r / \partial t$. Note that [Equation 13.5 \(p. 362\)](#) must be evaluated at retarded time τ . This equation can be used to find an expression for the sound from a point force in arbitrary harmonic motion.

Figure 13.1: Relative position of the source and the observer



The Lowson model enables the calculation, at the observer position, of the acoustic pressure generated by steady and unsteady efforts. The latter are considered as punctual sources and correspond to the loads exerting by the z blades of the rotor. Lowson integrated [Equation 13.5 \(p. 362\)](#) in time and space to get the m^{th} harmonic of the acoustic pressure generated by a periodic rotating loading:

$$p'_m = a_m + j b_m = \frac{\omega}{\pi} \int_0^{\frac{2\pi}{\omega}} \left[\frac{x_i - y_i}{4\pi a_0 r^2 (1 - M_r)^2} \left[\frac{\partial F_i}{\partial t} + \frac{F_i}{1 - M_r} \frac{\partial M_r}{\partial t} \right] \right] \exp(j m \omega t) dt \quad (13.6)$$

using the following equation:

$$dt = (1 - M_r) d\tau \quad (13.7)$$

and integrating Equation 13.6 (p. 362) by parts gives:

$$p'_m = -\frac{\omega}{4\pi^2 r} \int_0^{\frac{2\pi}{\omega}} \left\{ \frac{j m \omega F_r}{a_0} + \frac{F_i}{1-M_r} \left[-\frac{M_i}{r} + \frac{(x_i - y_i)}{r^2} \right] \right\} \exp[j m \omega (\tau + \frac{r}{a_0})] dt \quad (13.8)$$

as shown in Figure 13.1: Relative position of the source and the observer (p. 362) with x being the axis of rotation and the fluctuating loading and observer position being defined as:

$$F_i = \begin{cases} -F_x \\ -F_y \sin \theta \\ F_y \cos \theta \end{cases}$$

$$x_i - y_i = \begin{cases} x \\ y - R \cos \theta \\ -R \sin \theta \end{cases}$$

$$r = |x_i - y_i| \approx r_1 - \left(\frac{y R}{r_1} \right) \cos \theta$$

In

$$F_r = -\frac{x F_x}{r} - \left(\frac{y F_y}{r} \right) \sin \theta \quad (13.9)$$

- F_x and F_y are respectively the thrust and drag (torque) components of the aerodynamic unsteady force represented by a global force exerted on the blade.
- The terms in $1/r$ and $1/r^2$ are important only in the acoustic near field. Thus, in the acoustic far field, Equation 13.8 (p. 363) becomes:

$$p'_m = -\frac{\omega}{4\pi^2 r} \int_0^{\frac{2\pi}{\omega}} \left\{ \frac{j m \omega F_r}{a_0} \right\} \exp[j m \omega (\tau + \frac{r}{a_0})] dt \quad (13.10)$$

Taking into account of the thrust and drag periodicities, Lawson proposed the following formulation:

$$\begin{pmatrix} F_x \\ F_y \end{pmatrix} = \sum_{\lambda=-\infty}^{\lambda=\infty} \begin{pmatrix} F_x(\lambda) \\ F_y(\lambda) \end{pmatrix} \exp(-i\lambda\omega t) \quad (13.11)$$

where λ is the effort harmonic order or the mode.

Substituting the results obtained from Equation 13.9 (p. 363) and Equation 13.11 (p. 363) into Equation 13.10 (p. 363) gives:

$$p'_m = -\frac{j m \omega}{4\pi^2 r c_0} \int_0^{\frac{2\pi}{\omega}} \left(\sum_{\lambda=-\infty}^{\lambda=\infty} \left\{ \frac{x F_x(\lambda)}{r_1} + \frac{y F_y(\lambda)}{r_1} \sin \theta \right\} \right) \cdot \exp[j(m-\lambda)\theta - j m \left(\frac{y R}{r_1} \right) \cos \theta] d\theta \quad (13.12)$$

where the rotational Mach number is $M = \omega R / c_0$

The integrals in [Equation 13.12 \(p. 363\)](#) can be identified as Bessel functions, and, using the expressions:

$$\begin{aligned} \int_0^{2\pi} \exp[j(m\theta - z \cos\theta)] d\theta &= 2\pi j^{-m} J_m(z) \\ \int_0^{2\pi} \exp[j(m\theta - z \cos\theta)] \sin\theta d\theta &= (-2) \pi j^{-m} \left(\frac{m}{z}\right) J_m(z) \end{aligned} \quad (13.13)$$

[Equation 13.12 \(p. 363\)](#) can be evaluated directly to give the sound level radiated from z rotor blades:

$$\begin{aligned} p'_m &= \frac{j m z^2 \omega}{2\pi c_0 r_1} \sum_{\lambda=-\infty}^{\lambda=\infty} (-j)^{mz-\lambda} \left[\cos\phi \left(F_x(\lambda) - \left(\frac{mz-\lambda}{mzM}\right) F_y(\lambda) \right) \right] J_{mz-\lambda}(mzM \sin\theta) \end{aligned} \quad (13.14)$$

where:

- $J_{mz-\lambda}$ is a first Bessel Function of order $mz-\lambda$
- z is the number of blades
- $j^2 = -1$

The interest of this relation is the knowledge of the components of the fluctuating efforts $F_x(\lambda)$ and $F_y(\lambda)$.

Following the experimental work done on helicopter blades by Scheiman [Scheiman, J., 1964, "Sources of noise in axial flow fans", Journal of Sound and Vibration, Vol. 1, (3), 1964, pp. 302-322.], Lowson extended [Equation 13.14 \(p. 364\)](#) to an equation that relates the steady-state components of the force to the acoustic pressure.

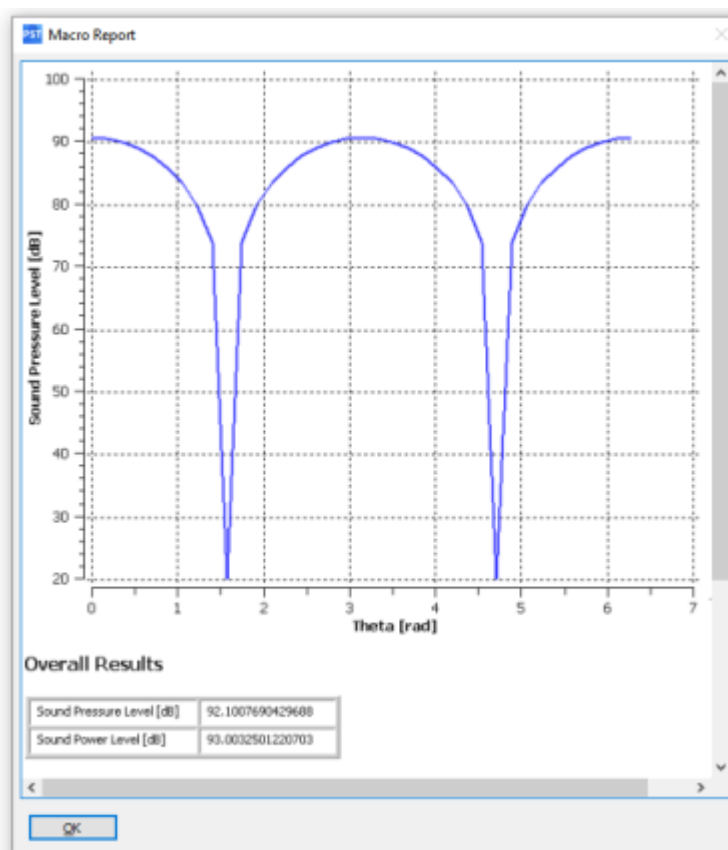
13.6.3.7.1.2. Fan Noise Macro Input

The Fan Noise macro calculates the tonal noise levels generated by a fan as heard at a specific location. To access the Fan Noise macro:

1. Load the .res file into CFD-Post.
2. Click the **Calculators** tab.
3. In the **Macro** field, select Fan Noise.
4. In the Macro Calculator, specify the information described in [Fan Noise Macro \(p. 359\)](#).
5. When the Macro Calculator fields are filled in, click **Calculate**.

13.6.3.7.1.3. Fan Noise Output (Reports)

The Fan Noise macro outputs a report; to view it, click **View Report**. The report displays the input values, the sound pressure levels, the sound power levels, the directivity of harmonic 1, and the overall results. Here is a partial sample:



The turbo noise report will be created in your working directory as `turboNoise_report.html` along with the tables (`turboNoise_*.csv`) and graphics (`turboNoise_*.png`) included in the report. This enables you to reuse these elements in other documents, if required.

13.6.3.7.1.4. Fan Noise Examples

There are two ways to perform turbo noise calculations; you can have:

- A case with a single blade passage (the Lowson model is based on this)
- A case with a multiple blade passage, including a 360° case.

As shown below, the only necessary differences in the two cases are the settings for **Blade Selection** and the custom blade fields.

Fan Noise Macro Values	Single Blade Passage	Multiple Blade Passage
Domain	Fan Block	Fan Block
Blade Selection	Automatic	Custom
> Custom Blade		Blade
> Custom # of Blades		9
# of Harmonics	6	6
Observer (r)	1 m	1 m
Observer (theta)	0 degree	0 degree

Fan Noise Macro Values	Single Blade Passage	Multiple Blade Passage
Theta Sectors	36	36
Directivity Harm. #	1	1
Loading Coeff.	2.2	2.2
Acou. Ref. Pressure	2e-005 Pa	2e-005 Pa
Acou. Ref. Power	1e-012 W m ⁻³	1e-012 W m ⁻³
Sound Speed	340 m/s	340 m/s

To view the report, click **Calculate** and then **View Report**.

13.6.4. User-defined Macros

You can write your own macros and make them available in the Macro Calculator.

The following topics are discussed:

[13.6.4.1. Writing a Macro](#)

[13.6.4.2. Macro GUI Definition](#)

13.6.4.1. Writing a Macro

Using a text editor, you can write a macro in a CFD-Post session file format (file extension .cse).

A macro can contain:

- Special commented lines that help define the graphical user interface for the macro
- Power Syntax commands

For details, see [Power Syntax in Ansys CFX](#).

- CCL (see [CFX Command Language \(CCL\) in CFD-Post \(p. 417\)](#))

A macro must contain at least one subroutine written using Power Syntax.

Following are two example subroutines written using Power Syntax:

```
! sub Hello1 {
! print "Hello !\n";
! }
! sub Hello2 {
! ($title, $name) = @_;
! print "Hello $title $name\n";
! }
```

Note that commas are used to separate arguments. Also note that strings are quoted.

You can embed graphical user interface controls into the macro by writing special comments between the # Macro GUI begin and # Macro GUI end lines. An example macro follows:

```
# Macro GUI begin
#
# macro name = Area Average Macro
# macro subroutine = mySub
```

```

#
# macro parameter = Var
#   type = variable
#   multiselect = true
#   default = Y
#
# macro parameter = Location
#   type = location
#   location category = surface
#
# Macro GUI end

! no warnings 'redefine';

! sub mySub {
! ($variable, $plane) = @_;

# Create an expression for the area average of the variable value on the plane
LIBRARY:
CEL:
  EXPRESSIONS:
! my $locationName = getObjectNames($plane);
  Average Value = areaAve($variable)@$locationName
  END
END
END

! my $average = getExprVal("Average Value");

! print "Var = $variable, Location = $locationName, Area Average = $average\n";

!}

```

In the macro above, the special comments between the `# Macro GUI begin` and `# Macro GUI end` lines specify:

- The macro name
- The subroutine to call
- The input parameters and details of them, including their types and default values

Macro Area Average Macro has two input parameters:

- **Var** - The variable to average. The default is Y
- **Location** - A surface locator.

When this macro is loaded, the entry Area Average Macro appears in the list of available macros (under the **Macro** setting).

When this macro is run:

- Expression Average Value is updated (created if necessary). This expression evaluates to the area average of the specified variable on the specified location.
- The specified values **Var** and **Location**, and the value of Average Value, are printed to the console window (which is viewable when starting CFD-Post in stand-alone mode).

The following example macro writes text output to a file.

```
! open(FH, ">myOut.txt");
! $val = ave("Pressure", "Point 1");
! $time = getValue( "DATA READER", "Current Timevalue");
! print FH "$time $val\n";
! close(FH);
```

In this example, the output filepath is specified using redirection of output from the Perl print command.

In order to better learn how to create your own macro, you can view the existing macros in `<CFDPOSTROOT>/etc/*.cse` and study the definitions.

13.6.4.2. Macro GUI Definition

The portion of a macro definition that falls between the `# Macro GUI begin` and `# Macro GUI end` lines contains special comments that provide basic information about the macro, including the information required to populate the graphical user interface of the Macro Calculator with the settings (input parameters) required by the macro. The following table lists the special comments that can be added to a macro:

Special Comment	Description
<code># macro name = <name></code>	The macro identifier to appear in the macro combo
<code># macro subroutine = <subname></code>	The subroutine to call. A macro must contain at least one subroutine.
<code># macro report file = <filename></code>	The file generated by the macro (if any). This enables the View Report button, which attempts to load the file in a text/html browser.
<code># macro related files = <file1>, <file2></code>	Other related files to load when loading this macro. The main use of this feature is to load subroutines from other files so that these subroutines can be used in the macro. Each specified file can contain one or more subroutines.
<code># macro parameter = <name></code> <code>#type = <type></code> <code>#<option1> = <val></code> <code>#<option2> = <val></code> <code>#..</code>	Defines an input parameter and its properties: its name, type, and other options as described in the next table. Specify this group of comments once for each input parameter that is required by the macro. The input parameters are displayed (in the order defined) in the graphical user interface, and, upon running the macro, are passed (in the order defined) to the macro's (main) subroutine as input arguments.

The following table provides details on the `# macro parameter` comment, including the options available for each parameter type:

Type	Option	Example	Notes
string	default	My String	
integer	default	10	

Type	Option	Example	Notes
	range	1, 100	
float	quantity type	0.1 [s]	The quantity type controls which units are allowed. A full list of quantity types can be found in <code><CFDPOSTROOT>/etc/<version>/common_units.cfx</code> . To specify that the float is dimensionless, omit the specification of a quantity type.
	default	0.1 [s], 0.4 [s]	
	range	Time	
triplet	default	0.5[m], 0[m], 1[m]	The quantity type controls which units are allowed. A full list of quantity types can be found in <code><CFDPOSTROOT>/etc/<version>/common_units.cfx</code> . To specify that the triplet is dimensionless, omit the specification of a quantity type.
	range	-1[m], 1[m]	
	quantity type	Length	
location	default	Inlet	The location type can be any object type listed in <code><CFDPOST-ROOT>/etc/CFX-PostRules.ccl</code> after an occurrence of the string "OBJECT:", provided that the object description contains the string "Category =" followed by a list of categories that includes the word "selectable". Valid location types include (but are not limited to):
	location type	Boundary	
	location category	surface	
			• boundary • chart • chart series • domain • isosurface • line • plane • point • polyline • streamline • turbo line

Type	Option	Example	Notes
			- turbo surface - user surface - volume The location category can be any category listed in <code><CFDPOST-ROOT>/etc/CFX-PostRules.ccl</code> after the string "Categories =". Valid location categories include: - geometry - line - plane - point - region2d - selectable - surface - variable - viewer-viewable - volume You can specify a location category to allow locations of various related types. For example, the "surface" category includes any locations of type "boundary", "isosurface", or "turbo surface". The location parameter returns either the object's CCL path or just its name. The object's name is returned for surfaces that are read from the file, for example, boundaries or regions. For other objects, the object's path is returned; to get the object's name, use getObjectName(Object Path). [a]
list	default	orange	

Type	Option	Example	Notes
	list	apple, orange, fig	
variable	default	Pressure	
domain	default	Stator	

^[a] CFD-Post normally refers to boundaries in other objects by name, not path (for example, a contour plot on a boundary), so that the object can be re-used in other files or for file comparisons. For example, a contour in "inlet" will automatically turn into two separate objects when two files are loaded and both have "inlet".

13.7. Mesh Calculator

The Mesh Calculator (**Tools > Mesh Calculator**) offers a variety of tools to check the quality of your mesh. The results of each calculation are performed over all domains^[1] and printed to the output window. Each calculated variable is also added to the list of available variables, which enables you to use them as a basis for creating new plots. It is important to note that these variables are evaluated on nodes rather than elements, based on the criteria described below.

You can select the following functions to calculate:

Maximum Face Angle

This calculates the largest face angle for all faces that touch a node. For each face, the angle between the two edges of the face that touch the node is calculated and the largest angle from all faces is returned for each node. Therefore, there is one maximum value for each node. The values that are reported are the smallest and largest of these maximums.

The maximum face angle can be considered to be a measure of skewness. For details, see [Mesh Visualization Advice](#) (p. 372).

Minimum Face Angle

This calculates the smallest face angle for all faces that touch a node. For each face, the angle between the two edges of the face that touch the node is calculated and the smallest angle from all faces is returned for each node. Therefore, there is one minimum value for each node. The values that are reported are the smallest and largest of these minimums. For details, see [Mesh Visualization Advice](#) (p. 372).

Edge Length Ratio

This is a ratio of the longest edge of a face divided by the shortest edge of the face. For each face:

$$\frac{\max(l_1, l_2)}{\min(l_1, l_2)} \quad (13.15)$$

is calculated for the two edges of the face that touch the node. The largest ratio is returned.

^[1] If multiple cases are loaded, the results of each calculation are performed over all domains in the specified cases.

Connectivity Number

Connectivity number is the number of elements that touch a node.

Element Volume Ratio

Element Volume Ratio is defined as the ratio of the maximum volume of an element that touches a node, to the minimum volume of an element that touches a node. The value returned can be used as a measure of the local expansion factor.

Mesh Information

The Mesh Information option returns the number of nodes and elements in your volume mesh. It also lists the number of elements of each element type. As an example, the mesh for the following output contains two domains: one using hexahedral elements and the other containing tetrahedral elements. The domains were connected using a domain interface:

```
Number of Nodes: 71680
Number of Elements: 139862
  Tetrahedra: 75265
  Wedges: 31395
  Pyramids: 0
  Hexahedra: 33202
```

When you click **Calculate**, the result window displays the results of the specified calculation. If the calculated variable does not already exist, it will be created. This enables you to create plots of the calculated variable.

Note:

When you compare the mesh information for a Fluent file in Fluent and in CFD-Post, the reported number of nodes (Fluent's "cells") will differ. In Fluent, each domain can have nodes at its boundaries that are not acknowledged as being shared with other domains. This causes Fluent mesh reports to contain duplicated nodes; however, the actual number of cells is the same as reported by CFD-Post.

13.7.1. Mesh Visualization Advice

The following table gives some guidelines for checking mesh quality. If there are elements that have mesh quality parameters greater or less than those listed, you may find problems with using the mesh in the CFX-Solver.

Element Type	Elements may be a problem if they have any of:
Tetrahedrons (4 nodes)	Edge Length Ratio > 100 Max Face Angle > 170° Min Face Angle < 10° Element Volume Ratio > 30 Connectivity Number > 50

Element Type	Elements may be a problem if they have any of:
Pyramids (5 nodes)	Edge Length Ratio > 100 Max Face Angle > 170° Min Face Angle < 10° Element Volume Ratio > 5
Prisms (6 nodes)	Edge Length Ratio > 100 Max Face Angle > 170° Min Face Angle < 10° Element Volume Ratio > 5 Connectivity Number > 12
Hexahedrons (8 nodes)	Edge Length Ratio > 100 Max/Min Edge Length > 100 Min Face Angle < 10° Element Volume Ratio > 5 Connectivity Number > 24

In many cases, the robustness of the CFX-Solver will not be adversely affected by high element volume ratios. However, you should be aware that accuracy will decrease as the element volume ratio increases. For optimal accuracy, you should try to keep the element volume ratio less than the value suggested in the above table.

13.8. Case Comparison

The **Compare Cases** command enables you to compare results from two distinct cases, or between two steps of a single case. The **Compare Cases** command is available in the **Tools** menu in any of the following situations:

- You have loaded two or more cases using the **Load Results File** dialog box option **Keep current cases loaded**
- You have loaded a single transient case (with results available for at least two time steps)
- You have loaded a multi-configuration case, or a case with run history, using the **Load Results File** dialog box option **Load complete history as** (either as a single case or as separate cases), so that results for two or more steps are available through the timestep selector.

Selecting **Compare Cases** displays the **Case Comparison** details view.

The following options are available:

Case Comparison Active

Enables the Case Comparison function; the comparison occurs when you click **Apply**.

In **Case Comparison** mode:

- Difference variables are computed as the variable values from Case 1 minus the variable values from Case 2. The latter are interpolated onto the mesh from Case 1 before the subtraction. As a result, the difference variables are located on the mesh from Case 1.

To reverse the order of subtraction, swap the specifications for Case 1 and Case 2 in the **Case Comparison** details view.

Note:

- CFD-Post does not support comparison of the following variables:
 - Connectivity Number
 - Edge Length Ratio
 - Element Volume Ratio
 - Force
 - Length
 - Mass Flow
 - Maximum Face Angle
 - Minimum Face Angle
 - When comparing variables on interior walls in cases where meshes are not identical, you may see unexpected differences in difference plots. This can happen because during mesh interpolation, variable values may get picked up from one or the other side of the interior boundary. If the two sides do not have the same values, the interpolated values could randomly oscillate between values of the two sides, producing additional difference in the plot.
 - When comparing cases with a large number of mesh nodes, CFD-Post may take a long time to produce the difference variable and as a result may appear to be unresponsive.
-
- A **Difference** view is shown in a new view (in addition to the **Case 1 (<case_name>)** view and the **Case 2 (<case_name>)** view). In that view, differences are shown on the mesh from Case 1.
 - Each difference variable is named by appending ".Difference" to the end of the variable name from which it was derived. For example, the difference variable for the variable Pressure is Pressure.Difference.

- The difference variables can be used anywhere that variables can normally be used. The Function Calculator and **Table Viewer** have special support for the difference variables, enabling you to easily see functions and tables (respectively) of difference values. In addition, a chart that is based on locators that exist in both Case 1 and Case 2 will have a "Difference" chart line. See [Example: Comparing Differences Between Two Files](#) (p. 326).
- CFD-Post refers to the cases as "Case 1" and "Case 2" rather than as the original case names (which are usually based on the results filename).

Case 1 and Case 2

Enables you to select the cases to be compared. If you want to compare two steps from within the same case (that is, two time steps from a transient case) then you should select the same case for both **Case 1** and **Case 2**. The timestep selector that is embedded into the **Case Comparison** details view then enables you to select which steps you want to compare. In this circumstance, CFD-Post needs to load the results from the selected case a second time, so you will see a second case appearing in the tree view. After the comparison has been initialized, the steps used for the comparison can be changed either by using the embedded timestep selector on the **Case Comparison** details view, or by using the usual timestep selector (which now has separate entries for each of the two copies of the case being compared).

Tip:

When comparing two 2D cases, set the case that is extruded less as Case 1. This enables CFD-Post to match nodes between the two cases for one of the symmetry boundaries and to define difference plots.

Options: Synchronize camera in displayed views

Causes changes in orientation of one view to be duplicated in the other. If the views are initially in different orientations, the first movement of any view will align all views to the same orientation.

Options: Use absolute difference for scalar variables

Causes all scalar variable differences to be reported as positive numbers.

Mesh Detection

Enables you to control whether or not CFD-Post needs to determine whether the meshes in the two cases are identical. If you know beforehand that the meshes are the same or different, you can save processing time by enabling the appropriate mesh detection setting. Your options are:

- **Auto-detect same mesh** causes CFD-Post to analyze the two meshes to determine whether they are the same or different before performing any interpolation.
- **Meshes are identical** and **Meshes are different** enable CFD-Post to perform interpolation immediately, which saves processing time when cases are large.

Note:

When you know meshes to be topologically identical but the node numbering may be different, use **Meshes are identical**. This setting causes CFD-Post to ignore node

numbering and just use the topology of the mesh. In such cases do not use the **Auto-detect same mesh** setting because this fails when node numbering is not the same.

For example, when comparing a case from Ansys CFX with a case from Fluent, the node numbering may differ even between apparently identical meshes, so the **Mesheres are identical** setting is required.

Note:

- If you run a case comparison on a file that contains solver-generated difference variables (such as `Volume Porosity.Difference`), these variables will become unviewable when you enter case comparison mode. However, the variables will be viewable again if you reload the results file.
- Global ranges of difference variables are updated as domains are used. For example, if a multi-domain case is loaded and a difference variable colors a locator that is in a single domain, the range displayed will reflect the range of the difference variable *only in that domain*. If the locator is moved to another domain (or a new locator colored by the same variable is added), the global range for that difference variable is updated to reflect both domains.

- When using expressions in case-comparison situations, the expression syntax is:

`<function>()@CASE:[1|2].<location>`

For example, `area()@CASE:2.mylane`

- Case comparison is supported only for General mode. As a result, case comparison initiated from the **Turbo** tab will revert to General mode.
 - When using **Variable Minimum** or **Variable Maximum** option on a point in multi-file or comparison mode, the point is placed at the location of the overall minimum/maximum. If you want to place the point at the minimum/maximum value for the individual cases, select the appropriate case in the point's **Domain List** selector.
-

13.8.1. Calculating Difference Variables

There are two ways of creating difference variables:

- You can use the CFX Interpolator.
- You can use CFD-Post in comparison mode.

In each case you can then view variables such as "`<vector variable>.Difference`" (such as `Velocity.Difference`) and "`<scalar variable>.Difference`" (such as `Temperature.Difference`). For a description of the general variable syntax, see [Quantitative CEL Functions in Ansys CFX in the CFX Reference Guide](#).

Difference variables are computed on the mesh of the first case by first interpolating the variable from the second mesh to the first mesh, and then subtracting the two variables.

The magnitude of a difference variable "<vector variable>.Difference" is always calculated as:

$$\begin{aligned} & ((\text{<vector variable>.Difference } X)^2 \\ & + (\text{<vector variable>.Difference } Y)^2 \\ & + (\text{<vector variable>.Difference } Z)^2)^{1/2} \end{aligned} \quad (13.16)$$

This is *not* the difference of the vector magnitudes between file 1 and file 2.

If you plot a vector plot such as Velocity.Difference, it is obvious that a real vector is being plotted. However, if you plot "<vector variable>.Difference" in plots that use a scalar variable, how the difference variable is calculated is an issue. For example, suppose in one file you have a velocity vector (1, 0, 0), so the velocity magnitude is 1 [m/s], and in the second file you have a velocity vector of (-1, 0, 0), so the velocity magnitude is also 1 [m/s]. The vector variable Velocity.Difference variable is (2, 0, 0), and the scalar variable that CFD-Post calls "Velocity.Difference" is equal to the magnitude of this vector variable (that is, it is 2 [m/s]). You might expect Velocity.Difference to be equal to "velocity magnitude in file 2" - "velocity magnitude in file 1", which would give a value of 0 [m/s], but this is incorrect.

13.9. Command Editor

To start the **Command Editor**:

1. Select **Tools > Command Editor**. Alternatively, right-click any object that can be modified using the **Command Editor** and select **Edit in Command Editor**.
 - If you select **Tools > Command Editor**, the **Command Editor** opens and displays the current state regardless of any selection.
 - If the **Command Editor** dialog box has not been used previously, it will be blank.
 - If the **Command Editor** dialog box has been used previously, it will contain CCL commands. If you do not want to edit the CCL that appears, click **Clear** to erase all content.
 - If you right-click an object and select **Edit in Command Editor**, the CCL definition of the specific object populates the **Command Editor** automatically. Modify or add parameters as required, then process the new object definition to apply the changes.
2. Click in the **Command Editor**.
3. Prepare the content of the **Command Editor** by adding new content, modifying the existing content, or both.

The types of content that may be prepared are CCL, action commands, and power syntax. Combinations of these types of content are allowed. For details, see:

- [CFX Command Language \(CCL\) Syntax in the CFX Reference Guide](#).
- [Command Actions in the CFD-Post User's Guide \(p. 421\)](#).
- [Power Syntax in Ansys CFX in the CFX Reference Guide](#).

Right-click in the **Command Editor** to access basic editing functions. These functions include **Find**, which makes a search tool appear at the bottom of the **Command Editor** dialog box. Enter a search term and click either **Next** or **Previous** to search upwards or downwards from the insertion point or text selection. To hide the search tool, press **Esc**.

4. Click **Process**.

The contents are processed: CCL changes will affect CCL object definitions, actions will be carried out, and power syntax will be run.

Chapter 14: CFD-Post Help Menu

The **Help** menu has the following commands:

CFD-Post

Opens [CFD-Post User's Guide \(p. 1\)](#).

Contents

Opens a page that lists various help resources associated with this product.

Ansys Product Improvement Program

Provides a brief description of, and enables you to control participation in, the Ansys Product Improvement Program.

About CFD-Post

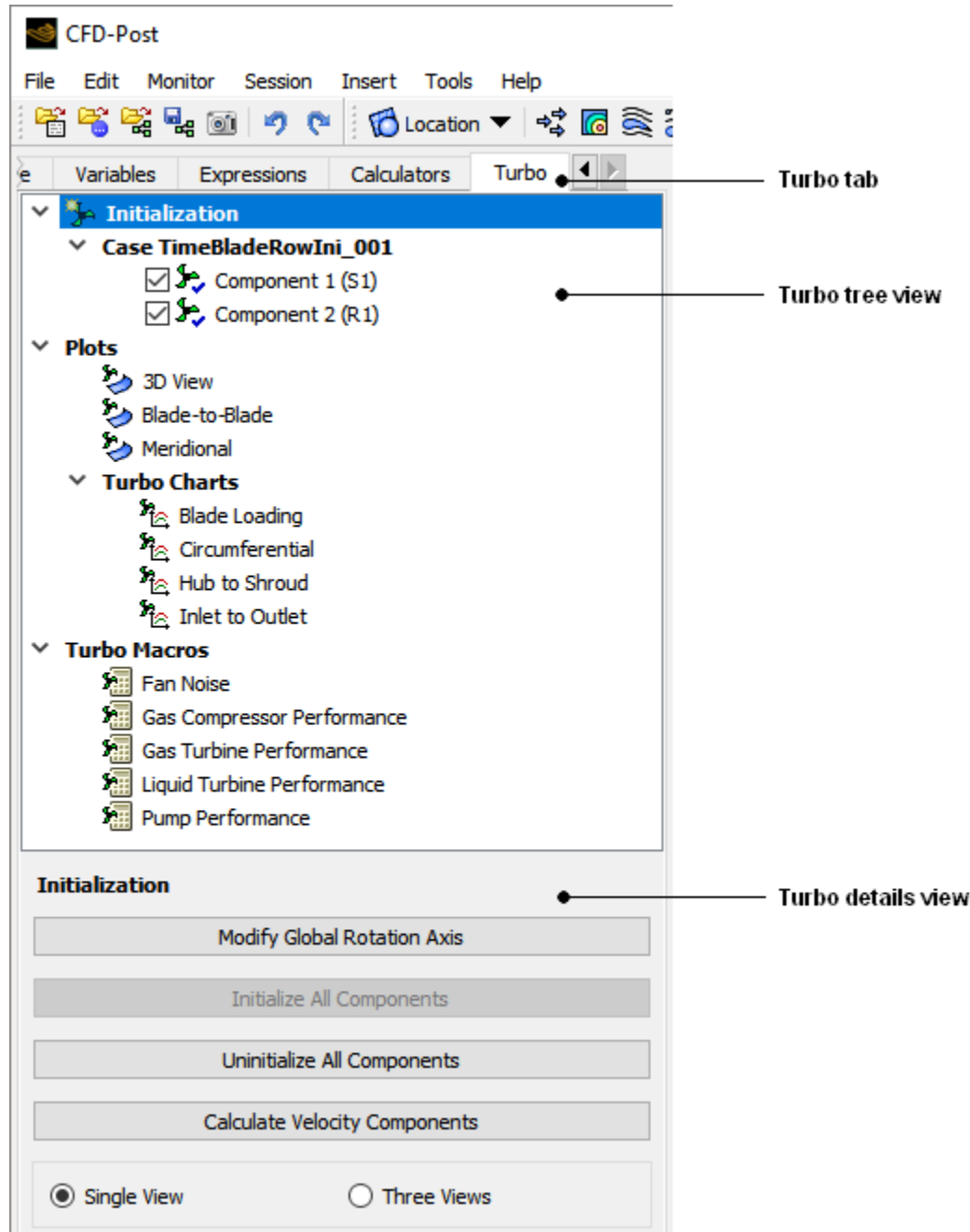
This gives the point releases and software patches that are installed.

Help on Help

Opens [Preface \(p. xxvii\)](#).

Chapter 15: Turbo Workspace

The **Turbo** workspace improves and speeds up postprocessing for turbomachinery simulations. To access the **Turbo** workspace, click the **Turbo** tab. The two main parts of the **Turbo** workspace interface are the **Turbo** tree view and the **Turbo** details view.



This chapter describes:

15.1. Visual Representation of Initialization Status

15.2. Define/Modify Global Rotation Axis

15.3. Turbo Initialization

15.4. Turbo View Shortcuts

15.5. Turbo Surface

15.6. Turbo Line

15.7. Turbo Plots

15.8. Turbo Macros

15.9. Calculate Velocity Components

15.1. Visual Representation of Initialization Status

Tip:

Load file `AxialIni_001.res` (provided with a tutorial) into CFD-Post so that the descriptions in this chapter are easier to follow.

When in the **Turbo** workspace, a wireframe representation of each component appears in the viewer. The currently selected turbo component appears as a green wireframe. If it also happens to be initialized, it will be accompanied by a visual depiction of the background mesh, shown as a transparent green surface with white mesh lines.

The **Turbo** tree view also indicates which components are initialized and which are not; if the component is uninitialized, the symbol next to a component name is grayed out.

After entering the **Turbo** workspace and initializing the turbo components, you are ready to start using the turbo-specific features offered in the **Turbo** workspace.

15.2. Define/Modify Global Rotation Axis

The **Define/Modify Global Rotation Axis** button is found on the **Turbo** workspace's **Initialization** view. When you click that button, the **Define/Modify Global Rotation Axis** dialog box appears.

Exactly one axis of rotation method must be specified. The axis definition can come from the results file, or it can be specified manually as either a **Rotation Axis** (six Cartesian coordinates) or a **Principal Axis** (X, Y, or Z). Upon changing the axis definition, the axial, radial, and Theta coordinates (and their dependent objects and expressions) are automatically updated. For details, see [Theta](#) (p. 394).

15.3. Turbo Initialization

Before using the **Turbo** workspace features, the components of the loaded case (such as rotor, stator, and so on^[1]) need to be initialized. Initialization causes, among other things, span, m' , a (axial) and r (radial) and Theta coordinates to be generated for each component.

^[1] Available components depend on the turbo setup in the preprocessor. There is a minimum of one component available for each domain.

The topics in this section include:

- [Requirements for Initialization \(p. 383\)](#)
- [Initialize All Components \(p. 383\)](#)
- [Uninitializing Components \(p. 384\)](#)
- [Individual Component Initialization \(Advanced Feature\) \(p. 384\)](#)
- [Details View for Individual Component Initialization \(p. 385\)](#)

Important:

Transient blade row cases that use the Fourier Transformation method will have two domains in CFD-Post, but only one domain will have data. Do not initialize Turbo Post for the non-data domain because this will cause some Turbo-related features to fail.

15.3.1. Requirements for Initialization

Initialization of a turbo component requires the following:

- Input for calculating a background mesh. For details, see [Purpose of Background Mesh \(p. 386\)](#).
- Specification of the number of instances of each turbo component (such as stator, rotor, and so on) required to represent the full geometry around the rotation axis.

Note:

CFD-Post can initialize turbo space only for domains that are enclosed with inlet, outlet, hub, and shroud regions. For more complex geometries, you must set up the problem to isolate the region of interest into a separate domain that has these regions.

15.3.2. Initialize All Components

To access the initialization options, double-click **Initialization** in the **Turbo** tree view. The **Initialize All Components** button that appears is used to set the region and instancing information for each of the domains contained in your results file.

Correctly defined turbo spaces, as described in [Requirements for Initialization \(p. 383\)](#), can be automatically initialized. To automatically initialize all components using the default (best guess) region assignment, you can do one of the following:

- Choose to auto-initialize all components when a message prompts you upon entering the **Turbo** workspace for the first time (after loading a case).
- Right-click a component in the **Turbo** tree view and select **Initialize All**.

- Use a CCL instruction; for details, see [Initializing all Turbo Components \(p. 431\)](#).

Tip:

For automatic 360° initialization, CFD-Post uses cut planes and then looks for intersections between these and the turbo regions. However, if gaps within the slice (due to the blade region) are large relative to complexity of the topology and curvature of the passage is high, automatic 360° initialization might fail as CFD-Post cannot reconstruct the passage curves. If your case has regions, you should be able to manually initialize by setting the turbo regions from any one of the passages. See [Individual Component Initialization \(Advanced Feature\) \(p. 384\)](#) for details.

15.3.3. Uninitializing Components

After a turbo component has been initialized, it is possible to change or even remove its initialization settings. An uninitialized component still has axial, radial and Theta coordinates generated for it, as long as the rotation axis is defined.

The **Uninitialize All Components** button is accessible in the **Turbo** details view after double-clicking **Initialization** in the **Turbo** workspace. A shortcut menu associated with a turbo component in the **Turbo** tree view enables uninitialization for that component, or for all components.

Uninitializing all turbo components can be followed by initializing only the components that will be studied. Keeping the number of initialized components to a minimum saves computer memory. It also saves computational effort when generating plots that span multiple components. For example, having only one component initialized in a domain with many components restricts calculations and plots to just the initialized component.

Uninitialization does not cause graphic objects to be deleted. A graphic object that disappears due to the uninitialization of a turbo component reappears if the component is initialized.

15.3.4. Individual Component Initialization (Advanced Feature)

To manually initialize or modify the initialization of a turbo component, double-click the component in the **Turbo** tree view. A details view for the component appears with two tabs: **Definition** and **Instantiating**.

1. Select the boundary names that correspond to the required turbo regions. To select multiple regions, click the icon to the right of the drop-down list and hold the **Ctrl** key while selecting the regions.
2. In the **Background Mesh** frame for each of the hub, shroud, inlet, and outlet curves, choose to specify each to be **From Turbo Region** or **From Line** (that is, from a predefined line). If **From Line** is chosen, choose the line locator.
3. Set the mesh **Method** to either **Linear** or **Quasi Orthogonal**.
4. Click **Apply** (or **Initialize**, for subsequent initializations).

Additional information on Individual Component Initialization is available in the [Details View for Individual Component Initialization \(p. 385\)](#) section; for details, see:

- [Turbo Regions Frame \(p. 385\)](#)
- [Background Mesh Frame \(p. 386\)](#).

15.3.5. Details View for Individual Component Initialization

The Individual Component Initialization view contains the following tabs:

- [Definition Tab \(p. 385\)](#)
- [Instancing Tab \(p. 387\)](#)

15.3.5.1. Definition Tab

The **Definition** tab is used to specify:

- The hub, shroud, inlet, and outlet curves and other regions for a turbo component (such as a rotor or stator). For details, see [Turbo Regions Frame \(p. 385\)](#).
- The parameters controlling the component's associated background mesh.

The background mesh is a mesh generated on a constant-Theta projection of the passage, used to define spanwise and meridional coordinates for the 3D geometry. For details, see [Background Mesh Frame \(p. 386\)](#).

15.3.5.1.1. Turbo Regions Frame

The **Turbo Regions** frame is used to assign 2D regions to the Hub, Shroud, Blade, Inlet, Outlet, and Periodic regions of a turbo component. These regions are not always required, but when provided, may be used in the following ways:

- The Blade region specification is used to enable macros and plots that deal with blades (for example, a blade loading macro).
- The intersections of the Hub, Shroud, Inlet and Outlet regions with Periodic 1 may be used in order to generate internal polylines that are then collapsed in the Theta direction to form the boundaries of the background mesh. Alternatively, or if any of these intersections are not possible, polylines/lines may be specified explicitly in the **Background Mesh** frame. For details, see [Background Mesh Frame \(p. 386\)](#).

In the special case of a turbo component that wraps 360 degrees around the rotation axis, there may be no periodic regions available. In this case, you may do one of the following:

1. Select the **360 Case Without Periodics** check box.
2. Specify the hub, shroud, inlet, and outlet regions. Create a rectangularly-bounded slice plane, using the point-and-normal method, such that it intersects the turbo component on only one side of the rotation axis. In this case, it may be helpful to temporarily set the plane type to Sample so that you can see the entire plane. After the plane is in the correct position, set the type to Slice. Finally, specify this slice plane as Periodic 1. You do not need to set Periodic 2.

3. Specify polylines for the hub, shroud, inlet, and outlet in the **Background Mesh** frame (described next).

15.3.5.1.2. Background Mesh Frame

15.3.5.1.2.1. Purpose of Background Mesh

In order to calculate **Streamwise Location** (m') and **Span** coordinates for a turbo component, a separate 2D mesh is created as an intermediate step. The mesh, here referred to as a *background mesh*, is formed by taking the 3D passage boundaries (hub, shroud, inlet, outlet) and collapsing them in the Theta direction, forming a 2D passage outline on an axial-radial plane. The outline is then filled in with a mesh consisting of lines of constant span and meridional coordinate. The resulting mesh is then used to associate **Streamwise Location** and **Span** coordinates with any 3D position in the passage.

15.3.5.1.2.2. Requirements for Setting Up a Background Mesh

The background mesh frame requires you to specify how the hub, shroud, inlet, and outlet curves will be obtained. The two available options are:

- From Turbo Region

When From Turbo Region is specified for a particular curve, that curve is automatically extracted by intersecting the corresponding turbo region (specified in the **Turbo Regions** frame) with the specified **Periodic 1** region (also specified in the **Turbo Regions** frame).

- From Line

When From Line is specified for a particular curve, you must provide a polyline/line locator for that curve. You must use the latter method for every curve that cannot be derived by the first method (for example, because one or more **Turbo Regions** are not specified).

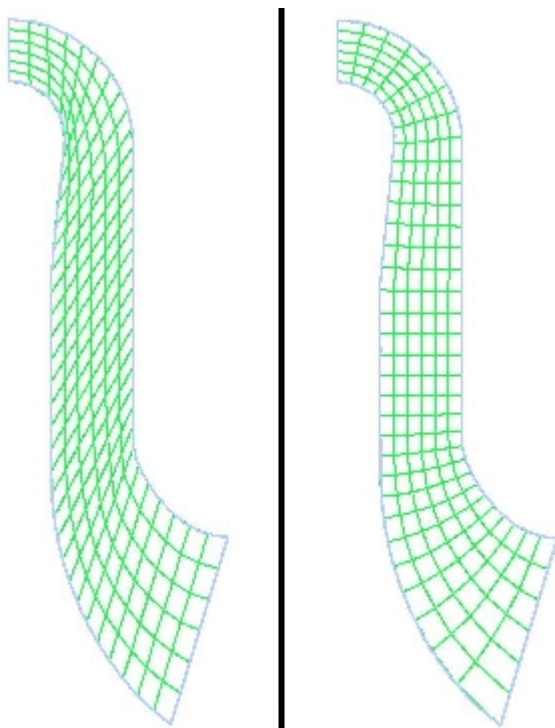
A line or polyline used to generate a background mesh must follow the entire surface it represents (along the component). One way in which a polyline can be created is by using the intersection between a bounded plane (such as a slice plane or a turbo surface of constant Theta) and the appropriate surface (for example, the hub surface). Before the polyline is used for initialization, is transformed by adjusting all Theta coordinates to the same value. The Theta coordinates of the polyline, therefore, have no effect; polylines obtained by intersection with a plane need not use a constant-Theta plane. If you cannot form the polyline easily, you can save pieces of the polyline to a series of files, use an editor to consolidate the parts, and then reload the edited file. For details, see:

- [Line Command \(p. 227\)](#)
- [Polyline Command \(p. 252\)](#).

15.3.5.1.2.3. Types of Background Mesh

Two methods are offered for creation of the background mesh:

- Linear
- Quasi Orthogonal



The figure on the left shows a background mesh (for clarity, **Density** was set to 200) using the **Linear** method, while the figure on the right shows the mesh using the **Orthogonal** (default) method. As can be seen from pictures, the **Quasi Orthogonal** method offers a higher-quality meridional space representation, especially in highly curved passages.

15.3.5.1.2.4. Density of the Background Mesh

The density of the background mesh influences the accuracy of the representation of the meridional space and, therefore, the accuracy of the nonlinear coordinate transformations. The default offered for **Density** should be sufficient in most cases.

15.3.5.2. Instancing Tab

Instancing settings are used to display multiple instances of objects. For example, if there are two turbo components, with the instancing information for component 1 specifying one copy, and the instancing information for component 2 specifying ten copies, then a turbo surface of constant span that covers both components will show, by default, one copy of the portion generated for component 1, and ten copies of the portion generated for component 2.

The **Instancing** tab for a turbo component is the same as the **Instancing** tab for a domain (see [Instancing Tab \(p. 62\)](#)) and similar to the **Definition** tab for an Instance Transform object (see [Instance Transform: Definition Tab \(p. 293\)](#)). (The **Definition** tab for an Instance Transform object is different in that its **Axis Definition** settings and **Instance Definition** settings cannot be set from a results file.)

By default, the **Axis Definition** and **Instance Definition** settings are automatically determined from the results file. To set your own axis definition, set **Axis Definition** to **Custom**. To set your own instance definition, set **Instance Definition** to **Custom**.

The instancing information specified for a component applies to graphic objects (or parts thereof) generated over the component. In order for this instancing information to apply to a graphic object:

- At least part of the graphic object must be generated using data from the component (that is, there must be an association between the graphic object and the component).
- The graphic object must have **Apply Instancing Transform** selected and **Transform** set to an Instance Transform that has **Instancing Info From Domain** selected.

The instancing information for a component is the same as the instancing information for the component's domain, and the instancing information for any other component in the same domain.

15.4. Turbo View Shortcuts

The following table shows commands that are specific to the **Turbo** tree view. To access these commands, right-click the appropriate elements in the **Turbo** tree view.

For a list of the other commands that appear in the **Turbo** tree view (and in most tree views), see [Common Tree View Shortcuts \(p. 49\)](#).

Command	Description
Initialize	Initializes the selected turbo components. For details, see Individual Component Initialization (Advanced Feature) (p. 384) .
Uninitialize	Uninitializes the selected turbo components. For details, see Uninitializing Components (p. 384) .
Initialize All	Initializes all turbo components. For details, see Initialize All Components (p. 383) .
Uninitialize All	Uninitializes all turbo components. For details, see Uninitializing Components (p. 384) .
Show in Separate Window	Displays the selected plot or chart in its own window.
Promote to General Mode	Copies the selected plot object and any required supporting objects (for example, a line locator) to the Outline workspace. This would enable, for example, the selected plot to be included in a report.

15.5. Turbo Surface

Turbo surfaces are graphic objects that can be viewed and used as locators, just like other graphic objects. To create a turbo surface, select **Insert > Location > Turbo Surface** from the menu bar. After you enter a name in the new **Turbo Surface** dialog box and click **OK**, the details view for the turbo surface will appear.

Note:

Blade Aligned Turbo Surfaces can fail due to the following limitations:

- The extraction of leading and trailing edges of the blade is sensitive to tip clearance and to the curvature of the edges.
- The normalization of coordinates is sensitive to blade extend comparing to inlet and outlet extend (that is, when the edges are too close to inlet/outlet).

You can always use the Streamwise Location coordinate when the quality of the blade aligned coordinates are in doubt.

15.5.1. Turbo Surface: Geometry

Options available for **Definition** are:

- Constant Span
- Constant Streamwise Location
- Constant Blade Aligned
- Constant Blade Aligned Linear
- Constant Theta
- Cone

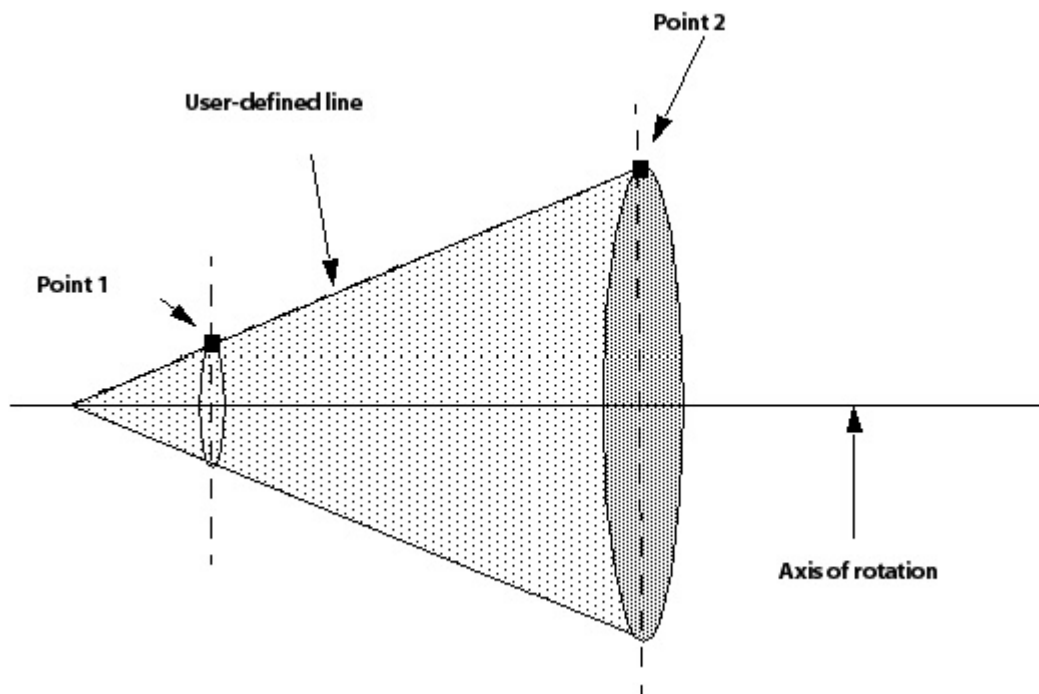
The Constant Span, Constant Streamwise Location, and Constant Theta options are similar to planes in that they can be bounded and have Slice or Sample types. For details, see [Type \(p. 391\)](#).

15.5.1.1. Domains

See [Selecting Domains \(p. 52\)](#).

15.5.1.2. Definition

- Constant Span creates a surface at a fractional span value between the hub and shroud. For details, see [Span Normalized \(p. 393\)](#).
- Constant Streamwise Location creates a surface at a fractional streamwise distance between the inlet and outlet. For details, see [Streamwise Location \(p. 393\)](#).
- Constant Blade Aligned creates surfaces that are aligned with the leading and trailing edges of the blade. If the blade is curved, the surfaces are similarly curved.
- Constant Blade Aligned Linear creates flat surfaces that approximate the leading and trailing edges of the blade.
- Constant Theta creates a surface at a specific Theta value. For details, see [Theta \(p. 394\)](#).
- Cone uses the two supplied points to create a line. The cone is created where the user-defined line intersects the axis of rotation and Point 2:



The user-defined line is then rotated about the axis of origin to create the cone. If the line is parallel to the axis of rotation, a cylinder is created. If the line is normal to the axis of rotation, a disc is created.

The line can be described by Cartesian or cylindrical components. When entering cylindrical coordinates, only the axial distance and radius are required. The points can be entered or picked directly from the viewer.

Note:

Constant Theta and Cone methods are available even before turbo initialization has been performed because these methods do not depend on span or streamwise coordinates.

15.5.1.3. Bounds

The available types of **Bounds** for the **Turbo Surface** to be created can be seen by clicking  next to the Type box.

- When **None** is selected, the **Turbo Surface** cuts through a complete cross-section of each domain specified in the **Domains** list. The **Turbo Surface** is bounded only by the limits of the domain.
- Using **Rectangular**, you can enter the maximum and minimum value for the two dimensions on the **Turbo Surface**. The **Turbo Surface** is undefined in areas where the rectangle extends outside of the domains specified in the **Domains** list.

The **Invert Surface Bounds** check box reverses the effect of the surface bound. The surface is defined only in regions outside the bounding constraints.

15.5.1.4. Type

You can set the **Type** to either **Slice** or **Sample**.

Slice extends the **Turbo Surface** in all directions until it reaches the edge of the domain. Points on the **Turbo Surface** correspond to points where the **Turbo Surface** intersects an edge of the mesh. As a result, the number of points in a slice **Turbo Surface** is indirectly proportional to the mesh spacing.

Sample creates the **Turbo Surface** with rectangular bounds. The density of points on the **Turbo Surface** corresponds to the size of the bounds for your **Turbo Surface** in each of the **Turbo Surface** directions, and the value in the **Samples** box for each of the two directions that describe the **Turbo Surface**. You can type in the value in the **Samples** box, increase or decrease the value by 1 by clicking  or  respectively, or use the embedded slider (which has a maximum value of 998 and a minimum value of 2). A sample **Turbo Surface** is a set of evenly-spaced points which are independent of the mesh spacing.

15.5.2. Turbo Surface: Common Tabs

You can adjust the other Turbo Surface settings on tabs that are common to other features in CFD-Post:

15.5.2.1. Turbo Surface: Color

To change the color settings, click the **Color** tab. For details, see [Color Tab \(p. 52\)](#).

15.5.2.2. Turbo Surface: Render

To change the rendering settings, click the **Render** tab. For details, see [Render Tab \(p. 55\)](#).

15.5.2.3. Turbo Surface: View

To change the view settings, click the **View** tab. For details, see [View Tab \(p. 60\)](#).

15.6. Turbo Line

Turbo lines are graphic objects that can be viewed and used as locators, just like other graphic objects. To create a turbo line, select **Insert > Location > Turbo Line** from the menu bar. After entering a name in the new **Turbo Line** dialog box, the details view for the turbo line will appear.

15.6.1. Turbo Line: Geometry

Set the following:

1. Set the applicable **Domains** as described in [Selecting Domains \(p. 52\)](#).
2. Define the line. The options available for the **Method** are:
 - **Inlet to Outlet**, which creates a line at specific [Span Normalized \(p. 393\)](#) and [Theta \(p. 394\)](#) value, over a range of streamwise values.

Select the number of points along the line per component with the value you enter in the **Samples/Comp** box. The sample line is a set of evenly-spaced sampling points that are independent of the mesh spacing.

- **Hub to Shroud**, which creates a line of a specific **Mode** at a specific **Theta** (p. 394) value. The method for creating a line in this way is the same as for the locator line in a hub-to-shroud turbo chart. For details on the possible **Mode** settings, see [Hub to Shroud Turbo Charts](#) (p. 398).

Tip:

If you set a mesh-density based turbo line and want to be able to see the points of analysis so that you can set an appropriate amount of reduction, you can create a vector (**Insert > Vector**) and define its **Location** to be the turbo line.

- **Circumferential**, which creates a line at specific streamwise and span values, over a range of Theta values. The number of samples is required. The number of points along the line will correspond to the value you enter in the **Samples** box. The sample line is a set of evenly-spaced sampling points that are independent of the mesh spacing. For details, see:

- [Streamwise Location](#) (p. 393)
- [Span Normalized](#) (p. 393).

3. For **Inlet to Outlet** and **Circumferential**, set the **Bounds**.

- When **None** is selected, the **Turbo Line** is restricted to only the parameters specified in the Definition section of the form. The **Turbo Line** is not bounded by the limits of the domain if the conditions you specify describe locations outside of the domain.
- When **End Points** is selected, you can define the ends of the **Turbo Line** by entering the maximum and minimum for the dimension making up the line. The **Turbo Line** is visible but will be colored with an undefined color in areas where the line extends outside of the domains specified in the **Domains** list.

15.6.2. Turbo Line: Common Tabs

You can adjust the other Turbo Surface settings on tabs that are common to other features in CFD-Post:

15.6.2.1. Turbo Line: Color

To change the color settings, click the **Color** tab. For details, see [Color Tab](#) (p. 52).

15.6.2.2. Turbo Line: Render

To change the rendering settings, click the **Render** tab. For details, see [Render Tab](#) (p. 55).

15.6.2.3. Turbo Line: View

To change the view settings, click the **View** tab. For details, see [View Tab](#) (p. 60).

15.7. Turbo Plots

The following topics will be discussed:

15.7.1. Introduction to Turbo Plots

15.7.2. Initialization Three Views

15.7.3. 3D View Object

15.7.4. Blade-to-Blade Object

15.7.5. Meridional Object

15.7.6. Turbo Charts

15.7.1. Introduction to Turbo Plots

Each turbo plot appears in the Turbo tree view under **Plots**, and can be edited.

15.7.1.1. Show Faces/Show Mesh Lines

Double-click 3D View and choose to view the faces (**Show Faces**) or the edges of the mesh elements (**Show Mesh Lines**) by enabling the appropriate toggles.

15.7.1.2. Graphical Instancing

The instancing information has already been entered during the initialization phase. You can opt to show instancing for the plots in each domain by changing the **# of Copies**. For details, see [Instancing Tab](#) (p. 387).

15.7.1.3. Turbo Measurements

15.7.1.3.1. Span

Span is the distance from hub region in length units.

15.7.1.3.2. Span Direction

Span Direction is a unit vector pointing along the direction of span coordinate for each mesh node, based on the background mesh.

15.7.1.3.3. Span Normalized

Span Normalized defines the dimensionless distance (between 0 and 1) from the hub to the shroud. For example, if the distance between the hub and shroud in a straight duct is 0.1 m, a span of 0.9 would describe a location 0.09 m from the hub and 0.01 m away from the shroud.

15.7.1.3.4. Streamwise Location

Streamwise location is the dimensionless distance from the inlet to the outlet. It ranges from 0 to 1 for the first component, 1 to 2 for the second, and so on. For example, in a single domain case, if the distance between the inlet and the outlet in a straight duct is 1 m, a streamwise location of 0.4 would describe a location 0.4 m from the inlet and 0.6 m away from the outlet. If the

same duct were the second component in a multi-component case, the same location would then be expressed as a streamwise location of 1.4.

15.7.1.3.5. Theta

Theta is the angular coordinate measured about the axis of rotation following the right-hand rule.

The Theta variable is intentionally generated by CFD-Post to have the following two properties:

- A minimum Theta value of zero (at the inlet).
- Continuously increasing values of Theta independent of the total blade wrap. This is particularly useful for high-wrap blades.

Because of these properties, the Theta variable generated in CFD-Post is most likely different than that of a user-defined expression based on the Cartesian coordinates.

After turbo initialization, the Theta range starts at a small but non-zero value. To set the Theta range to exactly zero, use the command `>turbo update_theta`.

15.7.1.3.6. Advanced: Position of Zero Theta

The position of zero Theta ($\text{Theta} = 0^\circ$) relative to the global coordinate system depends on the loaded case. For geometries that define a partial machine (not full 360°), zero Theta is at the geometry point with the lowest angle following the right-hand rule. For full 360° geometries, zero Theta is generally at an arbitrary position. You can specify zero Theta via an environment variable. For details, see [Setting CFD-Post Operation Through Environment Variables \(p. 39\)](#).

15.7.2. Initialization Three Views

If you double-click **Initialization** in the Turbo tree view, the Initialization editor appears and a **Three Views** toggle will be available. Selecting this toggle causes the viewer to show three viewports that have the following views:

- A **3D View**, which is described in [3D View Object \(p. 395\)](#), is the same as the standard viewer, with 3D manipulation available using the rotate, translate and zoom functions.
- A **Blade-to-Blade** 2D view, which is described in [Blade-to-Blade Object \(p. 395\)](#). The horizontal axis shows streamwise location and the vertical axis shows Theta. The 2D view enables translation, zoom, and rotation around the axis normal to the blade-to-blade view. Other rotations are not possible.
- A **Meridional** 2D view, which is described in [Meridional Object \(p. 396\)](#). The horizontal axis shows axial distance and the vertical axis shows the radius. The view will allow the same transformations as the blade-to-blade view, with rotation possible around the axis normal to the meridional view.

The three views listed above are also listed in the **Turbo** tree view as objects under **Plots**. The **Blade-to-Blade** and **Meridional** objects can be copied into the **Outline** tree view by right-clicking and selecting **Promote to General Mode**.

15.7.3. 3D View Object

3D View is used to draw regions of the turbo assembly for visualization purposes. It is not intended to be the basis for quantitative calculations. Select the regions that you want to draw.

After creating the blade-to-blade object (select the **Three Views** toggle in the Initialization object), you can view the blade-to-blade object in the **3D View** object by setting the appropriate option in the **3D View** object.

Note that you can view chart location lines in the **3D View** object by setting the appropriate option in the **3D View** object.

15.7.4. Blade-to-Blade Object

The Blade-to-Blade object is used to view plots on a surface of constant span. The surface is displayed in the Cartesian (X-Y-Z) and Blade to Blade views.

1. Select the **Domain(s)**.

To select more than one domain, click the multiple select icon and pick the entities.

2. Choose the fractional **Span** (0 to 1) where the plot is located. The **Plot Type** can be one of the following:

- [Color \(p. 396\)](#)
- [Contour \(p. 396\)](#)
- [Vector \(p. 396\)](#)
- [Stream \(p. 396\)](#)

3. Select a color if **Color By Variable** is not chosen as the **Plot Type**.

For details, see [Graphical Instancing \(p. 393\)](#).

15.7.4.1. Span

Set the fractional distance between the hub and shroud. For details, see [Span Normalized \(p. 393\)](#).

15.7.4.2. Angular Shift

The **Angular Shift** parameter moves the blade-to-blade plot along the Theta coordinate. This is useful to control the point of splitting in high wrap turbo cases. It does not affect the data; this is purely a rendering feature.

15.7.4.3. Plot Type

15.7.4.3.1. Color

The **Color** option displays variable values using a color legend. It requires the specification of a variable, range and the option of using hybrid or conservative values. For details, see:

- [Mode: Variable and Use Plot Variable \(p. 52\)](#)
- [Graphical Instancing \(p. 393\)](#).

15.7.4.3.2. Contour

Contour lines are drawn on the location described by the surface plot. Additional information on the option is available in [Contour Command \(p. 264\)](#).

15.7.4.3.3. Vector

A vector plot is created on the location described by the surface plot. For details, see [Vector Command \(p. 261\)](#).

15.7.4.3.4. Stream

A plot of streamlines are drawn on the location described by the surface plot. For details, see [Streamline Command \(p. 268\)](#).

15.7.5. Meridional Object

The **Meridional** object is used to view plots on an axial-radial plane. A surface of constant Theta at 0 degrees is created. The surface is displayed in the **Cartesian (X-Y-Z)** and **Meridional (A-R)** viewports.

1. Specify the applicable domains.

To select multiple domains, click the *Location editor* icon (beside the **Domains** setting) then use **Shift** and/or **Ctrl** while selecting domains.

2. Choose the number of **Stream Samples** and **Span Samples**.

Note that, for the Meridional object, sampling points are always distributed by equal distance.

3. Choose from: Out line, Color, Contour, or Vector plot types.

In order to obtain values for variables on the meridional surface, circumferential averaging is used. The types of circumferential averaging are:

- Length

Circumferential averaging is carried out in the same way as for a Hub to Shroud turbo chart (see [Circumferential Averaging by Length: Hub to Shroud Turbo Chart \(p. 404\)](#)) except that the sampling points are always distributed by equal distance.

- Area (default)

A variable value at each sampling point is calculated as an area average over the corresponding circular band. The band is constructed as for a Hub to Shroud turbo chart (see [Circumferential Averaging by Area: Hub to Shroud Turbo Chart \(p. 405\)](#)) except that the sampling points are always distributed by equal distance.

- Mass

A variable value at each sampling point is calculated as a mass flow average over the corresponding circular band. The band is constructed as for a Hub to Shroud turbo chart (see [Circumferential Averaging by Mass Flow: Hub to Shroud Turbo Chart \(p. 405\)](#)) except that the sampling points are always distributed by equal distance.

Toggles are available to show the following:

- Blade wireframe
- Sample mesh
- Chart location lines

15.7.6. Turbo Charts

The following turbo charts are available:

[15.7.6.1. Blade Loading Turbo Charts](#)

[15.7.6.2. Circumferential Turbo Charts](#)

[15.7.6.3. Hub to Shroud Turbo Charts](#)

[15.7.6.4. Inlet to Outlet Turbo Charts](#)

15.7.6.1. Blade Loading Turbo Charts

The **Blade Loading** feature plots pressure (or another chosen variable) on the blade at a given spanwise location. A polyline is created at the given spanwise location.

A special variable, `Streamwise (0-1)` is available as the `X Variable` used in blade loading plots. This is a streamwise coordinate that follows the blade surface; it can be used as a substitute for the axial coordinate (for example, `X`) or the variable `Chart Count`. The streamwise coordinate is based on the meridional coordinate, and is normalized so that it ranges from 0 at the leading edge to 1 at the trailing edge of the blade.

15.7.6.2. Circumferential Turbo Charts

Select a streamwise and spanwise location and a number of sampling points.

Note:

The Theta extents of the chart line are set to the Theta extents of the domain or, in the case of data instancing, the Theta extents of the expanded set of domains. Some of the sample points may fall outside the domain. To see the circumferential chart line, edit the `Plots > 3D View` object and turn on `Show chart location lines`.

15.7.6.3. Hub to Shroud Turbo Charts

Hub to Shroud has the following options:

- 15.7.6.3.1. Single Line vs. Two Lines
- 15.7.6.3.2. Display
- 15.7.6.3.3. Mode
- 15.7.6.3.4. Point Type
- 15.7.6.3.5. Theta
- 15.7.6.3.6. Samples
- 15.7.6.3.7. Streamwise
- 15.7.6.3.8. Distribution
- 15.7.6.3.9. X/Y Variable
- 15.7.6.3.10. Circumferential Averaging by Length: Hub to Shroud Turbo Chart
- 15.7.6.3.11. Circumferential Averaging by Area: Hub to Shroud Turbo Chart
- 15.7.6.3.12. Circumferential Averaging by Mass Flow: Hub to Shroud Turbo Chart
- 15.7.6.3.13. Constant Blade Aligned Linear Coordinates
- 15.7.6.3.14. Constant Blade Aligned Coordinates

15.7.6.3.1. Single Line vs. Two Lines

Select either **Single Line** or, to perform a comparison between two streamwise locations on a hub-to-shroud plot, to **Two Lines**.

15.7.6.3.2. Display

If you have selected **Two Lines**, you can set **Display** to:

- **Separate Lines**
Displays the two lines without performing any comparisons.
- **Difference (S2–S1)**
Displays the difference in the circumferentially averaged variable between the two locations, relative to the first line's location.
- **Ratio (S2/S1)**
Displays the ratio of the difference in the circumferentially averaged variable between the two locations, relative to the first line's location.

When **Display** is set to **Difference (S2–S1)** or **Ratio (S2/S1)**, you can set the **Compare** option to **X Values** or to **Y Values**. The selected values will be compared between the two lines.

15.7.6.3.3. Mode

Set **Mode** to one of the following options:

- Two Points Linear

The Two Points Linear option causes the hub-to-shroud line to be a straight line, specified by two points: one on the hub and one on the shroud. The **Point Type** setting (described below) specifies the coordinate system for interpreting the specified points.

- Blade Aligned Linear

The Blade Aligned Linear option causes the hub-to-shroud line to be specified by a curve of constant *Linear BA Streamwise Location* coordinate. For details, see [Constant Blade Aligned Linear Coordinates](#) (p. 405).

- Blade Aligned

The Blade Aligned option causes the hub-to-shroud line to be specified by a curve of constant *BA Streamwise Location* coordinate. For details, see [Constant Blade Aligned Coordinates](#) (p. 406).

- Streamwise Location

The Streamwise Location option causes the hub-to-shroud line to be specified by a curve of constant streamwise coordinate. Here, the streamwise coordinate system is derived from a "background mesh". For details, see [Background Mesh Frame](#) (p. 386).

Note:

- Blade Aligned coordinates may not always be available, depending on the case geometry. In particular, if the blade tip clearance is large or uneven between the leading and trailing edges, CFD-Post may not be able to detect the blade edge lines. In this case you will not be able to use Blade Aligned coordinates in turbo surface or turbo chart specification.
 - In turbo line, turbo surface, and related editors, the Blade Aligned coordinate values that you enter in the input fields (and the related CCL parameters) are normalized to the blade's leading and trailing edge locations with predefined constant references: 0.25 and 0.75 are taken to be the blades leading and trailing edges, respectively. The normalization of the input values is to enable a consistent reference to the leading and trailing edges regardless of specific cases. These values are conventions, not real blade aligned coordinated values; the normalized values are translated by the engine to create the real Blade Aligned coordinate values before constructing turbo lines and turbo surfaces.
-

15.7.6.3.4. Point Type

The **Point Type** setting is applicable when **Mode** is set to Two Points Linear. It controls the coordinate system for defining the specified hub and shroud point coordinates. The options for **Point Type** are:

- AR

When the AR option is selected, the hub and shroud points are specified in AR (axial, radial) coordinates.

- XYZ

When the XYZ option is selected, you specify the x, y, and z coordinates of the line's end points.

- Blade Aligned Linear

When the Blade Aligned Linear option is selected, the hub and shroud points are specified, each by a single *Linear BA Streamwise Location* coordinate. For details, see [Constant Blade Aligned Linear Coordinates \(p. 405\)](#).

- Streamwise Location

When the Streamwise Location option is selected, the hub and shroud points are specified, each by a single streamwise coordinate. Here, the streamwise coordinate system is derived from a "background mesh". For details, see [Background Mesh Frame \(p. 386\)](#).

15.7.6.3.5. Theta

The [Theta \(p. 394\)](#) setting is available with the **Hub to Shroud** methods.

15.7.6.3.6. Samples

The **Samples** setting controls the number of sampling points between the hub and shroud.

15.7.6.3.7. Streamwise

The **Streamwise** field sets the streamwise location for a chart line.

When **Display** is set to Difference or Ratio, the **Streamwise** fields set the locations to compare.

Note:

When there is a domain interface at a single streamwise location, for example a domain interface between the rotor and stator domains of a compressor stage, choosing a streamwise location exactly on the interface might lead to unexpected results. To cause CFD-Post to sample values from the intended side of the interface, choose a streamwise position slightly away from the interface. For example, if there is an interface at a streamwise position of 1, you could set **Streamwise** to 0.99999 or 1.00001 to cause CFD-Post to sample values from one side or the other.

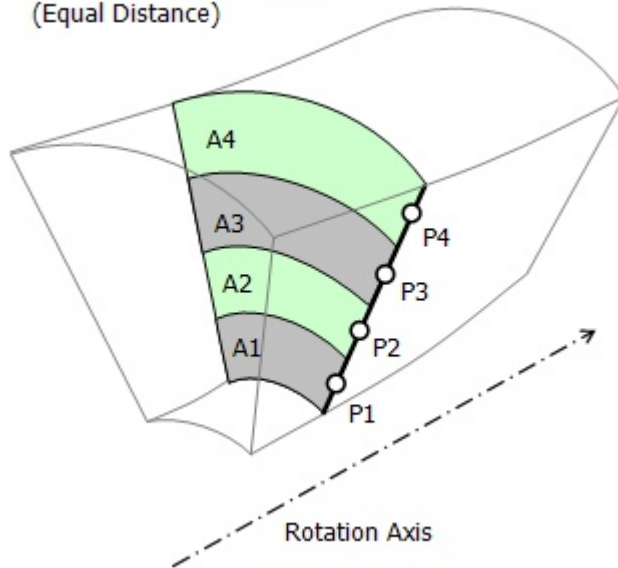
15.7.6.3.8. Distribution

Each sampling point value is evaluated from a corresponding circular band. The **Distribution** setting controls how the sampling points and their corresponding bands are distributed from hub to shroud (at the same streamwise coordinate).

The **Distribution** options are:

- Equal Distance

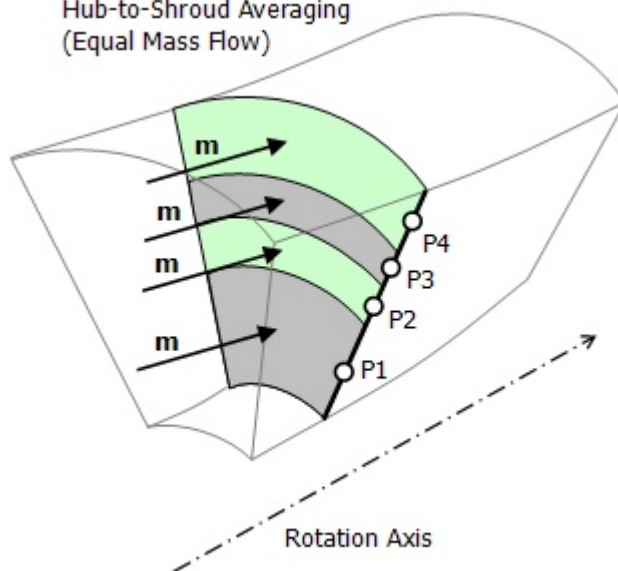
Hub-to-Shroud Averaging
(Equal Distance)



The **Equal Distance** option (default) causes the sampling points to be distributed at uniform distances along a hub-to-shroud path. For circumferential averaging purposes, contiguous circular bands are internally constructed, one for each sampling point, concentric about the rotation axis, width-centered (in the spanwise direction) about each sampling point, each band having the *same width* or spanwise extent.

- **Equal Mass Flow**

Hub-to-Shroud Averaging
(Equal Mass Flow)



The **Equal Mass Flow** option causes the sampling points to be distributed along a hub-to-shroud path such that contiguous circular bands can be internally constructed, one for each sampling point, concentric about the rotation axis, width-centered (in the spanwise direction)

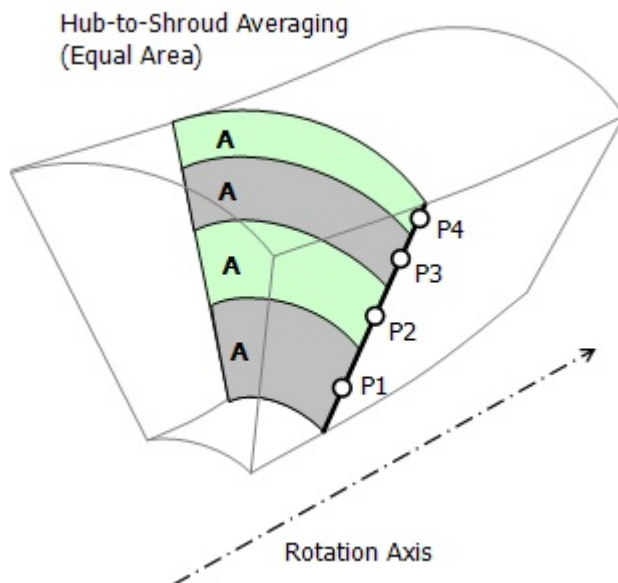
about each sampling point, *with an equal mass flow through each band* (except possibly the first and last bands). See **Include Boundary Points**, below.

Note:

CFD-Post cannot create an Equal Mass Flow point distribution for some cases:

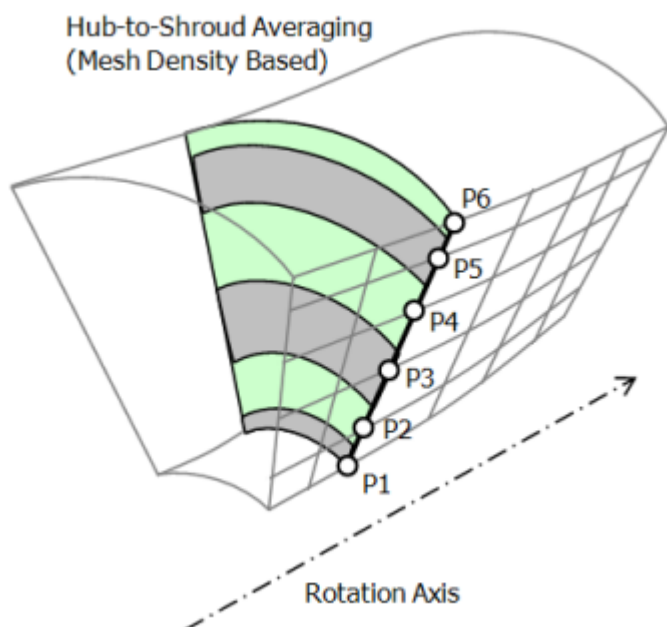
- When there is a cross-section recirculation and the total mass flow on the section is near zero, the point distribution will fail.
 - When there is a mass flow 'spike' on the section (usually this is caused by an ill-defined solution), the equal mass distribution will be impractical.
 - When too many sample points are requested over a small area.
-

• Equal Area



The Equal Area option causes the sampling points to be distributed along a hub-to-shroud path such that contiguous circular bands can be internally constructed, one for each sampling point, concentric about the rotation axis, width-centered (in the spanwise direction) about each sampling point, *with an equal area for each band* (except possibly the first and last bands). See **Include Boundary Points**, below.

• Mesh Density Based



The **Mesh Density Based** option causes the sampling points to be distributed along a hub-to-shroud path such that the sampling point density is proportional to the mesh node density along either

- the intersection of the inlet with the periodic surface, or
 - the intersection of the outlet with the periodic surface,
- whichever of these paths has a greater number of nodes.

The following settings are available:

- **Max. Number of Points** > **Max. Points**

The **Max. Points** setting controls the number of points of analysis.

- **Reduction Factor** > **Factor**

The **Factor** setting specifies the ratio of mesh nodes to sampling points along the hub-to-shroud path. A value of 1 causes one sampling point to be created per mesh node. You can reduce the computational time by setting a larger reduction factor.

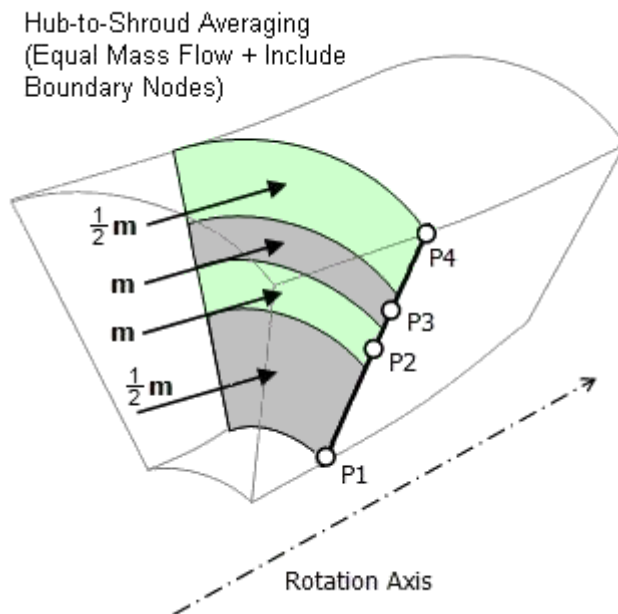
Note:

For Two-Line Hub to Shroud plots, you may not be able to create Difference and Ratio plots using **Reduction Factor** if the two lines are in different domains.

When **Distribution** is set to **Equal Mass Flow** or **Equal Area**, the **Include Boundary Points** option is available. This option shifts all of the bands so that the first and last sampling points are on the hub and shroud. The first and last bands are then "half" the size of the other bands

(in terms of the particular measure used in the band construction: distance, mass flow, or area). See [Figure 15.1: Sampling Point Distribution with Include Boundary Nodes Option](#) (p. 404).

Figure 15.1: Sampling Point Distribution with Include Boundary Nodes Option

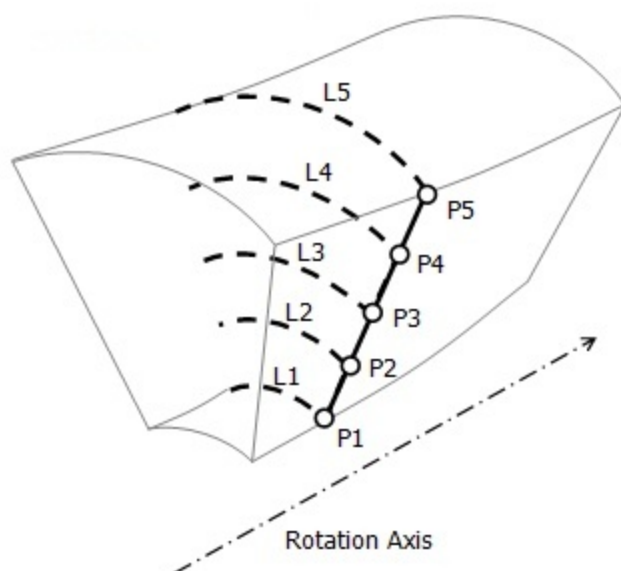


15.7.6.3.9. X/Y Variable

Choose X and Y variables for the chart axes from the list.

15.7.6.3.10. Circumferential Averaging by Length: Hub to Shroud Turbo Chart

When the **Circ. Average** setting is set to Length, circumferential averaging of values at a sampling point is carried out internally by forming a circular arc, centered about the rotation axis, passing through the sampling point. Values are interpolated to n equally-spaced locations along the arc, using values from nearby nodes, where n is a number that is inversely proportional to the mesh length scale, and limited by the **Max. Samples** setting. The n values are then averaged in order to obtain a single, circumferentially-averaged value for the sampling point.

Figure 15.2: Circumferential Averaging by Length

15.7.6.3.11. Circumferential Averaging by Area: Hub to Shroud Turbo Chart

When the **Circ. Average** setting is set to Area, a variable value at each sampling point is calculated as an area average over the corresponding circular band that was internally constructed as part of the process of distributing the sampling points. For details, see [Distribution \(p. 400\)](#).

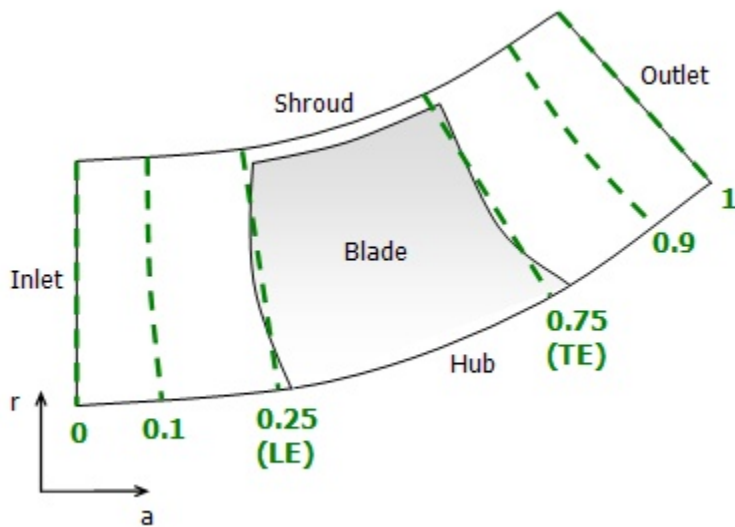
15.7.6.3.12. Circumferential Averaging by Mass Flow: Hub to Shroud Turbo Chart

When the **Circ. Average** setting is set to Mass, a variable value at each sampling point is calculated as a mass flow average over the corresponding circular band that was internally constructed as part of the process of distributing the sampling points. For details, see [Distribution \(p. 400\)](#).

15.7.6.3.13. Constant Blade Aligned Linear Coordinates

The Constant Blade Aligned Linear coordinates are defined as 0 (zero) at the inlet, 0.25 at a straight line that approximates the blade leading edge, 0.75 at a similar line for the trailing edge, and 1.0 at the outlet, adding 1.0 for each successive turbomachinery component downstream of the first. Dashed lines in [Figure 15.3: Blade Aligned Linear Coordinates \(p. 406\)](#) show constant values of Constant Blade Aligned Linear coordinate.

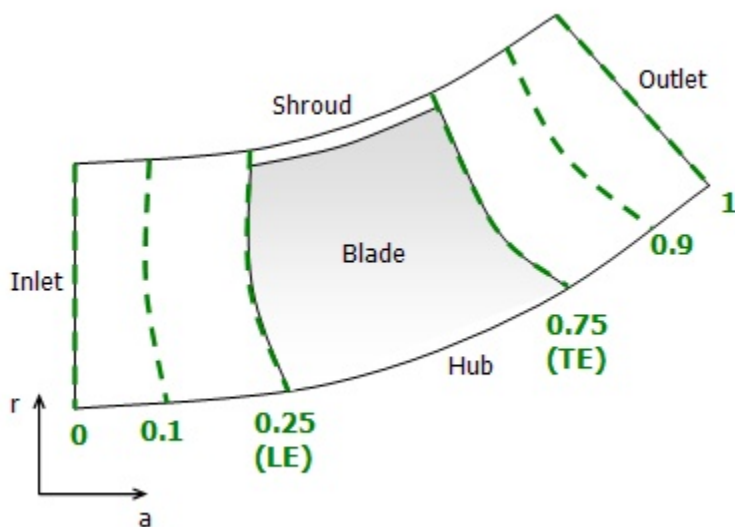
For more details on the convention that 0.25 and 0.75 are taken to be the blades leading and trailing edges, see [Mode \(p. 398\)](#).

Figure 15.3: Blade Aligned Linear Coordinates

15.7.6.3.14. Constant Blade Aligned Coordinates

The Constant Blade Aligned coordinates are defined as 0 (zero) at the inlet, 0.25 at the blade leading edge, 0.75 at the trailing edge, and 1.0 at the outlet, adding 1.0 for each successive turbomachinery component downstream of the first.

For more details on the convention that 0.25 and 0.75 are taken to be the blades leading and trailing edges, see [Mode \(p. 398\)](#).

Figure 15.4: Blade Aligned Coordinates

15.7.6.4. Inlet to Outlet Turbo Charts

The distance between sampling points between the inlet and outlet is controlled by the number you enter in the **Samples** box. Choose X and Y variables for the chart axes from the list.

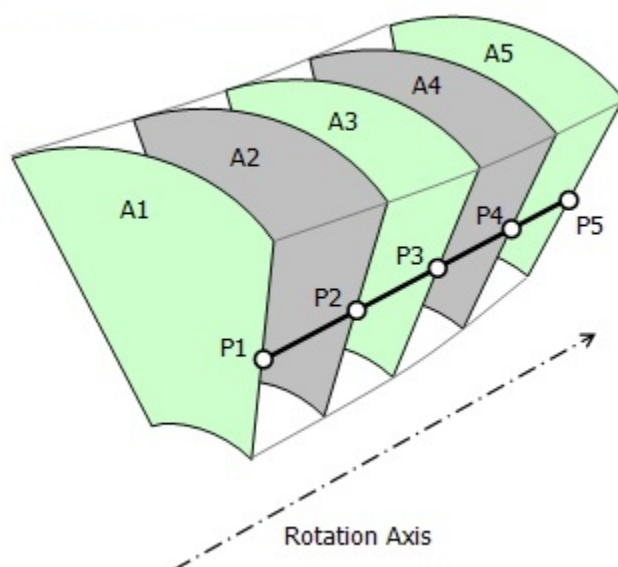
15.7.6.4.1. Circumferential Averaging by Length: Inlet to Outlet Turbo Chart

When the **Circ. Average** setting is set to Length, circumferential averaging of values is carried out internally by creating arcs through sampling about the rotation axis. Values are interpolated to n equally-spaced locations along the arc, using values from nearby nodes, where n is a number that is inversely proportional to the mesh length scale, and limited by the Max. Samples setting. The n values are then averaged in order to obtain a single, circumferentially-averaged value for the sampling point.

15.7.6.4.2. Circumferential Averaging by Area or Mass: Inlet to Outlet Turbo Chart

When performing area average or mass-flow average calculations, surfaces of constant-streamwise coordinate are used to carry out the averaging. Each surface passes through its associated sampling point, as shown in [Figure 15.5: Inlet to Outlet Sample Points](#) (p. 407).

Figure 15.5: Inlet to Outlet Sample Points



15.8. Turbo Macros

Select the macro of choice from the **Turbo** tree view.

Note:

Turbo initialization automatically sets up the performance macros in such a way that you have to define only a limited number of parameters. For details, see:

- [Gas Compressor Performance Macro \(p. 355\).](#)
- [Gas Turbine Performance Macro \(p. 356\)](#)
- [Liquid Pump Performance Macro \(p. 357\)](#)
- [Liquid Turbine Performance Macro \(p. 358\)](#)
- [Fan Noise Macro \(p. 359\).](#)

15.9. Calculate Velocity Components

The **Calculate Velocity Components** button on the Initialization object can be used to calculate velocity component (and other) variables pertinent to turbo simulations. These variables are listed in [Table 15.1: Generated Variables \(p. 408\)](#) and illustrated in [Figure 15.6: Axial, Radial, Circumferential, and Meridional Velocity Components \(p. 410\)](#), [Figure 15.7: Velocity Components in Meridional Plane \(p. 411\)](#), [Figure 15.8: Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components \(p. 412\)](#), [Figure 15.9: Velocity Components in Blade-To-Blade Plane \(p. 413\)](#), and [Figure 15.10: Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane \(p. 414\)](#). The relationship between the velocity components is described in [Equation 15.1 \(p. 410\)](#), [Equation 15.2 \(p. 412\)](#), and [Equation 15.3 \(p. 413\)](#).

Note:

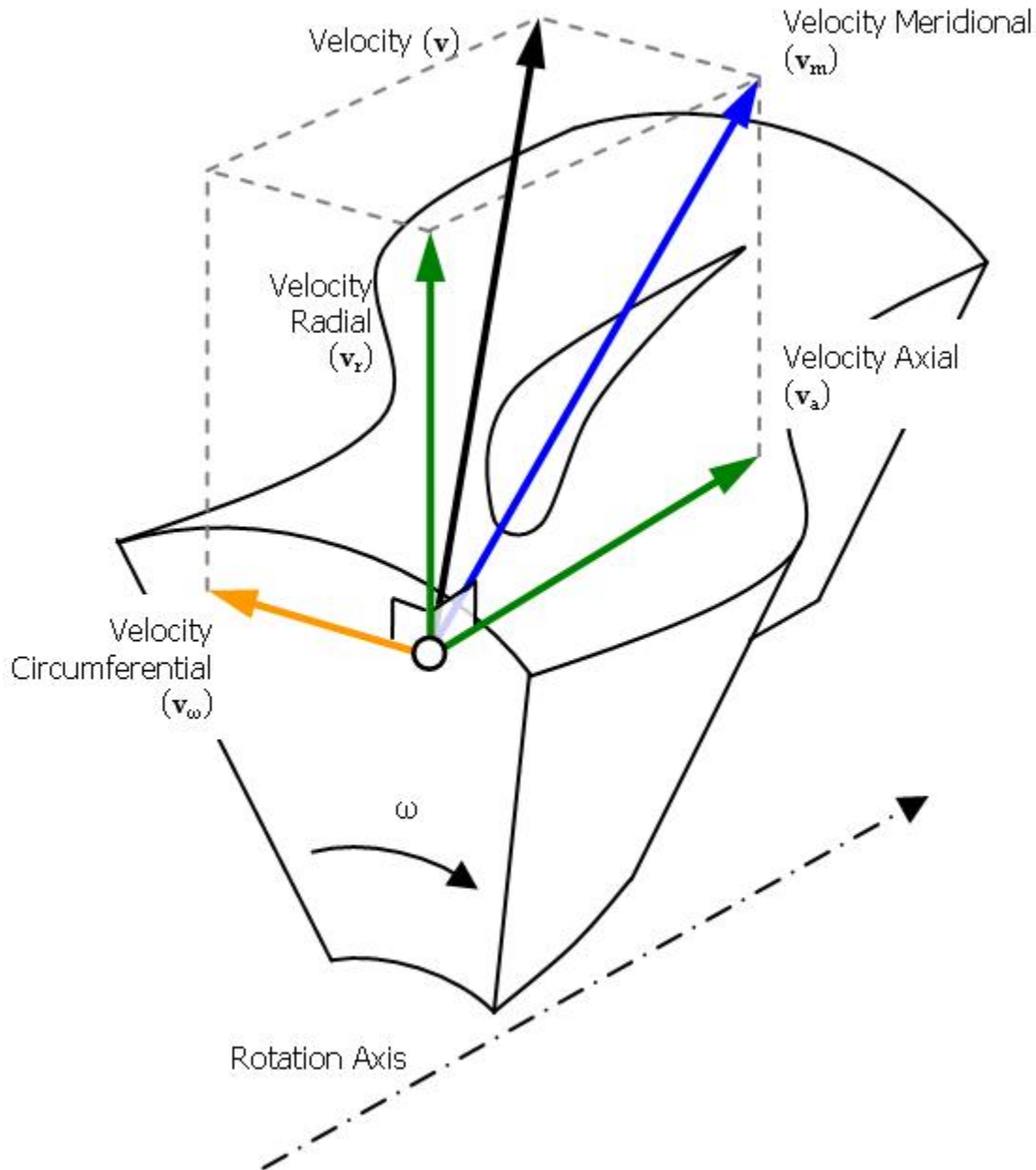
To get velocity units for tip speed derived from R and Omega quantities, you can divide the expression by 1 [rad] to eliminate the angle units from the expression. For example, use:

```
tipVel = Radius * omega / 1 [rad]
```

Table 15.1: Generated Variables

Variable Name	Type	Description
Velocity Axial	Scalar	The velocity component in the axial direction. It is positive when the velocity is in the direction of increasing axial coordinate. For details, see Figure 15.6: Axial, Radial, Circumferential, and Meridional Velocity Components (p. 410) and Figure 15.7: Velocity Components in Meridional Plane (p. 411) .
Velocity Radial	Scalar	The velocity component in the radial direction. It is positive when the velocity is in the direction of increasing radial coordinate. For details, see Figure 15.6: Axial, Radial, Circumferential, and Meridional Velocity Components (p. 410) and Figure 15.7: Velocity Components in Meridional Plane (p. 411) .
Velocity Circumferential	Scalar	The velocity component in the Theta direction. It is positive when the velocity is in the

Variable Name	Type	Description
		direction of increasing Theta (for details, see Theta (p. 394)). For details, see Figure 15.6: Axial, Radial, Circumferential, and Meridional Velocity Components (p. 410).
Velocity Spanwise	Scalar	The velocity component in the spanwise direction. It is positive when the velocity is in the direction of increasing spanwise coordinate. For details, see Figure 15.7: Velocity Components in Meridional Plane (p. 411) and Figure 15.8: Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components (p. 412).
Velocity Streamwise	Scalar	The velocity component in the streamwise direction. It is positive when the velocity is in the direction of increasing streamwise coordinate. For details, see Figure 15.7: Velocity Components in Meridional Plane (p. 411), Figure 15.8: Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components (p. 412), and Figure 15.9: Velocity Components in Blade-To-Blade Plane (p. 413).
Velocity Meridional	Vector	The vector sum of the axial and radial vector components of velocity. It lies in the meridional plane. For details, see Figure 15.6: Axial, Radial, Circumferential, and Meridional Velocity Components (p. 410), Figure 15.7: Velocity Components in Meridional Plane (p. 411), and Equation 15.1 (p. 410).
Velocity Blade-to-Blade	Vector	The vector sum of the circumferential and streamwise vector components of velocity. It lies in the blade-to-blade plane. For details, see Figure 15.8: Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components (p. 412), Figure 15.9: Velocity Components in Blade-To-Blade Plane (p. 413), Figure 15.10: Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane (p. 414), and Equation 15.2 (p. 412).
Velocity Flow Angle	Scalar	The angle between the blade-to-blade and circumferential velocity vector components. For details, see Figure 15.10: Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane (p. 414).

Figure 15.6: Axial, Radial, Circumferential, and Meridional Velocity Components

The velocity in the meridional plane can be represented by axial and radial components or streamwise and spanwise components:

$$\begin{aligned}\mathbf{v}_m &= \mathbf{v}_a + \mathbf{v}_r \\ &= \mathbf{v}_{st} + \mathbf{v}_s\end{aligned}\tag{15.1}$$

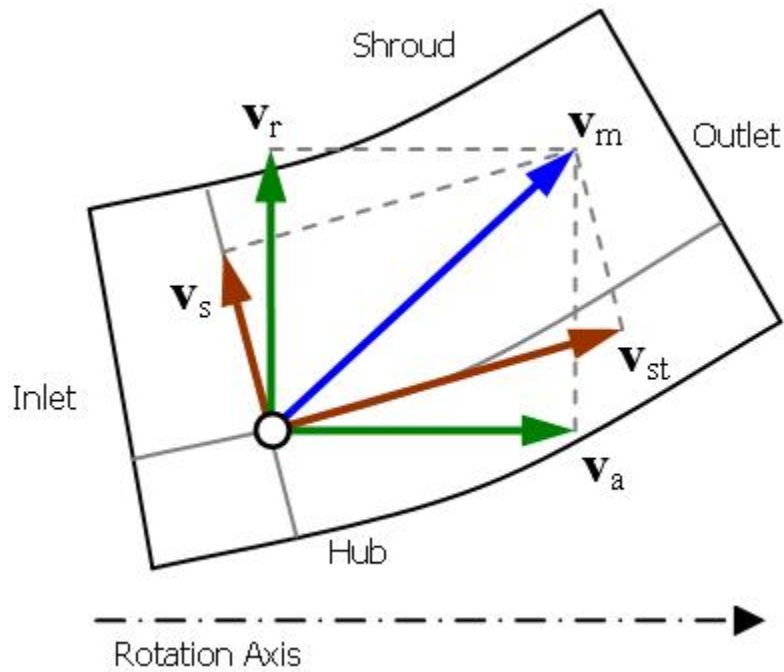
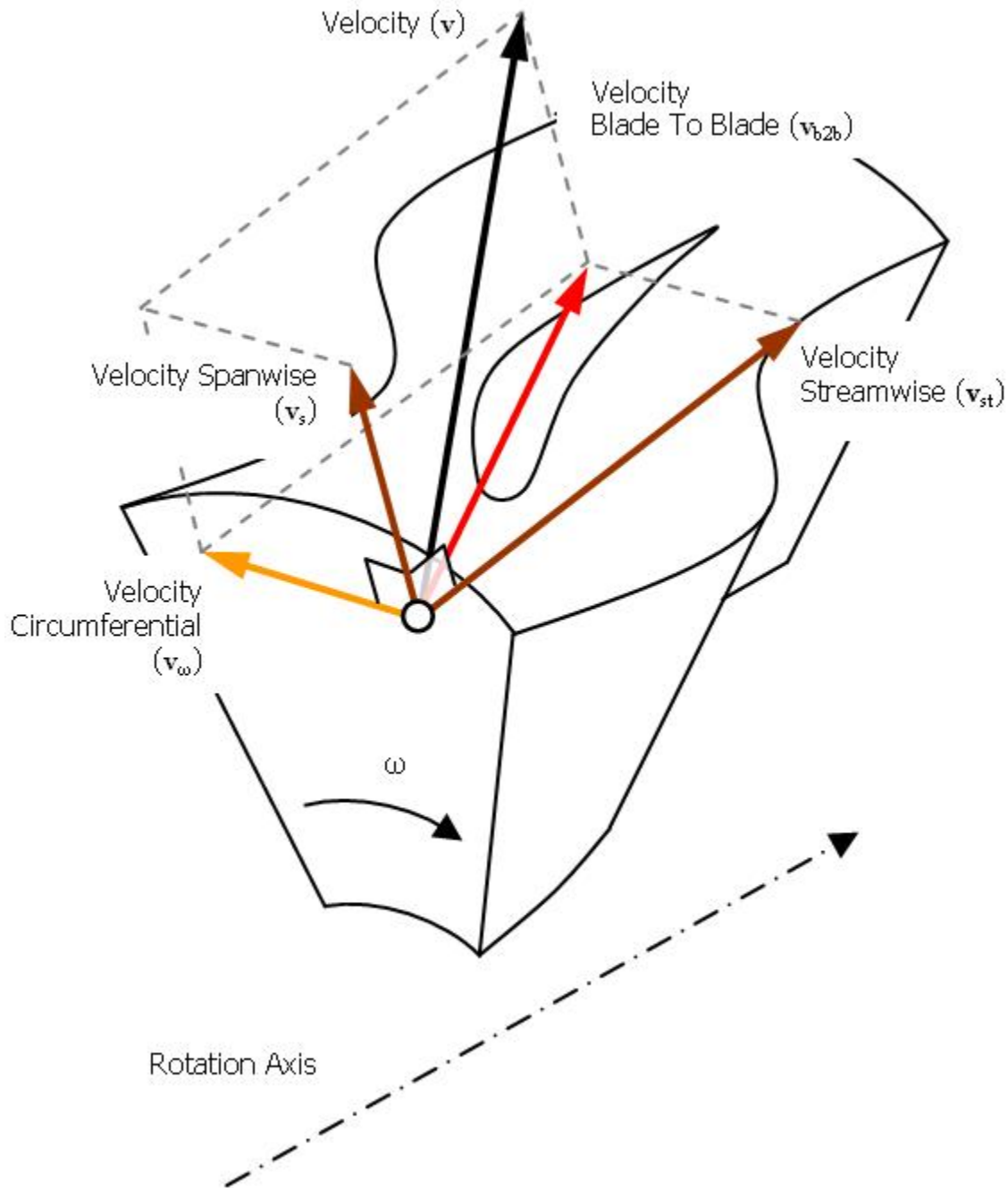
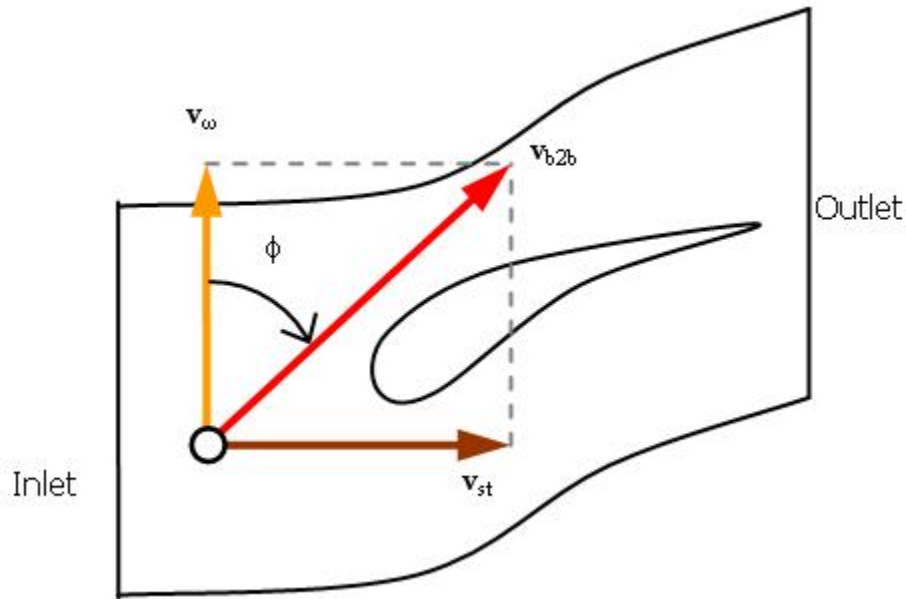
Figure 15.7: Velocity Components in Meridional Plane

Figure 15.8: Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components

The velocity in the blade-to-blade plane can be represented by streamwise and circumferential components:

$$\mathbf{v}_{b2b} = \mathbf{v}_{st} + \mathbf{v}_\omega \quad (15.2)$$

Figure 15.9: Velocity Components in Blade-To-Blade Plane

The velocity components are related as follows:

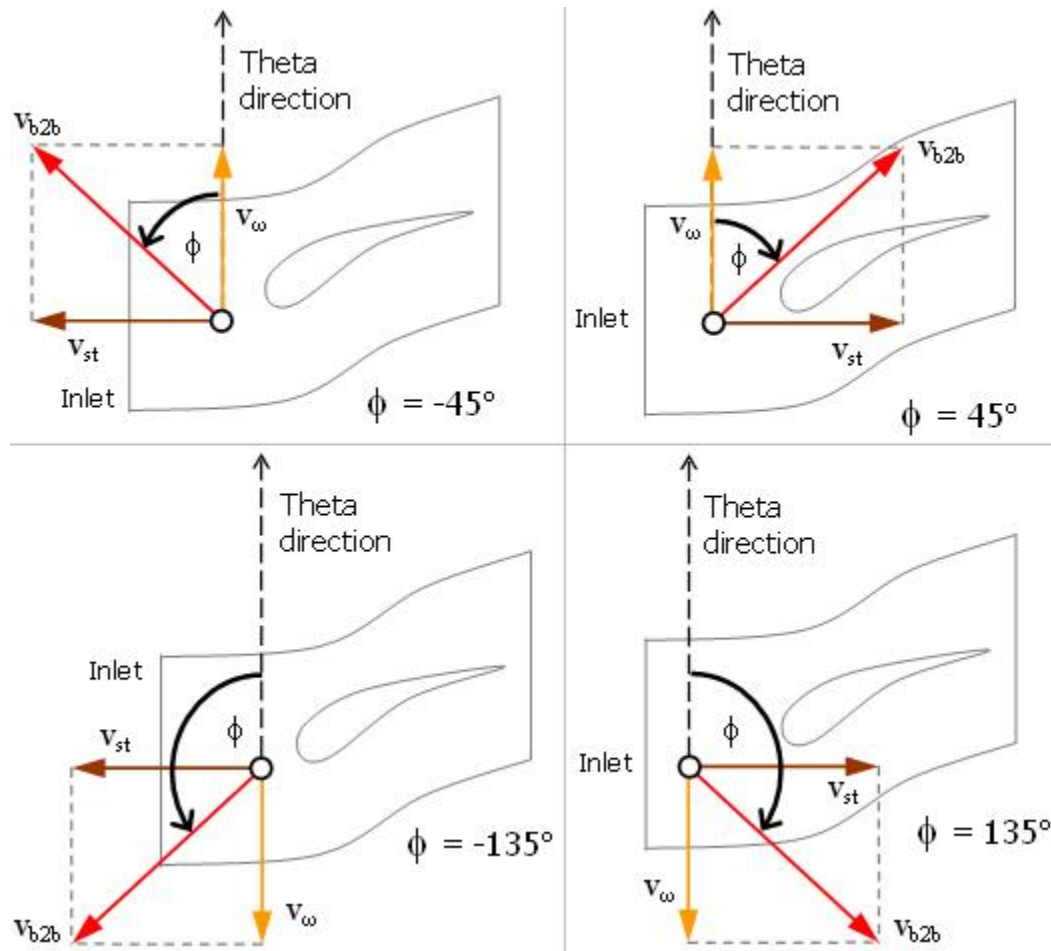
$$\begin{aligned}
 \mathbf{V} &= \mathbf{V}_a + \mathbf{V}_r + \mathbf{V}_\omega \\
 &= \mathbf{V}_{st} + \mathbf{V}_s + \mathbf{V}_\omega \\
 &= \mathbf{V}_m + \mathbf{V}_\omega \\
 &= \mathbf{V}_{b2b} + \mathbf{V}_s
 \end{aligned}
 \tag{15.3}$$

Axial, radial and meridional velocities are not calculated for **Velocity in Stn. Frame** because these components are not different from the regular **Velocity** components.

Information on calculating velocity components using CCL is available. For details, see [Calculating Velocity Components](#) (p. 431).

The range of **Velocity Flow Angle** is from -180° to $+180^\circ$. Four examples are shown in [Figure 15.10: Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane](#) (p. 414).

Figure 15.10: Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane



15.9.1. Calculating Cylindrical Velocity Components for Non-turbo Cases

If you want to calculate cylindrical velocity components for non-turbo cases, such as for swirling flows in axisymmetric geometries and mixing tank calculations, you can use the **Calculate Velocity Components** button on the **Turbo** workspace:

1. Load a results file for an axisymmetric simulation. (You can load a copy of StaticMixer_001.res (provided with a tutorial) to work through this example.)
2. Select the **Turbo** tab to open the Turbo workspace. A dialog box asks if you want to auto-initialize all components, but as this is unnecessary click **No**.
3. On the **Turbo** workspace's **Initialization** area, click **Define Global Rotational Axis**.
4. In the **Define Global Rotational Axis** dialog box, select the appropriate axis and click **OK**. (For the static mixer example, set **Axis** to Z.)
5. In the **Initialization** area, click **Calculate Velocity Components**. New variables such as **Velocity Circumferential** become available. (You can see these new variables in the **Variables** workspace.)
6. Create a plane so that you can display a cylindrical velocity component:

- a. From the menu bar, select **Insert** > **Location** > **Plane**. In the dialog box that appears, accept the default name and click **OK**.
- b. In the details view for Plane 1 on the **Geometry** tab, ensure that **Method** is YZ Plane.
- c. On the **Color** tab, set **Mode** to Variable and **Variable** to Velocity Circumferential.
- d. Click **Apply**. The plane is colored to show the velocity at each point.
- e. Right-click the viewer background and select **Predefined Camera** > **View From +X** so that the plane is easier to see.

Important:

Not all axisymmetric cases can have velocity components calculated in this way. In particular, cases that involve particles (such as smoke) will fail.

Chapter 16: CFX Command Language (CCL) in CFD-Post

[CFX Expression Language \(CEL\)](#) is an interpreted, declarative language that has been developed to enable CFX users to enhance their simulations without recourse to writing and linking separate external Fortran routines. You can use CEL expressions anywhere a value is required for input in Ansys CFX.

The CFX Command Language (CCL) is the internal communication and command language of CFD-Post. It is a simple language that can be used to create objects or perform actions in the postprocessor. All CCL statements can be classified into one of three categories:

- Object and parameter definitions, which are described in [Object Creation and Deletion \(p. 417\)](#).
- CCL actions, which are commands that perform a specific task (such as reading a session file), and which are described in [Command Actions \(p. 421\)](#).
- Power Syntax programming, which uses the Perl programming language to allow loops, logic, and custom macros (subroutines). Power Syntax enables you to embed Perl commands into CCL to achieve powerful quantitative postprocessing. For details, see [Power Syntax in Ansys CFX](#).

State files and session files contain object definitions in CCL. In addition, session files can also contain CCL action commands. You can view and modify the CCL in these files by using a text editor, and you can use CCL to create your own session and state files to read into CFD-Post.

Tip:

Advanced users can interact with CFD-Post directly by entering CCL in the **Command Editor** dialog box (see [Command Editor \(p. 377\)](#)), or by running CFD-Post in Line Interface mode (see [Line Interface Mode \(p. 433\)](#)).

For more information, see:

- [CFX Command Language \(CCL\) Syntax](#)
- [Object Creation and Deletion \(p. 417\)](#)

16.1. Object Creation and Deletion

You can create objects in CFD-Post by entering the CCL definition of the object into the **Command Editor** dialog box, or by reading the object definition from a session or state file. The object will be created and any associated graphics shown in the viewer.

You can modify an existing object by entering the object definition with the modified parameter settings into the **Command Editor** dialog box. Only those parameters that are to be changed need to be entered. All other parameters will remain unchanged.

There may be a significant degree of interaction between objects in CFD-Post. For example, a vector plot may depend on the location of an underlying plane, or an isosurface may depend on the definition of a CEL expression. If changes to one object affect other objects, the other objects will be updated automatically.

To delete an object, type `>delete <ObjectName>`. If you delete an object that is used by other objects, warnings will result, but the object will still be deleted.

Chapter 17: CFX Expression Language (CEL) in CFD-Post

This chapter provides information that is specific to CFX Expression Language (CEL) use in CFD-Post. For details on the CFX Expression Language, see [CFX Expression Language \(CEL\)](#). A list of variables available for use in CEL expression is available in [Variables and Predefined Expressions Available in CEL Expressions](#).

CEL Variables in CFD-Post

Within CFD-Post, you can:

- Create new expressions.
- Set any numeric parameter in a CFD-Post object based on an expression (and the object will update if the expression result changes).
- Create user-defined variables from expressions.
- Directly use the post-processor quantitative functions in an expression.
- Specify units as part of an expression.
- Use the variables x, y, and z in general CEL expressions. Additionally, you can use user-defined coordinate frames with the CEL functions. For details, see [Quantitative CEL Functions in Ansys CFX](#).

However, you cannot use CEL to solve systems of equations in CFD-Post—CEL expressions are purely algebraic operations.

All expressions in the post-processor are defined in the EXPRESSIONS singleton object (which is also a sub-object of LIBRARY:CEL). Each expression is a simple name = expression statement within that object. New expressions are added by defining new parameters within the expressions object (the EXPRESSIONS object is special in that it does not have a predefined list of valid parameters).

Note:

CFD-Post evaluates CEL expressions with single (not double) precision.

Important:

Because Power Syntax uses Perl mathematical operators, you should exercise caution when combining CEL with Power Syntax expressions. For example, in CEL, 2^2 is represented as 2^2 , but in Perl, would be written $2 * 2$. If you are unsure about the validity of an operator in Perl, consult a Perl reference guide.

17.1. Variables Created by CFD-Post

CFD-Post derives the following variables from the results file; you can use them in expressions or as plot variables:

Table 17.1: Variables Created by CFD-Post

Name	Description
Area	This is meaningful only for surface locators (user surface, plane, isosurface, boundary). The value at each node is equal to the sum of sector areas associated with the node (a sector area is the portion of area of a face touching a node that can be associated with that node). There is a function to sum this variable over a 2D locator to obtain the area of the locator; for details, see area in the CFX Reference Guide .
Force	There is a function for calculating force; for details, see force in the CFX Reference Guide .
Length	This is meaningful only for polyline and line objects. The value on each line node is equal to the sum of halves of the two line segments joined at the node. There is a function to sum this variable over a line locator to obtain the length of the locator; for details, see length in the CFX Reference Guide .
Mass Flow	There is a function for calculating mass flow; for details, see massFlow in the CFX Reference Guide .
Normal	This is meaningful only for surface locators (user surface, plane, isosurface, boundary). It is a vector variable defining the surface unit normal at each node in the locator.
Volume	This is defined only on volume locators (volume, domain, subdomain). The value at each node is equal to the sum of the sector volumes associated with the node (a sector volume is the portion of volume of an element touching a node that can be associated with that node). There is a function to sum this variable over a 3D locator to obtain the volume of the locator; for details, see volume in the CFX Reference Guide .

17.2. User Functions in CFD-Post

When a CFX results file is loaded, CFD-Post supports the evaluation of some user functions that were defined in the command language for the results file. Interpolation, profile data and table functions can all be evaluated.

- CFD-Post always evaluates profile data functions in the global coordinate frame.
- Profile data and table functions each reference data from a .csv file. CFX-Pre and CFX-Solver read from the .csv file specified in the function definition. CFD-Post reads not from a .csv file but from a copy of the .csv file that is stored within the results file.
- You can use expressions as user function arguments. Such expressions should have the same definition between CFD-Post and the CFX results file so that user function evaluations are consistent between CFD-Post and CFX-Solver.

Chapter 18: Command Actions

You can use command actions to edit or create graphic objects and to perform some typical actions (such as reading or creating session and state files).

This chapter describes:

[18.1. Overview of Command Actions](#)

[18.2. File Operations from the Command Editor Dialog Box](#)

[18.3. Quantitative Calculations in the Command Editor Dialog Box](#)

[18.4. Other Commands](#)

18.1. Overview of Command Actions

Command action statements are used to force CFD-Post to undertake a specific task, usually related to the input and output of data from the system. You can use command action statements in a variety of areas:

- You can enter command action statements into the **Tools > Command Editor** dialog box. All such actions must be preceded with the > symbol.

For details on the **Command Editor** dialog box, see [Command Editor \(p. 377\)](#). Additional information on editing and creating graphics objects using the CFX Command Language in the **Command Editor** dialog box is available in [CFX Command Language \(CCL\) in CFD-Post \(p. 417\)](#).

- Command actions also appear in session files (where they are also preceded by the > character).
- When running CFD-Post in **Line Interface** mode, the CFX> command prompt is shown in a Windows Command Prompt or UNIX shell. All the actions described in this section along with some additional commands can be entered at the command prompt. You do not have to precede commands with the > symbol when running in **Line Interface** mode. Additional information on using **Line Interface** mode is available in [Line Interface Mode \(p. 433\)](#).

Note:

In addition to command action statements, CCL takes advantage of the full range of capabilities and resources from an existing programming language, Perl. Perl statements can be embedded in between lines of simple syntax, providing capabilities such as loops, logic, and much, much more with any CCL input file. These *Power Syntax* commands are preceded by the ! symbol. Additional information on using Power Syntax in the **Command Editor** dialog box is available in [Power Syntax in Ansys CFX in the CFX Reference Guide](#).

Many actions require additional information to perform their task (such as the name of a file to load or the type of file to create). By default, these actions get the necessary information from a specific asso-

ciated CCL singleton object. For convenience, some actions accept a few arguments that are used to optionally override the commonly changed object settings. If multiple arguments for an action are specified, they must be separated by a comma (,). Lines starting with the # character are not interpreted and can be used for comments.

For example, all the settings for >print are read from the HARDCOPY: object. However, if you desire, you can specify the name of the hardcopy file as an argument to >print. The following CCL example demonstrates this behavior of actions:

```
# Define settings for printing
HARDCOPY:
  Hardcopy Format= jpg
  Hardcopy Filename = default.jpg
  Image Scale = 70
  White Background = Off
END
#Create an output file based on the settings in HARDCOPY
>print
#Create an identical output file with a different filename.
>print another_file.jpg
```

18.2. File Operations from the Command Editor Dialog Box

You can enter command action statements into the **Tools > Command Editor** dialog box. This section discusses the following actions:

- [Loading a Results File \(p. 422\)](#)
- [Reading Session Files \(p. 423\)](#)
- [Saving State Files \(p. 424\)](#)
- [Reading State Files \(p. 425\)](#)
- [Creating a Hardcopy \(p. 428\)](#)
- [Importing External File Formats \(p. 428\)](#)
- [Exporting Data \(p. 428\)](#)
- [Controlling the Viewer \(p. 429\)](#)

18.2.1. Loading a Results File

You load a results file by using the >load command. The parameter settings for loading the file are read from the DATA READER object. For simplicity, some parameters may be set via optional parameters as part of the load command.

```
>load [filename=<filename>][timestep=<timestep>]
```

If a timestep is not specified, a value of -1 is assumed (this corresponds to the Final state).

When a results file is loaded, all **Domain**, **Boundary**, and **Variable** objects associated with the results file are created or updated. **Variable** objects are created, but the associated data is not actually read

into the post-processor until the variables are used (load-on-demand). Variables will be pre-loaded if specified in the DATA READER.

18.2.1.1. load Command Examples

The following are example >load commands with the expected results.

```
>load filename=c:/CFX/tutorials/Buoyancy2DVMI_002.res, timestep=3
```

This command loads the specified results file at timestep 3.

Tip:

If going from a transient to steady-state results file, you should specify the timestep to be -1 (if this is not the current setting). If you do not explicitly set this, you will get a warning message stating that the existing timestep does not exist. The -1 timestep will then be loaded.

```
>load timestep=4
```

This command loads timestep 4 in the existing results file.

18.2.2. Reading Session Files

```
>readsession [filename=<filename>]
```

Performs session file reading and executing. The following option is available:

- `filename = <filename>`

Specifies the filename and path to the file that should be read and executed. If no filename is specified, the SESSION singleton object indicates the file to use. If no SESSION singleton exists, an error will be raised indicating that a filename must be specified.

18.2.2.1. readsession Command Examples

The following are example >readsession commands, and the expected results. If a SESSION singleton exists, the values of the parameters listed after the session command replaces the values stored in the SESSION singleton object. For this command, the filename command parameter value replaces the session filename parameter value in the SESSION singleton.

```
>readsession
```

Reads the session file specified in the SESSION singleton, and execute its contents. If the SESSION object does not exist, an error will be raised indicating that a filename must be specified.

```
>readsession filename=mysession.cse
```

Reads and execute the contents of the mysession.cse file.

18.2.3. Saving State Files

```
>savestate [mode=<none | overwrite>][filename=<filename>]
```

State files can be used to quickly load a previous state into CFD-Post. State files can be generated manually using a text editor, or from within CFD-Post by saving a state file. The commands required to save to these files from the **Command Editor** dialog box are described below.

The >savestate command is used to write the current CFD-Post state to a file. The >savestate action supports the following options:

- mode = <none | overwrite>

If mode is none, the executor creates a new state file, and if the specified file exists, an error will be raised. If mode is overwrite, the executor creates a new state file, and if the file exists, it will be deleted and replaced with the latest state information.

- filename = <filename>

Specifies the path and name of the file that the state is to be written to. If no filename is specified, the STATE singleton object will be queried for the filename. If the STATE singleton does not exist, then an error will be raised indicating that a filename must be specified.

18.2.3.1. savestate Command Examples

The following are example >savestate commands, and the expected results. If a STATE singleton exists, the values of the parameters listed after the >savestate command replaces the values stored in the STATE singleton object. For this command, the filename command parameter value replaces the state filename parameter value in the STATE singleton, and the mode command parameter value replaces the savestate mode parameter value in the STATE singleton.

```
>savestate
```

Writes the current state information to the filename specified in the STATE singleton. If the mode in the STATE singleton is none, and the filename exists, an error will be returned. If the mode in the STATE singleton is overwrite, and the filename exists, the existing file will be deleted, and the state information will be written to the file. If the STATE singleton does not exist, an error will be raised indicating that a filename must be specified.

```
>savestate mode=none
```

Writes the current state information to the file specified in the STATE singleton. If the file already exists, an error will be raised. If the STATE singleton does not exist, an error will be raised indicating that a filename must be specified.

```
>savestate mode=overwrite
```

Writes the current state information to the file specified in the STATE singleton. If the file already exists, it will be deleted, and the current state information will be saved in its place. If the STATE singleton does not exist, an error will be raised indicating that a filename must be specified.

```
>savestate filename=mystate.cst
```

Writes the current state information to the `mystate.cst` file. If the STATE singleton exists, and the `savestate` mode is set to `none`, and the file already exists, the command causes an error. If the `savestate` mode is set to `overwrite`, and the file already exists, the file will be deleted, and the current state information will be saved in its place. If the STATE singleton does not exist, then the system assumes a `savestate` mode of `none`, and behave as described above.

```
>savestate mode=none, filename=mystate.cst
```

Writes the current state information to the `mystate.cst` file. If the file already exists, the command causes an error.

```
>savestate mode=overwrite, filename=mystate.cst
```

Writes the current state information to the `mystate.cst` file. If the file already exists, it will be deleted, and the current state information will be saved in its place.

18.2.4. Reading State Files

```
>readstate [mode=<overwrite | append>][filename=<filename>, load=<true | false>]
```

The `>readstate` command loads a CFD-Post state from a specified file.

If a DATA READER singleton has been stored in the state file, the Load action will be invoked to load the contents of the results file.

If a state file contains BOUNDARY objects, and the state file is appended to the current state (with no new DATA READER object), some boundaries defined may not be valid for the loaded results. BOUNDARY objects that are not valid for the currently loaded results file will be culled.

`>readstate` supports the following options:

- `mode = <overwrite | append>`

If mode is set to `overwrite`, the executor deletes all the objects that currently exist in the system, and load the objects saved in the state file. Overwrite mode is the default mode if none is explicitly specified. If mode is set to `append`, the executor adds the objects saved in the state file to the objects that already exist in the system. If the mode is set to `append` and the state file contains objects that already exist in the system, the following logic will determine the final result:

If the system has an equivalent object (the name and type), then the object already in the system will be modified with the parameters saved in the state file. If the system has an equivalent object in name only, then the object that already exists in the system will be deleted, and replaced with that in the state file.

- `filename = <filename>`

The path to the state file.

- `load = <true | false>`

If load is set to `true` and a DATA READER object is defined in the state file, then the results file will be loaded when the state file is read. If load is set to `false`, the results file will not be loaded, and the DATA READER object that currently is in the object database (if any) will not be updated.

18.2.4.1. readstate Option Actions

The following table describes the options, and what will happen based on the combination of options that are selected.

Mode Selection	Load Data Selection	What happens to the objects?	What happens to the Data Reader
Overwrite	True	All user objects (planes, and so on) get deleted. The loading of the new results file changes the default objects (boundaries, wireframe, and so on) including deletion of objects that are no longer relevant to the new results. Default objects that are not explicitly modified by object definitions in the state file will have all user modifiable values reset to default values.	It gets deleted and replaced.
Overwrite	False	All user objects get deleted. All default objects that exist in the state file updates the same objects in the current system state if they exist. Default objects in the state file that do not exist in the current state will not be created. All user objects in the state file will be created.	If it exists, it remains unchanged regardless of what is in the state file.
Append	True	No objects are initially deleted. The default objects in the state file replaces the existing default objects. User objects will: • Be created if they have a unique name. • Replace existing objects if they have the same name but different type. • Update existing objects if they have the same name and type.	It is modified with new value from the state file.
Append	False	No objects are initially deleted. Default objects in the state file will only overwrite those in the system if they already exist. User objects have the same behavior as the Append/True option above.	If it exists, it remains unchanged regardless of what is in the state file.

18.2.4.2. readstate Command Examples

The following are example >readstate commands and their expected results. If a STATE singleton exists, the values of the parameters listed after the >readstate command replace the values stored in the STATE singleton object. For this command, the filename command parameter

value replaces the `state filename` parameter value in the STATE singleton, and the `mode` command parameter value replaces the `readstate mode` parameter value in the STATE singleton.

```
>readstate filename=mystate.cst
```

The `readstate mode` parameter in the STATE singleton determines if the current objects in the system are deleted before the objects defined in the `mystate.cst` file are loaded into the system. If the STATE singleton does not exist, then the system objects are deleted before loading the new state information.

```
>readstate mode=overwrite, filename=mystate.cst
```

Deletes all objects currently in the system, opens the `mystate.cst` file if it exists, and creates the objects as stored in the state file.

```
>readstate mode=append, filename=mystate.cst
```

Opens the `mystate.cst` file, if it exists, and adds the objects defined in the file to those already in the system following the rules specified in the previous table.

```
>readstate
```

Overwrites or appends to the objects in the system using the objects defined in the file referenced by the `state filename` parameter in the STATE singleton. If the STATE singleton does not exist, an error will be raised indicating that a filename must be specified.

```
>readstate mode=overwrite
```

Overwrites the objects in the system STATE using the objects defined in the file referenced by the `state filename` parameter in the STATE singleton. If the STATE singleton does not exist, an error will be raised indicating that a filename must be specified.

```
>readstate mode=append
```

Appends to the objects in the system using the objects defined in the file referenced by the `state filename` parameter in the STATE singleton. If the STATE singleton does not exist, an error will be raised indicating that a filename must be specified.

18.2.5. Solution Monitoring

Typically, you would start solution monitoring after reading a state file, see [Reading State Files \(p. 425\)](#). The following commands are available for solution monitoring:

```
>solmon update = [<true/false>]
```

This command reads solution data up to the last solved iteration into CFD-Post with different behavior depending on whether you specify `true` or `false`.

- `true`: Displays the most recent iteration.
- `false`: Remains at the current iteration.

```
>solmon autoupdate = [<on/off>]
```

This command starts or stops solution monitoring auto update. For more information on solution monitoring control, see [CFD-Post Monitor Menu \(p. 213\)](#).

18.2.6. Creating a Hardcopy

```
>print [<filename>]
```

Creates a file of the current viewer contents. Settings for output format, quality, and so on, are read from the HARDCOPY singleton object.

The optional argument *<filename>* can be used to specify the name of the output file to override that stored in HARDCOPY. HARDCOPY must exist before print is executed.

18.2.7. Importing External File Formats

Data import is controlled using the `>import` command. There are two file types that can be imported: Ansys (*.cdb) and Generic (*.csv). The CCL options associated with the `>import` command are:

```
>import type=<Ansys | Generic>,  
        filename=<filename>,  
        object name=<name of object>,  
        boundary=<associated boundary>,  
        conserve flux=<true | false>
```

type

Indicates whether to import the file as an Ansys file or Generic file.

filename

The name of the file to import.

object name

The name to give the USER SURFACE object that is created as a result of importing the file.

boundary

The name of the CFD-Post boundary/region to associate with the imported Ansys surface. This association is used during an Ansys file import to project data from the Ansys surface onto the CFD-Post boundary/region. The same association is used during an Ansys file export, when data from the CFD-Post boundary/region is projected back onto the Ansys surface.

conserve flux

Boolean to indicate whether or not to ensure that the heat fluxes associated with the imported Ansys geometry remain conservative relative to the fluxes on the associated CFD-Post Boundary.

18.2.8. Exporting Data

Data export is controlled using the `>export` command. The names of variables to export, locations to export, filenames, and so on, are defined in the EXPORT singleton object.

18.2.9. Controlling the Viewer

This section describes how multiple viewports can be accessed using Command Language, and how they are ordered and named.

The first (top-left) viewport is represented by the VIEWER singleton, while others are VIEWPORT objects. For example, to modify filtering in the first viewport, changes should be made to the VIEWER singleton. For all other viewports, changes are made to the VIEWPORT objects, which are numbered from 1-3 in a clockwise direction.

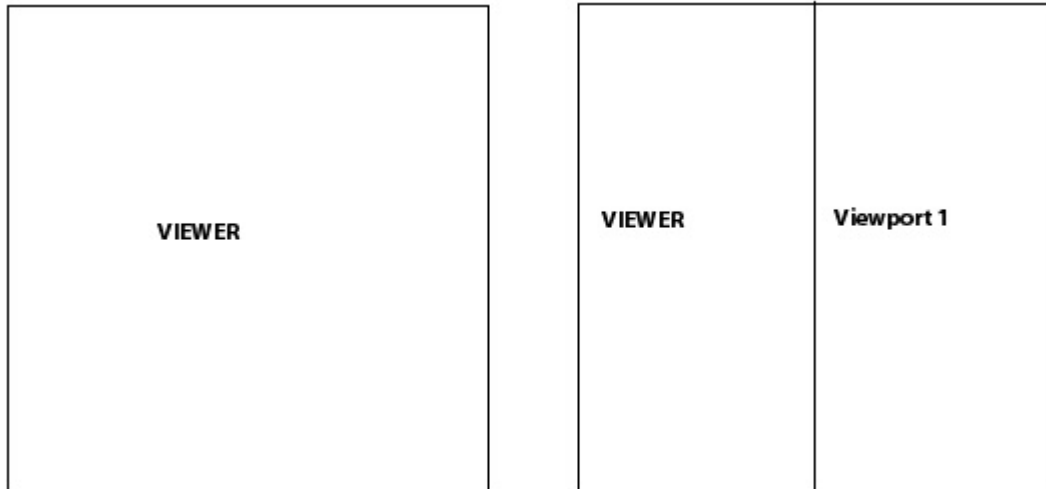
For example, to filter the top-left viewport:

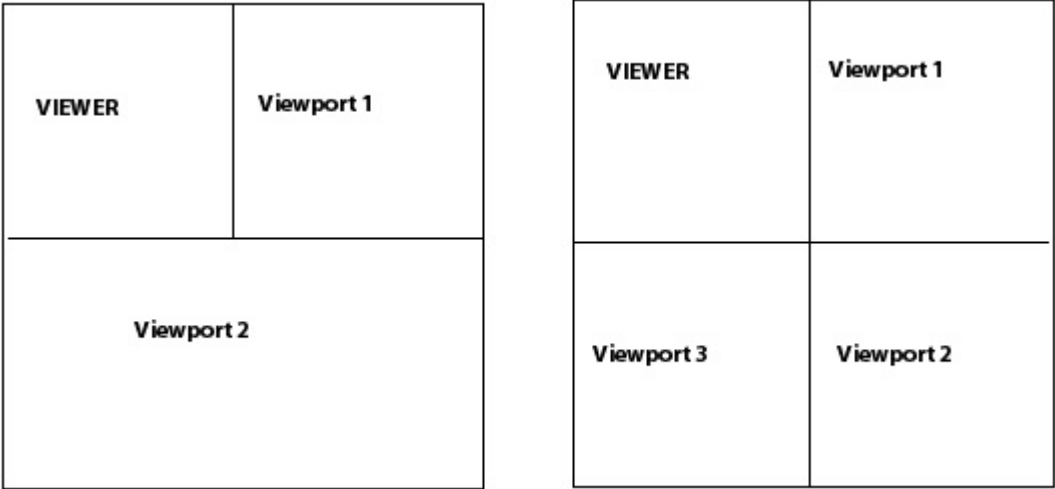
```
VIEWER
  Draw All Objects=false
  Object Name List=Wireframe
END
```

To filter the bottom-right viewport when all four viewports are active:

```
VIEWPORT:Viewport 2
  Draw All Objects=false
  Object Name List=Wireframe
END
```

The following are examples of viewport layouts:





18.3. Quantitative Calculations in the Command Editor Dialog Box

When executing a calculation from the **Command Editor** dialog box, the result is displayed in the **Calculator Window**.

The `>calculate` command is used to perform function calculations in the **Command Editor** dialog box. Typing `>calculate` alone performs the calculation using the parameters stored in the `CALCULATOR` singleton object. Entering `>calculate <function name>` will not work if required arguments are needed by the function.

18.4. Other Commands

The following topics will be discussed:

- [Deleting Objects \(p. 430\)](#)
- [Viewing a Chart \(p. 430\)](#)
- [Turbo Post CCL Command Actions \(p. 431\)](#)

18.4.1. Deleting Objects

```
>delete <objectnamelist>
```

The `>delete` command can be used in the **Command Editor** dialog box to delete objects. The command must be supplied with a list of object names separated by commas. An error message will be displayed if the list contains any invalid object names, but the deletion of valid objects in the list will still be processed.

18.4.2. Viewing a Chart

```
>chart <objectname>
```

Invokes the **Chart Viewer** and displays the specified **Chart** object. **Chart** objects and **Chart Lines** are created like other CCL objects.

18.4.3. Turbo Post CCL Command Actions

18.4.3.1. Calculating Velocity Components

```
>turbo more vars
```

Issuing the `>turbo more vars` command is equivalent to selecting the **Calculate Velocity Components** in the **Turbo** workspace. For details, see [Calculate Velocity Components \(p. 408\)](#).

18.4.3.2. Initializing all Turbo Components

```
>turbo init
```

Issuing the `>turbo init` command is equivalent to selecting **Initialize All Components** from the **Turbo** menu. For details, see [Initialize All Components \(p. 383\)](#).

Chapter 19: Line Interface Mode

This chapter contains information on how to perform typical user actions (loading, printing, and so on), create graphical objects, and perform quantitative calculations when running CFD-Post in Line Interface mode.

All of the functionality of CFD-Post can be accessed when running in *Line Interface* mode. In Line Interface mode, you are simply entering the commands that would otherwise be issued by the user interface. A viewer is provided in a separate window that will show the geometry and the objects that are created on the command line.

To run in Line Interface mode:

- **Windows:** Execute the command `<CFXROOT>\bin\cfdpost -line` (or `<CFDPOSTROOT>\bin\cfdpost -line`) at the Command Prompt (omitting the `-line` option will start the user interface mode).

You might want to change the size of the Command Prompt window to view the output from commands such as `getstate`. This can be done by entering `mode con lines=X` at the command prompt before entering CFD-Post, where X is the number of lines to display in the window. You may choose a large number of lines if you want to be able to see all the output from a session (a scroll bar will appear in the Command Prompt window). Note that once inside CFD-Post, filepaths should contain *forward slashes*.

- **UNIX:** Execute the command `<CFXROOT>/bin/cfdpost -line` (or `<CFDPOSTROOT>/bin/cfdpost -line`) at the command prompt (omitting the `-line` option will start the user interface mode).

In CFD-Post Line Interface mode, all commands are assumed to be actions, the `>` symbol required in the **Command Editor** dialog box is not needed. To call up a list of valid commands, type `help` at the command prompt.

All of the functionality available from the **Command Editor** dialog box in the user interface is available in **Line Interface** mode by typing `enterccl` or `e` at the command prompt. When in `e` mode, you can enter any set of valid CCL commands. The commands are not processed until you leave `e` mode by typing `.e`. You can cancel `e` mode without processing the commands by typing `.c`. For details, see [Command Editor \(p. 377\)](#).

An explanation and list of command actions are available. For details, see [Overview of Command Actions \(p. 421\)](#). (The action commands shown in this link are preceded by a `>` symbol. This should be omitted when entering action commands at the command prompt.)

You can create objects by entering the CCL definition of the object when in `e` mode, or by reading the object definition from a session or state file. For details, see [File Operations from the Command Editor Dialog Box \(p. 422\)](#).

In summary, **Line Interface** mode differs from the **Command Editor** dialog box because **Line Interface** action commands are not preceded by a > symbol. In the same way, when entering lines of CCL or Power Syntax, e must be typed (whereas this is not required in the **Command Editor** dialog box). It should be noted that these are the only principal differences, and all commands that work for the **Command Editor** dialog box will also work in Line Interface mode, providing the correct syntax is used.

19.1. Features Available in Line Interface Mode

The following features are available in line interface mode:

Viewer Hotkeys

The zoom, rotate, pan and other mouse actions available for manipulating the **Viewer** in the user interface perform identical functions in the **Viewer** in **Line Interface** mode. In addition to this, hotkeys can be used to manipulate other aspects of the **Viewer**. For a full list of all the hotkeys available, click in the **Viewer** to make it the active window and select the ? icon. To execute a hotkey command, click once in the **Viewer** (or on the object, as some functions are object-specific) and type the command.

Calculator

When functions are evaluated from the command line, the result is simply printed to standard output.

For a list of valid calculator functions and required parameters, type `calculate help` at the command prompt. Additional information is available; for details, see [Quantitative Calculations in the Command Editor Dialog Box \(p. 430\)](#).

Viewing All Currently Defined Objects (getstate Command)

The list of all currently defined objects can be obtained using the `getstate` command. To get details on a specific object, type `getstate <ObjectName>`.

Viewing a Chart

You can view a chart object in the **Chart Viewer** using the `chart <ChartObjectName>` command.

Repeating CCL Commands

If you want to repeat the most recent CCL command, type: `=`

Executing a UNIX Shell Command

If you want to carry out a UNIX shell command, type % directly before your command. For example, `%ls` will list all the files in your current directory.

Quitting a Command Line Interface Session

To end your CFD-Post command line interface session from the command prompt, enter: `quit`

Example

The following example provides a set of commands that you could enter at the CFX> command prompt. The output written to the screen when executing these commands is not shown.

```
CFX> load filename=c:/MyFiles/StaticMixer.res
CFX> getstate StaticMixer Default
CFX> e
BOUNDARY:StaticMixer Default
Visibility = On
Transparency = 0.5
END
.e
CFX> quit
```

Chapter 20: Fluent Field Variables Listed by Category

By default, CFD-Post does not modify the variable names in the Fluent file. If you want to use all of the embedded CFD-Post macros and calculation options, you need to convert variable names to CFX variable names. You can convert variable names to CFX variable names upon reading the file by first selecting **Edit > Options**, then, in the **Options** dialog box:

- For legacy Fluent files: selecting **CFD-Post > Files > Variables > Non CFX Files > Translate variable names to CFX-Solver style names**.
- For Common Fluids Format (CFF) files: setting **CFD-Post > Files > Variables > Common Fluids Format > Name Convention** to **CFD-Post**.

For details on the settings above, see [Variables \(p. 207\)](#).

Note:

Some variable names might not be translated.

Translation is carried out according to the tables that follow, which list the Fluent field variables and give the equivalent Ansys CFX variables where applicable. The variables are defined in [Alphabetical Listing of Field Variables and Their Definitions in the *Fluent User's Guide*](#).

The following restrictions apply to marked variables:

<i>2d</i>	available only for 2D flows
<i>2da</i>	available only for 2D axisymmetric flows (with or without swirl)
<i>2dasw</i>	available only for 2D axisymmetric swirl flows
<i>3d</i>	available only for 3D flows
<i>bns</i>	available only for broadband noise source models
<i>bnv</i>	node values available at boundaries
<i>cpl</i>	available only in the density-based solvers
<i>cv</i>	available only for cell values (Node Values option turned off)
<i>des</i>	available only when the DES turbulence model is used
<i>dil</i>	not available with full multicomponent diffusion
<i>do</i>	available only when the discrete ordinates radiation model is used
<i>dpm</i>	available only for coupled discrete phase calculations
<i>dtrm</i>	available only when the discrete transfer radiation model is used
<i>fwh</i>	available only with the Ffowcs Williams and Hawkings acoustics model
<i>e</i>	available only for energy calculations

<i>edc</i>	available only with the EDC model for turbulence-chemistry interaction
<i>emm</i>	available also when the Eulerian multiphase model is used
<i>ewt</i>	available only with the enhanced wall treatment
<i>gran</i>	available only if a granular phase is present
<i>h2o</i>	available only when the mixture contains water
<i>id</i>	available only when the ideal gas law is enabled for density
<i>ke</i>	available only when one of the k-epsilon turbulence models is used
<i>kw</i>	available only when one of the k-omega turbulence models is used
<i>les</i>	available only when the LES turbulence model is used
<i>melt</i>	available only when the melting and solidification model is used
<i>mix</i>	available only when the multiphase mixture model is used
<i>mp</i>	available only for multiphase models
<i>nox</i>	available only for NOx calculations
<i>np</i>	not available in parallel solvers
<i>nv</i>	uses explicit node value function
<i>p</i>	available only in parallel solvers
<i>p1</i>	available only when the P-1 radiation model is used
<i>pdf</i>	available only for non-premixed combustion calculations
<i>pmx</i>	available only for premixed combustion calculations
<i>ppmx</i>	available only for partially premixed combustion calculations
<i>r</i>	available only when the Rosseland radiation model is used
<i>rad</i>	available only for radiation heat transfer calculations
<i>rc</i>	available only for finite-rate reactions
<i>rsm</i>	available only when the Reynolds stress turbulence model is used
<i>s2s</i>	available only when the surface-to-surface radiation model is used
<i>sa</i>	available only when the Spalart-Allmaras turbulence model is used
<i>seg</i>	available only in the pressure-based solver
<i>sp</i>	available only for species calculations
<i>sr</i>	available only for surface reactions
<i>sol</i>	available only when the solar model is used
<i>soot</i>	available only for soot calculations
<i>stat</i>	available only with data sampling for unsteady statistics
<i>stcm</i>	available only for stiff chemistry calculations
<i>t</i>	available only for turbulent flows
<i>turbo</i>	available only when a turbomachinery topology has been defined
<i>udm</i>	available only when a user-defined memory is used
<i>uds</i>	available only when a user-defined scalar is used

v	available only for viscous flows

Table 20.1: Pressure and Density Categories

Category	Fluent Variable	CFX Variable
Pressure...	Static Pressure (<i>bnv</i>)	Pressure
	Pressure Coefficient	Pressure Coefficient
	Dynamic Pressure	Dynamic Pressure
	Absolute Pressure (<i>bnv</i>)	Absolute Pressure
	Total Pressure (<i>bnv</i>)	Total Pressure
	Relative Total Pressure	Relative Total Pressure
Density...	Density	Density
	Density All	

Table 20.2: Velocity Category

Category	Fluent Variable	CFX Variable
Velocity...	Velocity Magnitude (<i>bnv</i>)	Velocity
	Relative Velocity Magnitude (<i>bnv</i>)	
	X Velocity (<i>bnv</i>)	Velocity u
	Relative X Velocity (<i>bnv</i>)	
	Y Velocity (<i>bnv</i>)	Velocity v
	Relative Y Velocity (<i>bnv</i>)	
	Z Velocity (<i>3d, bnv</i>)	Velocity w
	Relative Z Velocity (<i>bnv</i>)	
	Swirl Velocity (<i>2dasw, bnv</i>)	Velocity Circumferential
	Tangential Velocity	
	Axial Velocity (<i>2da or 3d</i>)	Velocity Axial
	Radial Velocity	Velocity Radial
	Stream Function (<i>2d</i>)	Stream Function
	Tangential Velocity	Velocity Circumferential
	Mach Number (<i>id</i>)	Mach Number
	Relative Velocity Magnitude	Velocity Magnitude
	Relative Axial Velocity (<i>2da</i>)	Velocity Axial
	Relative Radial Velocity (<i>2da</i>)	Velocity Radial
	Relative Swirl Velocity (<i>2dasw, bnv</i>)	Velocity Circumferential
	Relative Tangential Velocity	
	Relative Mach Number (<i>id</i>)	Mach Number

Category	Fluent Variable	CFX Variable
	Mach Number	
	Mesh X-Velocity (<i>nv</i>)	Mesh Velocity u
	Mesh Y-Velocity (<i>nv</i>)	Mesh Velocity v
	Mesh Z-Velocity (<i>3d, nv</i>)	Mesh Velocity w
	Velocity Angle	Velocity Angle
	Relative Velocity Angle	
	Vorticity Magnitude (<i>v</i>)	Vorticity
	Helicity	Helicity
	X-Vorticity (<i>v, 3d</i>)	Vorticity in Stn Frame X
	Y-Vorticity (<i>v, 3d</i>)	Vorticity in Stn Frame Y
	Z-Vorticity (<i>v, 3d</i>)	Vorticity in Stn Frame Z
	Preconditioning Reference Velocity (<i>cpf</i>)	Reference Velocity (Preconditioning)

Table 20.3: Temperature, Radiation, and Solidification/Melting Categories

Category	Fluent Variable	CFX Variable
Temperature...	Static Temperature (<i>e, bnv, nv</i>)	Temperature
	Total Temperature (<i>e, nv</i>)	Total Temperature
	Enthalpy (<i>e, nv</i>)	Static Enthalpy
	Relative Total Temperature (<i>e</i>)	Total Temperature
	Rothalpy (<i>e, nv</i>)	Rothalpy
	Fine Scale Temperature (<i>edc, nv, e</i>)	Fine Scale Temperature
	Wall Temperature (Outer Surface) (<i>e, v</i>)	Wall Temperature Outer Surface
	Wall Temperature (Inner Surface) (<i>e, v</i>)	Wall Temperature Inner Surface
	Inner Wall Temperature	Inner Wall Temperature
	Total Enthalpy (<i>e</i>)	Total Enthalpy
	Total Enthalpy Deviation (<i>e</i>)	Total Enthalpy Deviation
	Entropy (<i>e</i>)	Static Entropy
	Total Energy (<i>e</i>)	Total Energy ^[a]
	Internal Energy (<i>e</i>)	Internal Energy
Radiation...	Absorption Coefficient (<i>r, p1, do, or dtrm</i>)	Absorption Coefficient
	Scattering Coefficient (<i>r, p1, or do</i>)	Scattering Coefficient
	Refractive Index (<i>do</i>)	Refractive Index
	Radiation Temperature (<i>p1 or do</i>)	Radiation Temperature
	Incident Radiation (<i>p1 or do</i>)	Incident Radiation
	Incident Radiation (Band n) (<i>do</i> (non-gray))	<Band n>.Incident Radiation
	Surface Cluster ID (<i>s2s</i>)	Surface Cluster ID
Solidification/ Melting...	Liquid Fraction (<i>melt</i>)	Mass Fraction

Category	Fluent Variable	CFX Variable
	Contact Resistivity (<i>melt</i>)	Contact Resistivity
	X Pull Velocity (<i>melt</i> (if calculated))	Pull Velocity $u^{[a]}$
	Y Pull Velocity (<i>melt</i> (if calculated))	Pull Velocity $v^{[a]}$
	Z Pull Velocity (<i>melt</i> (if calculated), <i>3d</i>)	Pull Velocity $w^{[a]}$
	Axial Pull Velocity (<i>melt</i> (if calculated), <i>2da</i>)	Pull Velocity Axial ^[a]
	Radial Pull Velocity (<i>melt</i> (if calculated), <i>2da</i>)	Pull Velocity Radial ^[a]
	Swirl Pull Velocity (<i>melt</i> (if calculated), <i>2dasw</i>)	Pull Velocity Circumferential ^[a]

[a] CFD-Post naming convention

Table 20.4: Turbulence Category

Category	Fluent Variable	CFX Variable
Turbulence...	Turbulent Kinetic Energy (k) (<i>ke</i> , <i>kw</i> , or <i>rsm</i> ; <i>bnv</i> , <i>nv</i> , or <i>emm</i>)	Turbulence Kinetic Energy
	Turbulent Kinetic Energy	
	UU Reynolds Stress (<i>rsm</i> ; <i>emm</i>)	Reynolds Stress uu
	VV Reynolds Stress (<i>rsm</i> ; <i>emm</i>)	Reynolds Stress vv
	WW Reynolds Stress (<i>rsm</i> ; <i>emm</i>)	Reynolds Stress ww
	UV Reynolds Stress (<i>rsm</i> ; <i>emm</i>)	Reynolds Stress uv
	UW Reynolds Stress (<i>rsm</i> , <i>3d</i> ; <i>emm</i>)	Reynolds Stress uw
	VW Reynolds Stress (<i>rsm</i> , <i>3d</i> ; <i>emm</i>)	Reynolds Stress vw
	Turbulence Intensity (<i>ke</i> , <i>kw</i> , or <i>rsm</i>)	Turbulence Intensity
	Turbulent Dissipation Rate (Epsilon) (<i>ke</i> or <i>rsm</i> ; <i>bnv</i> , <i>nv</i> , or <i>emm</i>)	Turbulence Eddy Dissipation
	Turbulent Dissipation Rate	
	Specific Dissipation Rate (Omega) (<i>kw</i>)	Turbulence Eddy Frequency
	Specific Dissipation Rate	
	Production of k (<i>ke</i> , <i>kw</i> , or <i>rsm</i> ; <i>emm</i>)	k Production ^[a]
	Modified Turbulent Viscosity (<i>sa</i>)	Eddy Viscosity (modified)
	Turbulent Viscosity (<i>sa</i> , <i>ke</i> , <i>kw</i> , <i>rsm</i> , or <i>des</i>)	Eddy Viscosity
	Effective Viscosity (<i>sa</i> , <i>ke</i> , <i>kw</i> , <i>rsm</i> , or <i>des</i> ; <i>emm</i>)	Effective Viscosity
	Turbulent Viscosity Ratio (<i>ke</i> , <i>kw</i> , <i>rsm</i> , <i>sa</i> , or <i>des</i> ; <i>emm</i>)	Eddy Viscosity Ratio
	Subgrid Kinetic Energy (<i>les</i>)	Kinetic Energy (subgrid)
	Subgrid Turbulent Viscosity (<i>les</i>)	Eddy Viscosity (subgrid)
	Subgrid Turbulent Viscosity Ratio (<i>les</i>)	Eddy Viscosity Ratio (subgrid)

Category	Fluent Variable	CFX Variable
	Effective Thermal Conductivity (<i>t, e</i>)	Effective Thermal Conductivity
	Effective Prandtl Number (<i>t, e</i>)	Effective Prandtl Number
	Wall Ystar (<i>ke, kw, or rsm</i>)	Ystar
	Wall Yplus (<i>t</i>)	Yplus
	Turbulent Reynolds Number (<i>Re_y</i>) (<i>ke</i> or <i>rsm</i> ; <i>ewt</i>)	Turbulent Reynolds Number
	Relative Length Scale (DES) (<i>des</i>)	Relative Length Scale (DES)

Table 20.5: Species, Reactions, Pdf, and Premixed Combustion Categories

Category	Fluent Variable	CFX Variable
Species...	Mass fraction of species-n (<i>sp, pdf, or ppmx; nv</i>)	<Species-n>.Mass Fraction
	Mole fraction of species-n (<i>sp, pdf, or ppmx</i>)	<Species-n>.Mole Fraction
	Molar Concentration of species-n (<i>sp, pdf, or ppmx</i>)	<Species-n>.Molar Concentration
	Lam Diff Coef of species-n (<i>sp, dil</i>)	<Species-n>.Laminar Diffusion Coefficient
	Eff Diff Coef of species-n (<i>t, sp, dil</i>)	<Species-n>.Effective Diffusion Diffusivity ^[a]
	Thermal Diff Coef of species-n (<i>sp</i>)	<Species-n>.Thermal Diffusion Coefficient
	Enthalpy of species-n (<i>sp</i>)	<Species-n>.Static Enthalpy
	species-n Source Term (<i>rc, cpl</i>)	<Species-n>.Source Term ^[a]
	Surface Deposition Rate of species-n (<i>sr</i>)	<Species-n>.Surface Deposition Rate
	Surface Coverage of species-n (<i>sr</i>)	<Species-n>.Surface Coverage ^[a]
	Relative Humidity (<i>sp, pdf, or ppmx; h2o</i>)	Relative Humidity
	Time Step Scale (<i>sp, stcm</i>)	Time Step Scale
	Fine Scale Mass fraction of species-n (<i>edc</i>)	<Species-n>.Fine Scale Mass Fraction
	Fine Scale Transfer Rate (<i>edc</i>)	Fine Scale Transfer Rate
	1-Fine Scale Volume Fraction (<i>edc</i>)	1-Fine Scale Volume Fraction
Reactions...	Rate of Reaction-n (<i>rc</i>)	<Reaction-n>.Molar Reaction Rate
	Net Reaction Rate of Species-n	<Species-n>.Net Molar Reaction Rate
	Kinetic Rate of Reaction-n	<Reaction-n>.Kinetic Rate of Reaction
	Kinetic Rate of Reaction-n(Porous)	<Reaction-n>.Kinetic Rate of Reaction(Porous)
	Arrhenius Rate of Reaction-n (<i>rc</i>)	<Reaction-n>.Arrhenius Rate of Reaction ^[a]
	Turbulent Rate of Reaction-n (<i>rc, t</i>)	<Reaction-n>.Turbulent Rate of Reaction ^[a]
	Heat of Reaction	Heat of Reaction
PDF...	Mean Mixture Fraction (<i>pdf</i> or <i>ppmx; nv</i>)	Mixture Fraction
	Secondary Mean Mixture Fraction (<i>pdf</i> or <i>ppmx; nv</i>)	Secondary Mixture Fraction ^[a]

Category	Fluent Variable	CFX Variable
	Mixture Fraction Variance (<i>pdf</i> or <i>ppmx</i> ; <i>nv</i>)	Mixture Fraction Variance
	Secondary Mixture Fraction Variance (<i>pdf</i> or <i>ppmx</i> ; <i>nv</i>)	Secondary Mixture Fraction Variance ^[a]
	Fvar Prod (<i>pdf</i> or <i>ppmx</i>)	Fvar Prod
	Scalar Dissipation (<i>pdf</i> or <i>ppmx</i>)	Scalar Dissipation
	PDF Table Adiabatic Enthalpy	Adiabatic Enthalpy (PDF Table)
	PDF Table Heat Loss/Gain	Heat Loss/Gain}, (PDF Table)
Premixed Combustion...	Progress Variable (<i>pmx</i> or <i>ppmx</i> ; <i>nv</i>)	Reaction Progress
	Damkohler Number (<i>pmx</i> or <i>ppmx</i>)	Damkohler Number ^[a]
	Stretch Factor (<i>pmx</i> or <i>ppmx</i>)	Stretch Factor ^[a]
	Turbulent Flame Speed (<i>pmx</i> or <i>ppmx</i>)	Turbulent Flame Speed ^[a]
	Static Temperature (<i>pmx</i> or <i>ppmx</i>)	Temperature
	Product Formation Rate (<i>pmx</i> or <i>ppmx</i>)	Product Formation Rate ^[a]
	Laminar Flame Speed (<i>pmx</i> or <i>ppmx</i>)	Laminar Flame Speed ^[a]
	Critical Strain Rate (<i>pmx</i> or <i>ppmx</i>)	Critical Strain Rate ^[a]
	Adiabatic Flame Temperature (<i>pmx</i> or <i>ppmx</i>)	Adiabatic Flame Temperature ^[a]
	Unburnt Fuel Mass Fraction (<i>pmx</i> or <i>ppmx</i>)	Unburnt Fuel Mass Fraction ^[a]

Table 20.6: NOx, Soot, and Unsteady Statistics Categories

Category	Fluent Variable	CFX Variable
NOx...	Mass fraction of NO (<i>nox</i>)	No.Mass Fraction
	Mass fraction of HCN (<i>nox</i>)	Hcn.Mass Fraction
	Mass fraction of NH3 (<i>nox</i>)	Nh3.Mass Fraction
	Mass fraction of N2O (<i>nox</i>)	N2o.Mass Fraction
	Mole fraction of NO (<i>nox</i>)	No.Molar Fraction
	Mole fraction of HCN (<i>nox</i>)	Hcn.Molar Fraction
	Mole fraction of NH3 (<i>nox</i>)	Nh3.Molar Fraction
	Mole fraction of N2O (<i>nox</i>)	N2o.Molar Fraction
	NO Density (<i>nox</i>)	No.Density
	HCN Density (<i>nox</i>)	Hcn.Density
	NH3 Density (<i>nox</i>)	Nh3.Density
	N2O Density (<i>nox</i>)	N2o.Density
	Variance of Temperature (<i>nox</i>)	Variance of Temperature
	Variance of Species-n (<i>nox</i>)	<Species-n>.Variance ^[a]
	Rate of NO	No.Source
	Rate of HCN	Hcn.Source
	Rate of NH3	Nh3.Source

Category	Fluent Variable	CFX Variable
	Rate of N2O (<i>nox</i>)	N2o Source ^[a]
	Rate of Thermal NO (<i>nox</i>)	Thermal No.Molar Reaction Rate
	Rate of Prompt NO (<i>nox</i>)	Prompt No.Molar Reaction Rate ^[a]
	Rate of Fuel NO (<i>nox</i>)	Fuel No.Molar Reaction Rate ^[a]
	Rate of N2OPath NO (<i>nox</i>)	N2oPath.Molar Reaction Rate ^[a]
	Rate of Reburn NO (<i>nox</i>)	Reburn No.Molar Reaction Rate ^[a]
	Rate of SNCR NO (<i>nox</i>)	SNCR No.Molar Reaction Rate ^[a]
	Rate of USER NO (<i>nox</i>)	User No.Molar Reaction Rate ^[a]
Soot...	Mass fraction of soot (<i>soot</i>)	Soot Mass Fraction
	Mass fraction of Nuclei (<i>soot</i>)	Soot Nuclei Specific Concentration
	Mole fraction of soot (<i>soot</i>)	Soot Molar Fraction ^[a]
	Soot Density (<i>soot</i>)	Soot.Density
	Rate of Soot (<i>soot</i>)	Soot Mass Source ^[a]
	Rate of Nuclei (<i>soot</i>)	Soot Nuclei Source ^[a]
	Heterogeneous Reaction Rate n	Heterogeneous Reaction Rate n
Unsteady Statistics...	Mean quantity-n (<i>stat</i>)	<variable>.Trnavg
	RMS quantity-n (<i>stat</i>)	<variable>.Trnrms

Table 20.7: Phases, Discrete Phase Model, Granular Pressure, and Granular Temperature Categories

Category	Fluent Variable	CFX Variable
Phases...	Volume fraction (<i>mp</i>)	<phase>.Volume Fraction
Discrete Phase Model...	DPM Mass Source (<i>dpm</i>)	<particle>.Particle Mass Source
	DPM Erosion (<i>dpm, cv</i>)	<particle>.Particle Erosion Rate Density ^[a]
	DPM Accretion (<i>dpm, cv</i>)	<particle>.Particle Wall Mass Flow Density ^[a]
	DPM X Momentum Source (<i>dpm</i>)	<particle>.Particle Momentum Source X
	DPM Y Momentum Source (<i>dpm</i>)	<particle>.Particle Momentum Source Y
	DPM Z Momentum Source (<i>dpm, 3d</i>)	<particle>.Particle Momentum Source Z
	DPM Swirl Momentum Source (<i>dpm, 2dasw</i>)	<particle>.Particle Swirl Momentum Source
	DPM Sensible Enthalpy Source (<i>dpm, e</i>)	<particle>.Particle Sensible Enthalpy Source
	DPM Enthalpy Source (<i>dpm, e</i>)	<particle>.Particle Energy Source
	DPM Absorption Coefficient (<i>dpm, rad</i>)	<particle>.Particle Absorption Coefficient
	DPM Emission (<i>dpm, rad</i>)	<particle>.Particle Radiative Emission
	DPM Scattering (<i>dpm, rad</i>)	<particle>.Particle Radiative Scattering ^[a]
	DPM Burnout (<i>dpm, sp, e</i>)	Particle Burnout
	DPM Evaporation/Devolatilization (<i>dpm, sp, e</i>)	Particle Evaporation-Devolatilization
	DPM Concentration (<i>dpm</i>)	<particle>.Volume Fraction
	DPM Mass Source	Particle Mass Source

Category	Fluent Variable	CFX Variable
	DPM Erosion	Particle Erosion Rate Density
	DPM Accretion	Particle Wall Mass Flow Density
	DPM X Momentum Source	Particle Momentum Source X
	DPM Y Momentum Source	Particle Momentum Source Y
	DPM Z Momentum Source	Particle Momentum Source Z
	DPM Swirl Momentum Source	Particle Swirl Momentum Source
	DPM Sensible Enthalpy Source	Particle Sensible Enthalpy Source
	DPM Enthalpy Source	Particle Enthalpy Source
	DPM Absorption Coefficient	Particle Absorption Coefficient
	DPM Emission	Particle Radiative Emission
	DPM Scattering	Particle Radiative Scattering
	DPM Burnout	Particle Burnout
	DPM Evaporation/Devolatilization	Particle Evaporation-Devolatilization
	DPM Concentration	Particle Mass Concentration
	DPM Turbulent Kinetic Energy Source	Particle Turbulent Kinetic Energy Source
	DPM Turbulent Dissipation Source	Particle Turbulent Dissipation Source
	DPM Species-n Concentration	<Species-n>.Particle Mass Concentration
	DPM Species-n Source (<i>dpm, sp, e</i>)	<Species-n>.Particle Mass Source
Granular Pressure...	Granular Pressure (<i>emm, gran</i>)	<phase>.Granular Pressure ^[a]
Granular Temperature...	Granular Temperature (<i>emm, gran</i>)	<phase>.Granular Temperature

Table 20.8: Properties, Wall Fluxes, User Defined Scalars, and User Defined Memory Categories

Category	Fluent Variable	CFX Variable
Properties...	Molecular Viscosity (<i>v</i>)	Dynamic Viscosity
	Diameter(<i>mix, emm</i>)	Mean Particle Diameter
	Granular Conductivity (<i>mix, emm, gran</i>)	Granular Conductivity ^[a]
	Thermal Conductivity (<i>e, v</i>)	Thermal Conductivity
	Specific Heat (Cp) (<i>e</i>)	Specific Heat Capacity at Constant Pressure
	Specific Heat Ratio (gamma) (<i>id</i>)	Specific Heat Ratio ^[a]
	Gas Constant (R) (<i>id</i>)	R Gas Constant
	Molecular Prandtl Number (<i>e, v</i>)	Prandtl Number ^[a]
	Mean Molecular Weight (<i>seg, pdf</i>)	Molar Mass ^[a]
	Compressibility Factor	Compressibility Factor
	Reduced Temperature	Reduced Temperature
	Reduced Pressure	Reduced Pressure
	Critical Temperature	Critical Temperature

Category	Fluent Variable	CFX Variable
	Critical Pressure	Critical Pressure
	Acentric Factor	Acentric Factor
	Critical Specific Volume	Critical Specific Volume
	Sound Speed (<i>id</i>)	Local Speed of Sound ^[a]
Wall Fluxes...	Wall Shear Stress (<i>v, cv, emm</i>)	Wall Shear
	X-Wall Shear Stress (<i>v, cv, emm</i>)	Wall Shear X
	Y-Wall Shear Stress (<i>v, cv, emm</i>)	Wall Shear Y
	Z-Wall Shear Stress (<i>v, 3d, cv, emm</i>)	Wall Shear Z
	Axial-Wall Shear Stress (<i>2da, cv</i>)	Wall Shear Axial
	Radial-Wall Shear Stress (<i>2da, cv</i>)	Wall Shear Radial
	Swirl-Wall Shear Stress (<i>2dasw, cv</i>)	Wall Shear Circumferential
	Skin Friction Coefficient (<i>v, cv, emm</i>)	Skin Friction Coefficient
	Total Surface Heat Flux (<i>e, v, cv</i>)	Wall Heat Flux
	Radiation Heat Flux (<i>rad, cv</i>)	Wall Radiative Heat Flux
	Solar Heat Flux (<i>sol, cv</i>)	Solar Heat Flux
	Absorbed Radiation Flux (Band-n) (<i>do, cv</i>)	Absorbed Radiation Flux (Band-n)
	Absorbed Visible Solar Flux (<i>sol, cv</i>)	Absorbed Visible Solar Flux
	Absorbed IR Solar Flux (<i>sol, cv</i>)	Absorbed IR Solar Flux
	Reflected Radiation Flux (Band-n) (<i>do, cv</i>)	Reflected Radiation Flux
	Reflected Visible Solar Flux (<i>sol, cv</i>)	Reflected Visible Solar Flux
	Reflected IR Solar Flux (<i>sol, cv</i>)	Reflected IR Solar Flux
	Transmitted Visible Solar Flux (<i>sol, cv</i>)	Transmitted Visible Solar Flux
	Transmitted IR Solar Flux (<i>sol, cv</i>)	Transmitted IR Solar Flux
	Beam Irradiation Flux (Band-n) (<i>do, cv</i>)	<Band-n>.Beam Irradiation Flux
	Surface Incident Radiation (<i>do, dtrm, or s2s; cv</i>)	Surface Incident Radiation
	Surface Heat Transfer Coef. (<i>e, v, cv</i>)	Surface Heat Transfer Coefficient
	Wall Func. Heat Tran. Coef. (<i>e, v, cv</i>)	Wall Heat Transfer Coefficient
	Surface Nusselt Number (<i>e, v, cv</i>)	Surface Nusselt Number
	Surface Stanton Number (<i>e, v, cv</i>)	Surface Stanton Number
User-Defined Scalars...	Scalar-n (<i>uds</i>)	<Scalar-n>
	Diffusion Coef. of Scalar-n (<i>uds</i>)	<Scalar-n>.Diffusion Coefficient
User-Defined Memory...	User Memory n (<i>udm</i>)	User Defined Memory <n>

Table 20.9: Cell Info, Grid, and Adaption Categories

Category	Fluent Variable	CFX Variable
Cell Info...	Cell Partition (<i>np</i>)	Cell Partition

Category	Fluent Variable	CFX Variable
	Active Cell Partition (p)	Active Cell Partition
	Stored Cell Partition (p)	Stored Cell Partition
	Cell Id (p)	Cell Id
	Cell Element Type	Cell Element Type
	Cell Zone Type	Cell Zone Type
	Cell Zone Index	Cell Zone Index
	Partition Neighbors	Partition Neighbors
Grid...	X-Coordinate (nv)	X
	Y-Coordinate (nv)	Y
	Z-Coordinate ($3d, nv$)	Z
	Axial Coordinate (nv)	Axial Coordinate
	Angular Coordinate ($3d, nv$)	Angular Coordinate
	Abs. Angular Coordinate ($3d, nv$)	Absolute Angular Coordinate
	Radial Coordinate (nv)	Radial Angular Coordinate
	Face Area Magnitude	Face Area Magnitude
	X Face Area	Face Area X
	Y Face Area	Face Area Y
	Z Face Area ($3d$)	Face Area Z
	Cell Equiangle Skew	Cell Equiangle Skew
	Cell Equivolume Skew	Cell Equivolume Skew
	Cell Volume	Cell Volume
	2D Cell Volume ($2da$)	2d Cell Volume
	Cell Wall Distance	Cell Wall Distance
	Face Handedness	Face Handedness
	Face Squish Index	Face Squish Index
	Cell Squish Index	Cell Squish Index

Table 20.10: Grid Category (Turbomachinery-Specific Variables) and Adaption Category

Category	Fluent Variable	CFX Variable
Grid...	Meridional Coordinate ($nv, turbo$)	Meridional Coordinate
	Abs Meridional Coordinate ($nv, turbo$)	Abs Meridional Coordinate
	Spanwise Coordinate ($nv, turbo$)	Spanwise Coordinate
	Abs (H-C) Spanwise Coordinate ($nv, turbo$)	Abs (H-C) Spanwise Coordinate
	Abs (C-H) Spanwise Coordinate ($nv, turbo$)	Abs (C-H) Spanwise Coordinate
	Pitchwise Coordinate ($nv, turbo$)	Pitchwise Coordinate
	Abs Pitchwise Coordinate ($nv, turbo$)	Abs Pitchwise Coordinate
Adaption...	Adaption Function	Adaption Function
	Adaption Curvature	Adaption Curvature

Category	Fluent Variable	CFX Variable
	Adaption Space Gradient	Adaption Space Gradient
	Adaption Iso-Value	Adaption Iso-Value
	Existing Value	Existing Value
	Boundary Cell Distance	Boundary Cell Distance
	Boundary Normal Distance	Boundary Normal Distance
	Boundary Volume Distance (<i>np</i>)	Boundary Volume Distance
	Cell Volume Change	Cell Volume Change
	Cell Surface Area	Cell Surface Area
	Cell Warpage	Cell Warpage
	Cell Children	Cell Children
	Cell Refine Level	Cell Refine Level

Table 20.11: Residuals Category

Category	Fluent Variable	CFX Variable
Residuals...	Mass Imbalance (<i>seg</i>)	Mass Imbalance
	Pressure Residual (<i>cpl</i>)	Residual Pressure
	X-Velocity Residual (<i>cpl</i>)	Residual u Velocity
	Y-Velocity Residual (<i>cpl</i>)	Residual v Velocity
	Z-Velocity Residual (<i>cpl, 3d</i>)	Residual w Velocity
	Axial-Velocity Residual (<i>cpl, 2da</i>)	Residual Velocity Axial
	Radial-Velocity Residual (<i>cpl, 2da</i>)	Residual Velocity Radial
	Swirl-Velocity Residual (<i>cpl, 2dasw</i>)	Residual Velocity Circumferential
	Temperature Residual (<i>cpl, e</i>)	Residual Temperature
	Species-n Residual (<i>cpl, sp</i>)	<Species-n>.Residual
	Time Step (<i>cpl</i>)	Time Step
	Pressure Correction (<i>cpl</i>)	Pressure Correction
	X-Velocity Correction (<i>cpl</i>)	Velocity Correction u
	Y-Velocity Correction (<i>cpl</i>)	Velocity Correction v
	Z-Velocity Correction (<i>cpl, 3d</i>)	Velocity Correction w
	Axial-Velocity Correction (<i>cpl, 2da</i>)	Velocity Correction Axial
	Radial-Velocity Correction (<i>cpl, 2da</i>)	Velocity Correction Radial
	Swirl-Velocity Correction (<i>cpl, 2dasw</i>)	Velocity Correction Circumferential
	Temperature Correction (<i>cpl, e</i>)	Temperature Correction
	Species-n Correction (<i>cpl, sp</i>)	<Species-n>.Correction

Table 20.12: Derivatives Category

Category	Fluent Variable	CFX Variable
Derivatives...	Strain Rate (<i>v</i>)	Strain Rate

Category	Fluent Variable	CFX Variable
	dX-Velocity/dx	du-Velocity-dx
	dY-Velocity/dx	dv-Velocity-dx
	dZ-Velocity/dx (3d)	dw-Velocity-dx
	dAxial-Velocity/dx (2da)	dAxial-Velocity-dx
	dRadial-Velocity/dx (2da)	dRadial-Velocity-dx
	dSwirl-Velocity/dx (2dasw)	dCircumferential-Velocity-dx
	d species-n/dx (cpl, sp)	d<Species-n>-dx
	dX-Velocity/dy	du-Velocity-dy
	dY-Velocity/dy	dv-Velocity-dy
	dZ-Velocity/dy (3d)	dw-Velocity-dy
	dAxial-Velocity/dy (2da)	dAxial-Velocity-dy
	dRadial-Velocity/dy (2da)	dRadial-Velocity-dy
	dSwirl-Velocity/dy (2dasw)	dCircumferential-Velocity-dy
	d species-n/dy (cpl, sp)	d<Species-n>-dy
	dX-Velocity/dz (3d)	du-Velocity-dz
	dY-Velocity/dz (3d)	dv-Velocity-dz
	dZ-Velocity/dz (3d)	dw-Velocity-dz
	d species-n/dz (cpl, sp, 3d)	d<Species-n>-dz
	dOmega/dx (2dasw)	dOmega-dx
	dOmega/dy (2dasw)	dOmega-dy
	dT/dx	dT/dx
	dT/dy	dT/dy
	dT/dz	dT/dz
	dp-dX (seg)	dp-dX
	dp-dY (seg)	dp-dY
	dp-dZ (seg, 3d)	dp-dZ

Table 20.13: Acoustics Category

Category	Fluent Variable	CFX Variable
Acoustics...	Surface dpdt RMS (fwh)	Surface dpdt RMS
	Acoustic Power Level (dB) (bns)	Acoustic Power Level (dB)
	Acoustic Power (bns)	Acoustic Power
	Jet Acoustic Power Level (dB) (bns, 2da)	Jet Acoustic Power Level (dB)
	Jet Acoustic Power (bns, 2da)	Jet Acoustic Power
	Surface Acoustic Power Level (dB) (bns)	
	Surface Acoustic Power (bns)	
	Lilley's Self-Noise Source (bns)	Lilley's Self-Noise Source
	Lilley's Shear-Noise Source (bns)	Lilley's Shear-Noise Source

Category	Fluent Variable	CFX Variable
	Lilley's Total Noise Source (<i>bns</i>)	Lilley's Total Noise Source
	LEE Self-Noise X-Source (<i>bns</i>)	LEE Self-Noise X-Source
	LEE Shear-Noise X-Source (<i>bns</i>)	LEE Shear-Noise X-Source
	LEE Total Noise X-Source (<i>bns</i>)	LEE Total Noise X-Source
	LEE Self-Noise Y-Source (<i>bns</i>)	LEE Self-Noise Y-Source
	LEE Shear-Noise Y-Source (<i>bns</i>)	LEE Shear-Noise Y-Source
	LEE Total Noise Y-Source (<i>bns</i>)	LEE Total Noise Y-Source
	LEE Self-Noise Z-Source (<i>bns, 3d</i>)	LEE Self-Noise Z-Source
	LEE Shear-Noise Z-Source (<i>bns, 3d</i>)	LEE Shear-Noise Z-Source
	LEE Total Noise Z-Source (<i>bns, 3d</i>)	LEE Total Noise Z-Source

Table 20.14: Sensitivities Category

Category	Fluent Variable	CFX Variable
Body Force...	Sensitivity to Body Force X-Component	Sensitivity to Body Force X
	Sensitivity to Body Force Y-Component	Sensitivity to Body Force Y
	Sensitivity to Body Force Z-Component	Sensitivity to Body Force Z
	Sensitivity to Mass Sources	Sensitivity to Mass Sources

Table 20.15: Film Category

Category	Fluent Variable	CFX Variable
Film...	Film Thickness	Film Thickness
	Film X Velocity	Film Velocity u
	Film Y Velocity	Film Velocity v
	Film Z Velocity	Film Velocity w
	Film Temperature	Film Temperature
	Film Weber Number	Film Weber Number
	Film Surface X Velocity	Film Surface Velocity u
	Film Surface Y Velocity	Film Surface Velocity v
	Film Surface Z Velocity	Film Surface Velocity w
	Film DPM Mass Source	Film DPM Mass Source
	Film DPM x-mom Source	Film DPM mom Source X
	Film DPM y-mom Source	Film DPM mom Source Y
	Film DPM z-mom Source	Film DPM mom Source Z
	Film DPM Energy Source	Film DPM Energy Source
	Film DPM Shed Mass	Film DPM Shed Mass
	Film DPM Stripped Mass	Film DPM Stripped Mass

Category	Fluent Variable	CFX Variable
	Film DPM Stripped Diameter	Film DPM Stripped Diameter

Note:

The Fluent variable XF_RF_REACTING_CHANNEL_DATA will not be read by CFD-Post.
