Aqwa User's Manual

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

Release 2025 R1
January 2025

Table of Contents

1. Introduction 15
2. The Ansys Product Improvement Program 17
3. Running Aqwa using RSM 21
3.1. Run Configurations 21
3.1.1. Configuring Individual Systems to Solve 21
3.1.2. Configuring Parametric Studies 27
3.2. Limitations and Unsupported Features 30
4. Aqwa Workflow and Systems 31
4.1. Hydrodynamic Diffraction Workflows 32
4.2. Hydrodynamic Diffraction System 33
4.3. Hydrodynamic Response System 35
5. Aqwa Approach 39
5.1. Create a Hydrodynamic Analysis System 39
5.2. Attach Geometry 40
5.2.1. General Modeling Requirements 42
5.2.2. Configuring the Geometry 42
5.2.2.1. Stability/Time Response Specific Options 43
5.2.2.2. Composite Cable Seabed Definition 44
5.2.2.2.1. For a Cable not Using Cable Dynamics 45
5.2.2.2.2. For a Cable Using Cable Dynamics 45
5.2.3. Add Fixed Points 46
5.3. Mesh 46
5.3.1. Global Mesh Options 51
5.3.1.1. Mesh Parameters 51
5.3.1.2. Waterline Node Generation 51
5.3.2. Local Mesh Sizing Controls 52
5.4. Add Project Information and Options 52
5.5. Define Parts Behavior 54
5.5.1. Surface Body 58
5.5.2. Line Body 58
5.5.3. Point Mass 60
5.5.4. Distributed Mass 60
5.5.4.1. Importing from a `.MSD` File 61
5.5.4.2. Importing from a `.SFM` File 62
5.5.4.3. Inputting Data Manually 62
5.5.4.4. Representation in the Graphical Window 62
5.5.5. Point Buoyancy 63
5.5.6. Disc 63
5.5.7. Internal Tank 64
5.5.7.1. Fluid Level Definition Mode 65
5.5.7.2. Internal Tank Properties 67
5.5.8. Moonpool 68
5.5.8.1. Geometry 68
5.5.9. Additional Hydrodynamic Stiffness 72
5.5.10. Additional Added Mass (Frequency Independent) 73
5.5.11. Additional Damping (Frequency Independent) 73
5.5.12. Nonlinear Roll Damping 74
5.5.13. Current Force Coefficients 77
5.5.14. Wind Force Coefficients 80

5.5.15. Morison Hull Drag Coefficients 83
5.5.16. Yaw-Rate Drag 84
5.5.17. Structure Connection Points 84
5.5.18. External Lid 86
5.5.19. Part Transform 88
5.5.19.1. Reference Point Definition 88
5.5.19.2. Transform Definition 88
5.5.19.3. Application of a Part Transform 89
5.5.19.3.1. Part 89
5.5.19.3.2. Surface Body 90
5.5.19.3.3. Line Body 90
5.5.19.3.4. Point Mass 90
5.5.19.3.5. Distributed Mass 91
5.5.19.3.6. Point Buoyancy 91
5.5.19.3.7. Disc 91
5.5.19.3.8. Internal Tank 91
5.5.19.3.9. Moonpool 91
5.5.19.3.10. Additional Hydrodynamic Stiffness, Added Mass and Damping, and Morison Hull Drag Coefficients 91
5.5.19.3.11. Nonlinear Roll Damping and Yaw-Rate Drag 92
5.5.19.3.12. Current and Wind Force Coefficients 92
5.5.19.3.13. Connection Points 92
5.5.19.3.14. Connection Point Groups 92
5.5.19.3.15. Connections, Hydrodynamic Diffraction, and Hydrodynamic Response Features 93
5.6. Attachment Point Groups 93
5.6.1. Origin Definition 94
5.6.2. Group Definition 95
5.6.3. Group Rotation 96
5.6.4. Hydrodynamic Response Nodal Motions Output 96
5.7. Dynamic Point Structures 96
5.7.1. Define Dynamic Points 97
5.7.1.1. Define Dynamic Point Position 97
5.7.1.2. Equivalent Mass Properties 98
5.7.1.3. Hydrodynamic Properties 98
5.7.2. Modelling Considerations 98
5.8. Line Body Data 99
5.8.1. Beam Section 99
5.9. Define Connections 100
5.9.1. Cables 101
5.9.1.1. Linear Elastic 103
5.9.1.2. Nonlinear Polynomial 103
5.9.1.3. Nonlinear Steel Wire 104
5.9.1.4. Nonlinear Catenary 104
5.9.2. Tethers/Risers 107
5.9.3. Connection Data 109
5.9.3.1. Catenary Section 109
5.9.3.2. Catenary Joint 110
5.9.3.3. Tether/Riser Section 111
5.9.4. Connection Stiffness 112
5.9.4.1. Import MFK File 112

5.9.4.2. Manual Input ... 114
5.9.5. Fenders ... 116
5.9.5.1. Examples of Use of Fenders ... 118
5.9.6. Joints ... 121
5.9.6.1. Initial Positioning of Jointed Structures ... 125
5.9.6.2. Moving Structures ... 125
5.9.6.3. Closed Loops ... 126
5.9.6.4. Removing a Joint ... 128
5.10. Establish Analysis Settings ... 129
5.10.1. Specifying External Pre- or Post-Solve Operations ... 129
5.10.2. Hydrodynamic Diffraction Settings ... 130
5.10.2.1. Common Analysis Options ... 131
5.10.2.2. QTF Options ... 132
5.10.2.3. Output File Options ... 132
5.10.3. Hydrodynamic Response Settings ... 133
5.10.3.1. Stability Analysis ... 134
5.10.3.1.1. Common Analysis Options ... 134
5.10.3.1.2. Output File Options ... 135
5.10.3.2. Time Response Analysis ... 135
5.10.3.2.1. Time Response Pressure Output ... 136
5.10.3.2.2. Common Analysis Options ... 137
5.10.3.2.3. QTF Options ... 139
5.10.3.2.4. Output File Options ... 139
5.10.3.2.5. Output Aqwa FMU Package ... 140
5.10.3.3. Frequency Statistical Analysis ... 140
5.10.3.3.1. Common Analysis Options ... 142
5.10.3.3.2. QTF Options ... 142
5.10.3.3.3. Output File Options ... 142
5.11. Applying Ocean Environment and Forces ... 143
5.11.1. Structure Selection ... 143
5.11.2. Wave Directions ... 144
5.11.3. Wave Frequencies ... 145
5.11.3.1. Encounter Frequencies for a Single Wave Direction ... 146
5.11.3.2. Encounter Frequencies for Multiple Wave Directions ... 147
5.11.4. Starting Conditions ... 148
5.11.4.1. Setting the Position Starting Conditions ... 150
5.11.4.2. Setting the Velocity Starting Condition ... 152
5.11.5. Structure Force ... 153
5.11.6. Regular Wave ... 155
5.11.7. Irregular Wave ... 156
5.11.7.1. JONSWAP (H_s or Alpha) ... 162
5.11.7.2. Pierson-Moskowitz ... 163
5.11.7.3. Ochi-Hubble ... 163
5.11.7.4. Bretschneider ... 163
5.11.7.5. TMA Spectrum ... 163
5.11.7.6. Gaussian ... 163
5.11.7.7. User Defined Spectrum ... 164
5.11.7.7.1. With Wave Spreading: None (Long-Crested Waves) ... 164
5.11.7.7.2. With Wave Spreading: Nth-Powered Cosine (Short-Crested Waves) ... 166
5.11.7.7.3. With Wave Spreading: Manual Definition (Carpet Spectrum) ... 167
5.11.7.8. User Time History ... 169

5.11.8. Irregular Wave Group ... 171
5.11.9. Current ... 172
5.11.9.1. Constant Velocity ... 174
5.11.9.2. Varies with Depth, Dimensional ... 174
5.11.9.3. Varies with Depth, Non-Dimensional ... 175
5.11.9.4. Formulated ... 176
5.11.9.5. Graphical Representation ... 177
5.11.10. Wind ... 178
5.11.10.1. Time Dependent Velocity ... 181
5.11.11. Cable Winch ... 183
5.11.11.1. Winch that Maintains Constant Tension ... 184
5.11.11.2. Winch that Changes Cable Length ... 184
5.11.12. Connection Failure ... 186
5.11.12.1. Cable Failure ... 187
5.11.12.2. Fender Failure ... 187
5.11.13. Point Force ... 189
5.11.14. Deactivated Freedoms ... 190
5.12. Solution ... 191
5.12.1. Hydrostatic Results ... 193
5.12.2. Hydrodynamic Graphical Results ... 195
5.12.2.1. Diffraction ... 198
5.12.2.1.1. Diffraction, Froude-Krylov, Diffraction + Froude-Krylov, Linearized Morison Drag, and Total Exciting Force Including Morison Drag ... 198
5.12.2.1.2. Response Amplitude Operators (RAOs) and RAOs with Linearized Morison Drag ... 199
5.12.2.1.3. Radiation Damping & Added Mass ... 201
5.12.2.1.4. Steady Drift ... 202
5.12.2.1.5. Sum QTF and Difference QTF ... 203
5.12.2.1.6. Splitting Forces (RAO) ... 203
5.12.2.1.7. Bending Moment and Shear Force ... 205
5.12.2.2. Time Response ... 207
5.12.2.2.1. Wave Surface Elevation ... 208
5.12.2.2.2. Structure Position ... 208
5.12.2.2.3. Structure Velocity ... 209
5.12.2.2.4. Structure Acceleration ... 210
5.12.2.2.5. Structure Forces ... 211
5.12.2.2.6. Fender Forces ... 212
5.12.2.2.7. Fender Motions ... 213
5.12.2.2.8. Joint Forces ... 213
5.12.2.2.9. Cable Forces ... 213
5.12.2.2.10. Tether/Riser Motions ... 214
5.12.2.2.11. Tether/Riser Tensions ... 215
5.12.2.2.12. Tether/Riser Shear Force/Bending Moment ... 216
5.12.2.2.13. Tether/Riser Stresses ... 216
5.12.2.2.14. Time Step Error ... 217
5.12.2.3. Frequency Domain ... 217
5.12.2.3.1. Wave Spectra ... 217
5.12.2.3.2. Structure Response ... 218
5.12.2.3.3. Connection Response ... 219
5.12.2.4. Stability Response ... 220
5.12.2.4.1. Structure Position ... 220

5.12.2.4.2. Structure Forces 221
5.12.2.4.3. Fender Forces 222
5.12.2.4.4. Fender Motions 222
5.12.2.4.5. Joint Forces 222
5.12.2.4.6. Cable Forces 223
5.12.2.4.7. Tether/Riser Tensions 223
5.12.2.4.8. Tether/Riser Shear Force/Bending Moment 224
5.12.2.4.9. Iteration Step Error 225
5.12.3. Frequency Domain Tabular Results 225
5.12.4. Pressures and Motions Results 227
5.12.5. Internal Tank Pressures Results 233
5.12.6. Animation Results 236
5.12.6.1. Animation Controls 237
5.12.6.2. Animation Settings 237
5.12.7. Dynamic Natural Modes 238
5.12.8. Time Domain Statistical Results 241
5.12.9. Time Domain Pressures Results 244
5.13. Aqwa Model Data Import 246
6. Aqwa Workbench Interface 253
6.1. The Aqwa Editor User Interface 253
6.2. Editing Objects 256
6.2.1. Copy and Paste 257
6.2.2. Duplicate 257
6.2.3. Propagate 258
6.2.4. Break Link 259
6.2.5. Delete 259
6.3. Displayed Positions and Features 260
6.4. Setting Aqwa Application Options 264
6.4.1. Setting Aqwa Parallel Processing Options 265
7. Aqwa Common Features 269
7.1. Generating Reports 269
7.2. Parameters 269
7.3. Comments, Images, and Figures 270
7.3.1. Comments 270
7.3.2. Images 271
7.3.3. Figures 272
8. Hydrodynamic Add-ons 273
8.1. The Hydrodynamic Pressure Add-on 273
8.1.1. Introduction 273
8.1.2. Loading the Hydrodynamic Pressure Add-on 274
8.1.3. Setting up your Workflow in Workbench 275
8.1.4. Importing Hydrodynamic Diffraction Pressures and Loads to Mechanical 276
8.1.4.1. Configuring the Hydrodynamic Pressure Object 276
8.1.4.1.1. Load Configuration 277
8.1.4.1.2. Mapping Configuration 278
8.1.4.1.3. Axis Transformation 279
8.1.4.1.4. Output: Imported Pressures and Loads 279
8.1.4.1.5. Advanced Options 280
8.1.4.2. Generating Hydrodynamic Pressures and Inertial Loads 280
8.1.4.3. Structural Analysis Load Steps 284
8.1.5. Associating a Hydrodynamic Structure from the Hydrodynamic Diffraction Analysis 285

8.1.5.1. Configuring the Hydrodynamic Structure Object 285
8.1.5.2. Structure Selection 286
8.1.5.3. External Surfaces 286
8.1.5.4. Line Bodies 286
8.1.5.5. Structural Acceleration Application 286
8.1.5.6. Output: Structure Acceleration at Center of Gravity, Imported Pressures and Loads 287
8.1.6. Importing Time Domain Hydrodynamic Response Pressures and Loads to Mechanical 287
8.1.6.1. Configuring the Hydrodynamic Pressure Object 287
8.1.6.1.1. Load Configuration 288
8.1.6.1.2. Mapping Configuration 289
8.1.6.1.3. Axis Transformation 289
8.1.6.1.4. Output: Imported Pressures, and Loads 290
8.1.6.1.5. Advanced Options 290
8.1.6.2. Generating Hydrodynamic Pressures and Inertial Loads 291
8.1.7. Associating a Hydrodynamic Structure from the Time Domain Hydrodynamic Response Analysis 294
8.1.7.1. Configuring the Hydrodynamic Structure Object 295
8.1.7.2. Structure Selection 296
8.1.7.3. External Surfaces 296
8.1.7.4. Line Bodies 296
8.1.7.5. Structural Acceleration Application 296
8.1.7.6. Output: Structure Velocity and Acceleration at Center of Gravity, Imported Pressures and Loads 296
8.1.8. Internal Tank Pressure from a Hydrodynamic Diffraction Analysis 297
8.1.8.1. Configuring the Internal Tank Pressure Object 297
8.1.8.1.1. Scope: Internal Tanks 298
8.1.8.1.2. Internal Tank Details 298
8.1.8.1.3. Displayed Pressures 298
8.1.8.1.4. Imported Internal Tank Pressures 298
8.1.8.2. Generating Internal Tank Pressures 299
8.1.9. Internal Tank Pressure from a Time Domain Hydrodynamic Response Analysis 301
8.1.9.1. Configuring the Internal Tank Pressure Object 301
8.1.9.1.1. Scope: Internal Tanks 302
8.1.9.1.2. Internal Tank Details 302
8.1.9.1.3. Displayed Pressures 302
8.1.9.1.4. Imported Internal Tank Pressures 302
8.1.9.2. Generating Internal Tank Pressures 303
8.1.10. Moonpool Pressure from a Hydrodynamic Diffraction Analysis 304
8.1.10.1. Configuring the Moonpool Pressure Object 304
8.1.10.1.1. Scope: Moonpool 305
8.1.10.1.2. Moonpool Details 305
8.1.10.1.3. Displayed Pressures 305
8.1.10.1.4. Imported Moonpool Pressures 305
8.1.10.2. Generating Moonpool Pressures 306
8.1.11. Cable Forces from a Time Domain Hydrodynamic Response Analysis 306
8.1.11.1. Configuring the Cable Force Object 307
8.1.11.1.1. Scope: Cable Attachment 307
8.1.11.1.2. Cable Details 308
8.1.11.1.3. Displayed Forces 308
8.1.11.1.4. Imported Cable Force 309
8.1.11.2. Generating Cable Forces 309

8.1.12. Tether Forces/Moments from a Time Domain Hydrodynamic Response Analysis 310
8.1.12.1. Configuring the Tether Force Object 310
8.1.12.1.1. Scope: Tether Attachment 311
8.1.12.1.2. Tether Details 311
8.1.12.1.3. Displayed Forces 311
8.1.12.1.4. Imported Tether Force/Moment 312
8.1.12.2. Generating Tether Forces/Moments 312
8.1.13. Joint Forces/Moments from a Time Domain Hydrodynamic Response Analysis 312
8.1.13.1. Configuring the Joint Force Object 312
8.1.13.1.1. Scope: Joint Attachment 313
8.1.13.1.2. Joint Details 313
8.1.13.1.3. Displayed Forces 314
8.1.13.1.4. Imported Joint Force/Moment 314
8.1.13.2. Generating Joint Forces/Moments 314
8.1.14. Ocean Current 315
8.1.15. Theory 315
8.1.15.1. Hydrodynamic and Hydrostatic Pressures on Surfaces 315
8.1.15.2. Inertia and Viscous Drag Forces on Line Bodies 316
8.1.16. Limitations 316
8.2. The Offshore Add-on 317
8.2.1. Introduction 318
8.2.2. Loading the Offshore Add-on 319
8.2.3. Defining an Ocean Environment Object 319
8.2.3.1. Definition Properties 320
8.2.3.2. General Properties 320
8.2.3.3. Structural Coefficients 321
8.2.3.4. Soil-Pile Interaction 322
8.2.3.5. Wave Options 322
8.2.3.6. Pressure Mapping 325
8.2.4. Ocean Current 326
8.2.5. User Defined Spectrum 327
8.2.6. Harmonic Wave Load 327
8.2.7. Geometry-Based Variation 328
8.2.8. Component-Based Variation 329
8.2.9. Pipe-in-Pipe 330
8.2.10. Added Mass 332
8.3. The Aqwa Co-simulation Add-on 332
8.3.1. Introduction 333
8.3.2. Loading the Aqwa Co-simulation Add-on 334
8.3.3. Aqwa Co-simulation Toolbar 335
8.3.3.1. Update/Import Aqwa File 336
8.3.3.2. Mapping Aqwa Data 336
8.3.3.3. Clear 336
8.3.3.4. Co-simulation Wizard 337
8.3.4. Aqwa Co-simulation Setup Wizard 337
8.3.4.1. AeroDyn Connection 337
8.3.4.2. Aqwa Modeling Data 340
8.3.4.3. Co-simulation Settings 341
8.3.4.3.1. Co-simulation Analysis Time 342
8.3.4.3.2. Aqwa Time Response Pressure Output 343
8.3.4.4. Run Co-simulation 343

8.3.4.4.1. Information Setup 343
8.3.4.4.2. Result Saving Directory Setup 344
8.3.4.4.3. Running the Co-simulation Analysis 344
8.3.5. Additional Notes 346
8.3.6. Bibliography 347
9. Aqwa Appendix 349
9.1. Information for Existing Aqwa Users 349
Index 353

List of Figures

4.1. Hydrodynamic Context Menu Options in the Project Schematic 31
4.2. Connected Hydrodynamic Systems 36
5.1. Objects Drawn with Default Sizing 54
5.2. Objects After Re-Scaling 54
5.3. Moonpool Object Details Panel 68
5.4. Moonpool Walls and Free Surface with Normal Vectors Pointing Inward (Wall Surfaces) and Upward (Free Surface) 69
5.5. Ball and Socket Joint 122
5.6. Universal Joint 122
5.7. Hinged Joint 123
5.8. Rigid Joint 123
5.9. **A Closed Loop with No Redundant Constraints** 127
5.10. A Closed Loop with Redundant Constraints 128
5.11. Possible Equilibrium Positions 150
5.12. Line Graph 197
5.13. Surface Graph 197
5.14. Surface Contour Graph 198
5.15. Natural Modes Result Object 239
5.16. Mode Match Indicator 240
5.17. Natural Modes Advanced Options 240
5.18. Mode List with Animation 241
6.1. Project Schematic of Linked Hydrodynamic Response Systems 258
6.2. Project Outline with Linked Environment Objects 259
6.3. Geometric Features Defined 262
6.4. Connections Defined 263
6.5. Setting Wave Directions 263
6.6. Stability Analysis Starting Conditions 264
6.7. Stability Analysis Final Conditions 264
7.1. Example of Parameters in the Details View 270
7.2. Comment Edit Pane 271
7.3. New Figure or Image Menu 271
7.4. New Figure or Image Menu 272
8.1. Hydrodynamic Pressure Add-on Showing Loaded Status 274
8.2. Typical Workflow for Hydrodynamic Diffraction Pressure Mapping 275
8.3. Typical Workflow for Time Domain Hydrodynamic Response Pressure Mapping 276
8.4. Details of Hydrodynamic Pressure 277
8.5. Hydrodynamic Pressures in the Static Structural System 282
8.6. Hydrodynamic Pressures in the Hydrodynamic Diffraction System 283
8.7. Minimum/Maximum Pressures and Beam Loads vs Phase Angle (Graph) 283
8.8. Minimum/Maximum Pressures and Beam Loads vs Phase Angle (Table) 284
8.9. Details of Hydrodynamic Structure 285
8.10. Details of Hydrodynamic Pressure 288
8.11. Hydrodynamic Pressures in the Static Structural System 292
8.12. Hydrodynamic Pressures in the Time Domain Hydrodynamic Response System 292
8.13. Minimum/Maximum Pressures vs Time (Graph) 293
8.14. Minimum/Maximum Pressures vs Time (Table) 294
8.15. Details of Hydrodynamic Structure 295
8.16. Details of Internal Tank Pressure 298
8.17. Internal Tank Pressures in the Static Structural System 300

8.18. Internal Tank Pressures in the Hydrodynamic Diffraction System 300
8.19. Details of Internal Tank Pressure 302
8.20. Internal Tank Pressures in the Static Structural System 303
8.21. Internal Tank Pressures in the Time Domain Hydrodynamic Response System 304
8.22. Details of Moonpool Pressure 305
8.23. Details of Cable Force 307
8.24. Imported Forces in the Static Structural System 309
8.25. Details of Tether Force 310
8.26. Details of Joint Force 313
8.27. Valid Workflows for the Offshore Add-on 319
8.28. Offshore Add-on Showing Loaded Status 319
8.29. Ocean Environment Details 320
8.30. Ocean Geometry 321
8.31. Wave Case Definition Tabular Data Input 323
8.32. Ocean Current Details 326
8.33. Current Stretching Formulations 327
8.34. Harmonic Wave Load Details 327
8.35. Geometry-Based Variation Details 328
8.36. Component-Based Variation Details 329
8.37. Precedence of Local Adjustments to Structural Coefficients 330
8.38. Pipe-in-Pipe Details 331
8.39. Added Mass Details 332
8.40. Aqwa-Rigid Dynamics Solver Relationship 333
8.41. Aerodyn-Rigid Dynamics Solver Relationship 334
8.42. **Aqwa Co-simulation Add-on Showing Loaded Status** 335
8.43. Aqwa Co-simulation Toolbar 335
8.44. AeroDyn Connection Setup 340
8.45. Aqwa Modeling Data Setup 341
8.46. Run Co-simulation Setup 346

List of Tables

4.1. Aqwa Hydrodynamic Diffraction Workflows ... 33
4.2. Valid System Connections ... 35
5.1. Fender Color Codes ... 117
5.2. Fender Color Codes ... 188
5.3. Animation Controls ... 237
6.1. Standard Workbench Toolbars ... 253
6.2. Aqwa Editor Toolbars ... 254
6.3. Availability of Editing Operations ... 256
6.4. Structure Positions and Features Displayed According to Tree Selection ... 260
8.1. Wave Properties for Non-Periodic Wave Theories ... 324
9.1. Cross reference of tree objects with Aqwa commands ... 349

Chapter 1: Introduction

Ansys Aqwa provides an engineering toolset for the investigation of the effects of wave, wind and current on floating and fixed offshore and marine structures, including: spars; floating production, storage, and offloading (FPSO) systems; semi-submersibles; tension leg platforms (TLPs); ships; renewable energy systems; and breakwater design.

Aqwa Hydrodynamic Diffraction provides an integrated environment for developing the primary hydrodynamic parameters required for undertaking complex motions and response analyses. Three-dimensional linear radiation and diffraction analysis may be undertaken with multiple bodies, taking full account of hydrodynamic interaction effects that occur between bodies. While primarily designed for floating structures, fixed bodies such as breakwaters or gravity-based structures may be included in the models. Computation of the second-order wave forces via the full quadratic transfer function matrices permits use over a wide range of water depths.

Aqwa Hydrodynamic Response provides dynamic analysis capabilities for undertaking global performance assessment of floating structures. A wide range of physical connections, such as mooring lines, fenders, and articulations, are provided to model the restraining conditions on the vessels. In addition, seakeeping simulation may be undertaken with the inclusion of forward speed effects. Slow-drift effects and extreme-wave conditions may be investigated, and damage conditions, such as line breakage, may be included to study any transient effects that may occur.

Aqwa Hydrodynamic Diffraction and time domain Hydrodynamic Response calculations can also generate pressure and inertial loading for use in a structural analysis as part of the vessel hull design process. The results from a diffraction or time response analysis can be mapped onto an Ansys Mechanical finite element model for further structural assessment and detailed design. Since the mapping function automatically accounts for mesh differences between the hydrodynamic and finite element models, they do not have to be topologically identical.

Chapter 2: The Ansys Product Improvement Program

This product is covered by the Ansys Product Improvement Program, which enables Ansys, Inc., to collect and analyze *anonymous* usage data reported by our software without affecting your work or product performance. Analyzing product usage data helps us to understand customer usage trends and patterns, interests, and quality or performance issues. The data enable us to develop or enhance product features that better address your needs.

How to Participate

The program is voluntary. To participate, select **Yes** when the Product Improvement Program dialog appears. Only then will collection of data for this product begin.

How the Program Works

After you agree to participate, the product collects anonymous usage data during each session. When you end the session, the collected data is sent to a secure server accessible only to authorized Ansys employees. After Ansys receives the data, various statistical measures such as distributions, counts, means, medians, modes, etc., are used to understand and analyze the data.

Data We Collect

The data we collect under the Ansys Product Improvement Program are limited. The types and amounts of collected data vary from product to product. Typically, the data fall into the categories listed here:

Hardware: Information about the hardware on which the product is running, such as the:

- brand and type of CPU
- number of processors available
- amount of memory available
- brand and type of graphics card

System: Configuration information about the system the product is running on, such as the:

- operating system and version
- country code
- time zone
- language used
- values of environment variables used by the product

Session: Characteristics of the session, such as the:

- interactive or batch setting
- time duration
- total CPU time used
- product license and license settings being used
- product version and build identifiers
- command line options used
- number of processors used
- amount of memory used
- errors and warnings issued

Session Actions: Counts of certain user actions during a session, such as the number of:

- project saves
- restarts
- meshing, solving, postprocessing, etc., actions
- times the Help system is used
- times wizards are used
- toolbar selections

Model: Statistics of the model used in the simulation, such as the:

- number and types of entities used, such as nodes, elements, cells, surfaces, primitives, etc.
- number of material types, loading types, boundary conditions, species, etc.
- number and types of coordinate systems used
- system of units used
- dimensionality (1-D, 2-D, 3-D)

Analysis: Characteristics of the analysis, such as the:

- physics types used
- linear and nonlinear behaviors
- time and frequency domains (static, steady-state, transient, modal, harmonic, etc.)
- analysis options used

Solution: Characteristics of the solution performed, including:

- the choice of solvers and solver options
- the solution controls used, such as convergence criteria, precision settings, and tuning options
- solver statistics such as the number of equations, number of load steps, number of design points, etc.

Specialty: Special options or features used, such as:

- user-provided plug-ins and routines
- coupling of analyses with other Ansys products

Data We Do Not Collect

The Product Improvement Program does *not* collect any information that can identify you personally, your company, or your intellectual property. This includes, but is not limited to:

- names, addresses, or usernames
- file names, part names, or other user-supplied labels
- geometry- or design-specific inputs, such as coordinate values or locations, thicknesses, or other dimensional values
- actual values of material properties, loadings, or any other real-valued user-supplied data

In addition to collecting only anonymous data, we make no record of where we collect data from. We therefore cannot associate collected data with any specific customer, company, or location.

Opting Out of the Program

You may *stop* your participation in the program any time you wish. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

The Ansys, Inc., Privacy Policy

All Ansys products are covered by the Ansys, Inc., Privacy Policy.

Frequently Asked Questions

1. *Am I required to participate in this program?*

   No, your participation is voluntary. We encourage you to participate, however, as it helps us create products that will better meet your future needs.

2. *Am I automatically enrolled in this program?*

   No. You are not enrolled unless you explicitly agree to participate.

3. *Does participating in this program put my intellectual property at risk of being collected or discovered by Ansys?*

   No. We do not collect any project-specific, company-specific, or model-specific information.

4. *Can I stop participating even after I agree to participate?*

   Yes, you can stop participating at any time. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

5. *Will participation in the program slow the performance of the product?*

   No, the data collection does not affect the product performance in any significant way. The amount of data collected is very small.

6. *How frequently is data collected and sent to Ansys servers?*

   The data is collected during each use session of the product. The collected data is sent to a secure server once per session, when you exit the product.

7. *Is this program available in all Ansys products?*

   Not at this time, although we are adding it to more of our products at each release. The program is available in a product only if this *Ansys Product Improvement Program* description appears in the product documentation, as it does here for this product.

8. *If I enroll in the program for this product, am I automatically enrolled in the program for the other Ansys products I use on the same machine?*

   Yes. Your enrollment choice applies to all Ansys products you use on the same machine. Similarly, if you end your enrollment in the program for one product, you end your enrollment for all Ansys products on that machine.

9. *How is enrollment in the Product Improvement Program determined if I use Ansys products in a cluster?*

   In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

10. *Can I easily opt out of the Product Improvement Program for all clients in my network installation?*

    Yes. Perform the following steps on the file server:

    a. Navigate to the installation directory: `[Drive:]\v251\commonfiles\globalsettings`

    b. Open the file **ANSYSProductImprovementProgram.txt**.

    c. Change the value from "on" to "off" and save the file.

Chapter 3: Running Aqwa using RSM

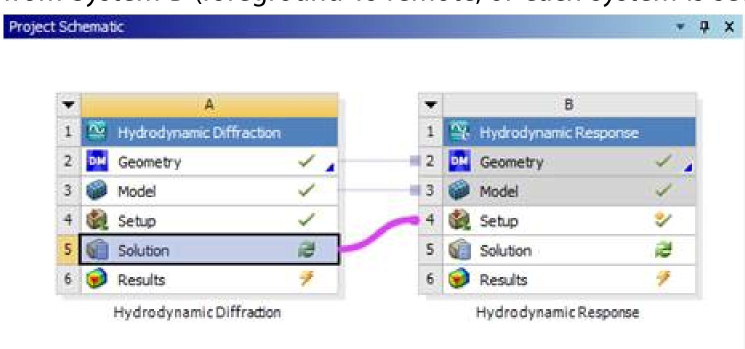
The Ansys Remote Solve Manager (RSM) allows you to configure and monitor job submission to HPC resources. This chapter describes how to use RSM to solve Aqwa models.

As a brief introduction, you have an option to install RSM when you install your Ansys software. Every RSM installation contains a default Ansys RSM Cluster (ARC) that can be used on the local machine or configured on a remote machine. RSM must be configured to define the resources where the jobs will be run, and the queues associated with those resources.

Generally, when you submit a job to RSM from a client application, job input files are transferred from the client working directory to an HPC staging directory. When the job has finished, all output files produced by the solver are transferred back to the client application.

3.1. Run Configurations

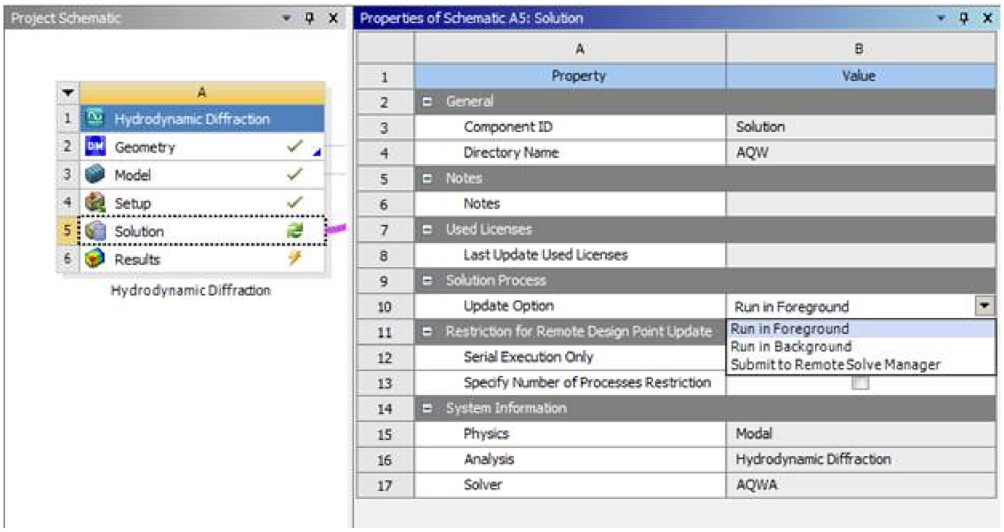
RSM works on individual systems in Workbench. This means that in the example below, System A will be solved independently from System B. Since B depends on the output of A, you must ensure that A finishes solving before B begins. Likewise, the solving method configuration for System A can be different from System B (foreground vs remote, or each system is set to run remotely on different queues).



3.1.1. Configuring Individual Systems to Solve

Because the solve is driven by Workbench, the queues need only to be defined once.

Solving Options



When you select the Solution cell in a system and open the Properties, these 3 choices are available for **Update Option** (running the solution) under **Solution Process**:

- Run in Foreground - (default) any properties you enter here will be superseded by the processing options specified in each Aqwa system.
- Run in Background - the solution runs in the background on the local machine. This option is appropriate for solutions that fit within the resources of your workstation but will take longer to execute.

> **Note:**
>
> The Aqwa editor will open to generate the input data file and store the results into the project following the completion of a run, similar to a normal RSM job. Therefore, background solutions can only be run on Windows machines.
>
> This mode requires that the RSM utilities are installed on your machine.

- Submit to Remote Solve Manager - this system will be solved using RSM.

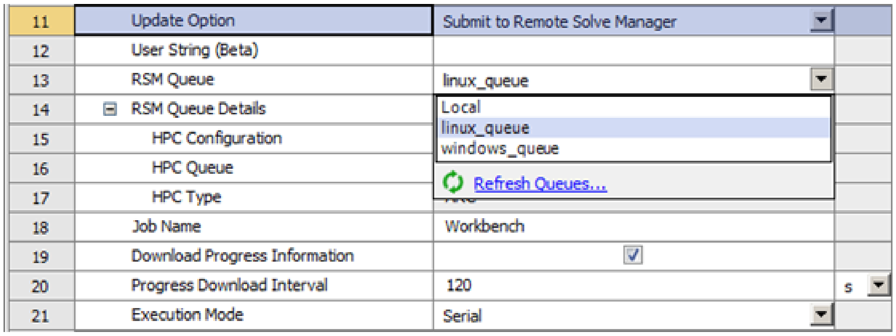
To update the solution, right-click the **Solution** cell and select Update. This will trigger the solving mechanism that you specified, creating the input files, running the solve and bringing the results back into the project directory if the solution was remote.

For more information on submitting a system for solution in Workbench, see Submitting Solutions in the *Workbench User's Guide*.

Solving Mechanism and Expectations

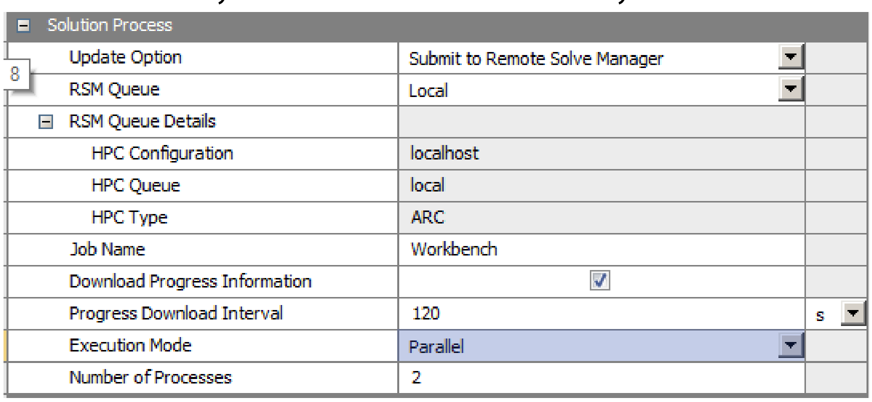
To enable RSM for a system, the **Update Option** of the solution process should be set to Submit to Remote Solve Manager.

1. Select and/or refresh the **RSM Queue** (the list of configured queues is directly read from your RSM configuration).



2. Set the **Execution Mode** to Serial or Parallel and set the number of physical cores you want to use to run Aqwa remotely. See Setting Aqwa Parallel Processing Options (p. 265) and The NUM_CORES Data Record in the *Aqwa Reference Manual* for more information on setting the number of cores. To find the number of cores in a cluster, see ????.

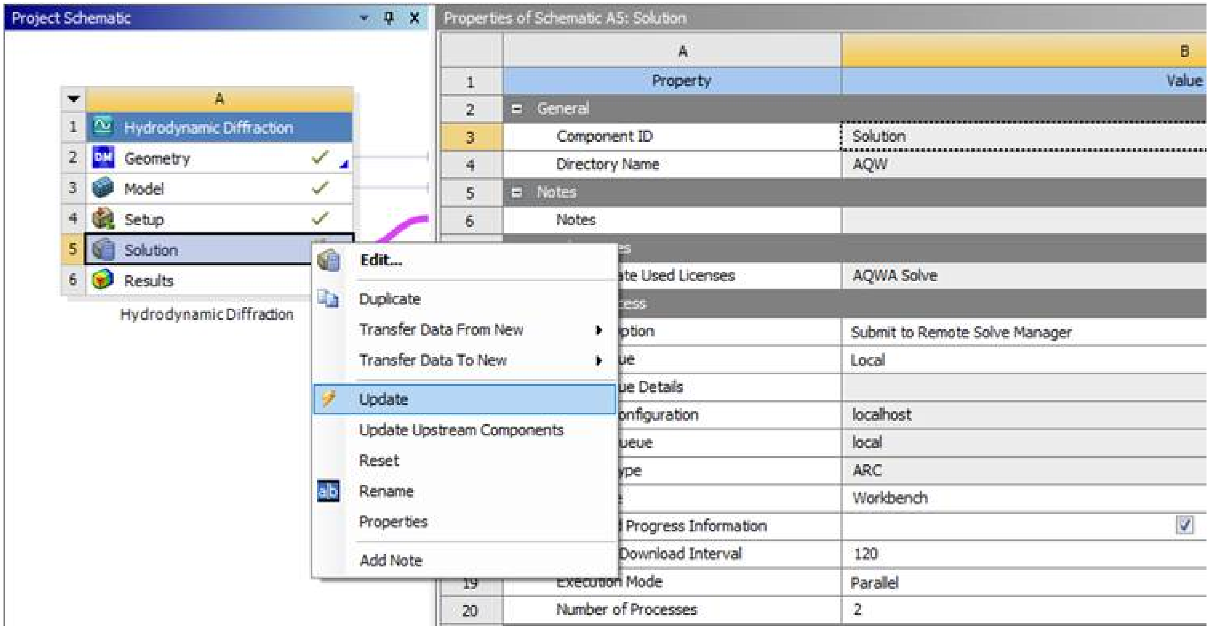
   - Serial will run with 1 core (default).
   - Parallel allows you to set the number of cores you want to use. Default is 2.



Note:

You need to ensure that the requested number of physical cores matches the host queue's properties. If you request more cores than the remote host has available, the job will fail.

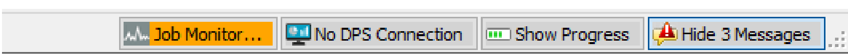
Update the **Solution** cell to initiate the solve.



If it is closed, the Aqwa editor will open to generate the solver input files. After the files are generated, the editor closes and is locked in this mode until the end of the solving process.

Meanwhile, the input files will be transferred to the remote host to start the solution.

Progress on the whole process can be followed by opening the Job Monitor window (at the bottom of the Workbench window).



When the solve is complete, the results files are transferred back into the project. The Aqwa editor can now be opened in order to read the messages issued by the solver and visualize the results when a run is successful.

Troubleshooting

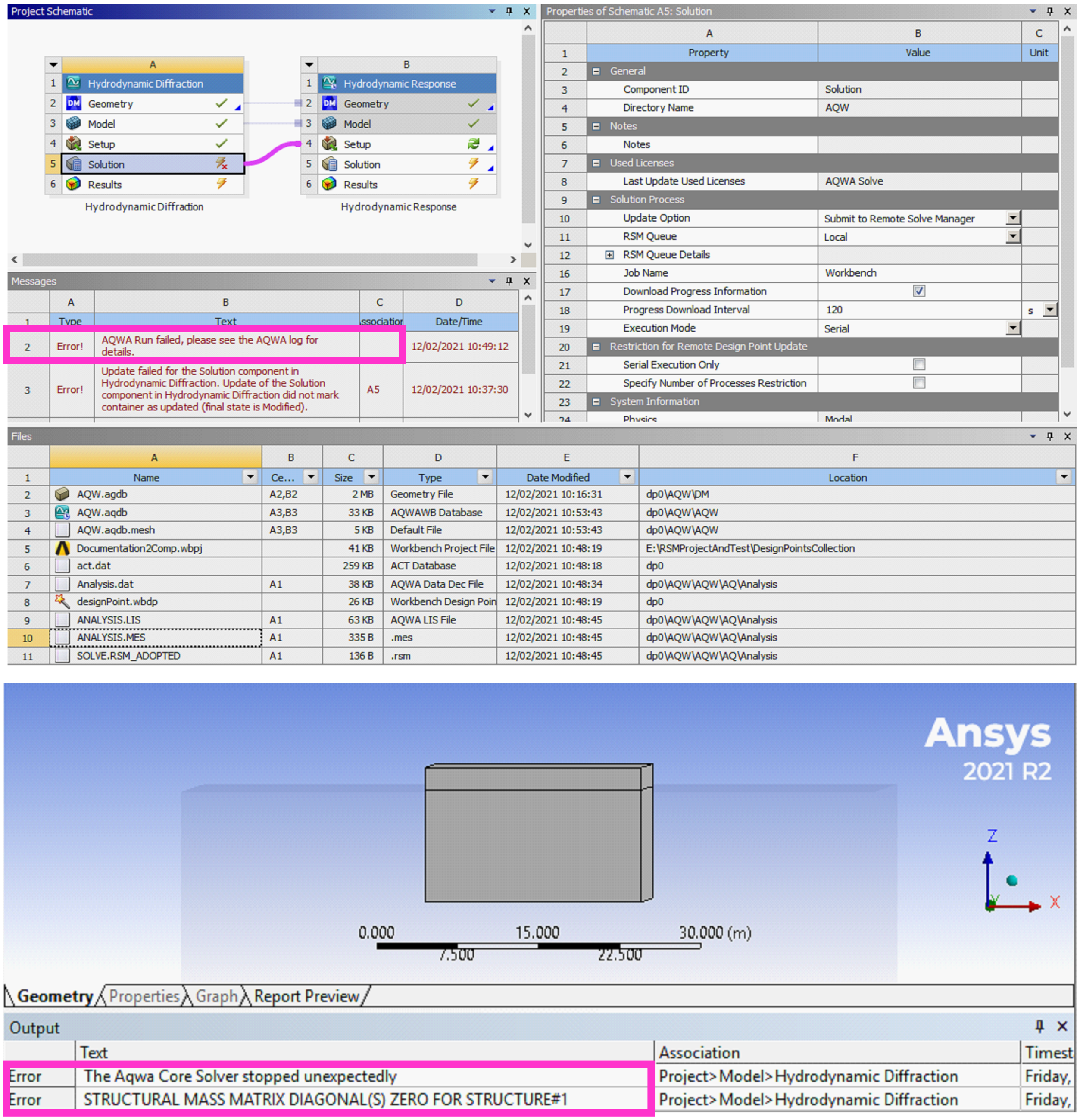
If your project is not set up correctly, you may get errors when you try to solve.

What if there is no mesh?

The setup cell is out of date and the editor will open as part of the solving process and generate the mesh if found missing.

What if there is no point mass and my model is incomplete?

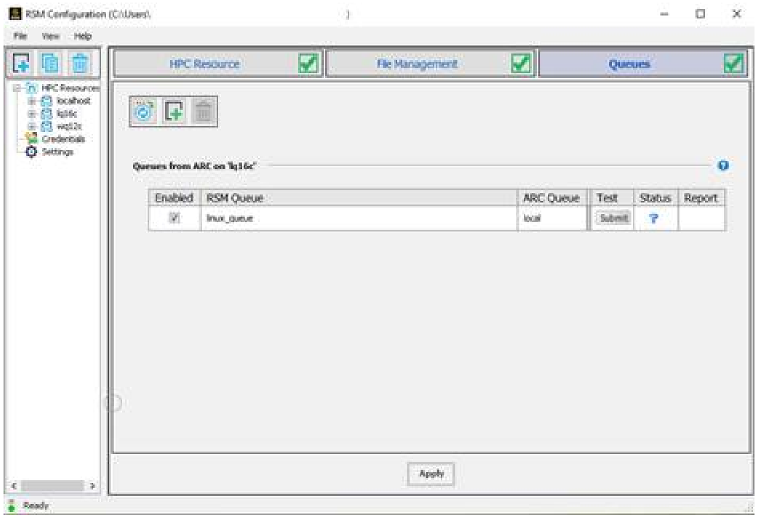
You will need to set a point mass if your structure is exclusively made of diffracting panels. If there is no point mass and the application runs in the foreground, the Aqwa editor will throw an error. If Aqwa runs through RSM, the application will attempt an RSM run but will fail. The following error message will be issued, requiring you to investigate the log files (`SOLVE.RSM` or `*.MES` files) in your project directory for further details or open the Aqwa editor to see the error message.



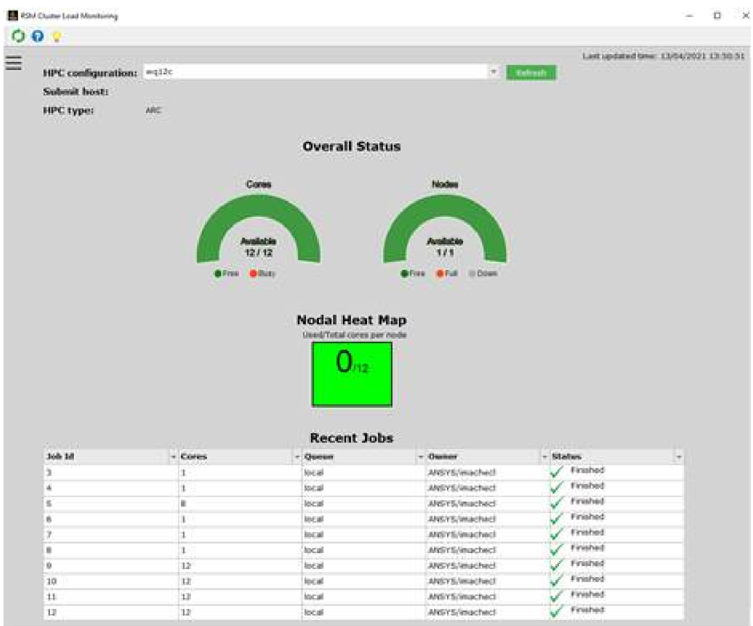
What if I ask for more cores than the host machine can provide?

- You need to ensure that the requested number of physical cores matches the host queue's properties. One way to check the number of available physical cores is by using **RSM Cluster Monitoring**, found under your Ansys installation on the Start Menu.

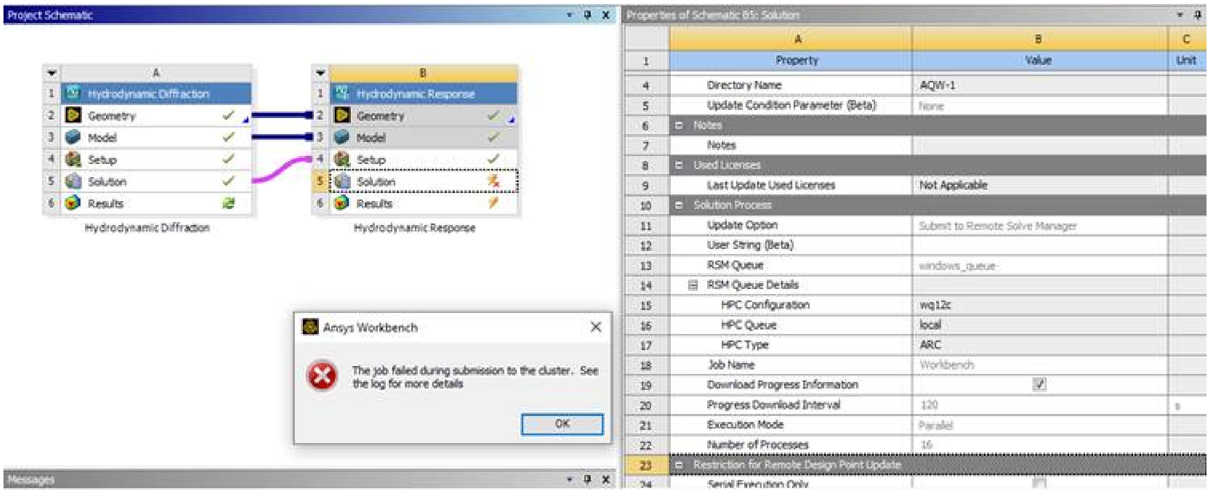
- To determine the properties of a specific queue, first run a test job on it. Open the **RSM Configuration** application from your Ansys installation on the Start Menu and submit a test job from the **Queues** tab.



- Then you can query its core and memory information. Open **RSM Cluster Monitoring** and select the queue you wish to submit your job to.



- If you require more cores than the remote host has available, the job will fail with the following error message:



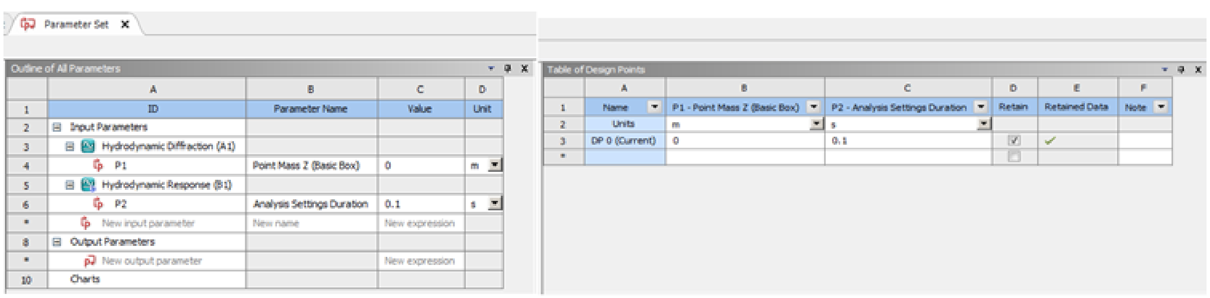
Supported Operating Systems

- Windows remote host
- Linux remote host

3.1.2. Configuring Parametric Studies

In Ansys applications, you can define key simulation properties to be parameters. The complete documentation on parametric studies in Workbench can be found in Working with Parameters and Design Points in the *Workbench User's Guide*:

To summarize, you can set input and output parameters corresponding to a series of "what-if" scenarios. The parameters are listed in a table, and you can interact with them through the project schematic.



The schematic is also responsible for the update process, which consists of taking a snapshot of your project with a given parameter set. Meaning, injecting into your project the input parameters and collecting the corresponding outputs.

This process can be carried out in 2 ways:

- Run in Foreground (local to the host machine, whether it is the user's or a remote machine)
- Run with Remote Solve Manager

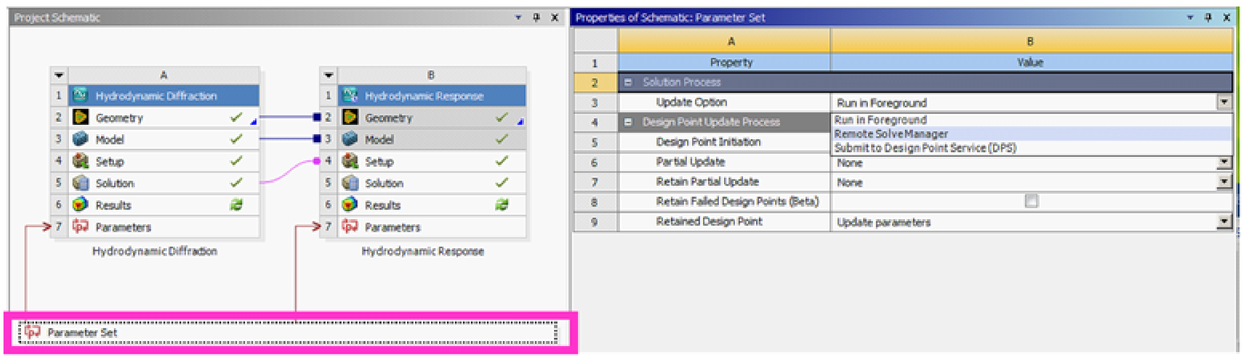
Since the program will need to open the Aqwa editor, the update of the design points can only take place on a Windows system, whether it is local or remote. This does not affect the solving configuration defined for each system as detailed in Configuring Individual Systems to Solve (p. 21).

There is one exception, though, which is when the whole parametric study is handled remotely via RSM, and RSM is also requested as the solving process for the individual system(s). This type of configuration is not permitted.

Review the examples (p. 28) of how to solve using the 2 different methods for more clarification.

Note:

The submission of a parametric study to the Design Point Service (DPS) is not supported for Aqwa at this time.



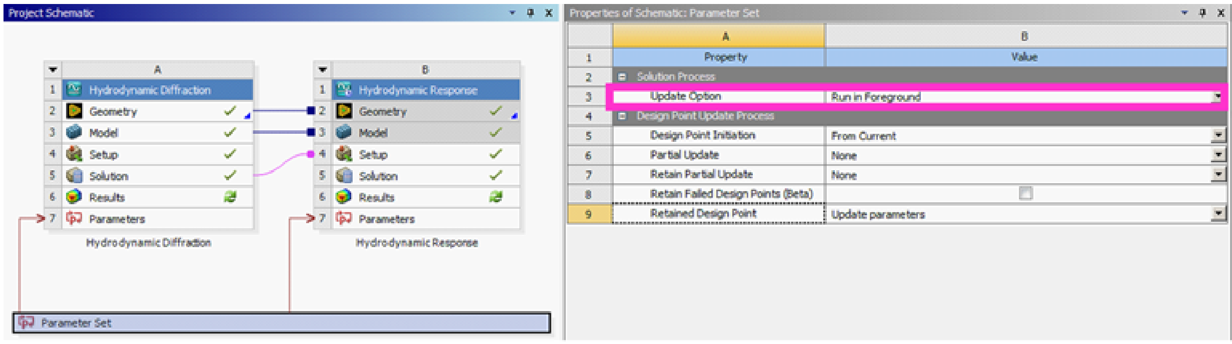
Supported Operating System and Configurations

	Solving Process for a System	
Parametric Study UpdateProcess	**Windows**	**Linux**
Run in Foreground	Y	Y
RSM	Y	N
DPS	n/a	n/a

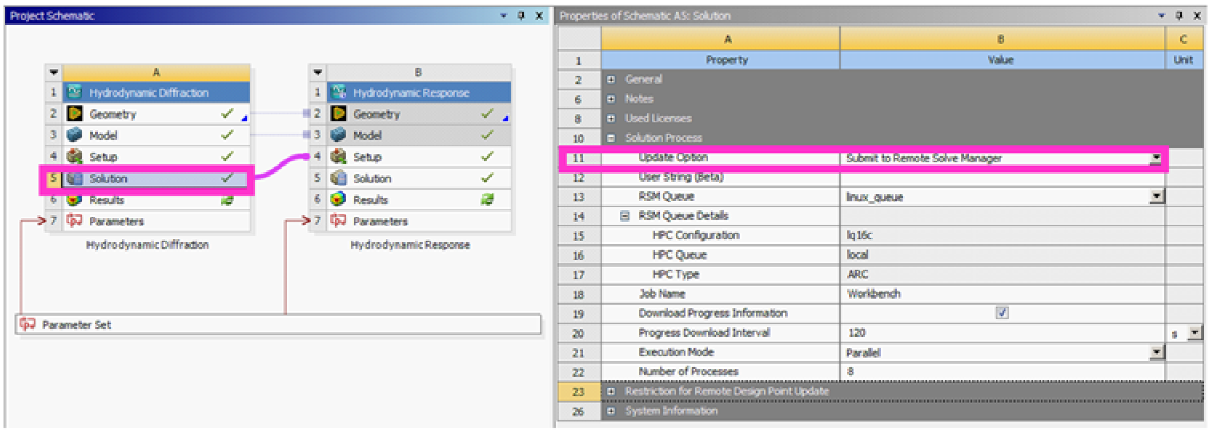
Examples of Possible Configurations

Run in Foreground

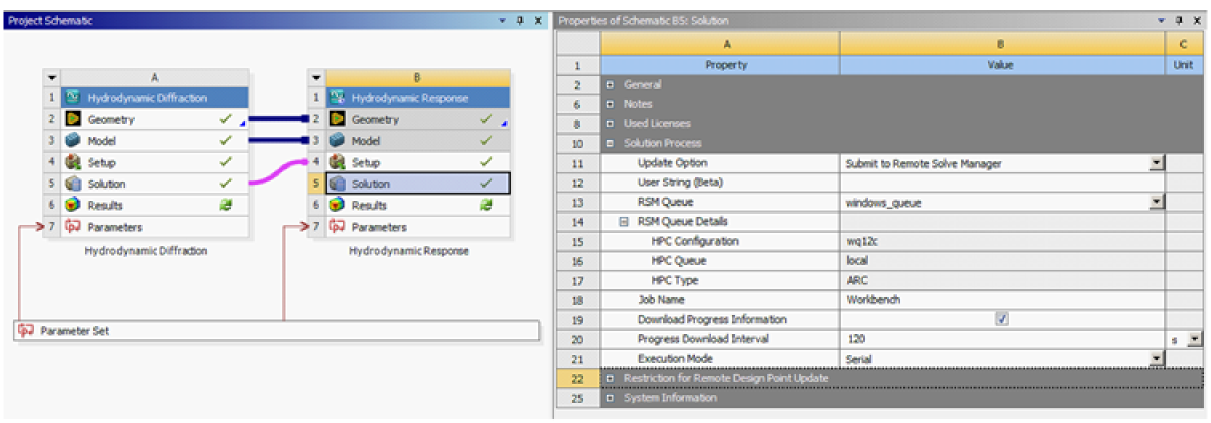
The update of the parameters takes place in the foreground (that is, on the local machine) so all the possible configurations for the remote solve of the systems A and B can be used as detailed in Configuring Individual Systems to Solve (p. 21). In this example, the Aqwa solve for the systems A and B is happening remotely on Linux and Windows, respectively.



Solve configuration for System A:

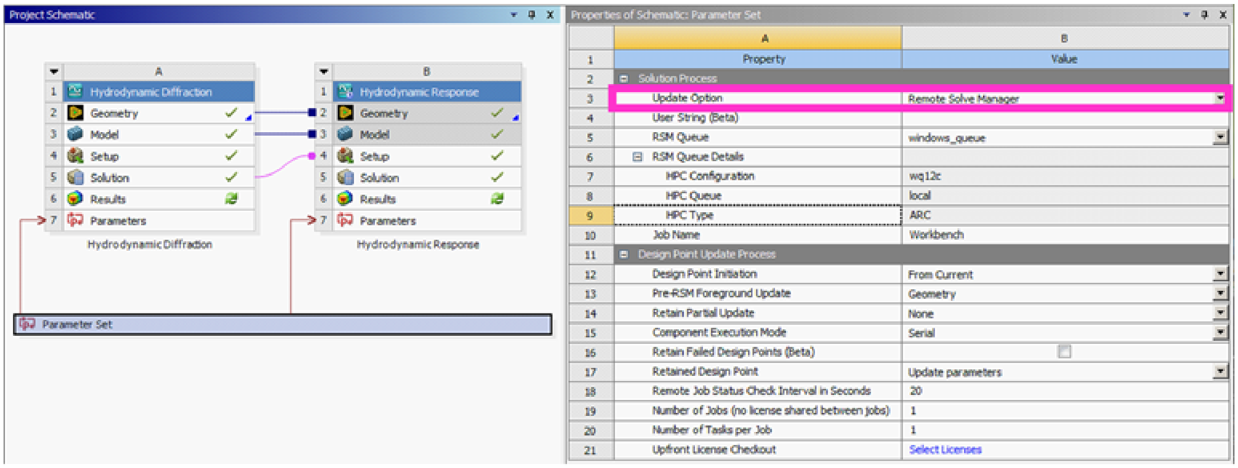


Solve configuration for System B:



Remote Solve Manager

The update of the parameters must be submitted to a Windows RSM queue. A a result, the only Solve Option that can be used for the update of systems A and B is Run in Foreground (meaning, locally on the remote machine).

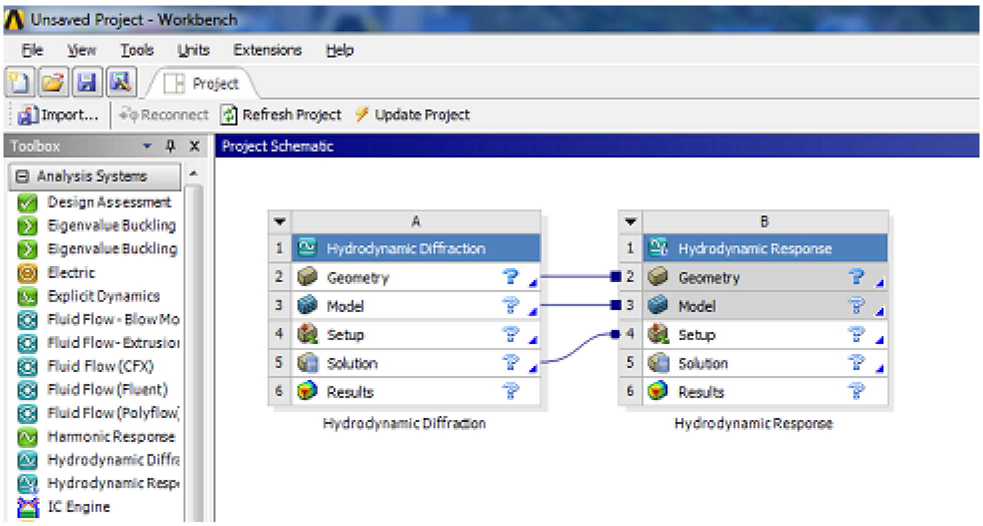


3.2. Limitations and Unsupported Features

- Presolve and postsolve commands: these commands are supported as long as when running on a remote host, they do not use non-Ansys software, non-standard versions of applications installed with Ansys software, or files that are located outside the project directory.
- External user force: not supported.
- Aqwa Editor on Linux: not supported.

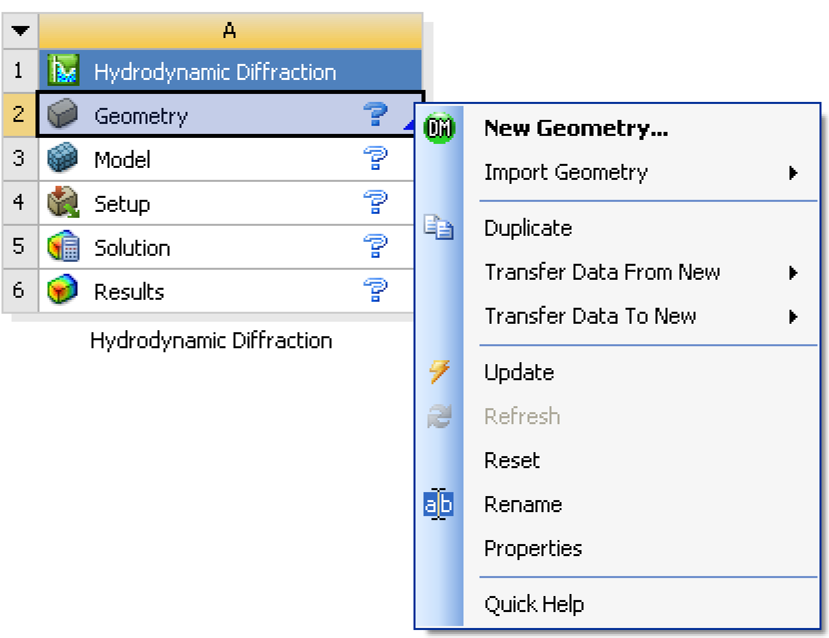
Chapter 4: Aqwa Workflow and Systems

Hydrodynamic systems can be dragged from the Analysis Systems Toolbox to the Project Schematic to create your analysis. See Create a Hydrodynamic Analysis System (p. 39) for more detailed information on creating your analysis using the Project Schematic.



Many tasks in the Schematic can be performed using Context Menu Options that display when you right-click a cell in a Hydrodynamic system in the Project Schematic.

Figure 4.1: Hydrodynamic Context Menu Options in the Project Schematic



Refer to Context Menu Options for more information on all of the available options, such as **Update**, **Refresh**, **Rename**, **Properties**, and **Quick Help**. Duplicate and Reset operations are not supported for Hydrodynamic Diffraction or Hydrodynamic Response systems.

Note:

The Hydrodynamics systems in Workbench support the import of elements from Aqwa models defined using the Aqwa Solver input format contained in files with the .DAT extension. See Aqwa Model Data Import (p. 246) for more information.

Note:

If your Geometry includes Moonpools that you wish to explicitly include in your analysis, you can only obtain results from the Hydrodynamic Diffraction system.

The Stability, Time Response, and Frequency Domain analyses, which are analyses available in the Hydrostatic Response system, do not support solving models where parts include active Moonpool objects.

4.1. Hydrodynamic Diffraction Workflows

There are two primary workflows in Aqwa to set up the Hydrodynamic Diffraction System which involve using the meshing tool provided internally in the Aqwa editor, or using the external Ansys Meshing tool.

When using the meshing tool provided internally by the Aqwa editor, you can access Ansys meshing capability with limited controls. For simple models, this tool may be sufficient, eliminating the need to add a separate meshing system in Workbench.

When using a Mesh system linked to a Hydrodynamics system in the Project Schematic in Workbench, you can access the Ansys Meshing Application. In this application, you may set the Physics Preference to Hydrodynamics, giving you similar settings as the internal Aqwa tool, but also providing access to the entire suite of features in the Meshing Application.

The following table describes the two workflows.

Note:

- Some of the data is only used in subsequent Hydrodynamic Response analyses and may be defined either before or after the Hydrodynamic Diffraction analysis is performed.
- A hydrodynamic add-on is provided which facilitates the transfer of hydrodynamic pressures calculated in a Hydrodynamic Diffraction system or time domain Hydrodynamic Response

to a Static Structural finite element analysis. For information on using this extension see Hydrodynamic Add-ons (p. 273).

Table 4.1: Aqwa Hydrodynamic Diffraction Workflows

Using Internal Mesher	Using External Mesher
1. Import/create Hydrodynamic Diffraction system (p. 39)	1a. Create mesh system
	1b. Attach geometry
2. Attach geometry (p. 40)	2a. Mesh using external tool
	2b. Link Hydrodynamic Diffraction system (p. 33)
3. Configure the geometry (p. 42)	
4. Configure mesh (p. 46) (if using Ansys Meshing tool, mesh is read-only in Aqwa)	
5. Add project information and options (p. 52)	
6. Define parts behavior (p. 54)	
7. Define connections (p. 100)	
8. Establish analysis settings (p. 129)	
9. Apply ocean environment and forces (p. 143)	
10. Solve and postprocess (p. 191)	

Aqwa Approach (p. 39) provides details on all procedures to perform an Aqwa analysis.

See Editing Objects (p. 256) for more information on interacting with objects in the tree view.

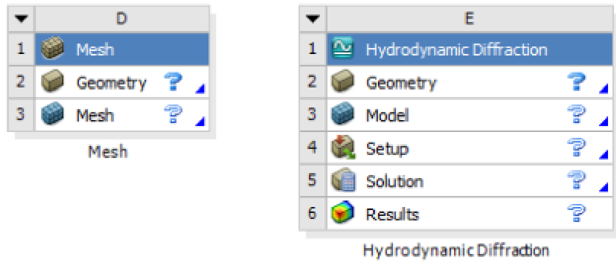
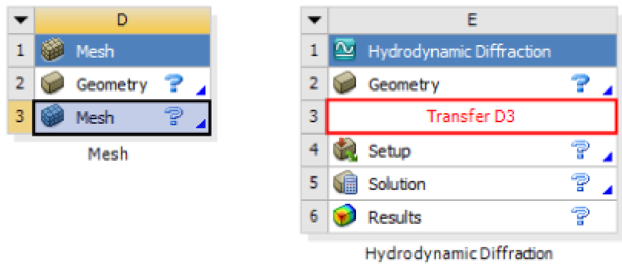
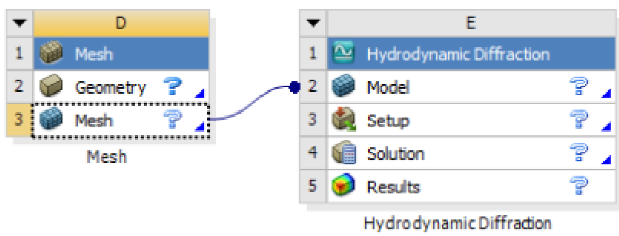
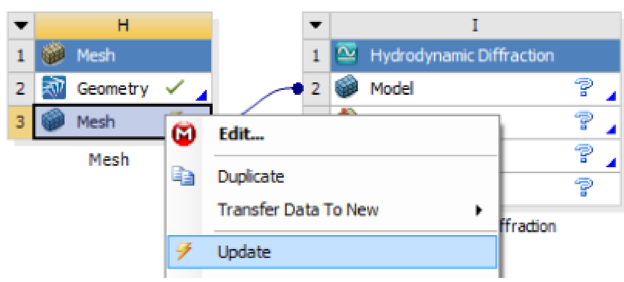
The following sections describe the hydrodynamic systems provided by Aqwa in Workbench:

- Hydrodynamic Diffraction System (p. 33)
- Hydrodynamic Response System (p. 35)

4.2. Hydrodynamic Diffraction System

Hydrodynamic Diffraction systems can be standalone systems, or attached to an upstream mesh system, depending on the analysis workflow (p. 31). In the external meshing workflow, the Hydrodynamic Diffraction system shares the **Mesh** of the Mesh system with the **Model** of the Hydrodynamic Diffraction system.

To share these systems in the Project Schematic, you must initially create two independent systems, then manually link the two relevant cells. The cell list updates once the link has been established.

Create Mesh and Hydrodynamic Diffraction systems in the Project Schematic	
Manually link the **Mesh** cell to the **Model** cell	
Hydrodynamic Diffraction system updates to reflect the connection	
After the geometry has been imported and the mesh is generated, update the **Mesh** cell from the Project Schematic	

There are two analysis options for a Hydrodynamic Diffraction analysis: either to calculate the Hydrostatics only (Aqwa stages 1 and 2), or to calculate the full Hydrodynamic results.

When the full hydrodynamic diffraction analysis is performed, Aqwa generates a database of hydrodynamic coefficients for each structure (or group of interacting structures) in the model. Generating this database is often the most computationally expensive part of the hydrodynamic workflow. Motion response amplitude operators (RAOs) and force/moment quadratic transfer functions (QTFs) are included in the database. These are derived from Aqwa's calculations of the first-order wave loads, frequency-dependent added mass, and radiation damping matrices.

After doing a full Hydrodynamic Diffraction analysis, you may occasionally need to modify a property of the model that affects the RAOs and QTFs, but has no effect on the wave loads or frequency-dependent added mass/radiation damping. For example, you may choose to introduce additional frequency-independent damping to a structure to tune the calculated RAOs to some experimental data. In these cases, the Aqwa Workbench editor re-uses the existing hydrodynamic database as much as possible to reduce the computational cost of re-solving the hydrodynamic diffraction analysis. A notification appears in the Messages panel to indicate when this has happened.

4.3. Hydrodynamic Response System

In a set of linked systems, the first Hydrodynamic Response system must be connected to an upstream Hydrodynamic Diffraction system.

For internal meshing, the Hydrodynamic Response system must share the Geometry, Model, and Solution cells of the Hydrodynamic Diffraction system (see Figure 4.2: Connected Hydrodynamic Systems (p. 36) with internal meshing).

For external meshing, the geometry cell is not available in the Hydrodynamic Diffraction system, so only the Model and Solution cells are shared (see Figure 4.2: Connected Hydrodynamic Systems (p. 36) with external meshing).

Note:

When you drop the Hydrodynamic Response system on the Hydrodynamic Diffraction system in the Project Schematic, you are given different options of sharing cells with the Hydrodynamic Response system. Only connect the systems to include transferring the Solution from the Hydrodynamic Diffraction system. If this cell is not shared, the subsequent analysis will not work.

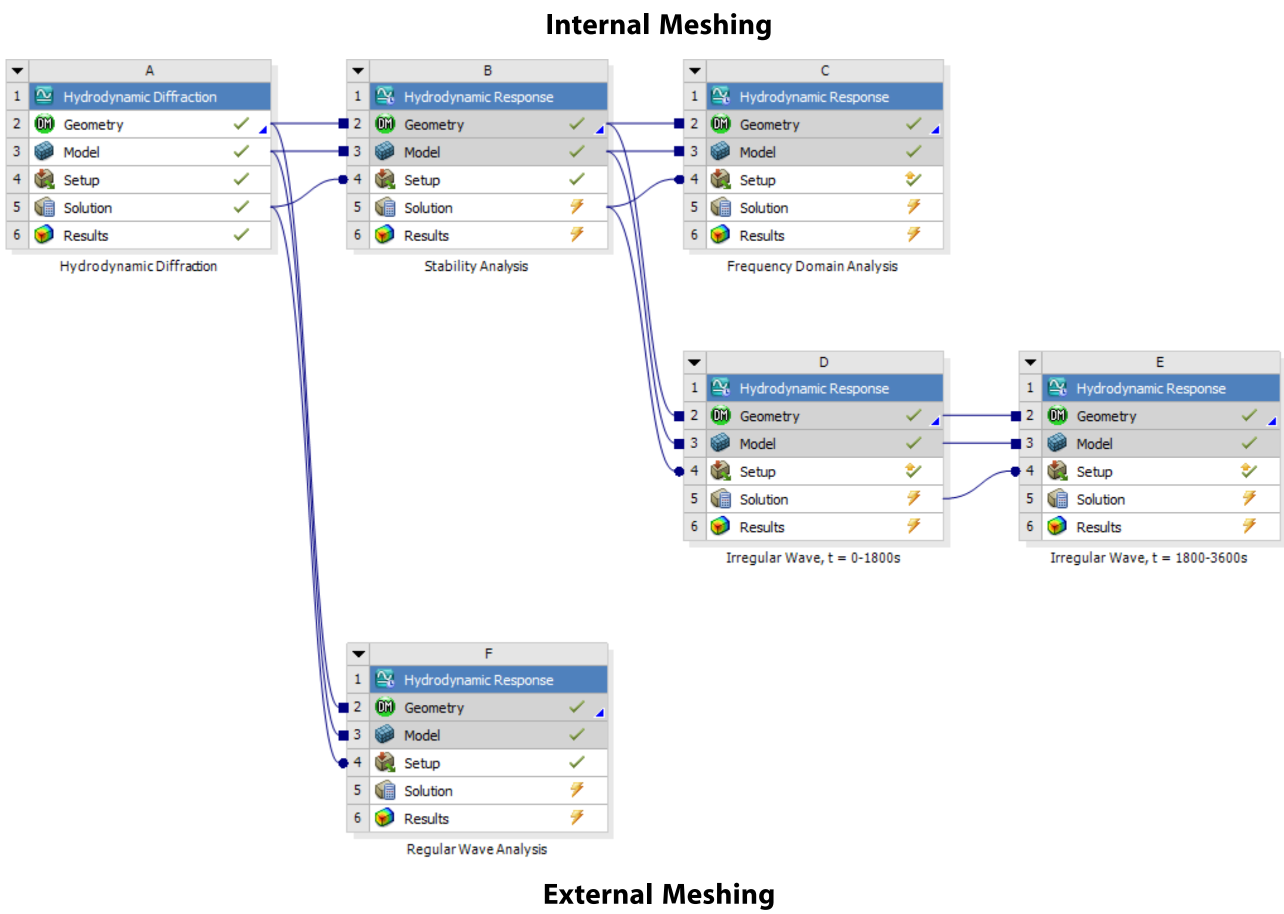
Hydrodynamic Response systems can be configured for a number of different analysis types using the **Computation Type** option. See Hydrodynamic Response Settings (p. 133) for more details on computation types. Changing the **Computation Type** setting for a Hydrodynamic Response analysis affects the types of systems to which you can share data downstream. Table 4.2: Valid System Connections (p. 35) shows how computation types can be connected.

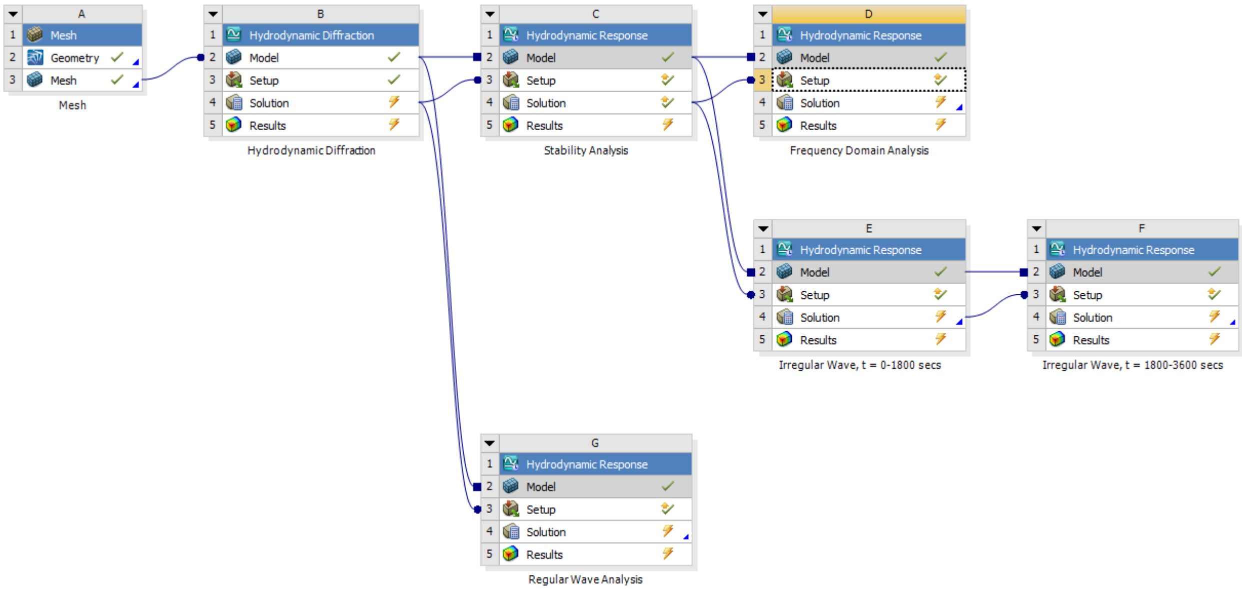
Table 4.2: Valid System Connections

Upstream System	Valid Downstream Systems	Notes
Hydrodynamic Diffraction (p. 130)	Hydrodynamic Diffraction (p. 130) Hydrodynamic Response: Time Response (p. 135) Hydrodynamic Response: Frequency Domain Analysis (p. 140) Hydrodynamic Response: Stability Analysis (p. 134)	When two Hydrodynamic Diffraction systems are connected, only the Geometry and Model information is shared (only Model information if external meshing is used).
Hydrodynamic Response: Stability Analysis (p. 134)	Hydrodynamic Response: Time Response (p. 135) Hydrodynamic Response: Frequency Domain Analysis (p. 140)	If you are performing a Stability Analysis and connect a downstream Stability Analysis, the downstream system uses the position computed in the upstream system as the starting position.

Upstream System	Valid Downstream Systems	Notes
	Hydrodynamic Response: Stability Analysis (p. 134)	
Hydrodynamic Response: Time Response (p. 135)	Hydrodynamic Response: Time Response (p. 135)	If you are performing a Time Response Analysis and connect a downstream Time Response Analysis, the downstream system begins its simulation at the end time step of the upstream system.
Hydrodynamic Response: Frequency Domain Analysis (p. 140)	None	

Figure 4.2: Connected Hydrodynamic Systems




A
1 Mesh
2 Geometry
3 Mesh
Mesh
B
1 Hydrodynamic Diffraction
2 Model
3 Setup
4 Solution
5 Results
Hydrodynamic Diffraction
C
1 Hydrodynamic Response
2 Model
3 Setup
4 Solution
5 Results
Stability Analysis
D
1 Hydrodynamic Response
2 Model
3 Setup
4 Solution
5 Results
Frequency Domain Analysis
E
1 Hydrodynamic Response
2 Model
3 Setup
4 Solution
5 Results
Irregular Wave, t = 0-1800 secs
F
1 Hydrodynamic Response
2 Model
3 Setup
4 Solution
5 Results
Irregular Wave, t = 1800-3600 secs
G
1 Hydrodynamic Response
2 Model
3 Setup
4 Solution
5 Results
Regular Wave Analysis

Chapter 5: Aqwa Approach

This chapter provides details on the procedures required to setup a hydrodynamic analysis.

5.1. Create a Hydrodynamic Analysis System

5.2. Attach Geometry

5.3. Mesh

5.4. Add Project Information and Options

5.5. Define Parts Behavior

5.6. Attachment Point Groups

5.7. Dynamic Point Structures

5.8. Line Body Data

5.9. Define Connections

5.10. Establish Analysis Settings

5.11. Applying Ocean Environment and Forces

5.12. Solution

5.13. Aqwa Model Data Import

5.1. Create a Hydrodynamic Analysis System

Each analysis type is represented by an *analysis system* that includes the individual components of the analysis such as the associated geometry and model properties. Most analyses are represented by one independent analysis system. However, an *analysis with data transfer* can exist where results of one analysis are used as the basis for another analysis. In this case, an analysis system is defined for each analysis type, where components of each system can share data.

- To create an analysis system, expand the **Analysis Systems** section in the **Toolbox** and drag an analysis object template onto the **Project Schematic**. The analysis system is displayed as a vertical array of cells (schematic) where each cell represents a component of the analysis system. Address each cell by right-clicking on the cell and choosing an editing option.

- To create an analysis system with data transfer to be added to an existing system, drag the object template representing the upstream analysis directly onto the existing system such that red boxes enclose cells that will share data between the systems. After you release the mouse button, the two systems are displayed, including an interconnecting link and a numerical designation as to which cells share data.

See Working Through a System for more information.

Note:

In hydrodynamic analysis systems, the Geometry cell is the only cell that can share data with other types of analysis systems.

5.2. Attach Geometry

There are no geometry creation tools in the Aqwa application so geometry must be attached to the hydrodynamic system. You can create the geometry from either of the following sources:

- From within Workbench using DesignModeler or Discovery Modeling. For details on the use of the creation tools, refer to the DesignModeler Help or Discovery Help.
- From a CAD system supported by Workbench. See the CAD Integration section for a complete list of the supported systems.

Note:

It is possible to import geometry from SpaceClaim. However, to edit a .scdoc file, you must install SpaceClaim from the Customer Portal, as it is no longer included in the unified installer.

Before attaching the geometry from either of these sources, you can specify several options that determine the characteristics of the geometry you choose to import by right-clicking the **Geometry** cell and choosing **Properties**. See the CAD Integration section for more details about the options, most of which do not apply in hydrodynamic analyses. Also see the General Modeling Requirements (p. 42) section for more information about the implications of Geometry import properties.

Note:

Aqwa only processes the Line Bodies and Surface Bodies in a geometry. Make sure that the boxes are checked for both of these Geometry options in the Properties view. You can attach a geometry that has no line or surface bodies, but Aqwa will not process the geometry.

Related Procedures

Procedure	Condition	Procedural Steps
Specifying geometry options	Optional task that can be done before attaching geometry.	1. In an analysis system schematic, perform either of the following: • Right-click the **Geometry** cell and choose **Properties** OR

Procedure	Condition	Procedural Steps
		• Select the **Geometry** cell in the schematic for a standard analysis, the from the **Workspace** toolbar dropdown menu, choose any option that includes **Properties** or **Components**. 2. Check boxes to specify **Default Geometry Options** and **Advanced Geometry Defaults**.
Attaching DesignModeler geometry to a hydrodynamic system	DesignModeler is running in an analysis system.	Double-click the **Model** cell in the same analysis system schematic. The Aqwa application opens and displays the geometry.
	DesignModeler is not running. Geometry is stored in an agdb file.	1. Select the **Geometry** cell in an analysis system schematic. 2. Browse to the agdb file from the following access points: • Right-click the **Geometry** cell in the **Project Schematic**, **Import Geometry** and choose **Browse**. 3. Double-click the **Model** cell in the schematic. The Aqwa application opens and displays the geometry.
Attaching CAD geometry to a hydrodynamic system	CAD system is running.	1. Select the **Geometry** cell in an analysis system schematic. 2. Right-click the **Geometry** cell listed there. 3. Double-click the **Model** cell in the same analysis system schematic. The Aqwa application opens and displays the geometry. 4. If required, set geometry options in the Aqwa application by highlighting the **Geometry** object and choosing settings under **Preferences** in the Details view.
	CAD system is not running. Geometry is stored in a native CAD system file, or in a CAD "neutral" file such as Parasolid or IGES.	1. Select the **Geometry** cell in an analysis system schematic. 2. Browse to the CAD file from the following access points: • Right-click the **Geometry** cell in the **Project Schematic** and choose **Import Geometry**. 3. Double-click the **Model** cell in the **Project Schematic**. The Aqwa application opens and displays the geometry.

CAD Interface Terminology

The CAD interfaces can be run in either plug-in mode or in reader mode.

- **Attaching geometry in plug-in mode:** requires that the CAD system be running.
- **Attaching geometry in reader mode:** does not require that the CAD system be running.

5.2.1. General Modeling Requirements

DesignModeler and Discovery Modeling are the Ansys tools used to create geometry for hydrodynamic systems. For information on importing geometry created in DesignModeler or Discovery Modeling, see the Attach Geometry (p. 40) section. When using either of these tools to define the geometry, there are a number of aspects that you should consider to ensure that your model is suitable for analysis with Aqwa, such as:

- The waterline must lie at **Z = 0** on the XY plane. Import or create each structure, then use **Translate** or **Move** operations to set the correct draft or depth for each structure.
- When using lines to create beams, only tubular sections are supported in Aqwa. All other sections will result in a slender tube element being formed and you will need to define additional information in the Aqwa application.
- Each vessel/structure should be a part, so all the bodies that you have should be grouped via the multibody part facility.
- The model is oriented with its Z axis vertical up.
- External surfaces must have normals pointing outward.
- Internal Tank surfaces must have normals pointing inward.
- Moonpool wall surfaces must have normals pointing inward, towards the enclosed fluid volume.
- The matching surface that corresponds to the hole in the hull should have its normal pointing upward (towards the enclosed sea surface so that $\overrightarrow{normal}.\vec{z}>0$).
- The free surface (which is the area where the sea surface would cross the Moonpool at the waterline z=0) should have its normal pointing upwards (away from the enclosed sea surface so that $\overrightarrow{normal}.\vec{z}>0$).

Hydrodynamic analysis systems only process the Line Bodies and Surface Bodies in a geometry. You can attach a geometry that has no Line Bodies or Surface Bodies, but note that this is not appropriate for an Aqwa analysis.

5.2.2. Configuring the Geometry

On opening a hydrodynamics analysis system, the Aqwa Editor will automatically attach the geometry. Each part becomes a separate structure in Aqwa.

The hydrodynamics systems assume the still water surface lies on the XY plane, and Z is positive up.

All structures are located in a global analysis space. Note that Hydrostatic Results (p. 193) and Hydrodynamic Graphical Results (p. 195) are also presented in global directions.

Within the geometry, you can select the types of bodies that will be attached.

Once attached, the diffracting behavior of surface geometry can be selected. Lines can be set to be Tubular or Slender Tube and additional Aqwa specific objects can be added, such as Point Mass (p. 60), Point Buoyancy (p. 63) and Disc (p. 63).

The Details panel provides you with options for setting up the sea geometry's **Water Depth** and size (**Water Size X**, **Water Size Y**), which can be used to alter the graphical view. The sea level lies at Z=0 in the XY plane.

It is important that the correct water depth is specified, especially with shallow water conditions, since the sea bed acts as a boundary condition to the diffraction analysis. The Water Size in the X and Y directions only modifies the extent of the graphical display of the water surface and sea bed.

The **Water Density** and **Gravity** can also be changed.

Note:

It is not yet possible to employ symmetry in the Aqwa Editor, hence the full model must be meshed.

5.2.2.1. Stability/Time Response Specific Options

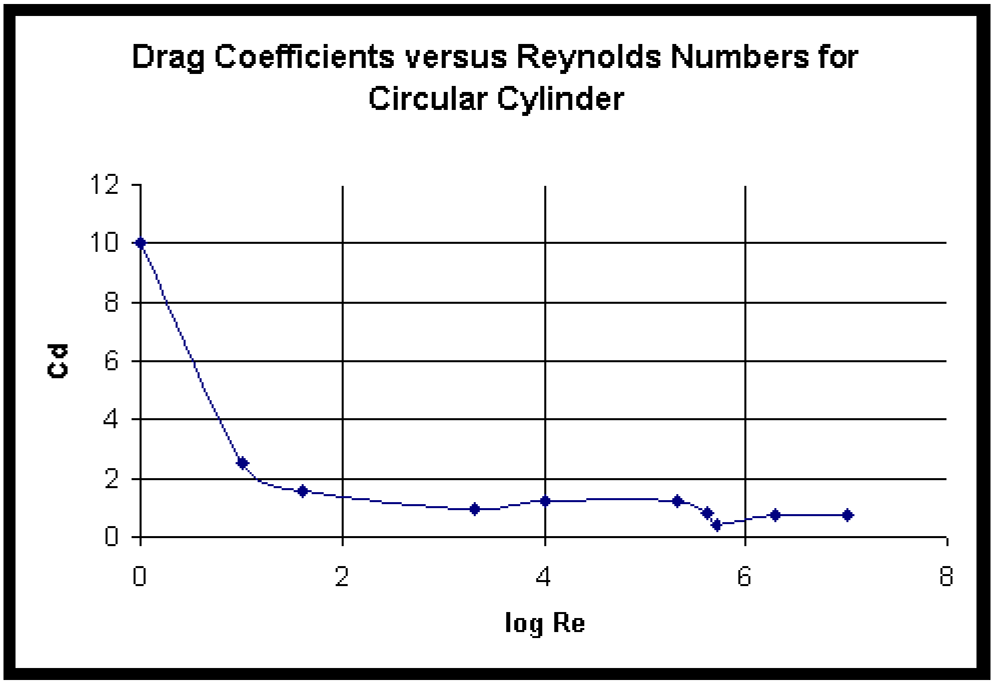
The **Tube Drag Coefficients** and **Scale Factor for Model Test Simulation** options apply to all Morison (TUBE and STUB) elements within your Geometry. These options affect only Stability and Time Response analyses; they are not used in Hydrodynamic Diffraction or Frequency Statistical analyses.

The **Seabed Inline/Lateral Friction Coefficient** options only affect Time Response analyses which include Cable Dynamics; they are not used in Hydrodynamic Diffraction, Stability, or Frequency Statistical analyses.

It is also possible to define the depth of the mud on the seabed onto which the cables lay. The default value is 2 meters (in SI and for a gravity value of 9.806m/s^2). See The SMUD Data Record - Seabed Mud Layer Depth for information on how it is defined.

Tube Drag Coefficients

While this option is set to Defined in Line Body Details, Aqwa calculates drag forces on Morison elements using the **Drag Coefficients** of the Line Body objects (via their associated Beam Section (p. 99) objects) in the Geometry. Alternatively, if set to **Reynolds Number Dependent**, drag coefficients will be calculated using the Wieselburger graph of drag coefficient versus Reynolds Number.



Scale Factor for Model Test Simulation

This option is only available while **Tube Drag Coefficients** is set to **Reynolds Number Dependent**. Set the **Scale Factor** to unity (default) to provide simple Reynolds Number-dependent drag calculations using the Wieselburger curve. If comparisons against scale-model tests are to be undertaken, then provide the scale factor of the model in this field (for example, if the scale-model is 1:10, then specify a factor of 10).

The scale factor (S_f) is used as follows:

$$\text{Local Reynolds Number} = \frac{\left(\frac{UD}{\nu}\right)}{S_f^{\frac{3}{2}}}$$

where:

U = local velocity transverse to the axis of the TUBE

D = diameter of the TUBE

ν = kinematic viscosity of water

Seabed Inline/Lateral Friction Coefficient

These options allow you to specify global inline and lateral seabed friction coefficients, which are applied for the calculation of seabed friction forces acting on any dynamic cables in a Time Response analysis.

5.2.2.2. Composite Cable Seabed Definition

In analyses employing Nonlinear Catenary Cables, it is possible to define a Composite Cable Seabed which may be used for any such cables that are expected to touch down. The Composite Cable

Seabed is never used in a Hydrodynamic Diffraction analysis, and is not used in any Hydrodynamic Response analysis that features only Linear, Nonlinear Polynomial, or Nonlinear Steel Wire Cables.

The behavior depends on whether a cable is dynamic or not.

5.2.2.2.1. For a Cable not Using Cable Dynamics

For a Nonlinear Catenary Cable which does not use Cable Dynamics, or in a Hydrodynamic Response analysis in which the **Use Cable Dynamics** option is set to No, the Composite Cable Seabed is used to create a quasi-static load/extension database for each Cable, which can take into account a **Global Seabed Slope**, if defined.

While **Seabed Type** is set to No Composite Cable Seabed, any Nonlinear Catenary Cables with Structure to Structure connectivity are treated as if they lie in infinite depth water. For Nonlinear Catenary Cables with Fixed Point to Structure connectivity, the seabed depth for each cable is considered as the depth of the selected Fixed Point.

Setting **Seabed Type** to Use Sea Geometry creates a horizontal Composite Cable Seabed at a depth equal to the Sea Geometry **Water Depth**. A Nonlinear Catenary Cable with Structure to Structure connectivity will only contact this seabed if the **Seabed Touchdown Expected** option of that cable is set to Yes. For Nonlinear Catenary Cables with Fixed Point to Structure connectivity, the depth of the Fixed Point must be equal to the Sea Geometry **Water Depth**.

Changing **Seabed Type** to Manual Definition allows the following options to be defined: the **Water Depth at Reference Point** option sets the depth of the seabed at the position specified by the **X-Position** and **Y-Position of Reference Point** fields, while the **Global Seabed Plane Azimuth** sets the (uphill) direction of the Composite Cable Seabed slope, and the **Global Seabed Slope** sets the slope angle.

5.2.2.2.2. For a Cable Using Cable Dynamics

When **Seabed Type** is set to No Composite Cable Seabed, a Nonlinear Catenary Cable with Structure to Structure connectivity may contact a horizontal seabed at the Sea Geometry **Water Depth** if any part of the line exceeds this depth. For Nonlinear Catenary Cables with Fixed Point to Structure connectivity, the seabed depth for each cable is considered as the depth of the selected Fixed Point.

Setting **Seabed Type** to Use Sea Geometry creates a horizontal Composite Cable Seabed at a depth equal to the Sea Geometry **Water Depth**. A Nonlinear Catenary Cable with Structure to Structure connectivity may contact this seabed if any part of the line exceeds this depth. For Nonlinear Catenary Cables with Fixed Point to Structure connectivity, the depth of the Fixed Point must be equal to the Sea Geometry **Water Depth**.

Changing **Seabed Type** to Manual Definition allows the following options to be defined: the **Water Depth at Reference Point** option sets the depth of the seabed at the position specified by the **X-Position** and **Y-Position of Reference Point** fields, while the **Global Seabed Plane Azimuth** sets the (uphill) direction of the Composite Cable Seabed slope, and the **Global Seabed Slope** is ignored, however, the Fixed Point of a Dynamic Nonlinear Catenary Cable with Fixed Point to Structure connectivity must still lie at the depth of the Composite Cable Seabed at the Fixed Point position.

The **Global Seabed Plane Azimuth** and **Global Seabed Slope** can be previewed by selecting **View Sea Surface and Composite Cable Seabed** from the View menu, or by clicking the corresponding icon from the View toolbar ().

5.2.3. Add Fixed Points

A Fixed Point is a non-moving attachment point defined as part of your geometry. Fixed Points are defined in the Details panel by entering coordinates in global space or specifying an offset from a vertex on a structure. The coordinates defining the Fixed Point can be parameterized. A Fixed Point does not move with any structure (even if it is initially defined as offset from a point on a structure). A Fixed Point can be used by more than one object (cable, fender, etc.), if required. The option to choose a Fixed Point vs. Connection Points on structures is controlled by the Connectivity field in the Details panel of the object using the attachments points.

To add a Fixed Point:

1. Select the **Fixed Points** object in the tree view.
2. Right-click the **Fixed Points** object and select **Add > Fixed Point**, or click the **Fixed Point** icon in the **Fixed Points** toolbar.
3. Click the Fixed Point object and configure it with one of the following sets of information:
   - Set **Definition of Position** to X, Y, and Z Coordinates, and set the Position Coordinates (**X**, **Y**, **Z**) in the Details panel.
   - Set **Definition of Position** to X and Y Coordinates, Z on Composite Cable Seabed, and set the Position Coordinates (**X**, **Y**) in the Details panel. The Z coordinate is automatically determined from the Composite Cable Seabed definition in Geometry, so the Composite Cable **Seabed Type** must be set to Use Sea Geometry or Manual Definition for this option to be used.
   - Set **Definition of Position** to Vertex Selection. Click **Select a Single Vertex** in the **Vertex** field, select a vertex on a structure, and click **Apply**. You can then set an **X Offset**, **Y Offset**, or **Z Offset** from the vertex if needed.

Note:

For a catenary cable connected to a Fixed Point, it is assumed that the seabed for that cable is at the same depth as the Fixed Point (even if this does not match the Water Depth defined in Geometry). A catenary cable will typically not lie lower than the depth of the Fixed Point, unless there is a local negative seabed slope, in which case the cable section can be lower than the anchor node location.

5.3. Mesh

If the external Mechanical Mesh component is part of your workflow on the Workbench Project Schematic, you will find that the Aqwa Workbench Mesh object is read-only, and any changes to the mesh must be made in the upstream Mesh component.

If you have imported an Aqwa input file using the **Insert from Aqwa Model Data File** operation, and there are no other active Parts in the model, then the Aqwa Workbench Mesh object is read-only. There is no function to modify the mesh imported from an Aqwa input file, currently.

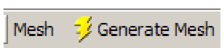
Otherwise, the options in the Details panel of the Aqwa Workbench Mesh object are applied to all un-suppressed Parts and Bodies in your model. Local Mesh Sizing objects can be added to override the global Mesh settings on selected Parts or Bodies.

Mesh elements are not generated for any suppressed Parts or Bodies, and they will not appear in any mesh-based results such as Pressures and Motions, Internal Tank Pressures, or Time Domain Pressures.

To generate the mesh:

1. Select the **Mesh** object in the **Outline** tree.

2. Right-click **Generate Mesh** or click the **Generate Mesh** button from the **Mesh** toolbar.



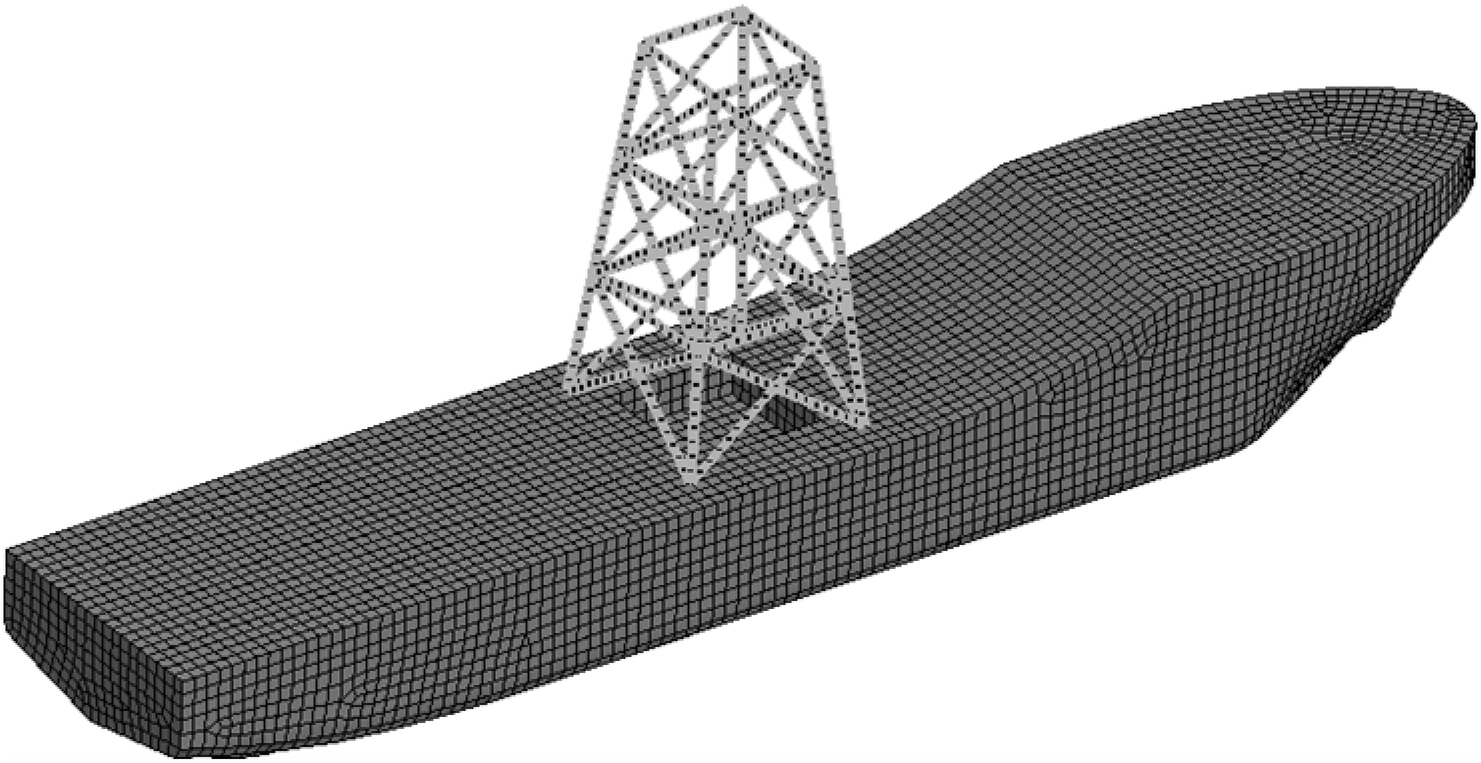
Note:

It is not yet possible to employ symmetry in Aqwa, hence the full model must be meshed.

The mesh controls available in the Aqwa Workbench editor are described in the following sections:

5.3.1. Global Mesh Options
5.3.2. Local Mesh Sizing Controls

Once a mesh has been generated successfully, the **Generated Mesh Information** section shows the Total Nodes and Total Elements in the mesh. These totals do not include elements that are generated by the Aqwa solver when a Part's **Generate Internal Lid** option is set to **Program Controlled**. By default, the Graphical window displays the mesh nodes and elements in a simple grey tone:

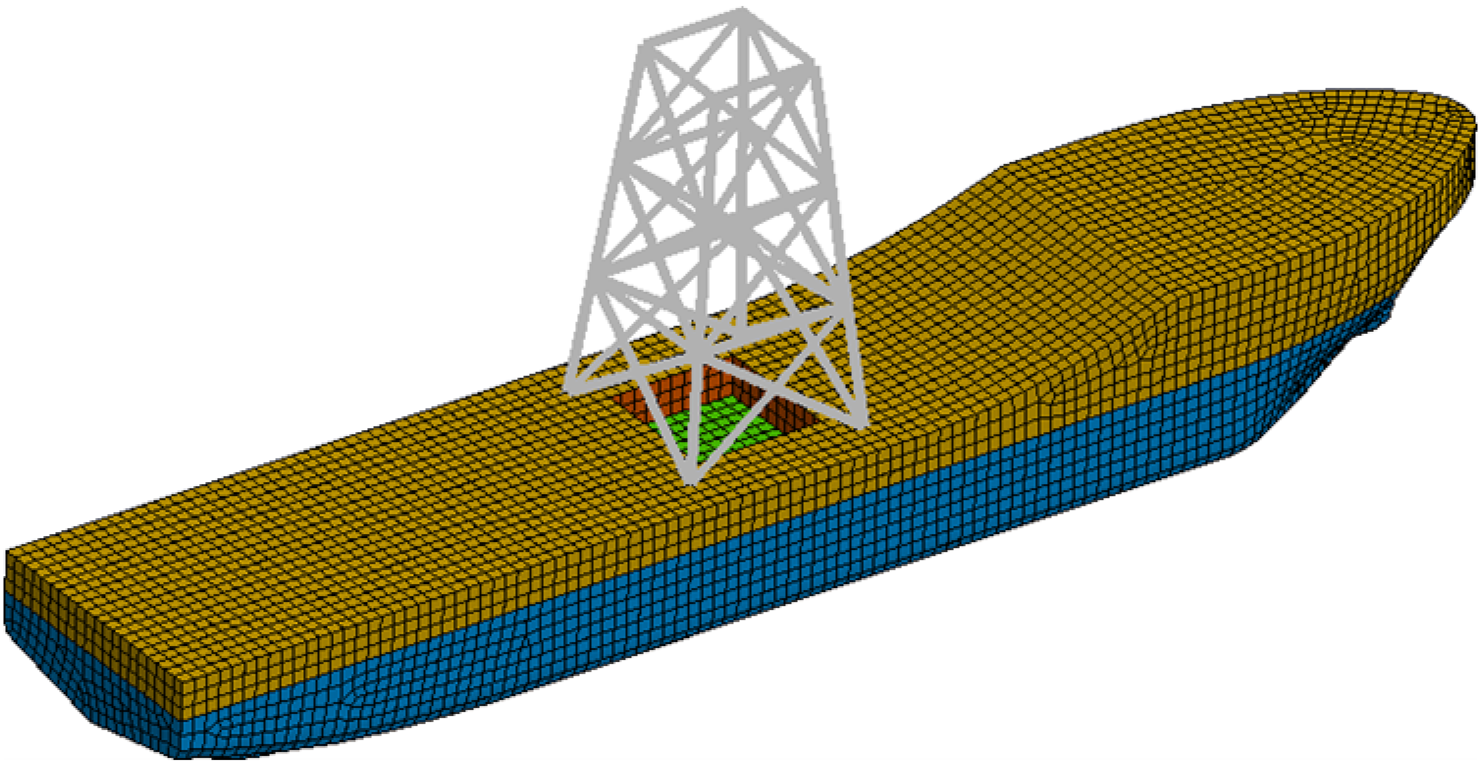


If you change the **Display Mesh Metric** option from **None** to **Element Types**, the **Generated Mesh Information** shows the breakdown of nodes and elements into:

- External Surface Diffracting Nodes
- External Surface Diffracting Elements
- External Surface Non-Diffracting Nodes
- External Surface Non-Diffracting Elements
- Internal Tank Diffracting Nodes
- Internal Tank Diffracting Elements
- Internal Tank Non-Diffracting Nodes
- Internal Tank Non-Diffracting Elements
- Abstract Lid Nodes
- Abstract Lid Elements
- Line Body Nodes
- Line Body Elements
- Field Points
- Moonpool Wall Surface Diffracting Nodes

- Moonpool Wall Surface Diffracting Elements
- Moonpool Wall Surface Non-Diffracting Nodes
- Moonpool Wall Surface Non-Diffracting Elements
- Moonpool Free and Matching Surfaces Nodes
- Moonpool Free and Matching Surfaces Elements

In the Graphical window, the elements are colored according to their type:



If you change the **Display Mesh Metric** option to **Failing Panels**, the Generated Mesh Information will display the **Diffracting Element Minor Failure Rate**. This is the percentage of diffracting panel elements that fail the mesh quality checks described in Mesh Quality Check, but whose failures can be avoided by turning on the **Ignore Modeling Rule Violations** option in the Hydrodynamic Diffraction Settings (p. 130). Generally, it is recommended that this percentage must be as small as possible. Reducing the Element Size usually helps, though naturally, this also increases the computational cost of the hydrodynamic analysis.

The Details panel also shows the breakdown of panel failures into:

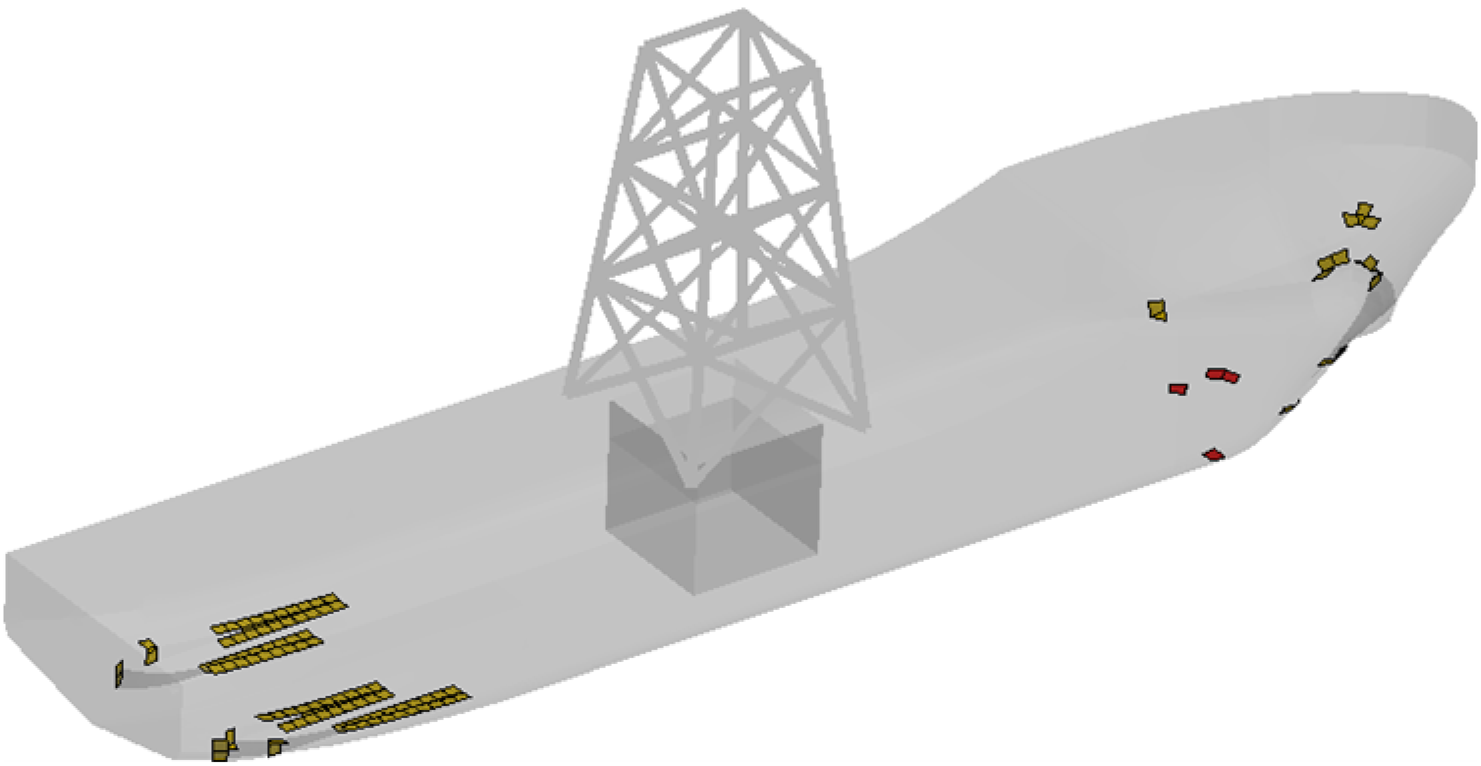
- Aspect Ratio Failures – where the aspect ratio of an element is less than 0.333
- Adjacent Element Area Ratio Failures – where the area ratio of adjacent elements is greater than 3
- Shape Factor Failures – where the shape factor of the element is less than 0.2
- Centroid Proximity Failures – where the distance between a pair of elements in the same interacting group is less than the effective radius of either of those elements

- Centroid Proximity Failures (FATAL) – as above, but with a stricter limit of 0.1x effective radius
- Element-Seabed Proximity Failures (FATAL) – where a diffracting element is less than half of its effective radius above the seabed
- Mesh Connectivity Failures (FATAL) – where an element edge is shared by more than two elements (so-called ‘T’ connectivity)
- Elements Cutting Fluid Surface (FATAL) – where diffracting elements cross the waterline or Internal Tank fluid level, as applicable

Note:

A field is not shown in the Details panel if there are no elements failing for that mesh quality check. You will not be able to run the Hydrodynamic Diffraction analysis if there are any fatally-failing elements in the mesh.

In the Graphical window, the elements suffering minor quality check failures are displayed in orange, while fatally-failing elements are shown in red, and all other elements are shown as transparent:



Note:

By default, the mesh information is displayed for all active Parts. However, where there are multiple active Parts in the Geometry, use the **Display Information For** option to display mesh information for a single Part.

5.3.1. Global Mesh Options

The following are global mesh options which are available in Aqwa Workbench:

5.3.1.1. Mesh Parameters

The overall density of the generated mesh is based on the Element Size parameter. The larger the Element Size is, the less accurate the results will be. However, the computational cost and memory requirement of the Hydrodynamic Diffraction calculation will scale with the square of the number of diffracting elements. This means that you may need to find a balance between accuracy and computational cost. The 64-bit version of the Aqwa solver is limited to 60,000 nodes and 40,000 elements, of which 30,000 may be diffracting.

Once the mesh has been generated, the **Maximum Allowed Frequency** field will display the maximum wave frequency that can be included in the Hydrodynamic Diffraction analysis. This is based on the Aqwa solver requirement of no less than 7 elements over the shortest wavelength in the hydrodynamic database.

5.3.1.2. Waterline Node Generation

Previously, it was mandatory to use the Geometry editor to split or slice your geometry over the waterline (at Z = 0 in the XY plane) to ensure that diffracting panel elements do not cross it. Internal Tank surfaces also had to be split or sliced at the internal tank fluid level. Now, you can generate a mesh which includes waterline and internal tank fluid level nodes automatically, by setting **Create Automatic Waterline Nodes** to **Yes**. When this option is on, you can adjust the Connection Tolerance between the water surface (or internal tank fluid surface) and the real vessel surfaces in your model, which is sometimes required to get a satisfactory mesh.

If the **Create Automatic Waterline Nodes** option is set to **No**, the meshing **Engine Selection** option is displayed. You can switch between **AnsMeshing** and **PRIME**, though in practice, the meshes created by these two engines are often similar. Without automatic waterline node generation, it is still required to split or slice your geometry at the waterline and internal tank fluid levels in the Geometry editor.

When **Create Automatic Waterline Nodes** is set to **No**, and the **Engine Selection** is set to **AnsMeshing**, the Aqwa Workbench mesh will be equivalent to a Mechanical Mesh with the **Physics Preference** set to **Hydrodynamics** and the **Batch Connections** option set to **No**. When using the AnsMeshing engine, you can adjust the Defeature Size to control how small details are treated in the mesh. For example, if a geometry detail is smaller than the Defeature Size, a single element may span over it. You can also choose to apply Advanced Options for sizing and defeaturing – for more information, see Sizing Group in the *Meshing User's Guide*.

When **Create Automatic Waterline Nodes** is set to **No**, and the **Engine Selection** is set to **PRIME**, the Aqwa Workbench mesh will be equivalent to a Mechanical Mesh with the **Physics Preference** set to **Hydrodynamics** and the **Batch Connections** option set to **Yes**. However, for an Aqwa analysis, you must still group rigidly-connected Bodies into a single Part with shared topology in the Geometry editor.

When **Create Automatic Waterline Nodes** is set to **Yes**, the PRIME meshing engine is always used. The Aqwa Workbench mesh will be equivalent to a Mechanical Mesh under these conditions:

- The **Physics Preference** is set to **Hydrodynamics**.

- The **Batch Connections** option is set to **Yes**.
- A mesh Connect object created between a flat face representing the water surface and the Surface Bodies in your model (with additional Connect objects for any Internal Tank Surface Bodies and their corresponding fluid surfaces).

In the configuration above, the fluid Surface Height (in Global Axes) for an Internal Tank can be defined directly and marked as an input parameter for a Design Point study.

Note:

There is no issue with turning on **Create Automatic Waterline Nodes** for a model, which has already been split or sliced at the waterline.

5.3.2. Local Mesh Sizing Controls

To add a Mesh Sizing object, follow these steps:

1. Select the **Mesh** in the **Outline** tree.
2. Right-click and select **Add Sizing Control**, or click the **Add Sizing Control** button from the Mesh toolbar.
3. Use the **Select Geometry** option to choose the Parts, Bodies or Faces desired for local Mesh Sizing.
4. Define the **Local Element Size** and **Local Defeature Size** for your geometry selection. The Local Element Size can be larger than the global Element Size defined in the Mesh details.

Any number of Mesh Sizing objects can be added to the tree as required. You cannot mix topology types in the Mesh Sizing selection – create separate Mesh Sizing objects, if you want to apply the same local sizing to surface bodies and line bodies.

Note:

Mesh Inflation Controls and Mesh Pinching Controls have now been withdrawn from Aqwa Workbench. Use the external Mesh component on the Project Schematic, if you need to apply more advanced mesh controls for your hydrodynamic analysis.

5.4. Add Project Information and Options

Selecting Project in the tree view displays basic project information and settings in the Details pane.

Details of Project

For convenience and identification, you can record basic details such as the **Author**, **Reference**, **Project Title**, and **Description**. The **Data Root Folder** shows where the project files are stored.

The **Hydrodynamic Solver Unit System** field allows you to change the unit system that the Aqwa solver will use for the solution. This is independent of the units currently defined in the Aqwa editor for input and results presentation. A selection of consistent Metric and U.S. Customary unit systems is available. Note that modifying this field will clear any existing Solution. The default is Metric (kg, m) units, with the derived force unit as Newtons.

The **Export CSV Separator Definition** field allows you to specify the field separator to use when exporting. The default option corresponds to your Windows Regional settings. The Manual Definition option allows you to manually enter a single non-digit value as the **Separator Value** to be used in the project when exporting.

Date Details

The **Date of Creation** and **Last Modified** fields indicate when a project was originally created and when it was last saved.

Graphics Size Factors

Objects displayed in the graphical window that are not sized according to their defined geometry are scaled appropriately for a large seafaring vessel or offshore installation.

In same cases, the default object sizing may be too large or too small for the given geometry. The **Graphics Size Factors** allow you to re-scale the objects. There are four controls:

- **Global Control** applies to all objects.
- **Geometric Features** applies to Fixed/Connection Points, Point Masses/Buoyancies, and Structure Axes.

  Note:

  Graphics properties such as size and color for the Fixed/Connection Points and Point Masses can be customized in the Graphics properties of each object's Details panel. However, if these Points are further used as a certain Connection type (such as a Connection Failure or a pulley), they will revert to their default color set in the Aqwa Editor. Their customized size will not be affected.

- **Connections** applies to Cable Thicknesses, Joints, Fender Contact Planes, and Joint/Fender Axes.
- **Environment Features** applies to Wave Directions, Current, Wind, Waves, and Structure Forces.

Each control consists of a slider with a range of 1-100 and a default of 50. Figure 5.1: Objects Drawn with Default Sizing (p. 54) and Figure 5.2: Objects After Re-Scaling (p. 54) show how the controls can be used to improve the display of a 1.5 m x 0.8 m buoy with catenary moorings in 20 m of water.

Figure 5.1: Objects Drawn with Default Sizing

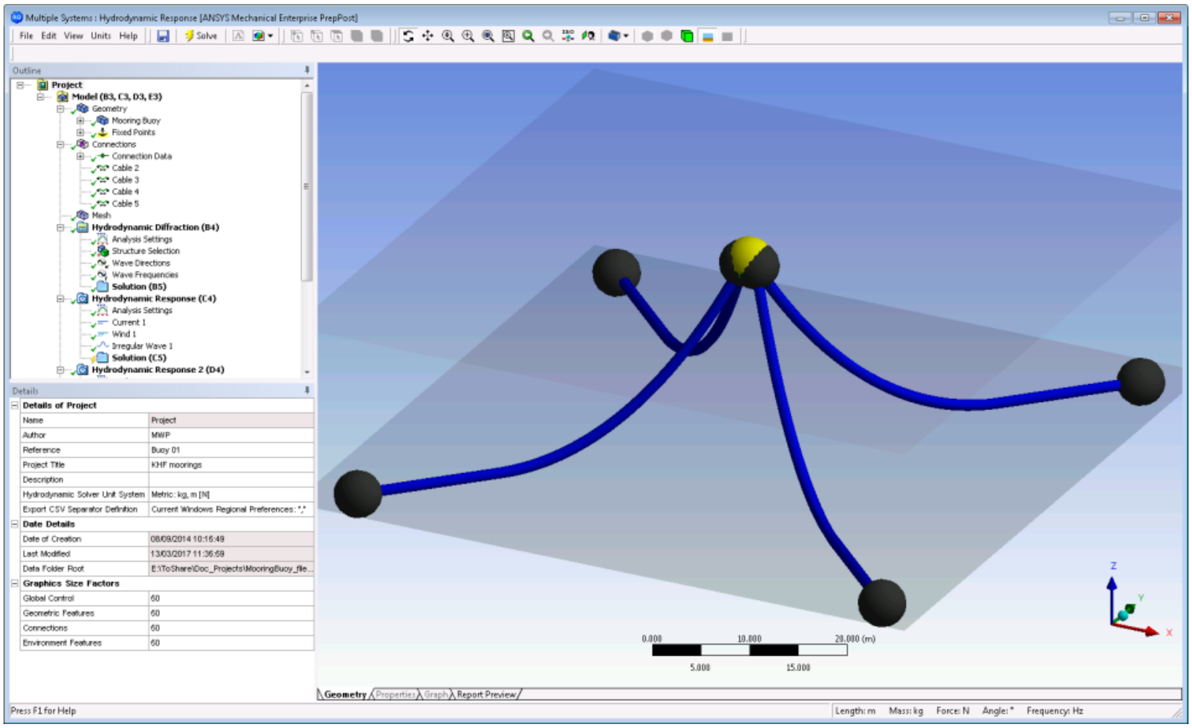
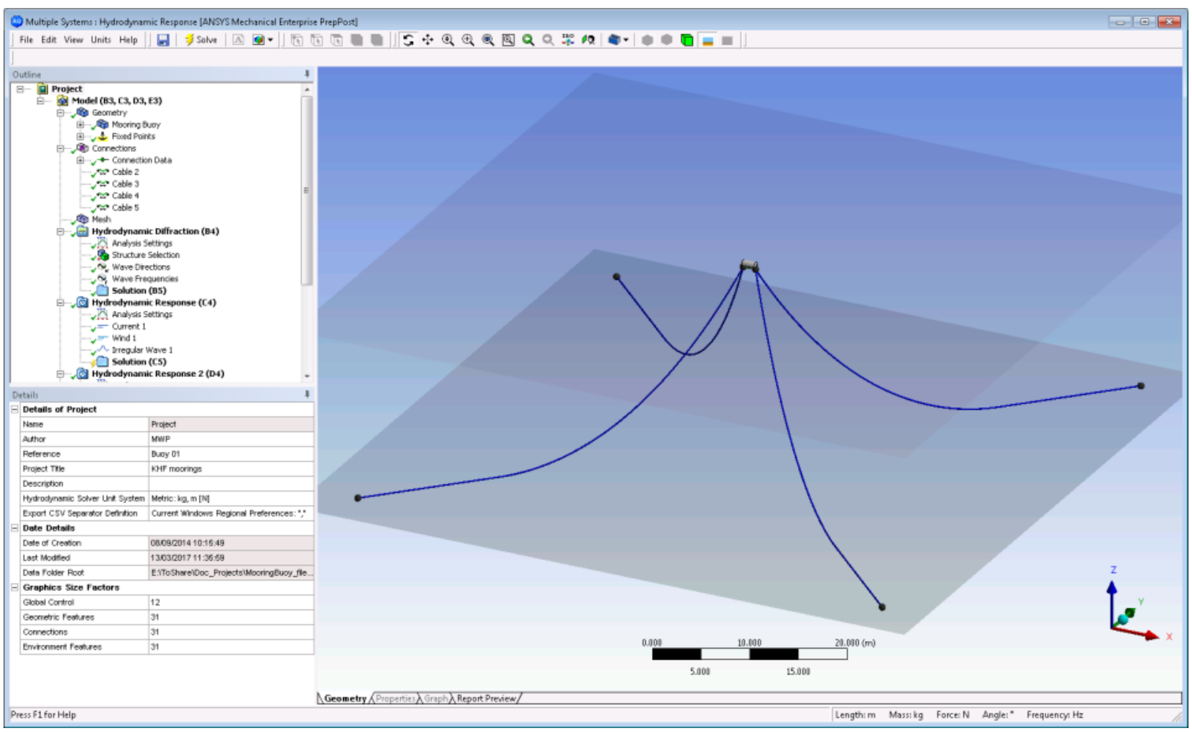


Figure 5.2: Objects After Re-Scaling



5.5. Define Parts Behavior

A part is a group of geometric entities that form a ship or other structure that is to be analyzed in Aqwa. The name is read in from the geometry database and the graphical view will show the part; the appropriate structure will be highlighted when the part in the tree is selected. Each part will be assigned a structure number for the analysis. The parts can be included or excluded from the analysis using the Structure Selection (p. 143).

For each imported part, a set of local structure axes is created. These axes are drawn at the part's center of gravity.

A number of options can be set for each part in the Details panel.

To help visualization, it is possible to show or hide specific parts using the **Part Visibility** option, while the **Part Color** option can be used to define the color of the surface bodies which make up each part. The **Part Activity** option is used to decide what structures are used in the analysis.

Note:

If a part is suppressed, it cannot be used in the analysis. However, when a part is unsuppressed, it must be added to the Structure Selection (p. 143) in order for it to be included in the analysis.

The Mass Properties section displays the **Total Mass**, the Center of Gravity **X**, **Y**, and **Z**, and the **Moment of Inertia Ixx**, **Ixy**, **Ixz**, **Iyy**, **Iyz**, and **Izz** for the Part. If the Part includes Internal Tanks, the **Total Structural Mass** and **Total Internal Tank Mass** will also be displayed; the **Center of Gravity** shown in the Details panel will be the combined center of gravity including the internal tank fluid. If the Part includes at least one Program Controlled Point Mass and the Mesh is not up-to-date, the **Total Mass** field will display Generate Mesh to Update and all other Mass Properties will be hidden.

If an internal lid is required to prevent irregular frequency problems then **Generate Internal Lid** can be set to Yes and it will be automatically generated during the Aqwa analysis. Note that an automatically generated lid will not be displayed. When **Lid Element Size Definition** is set to Program Controlled, the size of the internal lid elements will be set to the mean mesh size of the corresponding structure. Alternatively, the **Lid Element Size** can be defined manually by setting **Lid Element Size Definition** to Manual Definition.

A manually generated lid may also be used: create an appropriate plane surface as a Surface Body (p. 58); set **Structure Type** to Abstract Geometry and **Abstract Type** to Internal Lid. If you have a structure with a Moonpool where large resonant waves may occur, then you can form an external lid using a predefined geometry Surface Body with **Structure Type** set to Abstract Geometry and **Abstract Type** to External Lid.

Note:

The lid normal should be pointing up (away from the fluid). Aqwa will account for an improperly defined normal but will generate a warning.

Hull drag loads may either be accounted for by defining a 6 x 6 matrix of Morison Hull Drag Coefficients, or by including a table of Current Force Coefficients. Where Morison Hull Drag Coefficients are used, the current velocity is always measured at the structure center of gravity. Where Current Force Coefficients are used, a **Current Calculation Position** should be defined, at which the current velocity is measured for the hull drag loading calculation. When the **Current Calculation Position** is set to At Fixed Depth, a constant **Current Calculation Depth** can be set. Alternatively, when the **Current Calculation Position** is set to Moves with Structure, the **Current Calculation Position Definition** can be switched between

Vertex Selection and Manual Definition. If the position definition is set to Vertex Selection, you should select a **Current Calculation Vertex** from the geometry and (optionally) define a **Vertex X**, **Y**, and **Z Offset**. The **Vertex X**, **Y**, and **Z Position** will be read-only. If the position definition is set to Manual Definition, you should enter the **Current Calculation X**, **Y**, and **Z Position** directly.

By default the structure is set to be free to move. Alternatively, the whole structure can be fixed by setting **Structure Fixity** to Structure is Fixed in Place. Fixity primarily affects the results of a hydrodynamic diffraction analysis by affecting the structure's RAOs. It therefore also has an effect on a hydrodynamic time response result as the calculated drift forces depend on these RAOs. It is therefore necessary to impose the coherence between the setup of the two analyses by fixing the structure in the time response analysis. Since you can create joints to be used in a time response analysis, it is your responsibility to create a rigid joint when it is connected to a fixed point on any Part marked as Fixed in the Details dialog. Not doing so will result in an error when solving the time response analysis.

The **Mass Multiplying Factor** and **Drag Multiplying Factor** provide a way of modifying the added mass and drag coefficients defined for any Line Bodies (via their associated Beam Section objects) and Discs associated with this part. The Added Mass and Transverse Drag are multiplied by these factors after they have been calculated by Aqwa (the total coefficient applied for each quantity is the cumulative product of the factor specified by these settings with the factor specified for each Line Body or Disc). These factors may be used for parametric studies where the effects of Morison drag on Line Bodies and Discs are considered important (for example, simulating tests at model scale). These factors have no effect on any other object type in the part.

The **Slam Multiplying Factor** provides a way to enable the computation of slamming loads on Line Bodies. By default a factor of zero is specified which disables this computation. Any positive non-zero value will cause the program to compute the slam coefficient for each Line Body, based on the premise that the slam force is equal to the rate of change of the added mass tensor (with time) multiplied by the velocity. The resulting coefficient is then multiplied by this factor. This may be used for parametric studies where the effects of slamming loads on Line Bodies are considered important (for example, simulating tests at model scale).

Note:

Slamming loads are not calculated on Morison Disc elements, or on any other object type in the Part.

Slamming loads can also be included for slender tube elements by setting the Slam Factor to a positive non-zero value, but the magnitude of the factor is immaterial in this case because a value of unity is always employed in the analysis.

Note:

The method for computing the slam coefficient requires that the time-step used in a time history analysis must be sufficiently small to accurately represent the added mass at each stage of immersion/emergence. In general this will depend on the geometry of each element and its orientation to the water surface. In practice, this severe restriction of the size of the time-step means that this facility is only used when specifically investigating the effects of slam forces on individual elements during critical stages of the simulation period, as the momentum change due to slam forces are normally small and have little effect on the overall motion of the structure.

In analyses containing a single structure only, you may calculate the (static or RAO-based) shear forces and bending moments (p. 205) acting on that structure under different wave load conditions. Such calculations require the **Neutral Axis** of the structure to be defined, which can be any one of the Fixed Reference Axes (FRA) as well as the position of this axis in the perpendicular plane. By default, the Neutral Axis is assumed to pass through the center of gravity (COG) of the structure. Changing **Neutral Axis Position Definition** from Through COG to Manual Definition allows you to define the position manually. For example, if **Neutral Axis** is set to Global X, you should also specify the positions **Neutral Axis Y** and **Neutral Axis Z**.

Submerged Structure Detection is Program Controlled by default, and Aqwa will detect the highest point (greatest Z coordinate) and check whether it is below the water level; alternatively this automatic detection can be overridden.

The Metacentric Heights can be overridden about both the global X (**Override Calculated GMX** = Yes) or Y (**Override Calculated GMY** = Yes) axes to modify the hydrostatic stiffness of the vessel. When these are overridden, Aqwa first calculates the hydrostatic stiffness matrix based only on the cut water plane and displaced volume properties. It then adjusts the second moments of area IXX, IYY and recalculates its associated properties, PHI (principal axis), GMX/GMY, BMX/BMY etc. to give the required GM values. The associated additional hydrostatic stiffness is calculated automatically and stored in the hydrodynamic database. If the GM value input is less than that based on the geometry alone, the resulting additional stiffness will be negative. This would be the case if ballast tanks were being modeled, making the structure less stable, statically.

A number of Aqwa-specific elements and properties may be added to a Part:

- 5.5.1. Surface Body
- 5.5.2. Line Body
- 5.5.3. Point Mass
- 5.5.4. Distributed Mass
- 5.5.5. Point Buoyancy
- 5.5.6. Disc
- 5.5.7. Internal Tank
- 5.5.8. Moonpool
- 5.5.9. Additional Hydrodynamic Stiffness
- 5.5.10. Additional Added Mass (Frequency Independent)
- 5.5.11. Additional Damping (Frequency Independent)
- 5.5.12. Nonlinear Roll Damping
- 5.5.13. Current Force Coefficients
- 5.5.14. Wind Force Coefficients
- 5.5.15. Morison Hull Drag Coefficients
- 5.5.16. Yaw-Rate Drag
- 5.5.17. Structure Connection Points
- 5.5.18. External Lid
- 5.5.19. Part Transform

You can remove any of these objects by right-clicking them in the tree and selecting **Delete** from the context menu.

5.5.1. Surface Body

Surface bodies are areas that can be meshed (p. 46) to create diffracting or non-diffracting elements for the Aqwa analysis. The name of the surface body will be obtained from that given in DesignModeler/Discovery Modeling.

If a body is not required for the analysis it can be suppressed (**Body Activity**). Suppressed bodies will not be meshed and will be excluded from the analysis. You can hide the body in the graphic window (**Body Visibility**), in which case it will not be shown but will be included in the analysis. By default, each surface body inherits its color from the parent part. You can override this behavior by changing **Body Color Definition** to Manual Definition, which allows you to set the **Body Color** manually.

It is possible to change the type of surface from a Physical Geometry to an Abstract Geometry (**Structure Type**). For physical geometry, Program Controlled **Surface Type** will set all surface bodies below the water surface as diffracting and those above will be non-diffracting. If required, those below the water surface can be manually defined as non-diffracting elements for the analysis. This may be required, for instance, when part of the structure is in contact with the sea bed, or where contact occurs underwater between adjacent parts.

For an abstract geometry, **Abstract Type** provides a number of options to select how this geometry is to be used. If an area is of particular interest, then the Field Point Positions option enables a mesh to be applied and each node of the mesh will form a field point element; additional information will be available at these points. Alternatively, Internal Lid and External Lid can be used to suppress standing waves either between structures or within structures.

Note:

Automatic internal lids can be selected using the Part (p. 54) option.

If the generation of an external lid is specified, two additional parameters are required. The first is a **Lid Damping Factor**, set between 0 and 1. The factor represents how effective the lid is to be; 0 will result in no effect, while 1 will prevent any vertical water surface velocity under the lid.

The second parameter is the **Gap for Lid**. It is a representative size for the lid; typically the distance between the two vessels or the width of a moon-pool. It enables the lid properties to be tuned to the resonant frequency of waves in the gap.

5.5.2. Line Body

Line Bodies are used to create line elements for Aqwa. How they are interpreted depends on the cross section of the line; if it has a circular or circular tube cross section, then it will be automatically converted into a standard tubular (TUBE) element. All other sections with dual symmetry will create slender tube (STUB) elements. Cross sections which are not dual-symmetric are not permissible in Aqwa, as the centroid and geometric center of the cross section are required to be coincident.

Beam Properties

Each Line Body should be assigned a cross section and an orientation in DesignModeler/Discovery Modeling. In the Aqwa editor, when importing the geometry, each valid cross sectionis automatically paired with a Beam Section (p. 99) object. Beam Sections are created under the Line Body Data (p. 99) object in the Outline tree. Apart from storing the geometric properties of the cross section, a Beam Section object also allows mass and hydrodynamic properties to be defined.

Once the geometry has been imported, the name of the cross section assigned to a Line Body in the geometry editor is provided for reference in the **Cross Section** field of the Line Body details.

Each Line Body has a **Beam Section** associated with it. The Line Body inherits geometric, mass and hydrodynamic properties from the Beam Section. The cross section storedin the Beam Section must match the **Cross Section** of the Line Body.

Where multiple Line Bodies have the same cross section but different mass or hydrodynamic properties, a Beam Section can be Duplicated. The duplicate Beam Section will retain the same cross section geometry, but can have different mass or hydrodynamic properties defined. In the Line Body details, the **Beam Section** field can then be used to select which Beam Section the Line Body inherits its properties from.

Note:

It is not possible to change the geometric properties of a Line Body in the Aqwa editor. To do his you must edit the cross section in DesignModeler/Discovery Modeling and re-import the geometry.

For Line Bodies with circular tube cross sections, the **Tube Type** field allows you to make the tube Sealed or Floodable. By default, the tube is sealed, and in this case the tube is buoyant; however, the tube does not have longitudinal drag or added mass unless discs are created at the ends.

For Line Bodies with circular or circular tube cross sections, discs can automatically be applied at either (Created at End A Only, Created at End B Only) or both (Created at Both Ends) ends with the **Tube End Discs** option; if you choose one of these options, default disc parameters are used. If you require different parameters, discs can be excluded here and added manually. You cannot automatically add discs to Line Bodies with non-circular cross sections, but you can add them manually if needed.

Note:

All non-circular cross sections are graphically represented as solid rectangles in Aqwa.

Beam Mass Properties

The mass properties of the Line Body (**Mass** and **X, Y, Z Position of Body COG**) are displayed for information. The mass of the Line Body is determined from the cross section area and material density of its associated **Beam Section**, and its center of gravity is calculated from the positions of the nodes along the Line Body length.

Line Body Axes

The Local Tube Axes (LTA) are defined with their origin at the position of the first node of the element. The LTA X-axis points from this origin toward the second node of the element. If the orientation of the Line Body has not otherwise been defined in DesignModeler/Discovery Modeling, the LTA Y-axis is parallel to the Fixed Reference Axes (FRA) XY plane, and at right-angles to the LTA X-axis; where the LTA X-axis is vertical in the FRA, the LTA Y-axis is parallel to the FRA Y-axis. The LTA Z-axis follows the right-hand rule. For more information, see Morison Equation in the *Aqwa Theory Manual*.

5.5.3. Point Mass

Surface bodies do not have a mass directly associated with them. In order to include the mass effects of a surface body, one or more point mass elements should be included.

To add a Point Mass element:

1. Select a Part in the tree view.

2. Right-click the Part and select **Add > Point Mass**, or click the **Point Mass** icon in the **Parts** toolbar.

3. Select the **Point Mass** object in the tree and set the properties manually, or select Program Controlled.

If **Mass Definition** is set to Manual, you must input all the properties for the point mass (**Mass**, **X**, **Y**, and **Z Position** coordinates)

Alternatively, if a Program Controlled point mass is used, the mass and the horizontal position are calculated from the panel elements and any manually-defined point masses, distributed masses, point buoyancies, or Line Bodies in the structure. The total structure mass equals the mass of water displaced, and the horizontal position of the Program Controlled point mass is calculated such that the structure center of gravity lies on the same vertical line as the center of buoyancy.

If the point mass is Program Controlled, its horizontal position and mass will not be displayed until a mesh has been generated; the Mass field will show an informational message Generate Mesh to Update.

The moments of inertia (or radii of gyration) and vertical position **Z Position** cannot be determined by the program and must always be input. Moments of inertia can be defined directly or by inputting radii of gyration. If **Define Inertia Values By** is set to via Radius of Gyration, you need to enter **Kxx**, **Kyy**, and **Kzz**. If you select Direct Input of Inertia, you must enter the **Ixx**, **Iyy**, and **Izz** values. The off-diagonal terms **Ixy**, **Ixz**, and **Iyz** may optionally be entered.

If the point mass is Program Controlled, the inertia properties which are not entered by the user will not be displayed until the **Mass** has been updated. For example: if **Define Inertia Values By** is set to Radius of Gyration, the inertia components **Ixx**, **Iyy**, and **Izz** will not be shown until the mass is determined.

5.5.4. Distributed Mass

If you intend to perform calculations of Shear Forces/Bending Moments or Splitting Forces for a Part, it is necessary to define a distribution of mass across that Part. You can do this by defining multiple Point Masses, but it is usually more convenient to create a **Distributed Mass** object.

A mass distribution can be defined directly in a table, or by importing the information from an existing .MSD or .SFM file.

To add a Distributed Mass:

1. Select a Part in the tree view.

2. Right-click the Part and select **Add** → **Distributed Mass**, or click the Distributed Mass icon in the **Parts** toolbar. To define data directly in a table, select **Add** → **Distributed Mass** → **Manual Input**. To read data from a file, select **Add** → **Distributed Mass** → **Import MSD File** or **Add** → **Distributed Mass** → **Import SFM File**.

Note:

After the data is imported from a `.MSD` or `.SFM` file, no record of the file used to import the data is retained.

The **Distributed Mass Definition Data** table allows you to define the mass and inertia properties of any number of mass contributions across the Part. Each contribution is defined by its:

- Mass
- X/Y/Z Start, defining the minimum location of the mass contribution
- X/Y/Z CoG, defining the center of gravity of the mass contribution
- X/Y/Z End, defining the maximum location of the mass contribution
- Inertia terms Ixx, Ixy, Ixz, Iyy, Iyz, Izz

Each mass contribution is evenly distributed over its defined extents (from X/Y/Z Start to X/Y/Z End).

The usage of this data depends on the type of calculation that you are performing. For a Splitting Forces calculation all of the mass, center of gravity, and inertia terms are used, but the extents of the mass contribution (X/Y/Z Start and X/Y/Z End) are not relevant. For a Shear Forces/Bending Moments calculation the extents are used, but only the inertia term about the Part’s Neutral Axis is required.

Note:

If the **Calculate Shear Force/Bending Moment** option under the Details of the Part is set to No, the extents of each mass contribution (X/Y/Z Start and X/Y/Z End) will be hidden in the Distributed Mass Definition Data table, as only Splitting Forces calculations will be permitted.

5.5.4.1. Importing from a `.MSD` File

When a mass distribution is imported from a `.MSD` file, that file will also contain a neutral axis direction which defines how the imported data is interpreted. The data will only include the extents of each mass contribution in the neutral axis direction, and the principal inertia term for each contribution about that neutral axis. The extents in other directions will be assumed from the dimensions of the Part, and other inertia terms should be defined manually.

The unit system of the data in the `.MSD` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data** and **Mass Unit for Imported Data** to the required unit system.

5.5.4.2. Importing from a `.SFM` File

When a mass distribution is imported from a `.SFM` file, the data will only include the mass, center of gravity position and inertia terms for each mass contribution. The extents of each mass contribution will be assumed from the dimensions of the Part.

The unit system of the data in the `.SFM` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data** and **Mass Unit for Imported Data** to the required unit system.

5.5.4.3. Inputting Data Manually

Distributed Mass data can be entered directly in the Distributed Mass Definition Data table, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension
- Contains values separated by commas, tabs, or single spaces (not multiple spaces)
- Contains exactly 16 columns, which are (1) Mass, (2-4) X/Y/Z Start, (5-7)X/Y/Z CoG, (8-10) X/Y/Z End and (11-16) Inertia terms Ixx, Ixy, Ixz, Iyy, Iyz, Izz

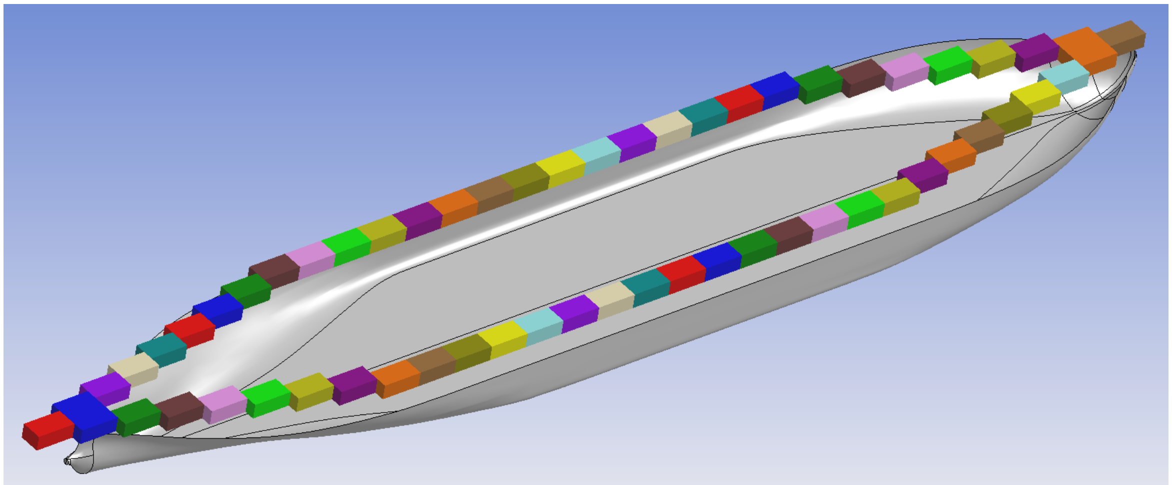
The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data** and **Mass Unit for Imported Data** to the required unit system.

Note:

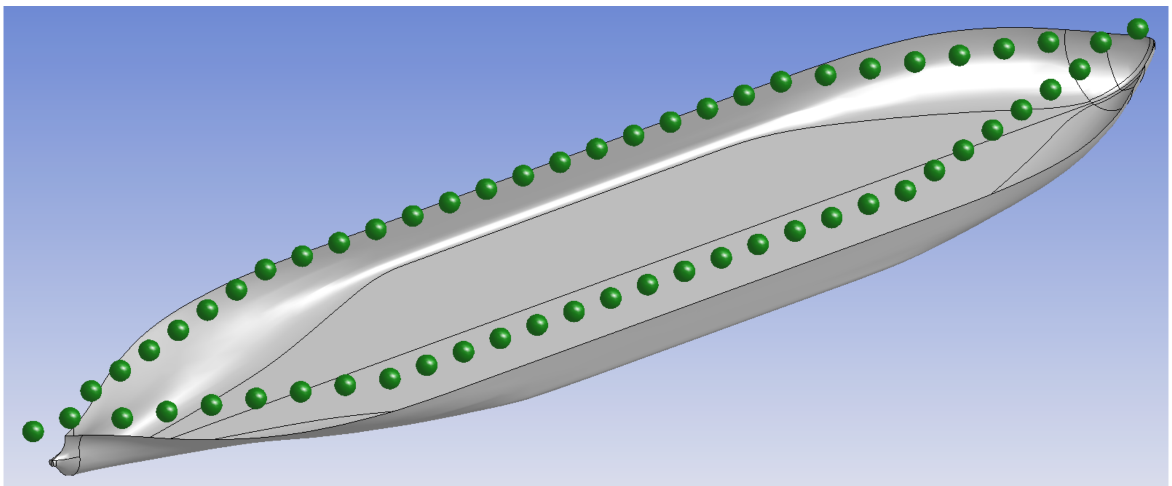
Imported `.CSV` data should still include columns for mass contribution extents (X/Y/Z Start and X/Y/Z End) even if the Part's **Calculate Shear Force/Bending Moment** option is set to No.

5.5.4.4. Representation in the Graphical Window

The graphical representation of a Distributed Mass depends on the Part’s **Calculate Shear Force/Bending Moment** option. If this is set to Yes, each mass contribution will be drawn as a colored cuboid over its defined extents (X/Y/Z Start and X/Y/Z End):



If the Part's **Calculate Shear Force/Bending Moment** option is set to No, each mass contribution will be drawn as a dark green sphere at its defined center of gravity position:



5.5.5. Point Buoyancy

If additional buoyancy is required for a part, then Point Buoyancy elements may be included.

To add a Point Buoyancy element:

1. Select a Part in the tree view.
2. Right-click the Part and select **Add > Point Buoyancy**, or click the **Point Buoyancy** icon in the **Parts** toolbar.
3. Select the **Point Buoyancy** object in the tree and set the properties. Point Buoyancy objects require a position (**X**, **Y**, **Z**) and a **Volume**.

5.5.6. Disc

Disc elements can be used to create an area that has drag and added mass in the direction perpendicular to the disc.

To add a Disc element:

1. Select a Part in the tree view.

2. Right-click the Part and select **Add** > **Disc**, or click the **Disc** icon in the **Parts** toolbar.

3. Select the **Disc** object in the tree and set the properties.

The **Diameter** of the disc is required along with the centroid and the definition of the normal direction. If the centroid is at the position of an existing vertex, set **Centroid Definition** to Select Vertex and click Pick in the **Vertex** field; then select the vertex on the model and click **Apply**. To enter the co-ordinates of the vertex directly, set **Centroid Definition** to Specify Coordinates, and enter the **X**, **Y**, and **Z** values.

You can specify the normal by picking a second vertex or specifying the direction of a normal vector. To use an existing vertex to define the normal, set **Normal Definition** to Select Second Vertex and click **Pick** in the **Normal Vertex** field; then select the vertex on the model and click **Apply**. To enter the vector for the normal directly, set **Normal Definition** to Specify Vector Components, and enter the **Normal X**, **Normal Y**, and **Normal Z** component values.

The default values of the added mass (**Added Mass Coefficient**) and viscous drag coefficients (**Drag Coefficient**) can be modified if desired. See Properties of Typical Morison Elements in the *Aqwa Theory Manual* to see how these coefficients are used in the program.

Note:

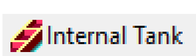
Drag is not used in Hydrodynamic Diffraction (p. 130) or Frequency Statistical Analyses (p. 140) unless the **Linearized Morison Drag** option is used from the **Common Analysis Options**.

5.5.7. Internal Tank

Internal tanks are filled with fluid for storage purposes or stability, particularly in marine structures such as ships and submarines. Ballast tanks, oil tanks and fresh water tanks are examples of internal tanks. Aqwa Workbench can define and simulate internal tanks for hydrostatic and hydrodynamic analyses.

To include an Internal Tank object in your model:

1. Select a Part in the tree view.

2. Right-click the Part and select **Add** → **Internal Tank**, or click the **Internal Tank** icon in the **Parts** toolbar 

.

3. Select the **Internal Tank** object in the tree.

4. Select a Fluid Level Definition Mode (p. 65) and set the properties described in Internal Tank Properties (p. 67).

You must define the Internal Tank geometry by selecting the **Internal Tank Surfaces** in the Graphical Window, where those surfaces must be selected from the Surface Bodies that make up the associated

Part. Both the diffracting and non-diffracting internal tank surfaces should be selected here. The surface normals should point inwards into the internal tank fluid.

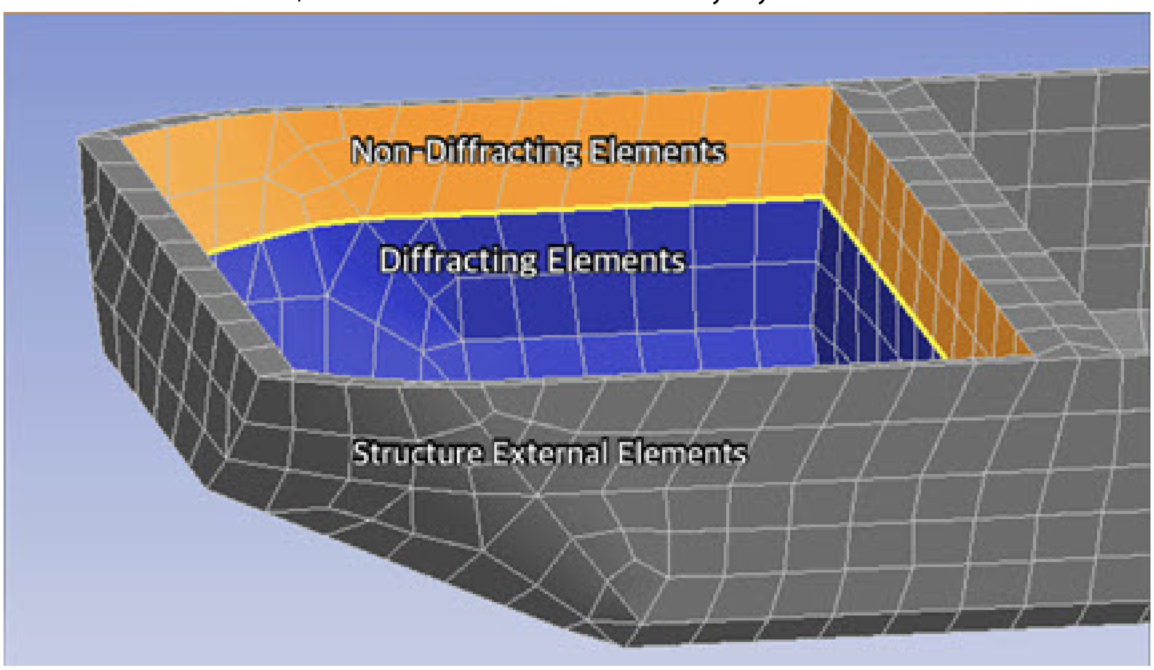
At this point, the selected Surface Bodies are now linked with the Internal Tank. The name of the Internal Tank is appended to the name of the Surface Body in the tree view to indicate this. If you change the **Activity** of the Internal Tank, the activity of any linked Surface Bodies will also be changed. If the Surface Bodies are deselected from the **Internal Tank Surfaces** or the Internal Tank object is deleted, this linkage is removed.

5.5.7.1. Fluid Level Definition Mode

The Internal Tank definition requires you to specify the level of the fluid surface inside the tank. Previously, it was mandatory to use the Geometry editor to split or slice your internal tank surfaces at the fluid level, to ensure that diffracting panel elements do not cross it. However, you can now generate a mesh which automatically includes internal tank fluid level nodes for some modes. The corners of the elements are known as nodes, so the nodes on the fluid level are named as the fluid level nodes. In relation to the fluid level, note that all mesh nodes:

- Must lay on or below the fluid level for diffracting elements.
- Must lay on or above the fluid level for non-diffracting elements.

This is illustrated below, where the fluid level is shown by a yellow line:



The following are the various modes you can set to determine the fluid level within the internal tank:

Edge Selection

This defines the fluid surface height by picking a horizontal edge in the Graphical Window, from the selected internal tank geometry. The Aqwa Workbench editor uses the vertical position of the edge as the fluid surface height. You can define the position of the internal tank fluid free surface by selecting an edge at the interface between the diffracting and non-diffracting surfaces of the internal tank, using the **Fluid Level Definition**. You do not need to select a closed loop of edges; a single edge is sufficient. The selected edge must be horizontal (parallel to the XY plane) and must be connected to at least one of the selected **Internal Tank Surfaces**. The **Surface Height (in Global Axes)** field shows the Global Z position of the free surface.

To set **Edge Selection**:

1. Select your Internal Tank of choice in the Graphical Window.
2. Click **Apply** for **Internal Tank Surfaces** in the Details Panel.
3. Change the **Fluid Level Definition Mode** to **Edge Selection**.
4. Click **Fluid Level Definition**, select an edge on the model, and click **Apply**.

Note:

Once a surface is associated with the Moonpool, it stays linked to this object, whether it is supressed or not. If you want to use the surface as part of your active model for further analysis, you must release this surface from your Moonpool object by carrying out a new surface selection that excludes this particular surface. Then, your surface becomes ready for further use, and you can suppress your Moonpool object.

Surface Height

This measures the vertical depth of the fluid in the internal tank. You can set the **Fluid Level Definition Mode** to **Surface Height** and enter a value directly into the **Surface Height (in Global Axes)** field. This can also be marked as an input parameter for a Design Point study.

To set **Surface Height**:

1. Select your Internal Tank of choice in the Graphical Window.
2. Click **Apply** for **Internal Tank Surfaces** in the Details Panel.
3. Change the **Fluid Level Definition Mode** to **Surface Height**.
4. Provide a numerical value in the **Surface Height (in Global Axes)** field.

Note:

To automatically include fluid level nodes, set **Mesh>Waterline Node Generation>Create Automatic Waterline Nodes** to **Yes** before setting the **Fluid Level Definition Mode** to **Surface Height**.

Fluid Volume

This capability allows you to set a target fluid volume for the internal tank. When a value is entered in the **Fluid Volume (Target)** field, Aqwa Workbench produces an estimate in the **Surface Height (in Global Axes)** and **Total Capacity (Estimated)** fields. This mode considers the varying shapes of marine structures which can make it more complex to calculate the surface height of the fluid level.

To set **Fluid Volume**:

1. Select your Internal Tank of choice in the Graphical Window.
2. Click **Apply** for **Internal Tank Surfaces** in the Details Panel.
3. Change **Fluid Level Definition Mode** to **Fluid Volume**.
4. Provide a numerical value in the **Fluid Volume (Target)** field.

Fluid Volume can also be set as a parameter when observing the hydrostatic behavior and hydrodynamic properties of marine structures to measure their stability. This can be done by setting the **Fluid Volume** as an input parameter and the **Metacentric Height** as an output parameter, such that the **Metacentric Height** is directly affected when the Fluid Volume is modified.

You can find the Parameter Set in the **Project Schematic**. The **Outline of All Parameters** displays a table which contains all defined input and output parameters. The **Table of Design Points** enables you to list multiple values under **Fluid Volume** and Aqwa Workbench will produce the corresponding metacentric heights for each of these values.

Note:

To automatically include fluid level nodes, set **Mesh>Waterline Node Generation>Create Automatic Waterline Nodes** to **Yes** before setting the **Fluid Level Definition Mode** to **Fluid Volume**.

5.5.7.2. Internal Tank Properties

The following are properties of the internal tank and its contained fluid that must be defined in Aqwa Workbench:

- **Permeability**: This is the empty proportion of the defined space, accounting for stiffeners, baffles etc. that may not have been included in the hydrodynamic model. It typically takes a value of 0.95 or higher.
- **Fluid Density**: This is used to set the density of the fluid contained in the internal tank.
- **Damping Factor**: Due to the linear potential theory used in the Hydrodynamic Diffraction analysis, which does not account for fluid viscosity in the solution, a damping model is applied to attenuate any unphysical resonant liquid motion. This property usually takes a value in the range [0, 0.1].

The Internal Tank hydrostatic properties are automatically determined from the mesh and the internal fluid density and tank permeability. The Internal Tank **Fluid Mass**, **Fluid Center of Gravity X, Y** and **Z** and the **Fluid Center of Floatation X** and **Y** are displayed once a mesh has been generated; otherwise, the **Fluid Mass** field will show an informational message, "**Generate Mesh to Update**".

5.5.8. Moonpool

Moonpools are features of marine structures such as drilling platforms, drill ships, diving support vessels as well as some marine and underwater exploration/research vessels. In underwater habitats, they are also known as a wet porch.

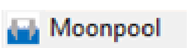
Simply described, a moonpool is an opening in the floor or base of the hull, platform, or chamber, which gives sheltered access to the water body below to enter or leave the water in a more protected environment compared to the deck.

Note:

Active Moonpool objects are only supported when performing Hydrodynamic Diffraction analyses.

Hydrodynamic Response analyses (Stability, Time Response, and Frequency Domain analyses) will not solve if they are active in the Project tree. In addition, you cannot transfer the pressure distribution field to a Mechanical system via the ACT load mapping.

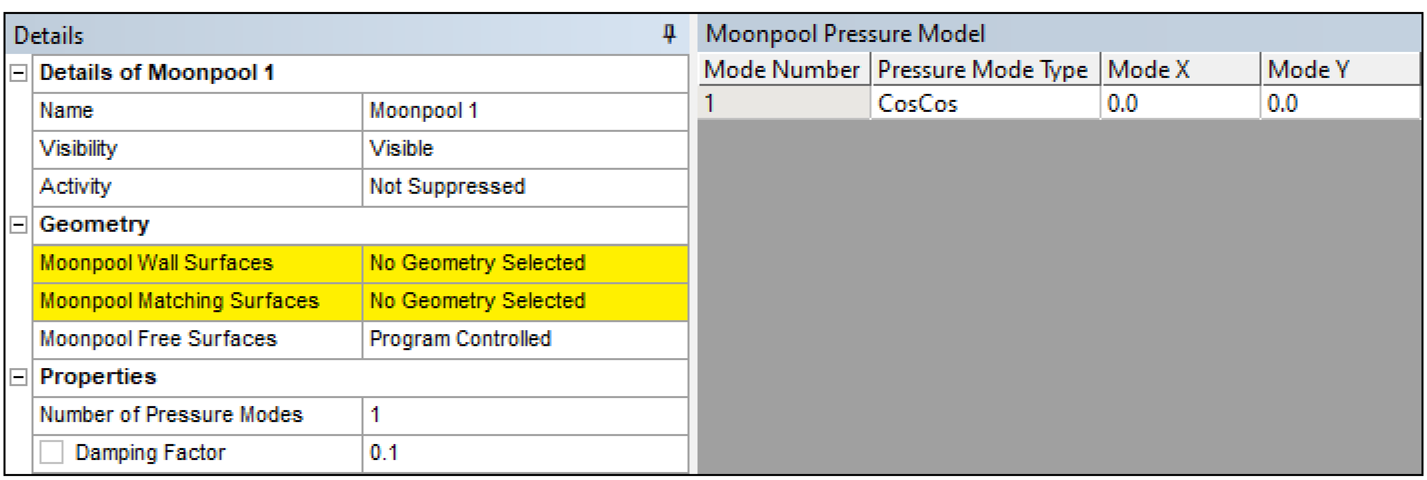
To include a Moonpool object in your model:

1. Select a **Part** in the tree view.
2. Right-click the Part and select **Add** > **Moonpool**, or click the **Moonpool** icon in the Parts toolbar 

.
3. Select the **Moonpool** object in the tree.

You can define up to 10 Moonpool objects per structure, and each object's Details panel must be populated.

Figure 5.3: Moonpool Object Details Panel



5.5.8.1. Geometry

Following the model definition of a Moonpool described in Prescribed Oscillatory Pressure Distribution Approach, you must define the Moonpool geometry by selecting the **Moonpool Surfaces** in the Graphical Window, where those surfaces must be selected from the Surface Bodies that make up the associated Part.

Your Moonpool should contain at least three types of surfaces:

- The **wall surfaces** that physically correspond to the Moonpool location.

  If a portion of the surfaces was made at the waterline in the Geometry editor, both the diffracting and non-diffracting Moonpool wall surfaces should be selected here. Alternatively, the water nodes can be automatically created if the **Create Automatic Waterline Nodes** option is turned on in the Mesh (see Waterline Node Generation (p. 51)). This prevents cutting the wall surface. The surface normal should point toward the enclosed sea volume.

- The free surface is an abstract surface which corresponds to the surface of the Moonpool at the water level. By default, the **Moonpool Free Surfaces** option displays **Program Controlled**, and the free surface is automatically created when the mesh is generated. However, you can still manually select a free surface, in which case the surface normal should point upwards so that $\overrightarrow{normal}.\vec{z}>0$. In the case of a normal pointing downward, the application flips the normal in the background when generating the mesh, so that the normal of the panels for this surface is pointing upward as required by the solver.

- The matching surface is also an abstract surface and corresponds to the portion of the Moonpool walls through the structure's hull. The surface normal should point towards the enclosed sea volume, which will be upwards in most cases so that $\overrightarrow{normal}.\vec{z}>0$. Similar to the free surface, if the normal is pointing downward, a flip is performed in the background when creating the mesh, so that the normal of each panel for this surface is pointing upward.

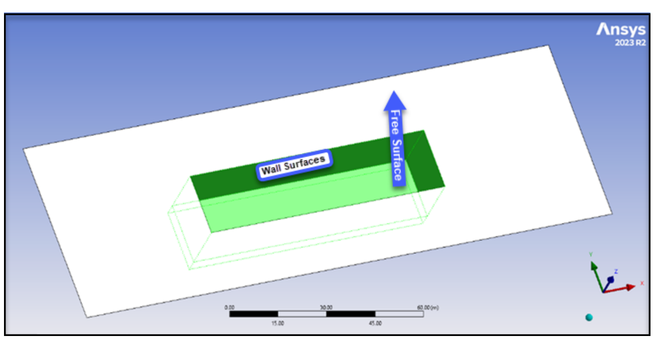
When selecting a surface:

1. Ensure that your target surface is visible or it will not be displayed.

2. Enter the selection mode and/or select the Moonpool. A bright green color will appear and indicate the direction the normal to the surface is pointing to.

For example, normal vectors are added to the free and wall surfaces, as indicated by the pointers in Figure 5.4: Moonpool Walls and Free Surface with Normal Vectors Pointing Inward (Wall Surfaces) and Upward (Free Surface) (p. 69).

Figure 5.4: Moonpool Walls and Free Surface with Normal Vectors Pointing Inward (Wall Surfaces) and Upward (Free Surface)



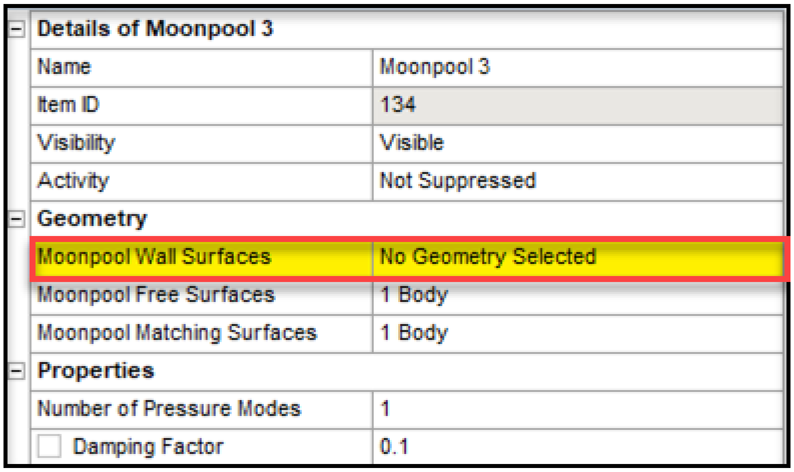
Selecting and Releasing Moonpool Surfaces

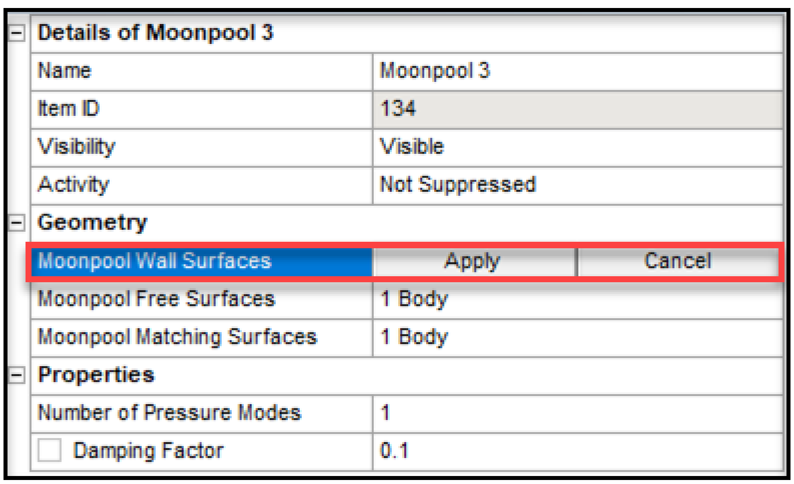
To select your bodies:

1. Click **Moonpool Wall Surfaces** (whose field may contain **No Geometry Selected** or the number of bodies previously selected) to enter the selection mode.

2. Select your target surfaces in your model in the Graphical editor.

3. Click **Apply** once your selection is made.





Any new valid selection will replace the previous one. If you need to add a body to your previous selection, add and group the pre-associated bodies.

Moonpool surfaces must **belong to the same Part**. They must also be **associated to one unique Moonpool** and **one type of surface, once selected**.

To release the surface selection associated to a type of surface:

1. Enter the selection mode (as you would to select new bodies).

2. Click the Graphical editor background.

3. Click **Apply**.

This will clear your current selection, and the **Moonpool Wall Surfaces** field with **No Geometry Selected** will turn yellow.

At this point, the selected Surface Bodies are now linked with the Moonpool. The Surface Bodies are grouped under the Moonpool in the tree view to indicate this. If you change the activity of the Moonpool, the activity of any linked Surface Bodies also changes. If the Surface Bodies are deselected from the Moonpool Surfaces or the Moonpool object is deleted, this linkage is removed.

Note:

Once a surface is associated with the Moonpool, it stays linked to this object, whether it is supressed or not. If you want to use the surface as part of your active model for further analysis, you must release this surface from your Moonpool object by carrying out a new surface selection that excludes this particular surface. Then, your surface becomes ready for further use, and you can suppress your Moonpool object.

Moonpool Properties

The following Moonpool properties must be defined in Aqwa Workbench:

- **Pressure Modes**: They correspond to the pressure distribution on the mean free surface inside the Moonpool. You can define **up to 20 pressure modes per Moonpool**. Each must be unique within their associated Moonpool. A comprehensive definition of the possible non-dimensional pressure distribution model shapes is found in Prescribed Oscillatory Pressure Distribution Approach.

  In Figure 5.3: Moonpool Object Details Panel (p. 68), Mode X and Mode Y values must be of a non-negative integer, corresponding to the modal number *n* in X-direction and the modal number *m* in Y-direction respectively in Equation 4.125 through Equation 4.127 in the Aqwa Theory Manual.

  The default definition corresponds to the piston mode defined as a combination of sine and cosine functions. Piston-type pressure modes are also defined as:

  – A Legendre function where the *x* and *y* values equal 0.

  – A Bessel function where the *x* value equals 0.

  No more than one piston definition is allowed in a Moonpool model, regardless of their formulation.

  Note:

  Results selection based on Moonpool pressure modes (whether as a SubType or Component) becomes invalid if the number of pressure modes is modified for any Moonpool. Manually select a new pressure mode to use to query the line results.

- **Damping Factor**: Due to the linear potential theory used in the Hydrodynamic Diffraction analysis, which does not account for fluid viscosity in the solution, a damping model is applied to attenuate any unphysical standing wave motion within the restricted fluid region. The default value is set to 0.1.

5.5.9. Additional Hydrodynamic Stiffness

This object may be used to input an additional linear hydrostatic stiffness matrix using tabular input in the Matrix Definition Data window. The linear stiffness matrix relates to the hydrostatic forces contributing to the equations of static equilibrium of a structure. Specifically, the net linear hydrostatic forces F_s, acting at the center of gravity of a structure, when the structure is at an arbitrary position X, are given by:

$$F_s = (K + dK)(X_e - X) + B_e$$

where:

K = stiffness matrix

dK = additional hydrodynamic stiffness matrix input on this object

X_e = equilibrium position

B_e = buoyancy force at equilibrium

If additional hydrodynamic stiffness is used it should be checked that the above expression, which is used to calculate the linear hydrostatic forces throughout the Aqwa suite, produces the forces on the structure intended by the user.

Note:

In this context, hydrostatic forces can act in all 6 degrees of freedom.

In the equation above, the term X(e) is the diffraction analysis defined position. If the initial position in a subsequent motions analysis is not as defined in the diffraction run, then there will be restoring forces which will try to return the structure to the diffraction defined position.

To add an Additional Hydrodynamic Stiffness Matrix:

1. Select a Part in the tree view.

2. Right-click the Part and select **Add > Additional Hydrodynamic Stiffness**, or click the **Hydrodynamic Properties** icon in the **Parts** toolbar and select **Additional Hydrodynamic Stiffness** from the dropdown menu.

Matrix Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.

- Contains values separated by commas, tabs, or single spaces (not multiple spaces).

- Contains exactly 6 columns and 6 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Force Unit for Imported Data**, and **Rotation Unit for Imported Data** to the required unit system.

5.5.10. Additional Added Mass (Frequency Independent)

This object may be used to input frequency independent additional added mass in global directions using tabular input. Only one definition of Additional Added Mass per structure can be active or not suppressed for the analysis and the values are added to those calculated automatically during the analysis.

To add Additional Added Mass:

1. Select a Part in the tree view.
2. Right-click the part and select **Add** > **Additional Added Mass**, or click the **Hydrodynamic Properties** icon in the **Parts** toolbar and select **Additional Added Mass** from the dropdown menu.

Matrix Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 6 columns and 6 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Mass Unit for Imported Data**, and **Rotation Unit for Imported Data** to the required unit system.

5.5.11. Additional Damping (Frequency Independent)

This object may be used to input frequency independent additional damping in global directions using tabular input. Only one definition of Additional Damping per structure can be active or not suppressed for the analysis and the values are added to those calculated automatically during the analysis.

To add Additional Damping:

1. Select a Part in the tree view.
2. Right-click the Part and select **Add** > **Additional Damping**, or click the **Hydrodynamic Properties** icon in the **Parts** toolbar and select **Additional Damping** from the dropdown menu.

Matrix Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.

- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 6 columns and 6 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Force Unit for Imported Data**, and **Rotation Unit for Imported Data** to the required unit system.

5.5.12. Nonlinear Roll Damping

Nonlinear roll damping moment can be calculated in Time Response and Frequency Statistical analyses to take into account the effect of vortex shedding from the bilges and bilge keels of a vessel.

To add Nonlinear Roll Damping properties:

1. Select a Part in the tree view.
2. Right-click the Part and select **Add > Nonlinear Roll Damping**, or click the **Nonlinear Roll Damping** icon in the **Parts** toolbar.
3. Select the **Nonlinear Roll Damping** object in the tree and set the properties.

The method for bilge vortex shedding is based on "An Engineering Assessment of the Roll of Non-linearities in Transportation Barge Roll Response," Robinson and Stoddart, Trans. R.I.N.A., 1986.

The method for bilge keel roll damping is based on "ITTC recommended procedures 7.5-02-07-04.5 - Numerical estimation of roll damping", 2011.

Whether vortex shedding is occurring or not is calculated by the program based on the relative flow velocity at the bilge, Keulegan-Carpenter number, roll natural frequency of the vessel, and the radius of the bilge. The roll damping coefficient used in the nonlinear roll damping force calculation is also calculated by Aqwa based on a database stored within the program.

To compute the effects of nonlinear roll damping, select one of the following options from the **Nonlinear Roll Damping** drop-down menu:

- Use a Coefficient
- Define Bilge Parameters
- Combine Coefficient and Bilge Parameters

For all three options, you must define an axis on which the damping moment is applied. A central line must be defined by an Aft End Position and a Forward End Position; each has an **End Position Definition** which can be switched between Vertex Selection and Manual Definition. If the **End Position Definition** is set to Vertex Selection, you should select an **End Vertex** from the geometry and (optionally) define the **End Vertex X Offset** and **End Vertex Y Offset**. The **End X Position** and **End Y Position** will be read-only. If the **End Position Definition** is set to Manual Definition, you should enter the **End X Position** and **End Y Position** directly.

When defining a coefficient (Use a Coefficient or Combine Coefficient and Bilge Parameters), use the **Quadratic Roll Damping Coefficient** field.

Details	
Details of Non-Linear Roll Damping	
Name	Non-Linear Roll Damping
Visibility	Visible
Activity	Not Suppressed
Non-Linear Roll Damping	
Non-Linear Roll Damping	Use a Coefficient
Quadratic Roll Damping Coefficient	30000 N.m/(°/s)^2
Bilge Central Line Definition	
Aft End Position Definition	Vertex Selection
Aft End Vertex	Vertex Selected (Hull)
Aft End Vertex X Position	-55 m
Aft End Vertex Y Position	-10 m
Aft End Vertex X Offset	15 m
Aft End Vertex Y Offset	10 m
Aft End X Position	-40 m
Aft End Y Position	0.0 m
Forward End Position Definition	Vertex Selection
Forward End Vertex	Vertex Selected (Hull)
Forward End Vertex X Position	25 m
Forward End Vertex Y Position	-10.0000000000009 m
Forward End Vertex X Offset	7 m
Forward End Vertex Y Offset	10 m
Forward End X Position	32 m
Forward End Y Position	0.0 m

When defining bilge parameters, it is assumed that the two bilges have symmetric properties about the center line of the vessel. The **Bilge Type** field allows you to control the type of damping calculated. By default, this field is set to Vortex Shedding Only. You can also select Bilge Vortex Shedding and Bilge Keel Roll Damping to calculate the roll damping effects of the bilge keel. The **Bilge Radius** defines the local radius of the bilge corner. The **Depth to Bilge** is the vertical position of the bilge corner on the vessel in the global axis system. The **Offset of Bilge from Central Line** is the lateral offset of the bilge corner from the center line of the vessel in the global axis system.

Note:

When an **End Position Definition** is set to Vertex Selection, the vertical position of the vertex selected for that central line end point is irrelevant; the vertical position of the central line is always set by **Depth to Bilge**.

Details	
Details of Non-Linear Roll Damping	
Name	Non-Linear Roll Damping
Visibility	Visible
Activity	Not Suppressed
Non-Linear Roll Damping	
Non-Linear Roll Damping	Define Bilge Parameters
Bilge Central Line Definition	
Aft End Position Definition	Manual Definition
Aft End X Position	-40 m
Aft End Y Position	0.0 m
Forward End Position Definition	Manual Definition
Forward End X Position	32 m
Forward End Y Position	0.0 m
Bilge Roll Damping Details	
Bilge Type	Bilge Vortex Shedding Only
Bilge Radius	0.8 m
Depth to Bilge	8.25 m
Offset of Bilge from Central Line	9.8 m

If you set **Bilge Type** to Bilge Vortex Shedding and Bilge Keel Roll Damping, the **Bilge Keel Details** submenu becomes available. You can set the **Bilge Keel Length** and **Bilge Keel Breadth**. Changing **Advanced Bilge Keel Options** to Manual Definition allows you to set the **Draft of Vessel** and **Breadth of Vessel**.

Details	
Details of Non-Linear Roll Damping	
Name	Non-Linear Roll Damping
Visibility	Visible
Activity	Not Suppressed
Non-Linear Roll Damping	
Non-Linear Roll Damping	Define Bilge Parameters
Bilge Central Line Definition	
Aft End Position Definition	Manual Definition
Aft End X Position	-40 m
Aft End Y Position	0.0 m
Forward End Position Definition	Manual Definition
Forward End X Position	32 m
Forward End Y Position	0.0 m
Bilge Roll Damping Details	
Bilge Type	Bilge Vortex Shedding and Bilge Keel Roll Damping
Bilge Radius	0.8 m
Depth to Bilge	8.25 m
Offset of Bilge from Central Line	9.8 m
Bilge Keel Details	
Bilge Keel Length	40 m
Bilge Keel Breadth	1 m
Midship Cross-Section Coefficient	0.98
Advanced Bilge Keel Options	Program Controlled

5.5.13. Current Force Coefficients

This object may be used to include the viscous drag of the current on the hull of a fixed or floating structure, using tabular input in the Current Force Coefficients window. The term current force coefficient is used to differentiate these coefficients from traditional drag coefficients and from coefficients of wind force.

To add a Current Force Coefficients object:

1. Select a part in the tree view.
2. Right-click the Part and select **Add > Current Force Coefficients**, or click the **Force Coefficients** icon in the **Parts** toolbar and select **Current Force Coefficients** from the dropdown menu.
3. Select the **Current Force Coefficients** object in the tree view and enter the coefficients in the Coefficient Data window that appears below the model. Enter the **Direction** of the current (-180 to +180 degrees), the X force coefficient (**Translation X**), Y force coefficient (**Translation Y**), Z force coefficient (**Translation Z**), Rotation about X coefficient (**Rotation X**), Rotation about Y coefficient (**Rotation Y**), and Rotation about Z coefficient (**Rotation Z**). The number of rows is increased as each entry is made, up to a maximum of 41 rows.

The current force coefficients are defined as the force or moment per unit velocity squared. The moment is about the center of gravity of the structure. These forces are a function of the relative velocity between the structure and the water. This means that the current coefficient should still be input even when there is no current present, as the relative velocity is generally non-zero for a dynamic analysis.

The Coefficient Data table must include an entry for the direction of -180 degrees. The coefficients defined for -180 degrees are automatically copied to the +180 degrees row, and these copied values cannot be modified (but will update if the -180 degree values are subsequently modified). **Direction** entries outside the range of -180 to +180 degrees are automatically adjusted to this range, and rows are sorted in ascending order of **Direction**. Duplicate **Direction** entries are not valid and may be removed by selecting the entire row and pressing the Delete key.

Coefficient Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 7 columns, and no more than 41 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Rotation Unit for Imported Data** , and **Force Unit for Imported Data** to the required unit system.

Note that the forces are in the directions of the axes, not in the direction of the current. For example, for relative current velocity V in direction φ:

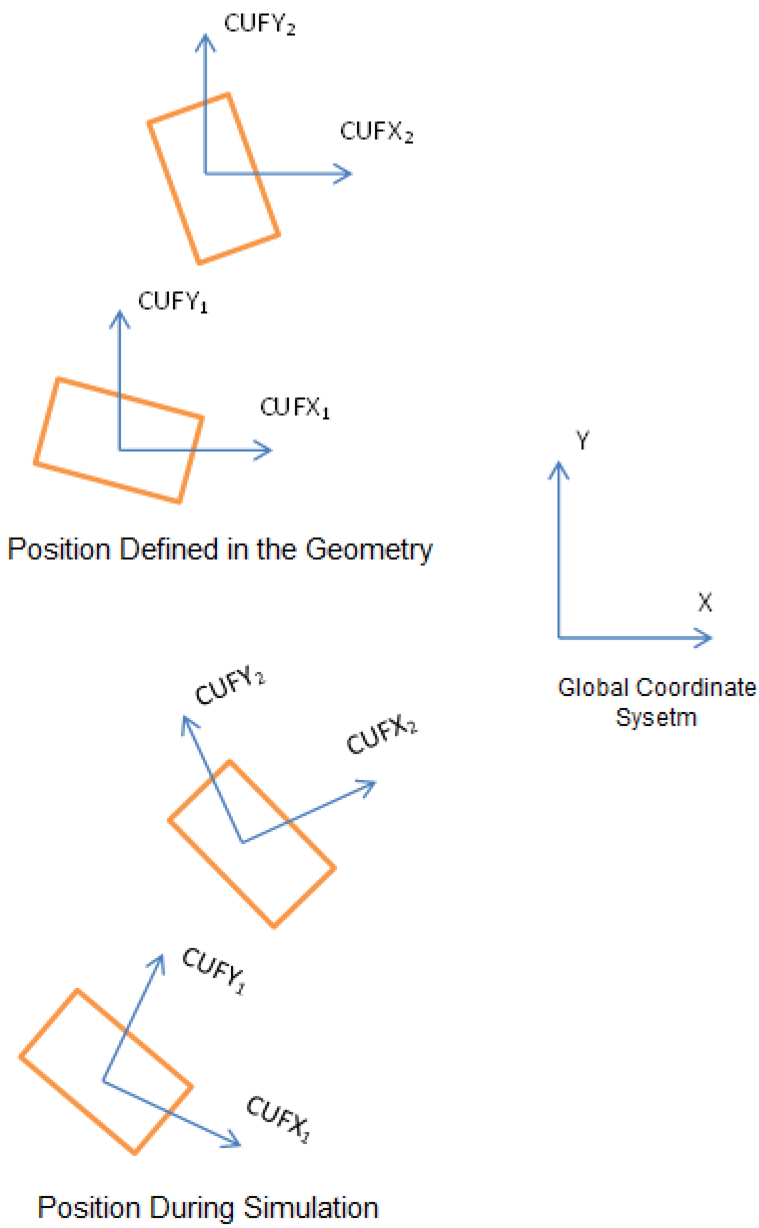
force in X direction = $CUFX_{\varphi}.V^2$

force in Y direction = $CUFY_{\varphi}.V^2$

moment about Z axis = $CURZ_{\varphi}.V^2$

where CUFX, CUFY and CURZ are the coefficients. Similar equations apply for force in Z and moments about X and Y.

The coefficients are applied in a moving axis system; that is, the axes move with the structure. Since there is no structure local axis system available, the initial coefficient data must be defined relative to the global coordinate system *when the structure is in the position as defined in the geometry.* For example:



Note:

- Only one active Current Force Coefficient object may be included for each structure.
- There is a limit of 41 unique directions for the enabled Wind Force Coefficient and Current Force Coefficient tables combined. Each table may have an entry for the same Direction value.
- If a direction is specified for Wind Force Coefficients that does not exist for Current Force Coefficients then linearly interpolated values will be utilized for the Current Force Coefficients for that direction, based upon adjacent defined directions.

5.5.14. Wind Force Coefficients

This object may be used to include the viscous drag of the wind on the superstructure of a fixed or floating structure using tabular input in the Wind Force Coefficients window. The term wind force coefficient is used to differentiate these coefficients from traditional drag coefficients and from coefficients of current force.

To add a Wind Force Coefficients object:

1. Select a Part in the tree view.

2. Right-click the Part and select **Add > Wind Force Coefficients**, or click the **Force Coefficients** icon in the **Parts** toolbar and select **Wind Force Coefficients** from the dropdown menu.

3. Select the Wind Force Coefficients object in the tree view and enter the coefficients in the Wind Force Coefficients window that appears below the model. Enter the **Direction** of the wind (-180 to +180 degrees), the X force coefficient (**Translation X**), Y force coefficient (**Translation Y**), Z force coefficient (**Translation Z**), Rotation about X coefficient (**Rotation X**), Rotation about Y coefficient (**Rotation Y**), and Rotation about Z coefficient (**Rotation Z**). The number of rows is increased as each entry is made, up to a maximum of 41 rows.

The wind force coefficients are defined as the force or moment per unit velocity squared. The moment is about the center of gravity of the structure. These forces are a function of the relative velocity between the structure and the air. This means that the wind coefficient should still be input even when there is no wind present, as the relative velocity is generally non-zero for a dynamic analysis.

The Coefficient Data table must include an entry for the direction of -180 degrees. The coefficients defined for -180 degrees are automatically copied to the +180 degrees row, and these copied values cannot be modified (but will update if the -180 degree values are subsequently modified). **Direction** entries outside the range of -180 to +180 degrees are automatically adjusted to this range, and rows are sorted in ascending order of **Direction**. Duplicate **Direction** entries are not valid and may be removed by selecting the entire row and pressing the Delete key.

Coefficient Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 7 columns, and no more than 41 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Rotation Unit for Imported Data** , and **Force Unit for Imported Data** to the required unit system.

Note that the forces are in the directions of the axes, not in the direction of the wind. For example, for relative wind velocity V in direction φ:

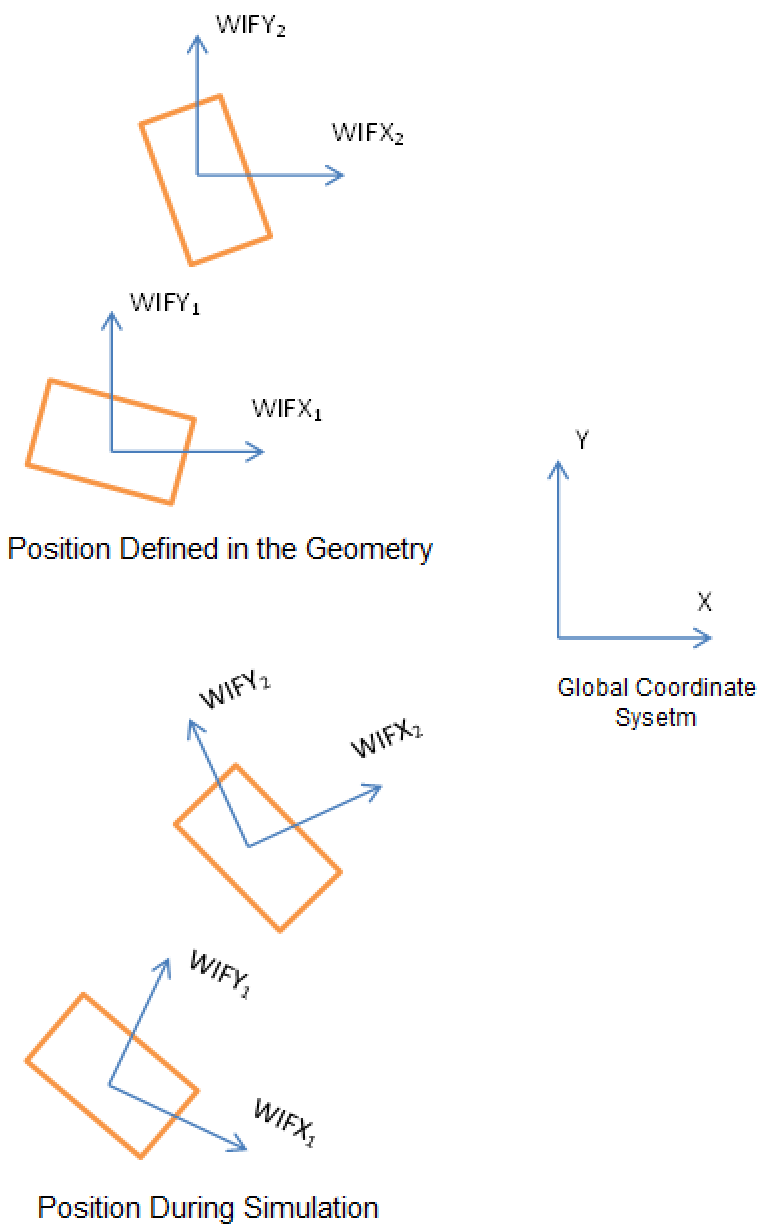
force in X direction = $WIFX_{\varphi}.V^2$

force in Y direction = $WIFY_{\varphi}.V^2$

moment about Z axis = $WIRZ_{\varphi}.V^2$

where WIFX, WIFY and WIRZ are the coefficients. Similar equations apply for force in Z and moments about X and Y.

The coefficients are applied in a moving axis system. This means that the axes move with the structure. Since there is no structure local axis system available, the initial coefficient data must be defined relative to the global coordinate system *when the structure is in the position as defined in the geometry.* For example:



Note:

- Only one active Wind Force Coefficient object may be included for each structure.
- There is a limit of 41 unique directions for the enabled Wind Force Coefficient and Current Force Coefficient tables combined. Each table may have an entry for the same Direction value.
- If a direction is specified for Current Force Coefficients that does not exist for Wind Force Coefficients then linearly interpolated values will be utilized for the Wind Force Coefficients for that direction, based upon adjacent defined directions.

5.5.15. Morison Hull Drag Coefficients

This object may be used to input a Morison Hull Drag Coefficients matrix using tabular input in the Matrix Definition Data window. In a Hydrodynamic Response analysis these coefficients are used to calculate hull drag forces and moments in a similar way to that for a Morison element. Specifically, the drag forces and moments acting at the center of gravity of the structure, when the structure experiences a relative fluid velocity, are given by:

$$F_d = C_d \left| u \right| u$$

Where C_d is the 6 x 6 Morison Hull Drag Coefficient matrix. The relative fluid velocity is determined as the difference between any current velocity and the structure velocity; the fluid particle velocity due to waves is not included.

Note:

Morison Hull Drag Coefficients are not used in a Hydrodynamic Diffraction analysis, even when the **Linearized Drag** option has been turned on.

In a Frequency Statistical Analysis, Morison hull drag forces can be included using the **Linearized Drag** option.

The relative fluid velocity is always evaluated at the structure's center of gravity; the Part **Current Calculation Position** has no effect on this drag calculation.

To add a Morison Hull Drag Coefficients matrix:

1. Select a Part in the tree view.
2. Right-click the Part and select **Add > Morison Hull Drag Coefficients**, or click the **Hydrodynamic Properties** icon in the **Parts** toolbar and select **Morison Hull Drag Coefficients** from the dropdown menu.
3. Select the **Morison Hull Drag Coefficients** object in the tree and enter the matrix coefficients in the Matrix Definition Data window that appears below the model.

Matrix Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-seperated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values seperated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 6 columns and 6 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Force Unit for Imported Data**, and **Rotation Unit for Imported Data** to the required unit system.

5.5.16. Yaw-Rate Drag

Although current and wind loads on a structure can be modeled in Aqwa by including Current Force Coefficients (p. 77) and Wind Force Coefficients (p. 80), these loads have no dependence on the yaw rotational velocity. The yaw-rate drag moment can be modeled separately by the addition of a Yaw-Rate Drag object to the Part.

To add Yaw-Rate Drag properties:

1. Select the Part in the tree view.
2. Right-click the **Part** and select **Add > Yaw-Rate Drag**, or click the **Yaw-Rate Drag** icon in the **Parts** toolbar.
3. Select the **Yaw-Rate Drag** object in the tree and set the properties.

The yaw-rate drag calculation requires the definition of a geometric central line for the structure, and a **Yaw-Rate Drag Force Coefficient**. The yaw-rate drag loads are calculated from the input properties using the approach described in Yaw Rate Drag Force in the *Aqwa Theory Manual*.

The central line must be defined by an Aft End Position and a Forward End Position; each has an **End Position Definition** which can be switched between Vertex Selection and Manual Definition. If the **End Position Definition** is set to Vertex Selection, you should select an **End Vertex** from the geometry and (optionally) define the **End Vertex X Offset** and **End Vertex Y Offset**. The **End X Position** and **End Y Position** will be read-only. If the **End Position Definition** is set to Manual Definition, you should enter the **End X Position** and **End Y Position** directly.

Note:

- Only one active Yaw-Rate Drag object may be included for each structure.
- As Yaw-Rate Drag depends on the relative structure velocity, it is used in time domain Hydrodynamic Response analyses. It is not included in a Hydrodynamic Diffraction calculation, a stability Hydrodynamic Response analysis, or a frequency statistical Hydrodynamic Response analysis if **Linearized Morison Drag** is set to No.

5.5.17. Structure Connection Points

A structure Connection Point is defined as an attachment point connected to a Part that moves with the Part. Connection Points are defined in the Details panel by entering coordinates in global space or specifying an offset from a vertex on a structure. The coordinates defining the Connection Point can be parameterized. A structure Connection Point can be used by more than one object (cable, fender, etc.), if required. The option to choose a Fixed point vs. Connection points on structures is controlled by the Connectivity field in the Details panel of the object using the connection points.

To add a Connection Point:

1. Select a Part in the tree view.
2. Right-click the Part and select **Add > Connection Point**, or click on the **Connection Point** icon in the **Parts** toolbar.

3. Select the **Connection Point** object in the tree and set the initial position by doing one of the following:

   - Set **Definition of Position** to X, Y, and Z Coordinates, and set the Position Coordinates (**X**, **Y**, **Z**) in the Details panel.
   - Set **Definition of Position** to X and Y Coordinates, Z on Composite Cable Seabed, and set the Position Coordinates (**X**, **Y**) and a positive **Z Offset** from the Composite Cable Seabed Water Depth at the given X, Y position.
   - Set **Definition of Position** to Vertex Selection. Click **Select a Single Vertex** in the **Vertex** field, select a vertex on a structure, and click **Apply**. You can then set an **X Offset**, **Y Offset**, or **Z Offset** from the vertex if needed.

Note:

Even if you define the initial position of the Connection Point using Coordinates, or with an offset from a vertex on the Part, the Connection Point is "attached" to the part and will move with the part, maintaining the relative position that you defined when you created it.

The **Include in Results** option controls whether the Connection Point is available to be used as a **Reference Point** in a Hydrodynamic Frequency result, or as a **Reference Point** for Wave Surface Elevation or with motions relative to the wave surface in a Hydrodynamic Time Response result.

Note:

In a Hydrodynamic Equilibrium result, or a Hydrodynamic Time Response result where the requested motion is not relative to the wave surface, the **Include in Results** option is not required for the Connection Point to be used as a **Reference Point**.

You may have up to 99 Connection Points in which the **Include in Results** option is set to Yes.

Important:

When resuming a project that has been created with a version of Aqwa prior to Ansys Release 17.0, the first 17 Connection Points appearing in the Outline will have the **Include in Results** option set to Yes. Any subsequent Connection Points will have **Include in Results** default to No. If you wish to add Hydrodynamic Frequency results at a Connection Point, or Hydrodynamic Time Response result for Wave Surface Elevation or at a Connection Point with motions relative to the wave surface, you will need to perform a fresh Solve of the analysis.

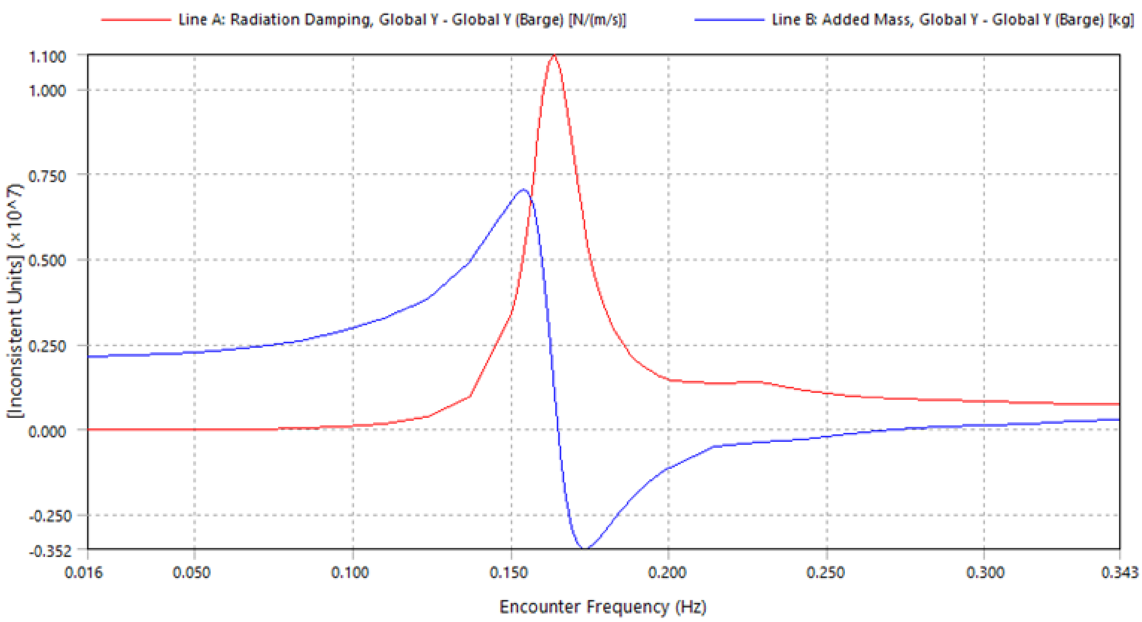
When creating a new project, the **Include in Results** option of newly-created Connection Points will default to Yes as long as the limit of 99 has not been reached. Any Connection Points created after the first 99 will have the **Include in Results** default to No.

5.5.18. External Lid

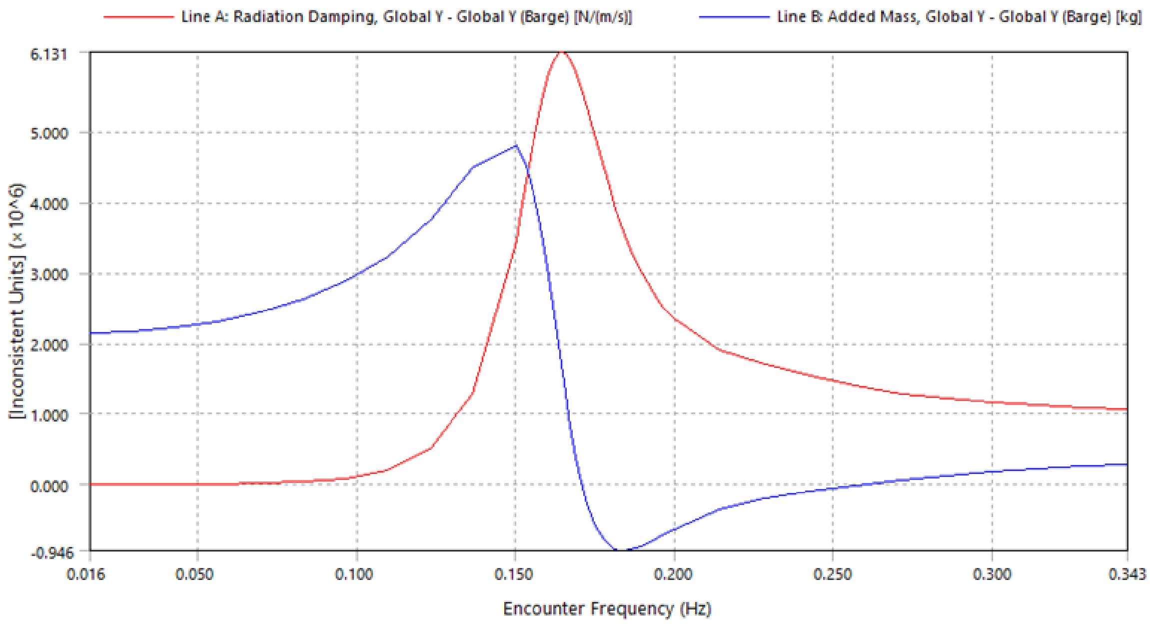
External Lids attenuate or suppress the standing waves that form in a gap between hydrodynamically interacting structures. They consist of a flat surface of panel elements, positioned at the mean water level (Z=0 in the XY plane).

Standing waves are a physical effect that is necessary to capture in our hydrodynamic analysis, but the absence of fluid viscosity in the solution means that these waves may become unrealistically large. External Lids are therefore used to model the dampening effect of fluid viscosity on the wave surface. They may also affect the radiation properties of the hydrodynamically interacting structures by removing any non-physical spikes in the radiation damping and added mass around the standing wave resonant frequency.

To illustrate the effect of an External Lid, consider the Hydrodynamic Diffraction analysis of a barge positioned next to a pier. Here are the sway-sway (Y-Y) radiation damping (red) and added mass (blue) values without an External Lid:



Here is the same analysis, now with an External Lid included, where the sharp peaks and troughs in the plots have been reduced:



In this way, External Lids are used to improve the accuracy of the Hydrodynamic Diffraction results and any linked Hydrodynamic Response calculations.

External Lids can be created as Surface Bodies (p. 90) in the geometry editor. However, it may be more convenient to add them directly in the Aqwa Workbench editor — for example, when a Part Transform (p. 88) will be applied, and the transform translates or rotates the structure out of the horizontal plane. This cannot be done with an External Lid which is modeled as geometry because the lid surface must always be positioned on the mean water surface (Z=0 in the XY plane).

To add an External Lid in the Aqwa Workbench editor:

1. Select a Part in the Outline tree view.
2. Right-click the Part and select **Add** > **External Lid**, or click the **External Lid** icon in the **Parts** toolbar.
3. Select the **External Lid** object in the Outline tree and set the properties.

An External Lid that is added in the Aqwa Workbench editor may be triangular or quadrilateral in shape. In the Details panel, use the External Lid Surface Definition to specify the points that make up the corners of the surface. Each point can be defined manually, where X and Y coordinates are entered directly or by selecting a vertex from the geometry, with an optional X or Y offset.

The defined points are automatically sorted into anti-clockwise order by the Aqwa Workbench editor. To create a triangular External Lid, set two of the points to share the same location. For more complex shapes, create multiple External Lids on the same Part.

Finally, two External Lid Hydrodynamic Properties must be defined. The first is the **Lid Damping Factor**, set between 0.0 and 1.0, which represents how effective the lid will be: 0.0 will result in no effect, while 1.0 will prevent any diffracted or radiated component of vertical water surface velocity under the lid. The second parameter is the **Gap**, which is a representative size for the lid: this is typically the distance between the two interacting vessels or the width of a moonpool. The Aqwa solver uses this value to tune the lid damping properties to the resonant frequency of waves in the gap.

5.5.19. Part Transform

When vessel geometry is first imported from Discovery Modeling or DesignModeler into the Aqwa Workbench editor, the positions of the imported Parts are consistent with the positions defined in the geometry editor. However, adding a Part Transform to a Part in Aqwa Workbench allows you to modify the position or rotation of that Part, without modifying and re-importing from the geometry editor. This makes it simpler to configure your model, and allows parametric studies of vessel draft or trim (for example) to be performed directly in Aqwa Workbench.

To include a Part Transform object in your model:

1. Select a Part in the Outline tree view.
2. Right-click the Part and select **Add** > **Part Transform**, or click the **Part Transform** icon in the **Parts** toolbar .
3. Select the **Part Transform** object in the tree.
4. Define the Reference Point and Transform properties, as described in Reference Point Definition (p. 88) and Transform Definition (p. 88).
5. Right-click the Part Transform and select **Apply**, or click the **Apply** icon in the Part Transform toolbar . At this point, any up-to-date Mesh or Solution data will be cleared.

5.5.19.1. Reference Point Definition

The Reference Point represents the position in space that the rotations of the Part Transform are applied about. Setting **Reference Point** to Manual Definition allows you to define directly the **Reference PointX/Y/Z** coordinates. In this mode, you can also use the **Move Reference Point to Center of Gravity** option to automatically position the Reference Point at the Part's center of gravity position.

Note:

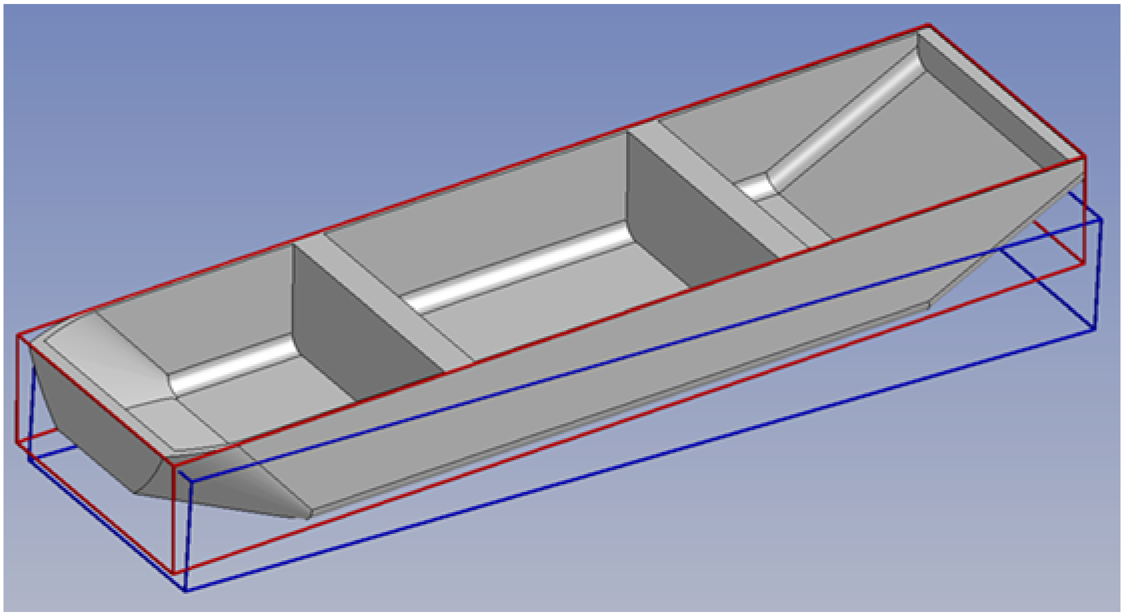
If the Part has a Program Controlled Point Mass, or it includes one or more Internal Tanks, and the Mesh has not yet been generated, the **Move Reference Point to Center of Gravity** option will not have any effect as the Part's center of gravity has not yet been determined by the program.

Alternatively, you can set **Reference Point** to Vertex Selection, then use the **Vertex** option to select a vertex from the geometry in the Graphical Window. The **Reference Point X/Y/Z** values will be updated to show the position of the selected vertex.

5.5.19.2. Transform Definition

The transformation to be applied to the Part is defined by entering the **Translation X/Y/Z** and **Rotation about Reference Point X/Y/Z** values into the appropriate fields. The translations are also applied to the Reference Point. For example, if you want to modify both the draft and the trim of the Part, the trim angle is applied about the translated Reference Point position (not the initial Reference Point position).

As the transform is modified, a preview of the Part's transformed position is shown as a blue bounding box in the Graphical Window, while the original position is shown by a red bounding box:



5.5.19.3. Application of a Part Transform

Once the Part Transform has been applied, the Surface Bodies and Line Bodies in Aqwa Workbench are transformed so that the generated Mesh reflects the new Part position, including the adjusted location of the waterline on the Part (provided that the Mesh **Create Automatic Waterline Nodes** option is set to **Yes**).

Note:

The Part Transform in Aqwa Workbench does not have any effect on the original geometry defined in Discovery Modeling or DesignModeler.

The Part Transform is also applied to any unsuppressed (active) Aqwa-specific features under the Part in the Outline tree, as well as some properties of the Part itself. However, there are some exceptions; the applicability of the Part Transform is described in detail here.

Note:

Suppressed (inactive) features are not modified by the Part Transform. For all other properties that are not modified by the Part Transform, the Aqwa Workbench editor will issue a warning message in the Output window.

5.5.19.3.1. Part

When the **Current Calculation Position** is set to **Moves with Structure**, the **Current Calculation X**/**Y**/**ZPosition** is transformed. However, when **Current Calculation Position** is set to **At Fixed Depth**, the **Current Calculation Depth** is not modified by the Part Transform.

When either of the **Override Calculated GMX** or **Override CalculatedGMY** options are set to **Yes**, the user-defined **GM about Global X/Y Axis** values are not modified by the Part Transform.

When the **Calculate Shear Force/Bending Moment** option is set to **Yes**, and the **Neutral Axis Position Definition** is set to Manual Definition, the **Neutral Axis X/Y/Z Positions** are not modified by the Part Transform.

5.5.19.3.2. Surface Body

Surface Bodies are always transformed, but if any surface has its **Structure Type** set to **Abstract Geometry** and its **Abstract Type** set to **Internal Lid** or **External Lid**, the Part Transform is restricted to translations and rotation in the horizontal XY plane only. If you want to change the draft, list, or trim of the vessel, and an Internal or Exterrnal Lid is included, then you will need to modify the geometry in Discovery Modeling or DesignModeler. (This restriction does not apply to the automatic internal lid, which is created when the Part's **Generate Internal Lid** option is set to **Yes**.)

5.5.19.3.3. Line Body

Line Bodies are always transformed.

5.5.19.3.4. Point Mass

When the **Mass Definition** is set to **Manual Definition**, the mass **X**/**Y**/**Z** position and the inertia matrix values **Ixx**, **Ixy**, **Ixz**, **Iyy**, **Iyz**, and **Izz** are transformed. If **the Define Inertia Values By** option is set to **Radius of Gyration**, the leading diagonal terms of the inertia matrix are determined, the matrix is transformed, and the radius of gyration values **Kxx**, **Kyy**, and **Kzz** are re-calculated from the transformed inertia matrix and the defined **Mass** value.

Note:

The radius of gyration values are defined about the Global Axes, rather than the Part principal axes.

However, when the **Mass Definition** is set to **Program Controlled**, none of the properties is modified by the Part Transform.

If you want the program to determine the Point Mass Properties automatically, and then apply a Part Transform without the Point Mass Properties being recalculated, you must:

- Generate Mesh with the Part in its original position, so that the Program Controlled Point Mass is determined by the program;
- Change the **Mass Definition** of the Point Mass to Manual Definition, which retains the existing Point Mass Properties;
- Apply the Part Transform, which moves the Point Mass with the Part (without changing the **Mass** value).

If any of the Point Mass **X**/**Y**/**Z** position values are selected as a parameter P, none of the **X**/**Y**/**Z** position values will be modified by the Part Transform. You must parameterize all of the **X**/**Y**/**Z** position values in this case.

Similarly, if any of the Point Mass Inertia Properties are selected as a parameter, none of the Inertia Properties is modified by the Part Transform. You must parameterize all of the Inertia Properties in this case.

5.5.19.3.5. Distributed Mass

When the Part's **Calculate Shear Force/Bending Moment** option is set to **No**, the Distributed Mass data is considered as a sequence of discrete point masses, and the mass X/Y/Z positions and inertia terms are transformed.

However, if the **Calculate Shear Force/Bending Moment** option is set to **Yes**, the Distributed Mass data is considered as a sequence of continuous masses, and the Distributed Mass Definition Data table also includes the X/Y/Z extents of each mass. In this case, the mass and inertia properties are not transformed, as the mass extents are not meaningful if the Part's principal axis is not aligned with one of the Global Axis directions.

5.5.19.3.6. Point Buoyancy

Point Buoyancy **X**/**Y**/**Z** positions are always transformed, unless any of the **X**/**Y**/**Z** values are selected as a parameter, in which case none of the properties will be modified by the Part Transform.

5.5.19.3.7. Disc

Disc **X**/**Y**/**Z** positions and **Normal X**/**Y**/**Z** values are always transformed.

5.5.19.3.8. Internal Tank

When the **Fluid Level Definition Mode** is set to **Fluid Volume**, the **Surface Height** estimate is updated when the Part Transform is applied.

When the **Fluid Level Definition Mode** is set to **Surface Height**, the user-defined **Surface Height** is not modified by the Part Transform. The height value is still defined in the Global Axes.

When the **Fluid Level Definition Mode** is set to **Edge Selection**, the selected edge is transformed as an entity within the geometry, but the **Fluid Level Definition** becomes invalid if the edge is no longer horizontal.

5.5.19.3.9. Moonpool

Moonpools are also transformed alongside their associated Part, but the Part Transform for this object is restricted to translations and rotation in the horizontal XY plane only.

5.5.19.3.10. Additional Hydrodynamic Stiffness, Added Mass and Damping, and Morison Hull Drag Coefficients

Any 6x6 matrices defined under the Part in the Outline tree are always transformed, where the rotation matrix associated with the Part Transform is applied to each 3x3 quadrant of the 6x6 matrix.

5.5.19.3.11. Nonlinear Roll Damping and Yaw-Rate Drag

The Central Line Definitions associated with these features, as specified by the **Aft End X/Y Position**, and **Forward End X/Y Position**, are always transformed, unless any of the **Aft** or **Forward End X/Y** values are selected as a parameter, in which case none of the properties will be modified by the Part Transform.

The formulation of these features assumes that the Central Line is horizontal, so the position and orientation of the Central Line are only modified in the horizontal XY plane. The **Depth to Bilge** and **Offset of Bilge from Central Line** properties are not modified by the Part Transform.

5.5.19.3.12. Current and Wind Force Coefficients

The **Directions** defined in the Coefficient Data table are rotated by the **Rotation about Reference Point Z** angle defined in the Part Transform, and are then wrapped back into the range ± 180° as necessary. If the transformed data does not include a set of coefficients in the ± 180° direction, a new set of coefficients are added at ± 180° by linearly interpolating between the coefficient values at the nearest adjacent directions.

If the Part Transform is subsequently modified, suppressed, or deleted, the original Current or Wind Force Coefficients are restored until a Part Transform is re-applied.

5.5.19.3.13. Connection Points

Connection Point **X/Y/Z** positions are always transformed, unless any of the **X/Y/Z** values are selected as a parameter, in which case none of the properties will be modified by the Part Transform.

Where the **Definition of Position** is set to **Vertex Selection**, the **X/Y/Z Offset** values are also transformed.

Where the **Definition of Position** is set to **X and Y Coordinates**, **Z on Composite Cable Seabed**, the relationship of the position of the Connection Point relative to the structure is maintained through the transform, but the Connection Point **Z** position is always set to the **Composite Cable Seabed** depth at the transformed **X/Y** position.

5.5.19.3.14. Connection Point Groups

The connection points generated or imported within a Connection Point Group are always transformed, unless any of the **Origin X/Y/Z** values are selected as a parameter, in which case none of the properties will be modified by the Part Transform.

The group **Origin X/Y/Z** values displayed in the Details panel reflect the transformed position. The Group Definition and Group Rotation values remain in the Connection Point Group reference axis system.

Where the **Position Points on Composite Cable Seabed** option is set to **Yes**, the relationship of the positions of the connection points relative to the structure is maintained through the transform, but the vertical positions are always set to the **Composite Cable Seabed** depth at the transformed lateral positions.

5.5.19.3.15. Connections, Hydrodynamic Diffraction, and Hydrodynamic Response Features

All other features defined in Aqwa Workbench are not modified by the application of a Part Transform. The Aqwa Workbench editor will issue a warning message in these cases. Specifically, the unaffected features include:

- Under Connections:
  - **Fender Axes** and **Joint Axes**
  - **Connection Stiffness** and any associated user-defined **Equilibrium Position**
- Under Hydrodynamic Diffraction:
  - The **Bounding Box** and **Calculation Coordinate** defined for any **Splitting Forces** result
- Under Hydrodynamic Response:
  - **Starting Position, Point Force**, **Structure Force**, and **Deactivated Freedom** objects
  - Stability Analysis **Movement Limitations** and **Maximum Error in Equilibrium Position** settings
  - Time Response Analysis **Co-simulation Coordinate System Offset** settings

5.6. Attachment Point Groups

Fixed Points and Connection Points are introduced in Add Fixed Points (p. 46) and Structure Connection Points (p. 84), respectively. Multiple Fixed Points or Connection Points in a user-defined arrangement can be created, using a Fixed Point Group or Connection Point Group. These can be generated in a pattern or imported from a `.CSV` file.

To add a Fixed Point Group:

1. Select the **Fixed Points**object in the Outline tree view.
2. Right-click the **Fixed Points** object and select **Add > Fixed Point Group**, or click the **Fixed Point Group** icon in the **Fixed Points** toolbar.

To add a Connection Point Group:

1. Select a Part in the Outline tree view.
2. Right-click the Part and select **Add > Connection Point Group**, or click the **Connection Point Group** icon in the Parts toolbar.

In this section, the term ‘attachment point group’ refers to both Fixed Point Groups and Connection Point Groups. Most of the user options are the same for each type of object.

Details	
Details of Connection Point Group 1	
Name	Connection Point Group 1
Visibility	Visible
Type	Attached to Structure
Structure	Part
Origin Definition	
Origin Type	Global Origin
Position Points on Composite Cable Seabed	No
Origin X	0.0 m
Origin Y	0.0 m
Origin Z	0.0 m
Group Definition	
Group Type	Linear Pattern
Number of Points	3
Spacing	10 m
Group Rotation	
Rotation about Origin X	0.0°
Rotation about Origin Y	0.0°
Rotation about Origin Z	0.0°
Hydrodynamic Response Nodal Motions Output	
Include in Results	No

Set the **Name** of the attachment point group, then set the **Visibility** to choose whether the associated attachment points are shown in the Graphical Window.

For a Connection Point Group, the **Type** field will display 'Attached to Structure', and the **Structure** field will show the name of the parent Part. For a Fixed Point Group, the **Type** field will display 'Fixed'.

5.6.1. Origin Definition

Each attachment point group requires an Origin Definition. The meaning of the group origin depends on the **Group Type**, which will be described in the following section. The **Origin X**, **Origin Y** and **Origin Z** fields show the location of the group origin in the Global Axes. Select the **Origin Type** from the following options:

- ***Global Origin***: matching the Global Axes origin.
- **Manual Definition**: defined by the user, entering values directly into the **Origin X/Y/Z** fields.
- **Vertex Selection**: based on the position of a **Vertex**, which is picked from the geometry in the Graphical Window.
- **Fixed/Connection Point**: based on the position of a **Fixed/Connection Point**, which is selected from a drop-down menu of existing attachment points.

Where a Composite Cable Seabed is defined, you can force the attachment points to lie on this seabed by setting **Position Points on Composite Cable Seabed** to **Yes**. The **Origin Z** position will be updated to display the seabed Z location at the defined **Origin X/Y** position. This may be useful when you want to position an arrangement of Fixed Points on a sloped seabed, for example.

5.6.2. Group Definition

Select the **Group Type** to specify the arrangement of attachment points that you are defining:

- **Linear Pattern**: attachment points are created in a line, starting from the **Origin X/Y/Z** position. Use **Number of Points** to set the total number of attachment points in the group, and enter a **Spacing** to set the distance between each attachment point.
- **Circular Pattern**: attachment points are created in a circle, with the **Origin X/Y/Z** position at its center. Enter an **Angular Spacing** to set the angular distance between each attachment point around the circumference of the circle. Set the **Radius** to define the distance from each attachment point to the group origin. The points start from the origin X axis on the circumference of the circle prior to Group Rotation (p. 96).
- **Rectangular Pattern**: attachment points are created in a two-dimensional grid, starting from the **Origin X/Y/Z** position. Use **Number of Points in X** and **Number of Points in Y** to set the number of attachment points in each direction, and enter **Spacing in X** and **Spacing in Y** to set the distances between each attachment point.
- **Import from CSV File**: click the **Import CSV File** field to select a comma-separated values (`.CSV`) file from a location on the disk. The selected file must meet the following requirements:
  - Has the `.CSV` extension.
  - Contains values separated by commas, tabs, or single spaces (not multiple spaces).
  - When you are importing into a Fixed Point Group, the `.CSV` file should contain 4 columns, which are the **Name**, **Position X**, **Position Y** and **Position Z** of each point in the group.
  - When you are importing into a Connection Point Group, the `.CSV` file should contain 5 columns, which are the **Name**, **Position X**, **Position Y**, **Position Z** and **Include in Results** settings for each point in the group.

  The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data** to the required unit system.

  The imported **Position X/Y/Z** values are considered as offsets from the **Origin X/Y/Z** position.

  Note:

  The Group Definition table can also be exported as a `.CSV` file. To reimport the file as a new group, ensure that the file matches the specified column format: **Name**, **Position X**, **Position Y**, **Position Z**, and **Include in Results**.

Once the Origin Definition and Group Definition have been fully defined, the **Attachment Point Group Data** table will show the names and positions of each generated or imported attachment point.

Attachment Point Group Data				
Point Name	Position X (m)	Position Y (m)	Position Z (m)	Include in Results
Point 1	0.0	0.0	0.0	No
Point 2	10	0.0	0.0	No
Point 3	20	0.0	0.0	No

When you are generating attachment points in a Linear, Circular or Rectangular Pattern, each point will be given a default **Point Name**. You can edit this name directly in the Attachment Point Group Data table. For example, the third attachment point in this group would be referred to as 'Connection Point Group 1, Point 3' throughout the Aqwa Workbench interface. For imported attachment points, the **Point Name** is read from the `.CSV` file.

5.6.3. Group Rotation

You can apply a rotational transform to the attachment points in the group by specifying values in the **Rotation about Origin X/Y/Z** fields.

5.6.4. Hydrodynamic Response Nodal Motions Output

When creating a Connection Point Group, choose whether to make the associated Connection Points available for selected results output - see 5.5.16. Structure Connection Points for more information.

Select **Include in Results** from the following options:

- **No**: none of the Connection Points in this group will be available for results output.
- **Define in Table**: in the Attachment Point Group Data table, the **Include in Results** column will be editable, allowing you to select **Yes** or **No** for each Connection Point.
- ***Yes***: all of the Connection Points in this group will be available for results output.

In total, you may have up to 99 Connection Points for which the **Include in Results** option is set to **Yes**.

5.7. Dynamic Point Structures

A Dynamic Point is a moving point that is not associated with a Part.

Unlike Fixed Points (p. 46) and Connection Points (p. 84), Dynamic Points introduce additional equations of motion to the hydrodynamic solution and therefore must have their own mass and rotational inertia defined. They can also include hydrodynamic properties such as linear stiffness, added mass, damping and Morison drag.

Dynamic Points can be used to represent something physical such as the shackle on a bridle mooring, or a derrick hoist block; or they can be used as a modeling tool such as a reference structure in the time-domain user-defined force function. The mass and hydrodynamic properties assigned to a Dynamic Point can also represent something physical such as the mass and drag of a linear cable (which is otherwise without mass or drag) or they can be abstract.

Dynamic Points do not comprise any surfaces so they do not have frequency-dependent hydrodynamic properties. They are included in the Hydrodynamic Diffraction analysis because they must be written into the hydrodynamic database generated by the Aqwa solver. However, you cannot review any results for Dynamic Points under the Hydrodynamic Diffraction Solution as these results would not have any physical significance.

The initial position of a Dynamic Point is defined in the same way as a Fixed Point or Connection Point. Then, when setting the Connections, the Dynamic Point can be selected in place of a Connection Point. For example, a Cable with structure-to-structure connectivity could either be defined between two Dynamic Points, or a Connection Point and a Dynamic Point. A Dynamic Point can be used by more than one connection (Cable, Fender, and so on). For steps on adding and defining Dynamic Points, see Define Dynamic Points (p. 97).

In the setup of a Hydrodynamic Response analysis, you can apply Deactivated Freedoms, Starting Conditions, Point Forces and Structure Forces to a Dynamic Point, in the same way that you would for a Part.

Similarly, in the results of a Hydrodynamic Response analysis, the Dynamic Point can be considered as a structure. Meaning any result that can be generated for a Part such as a graph plotting Structure Position vs Time can also be produced for a Dynamic Point.

5.7.1. Define Dynamic Points

To add a Dynamic Point:

1. Select **Dynamic Point Structures** in the Outline tree.
2. Right-click and select **Add** **Dynamic Point** .

   OR

   Click the **Dynamic Point** icon in the **Dynamic Point Structures** toolbar.
3. Select the **Dynamic Point** object in the Outline tree and then, in the **Details** panel, define the dynamic point position, equivalent mass, and hydrodynamic properties as described in the following sub-headings.

5.7.1.1. Define Dynamic Point Position

The initial position of the Dynamic Point can be defined in one of three ways:

- Set **Definition of Position** to X, Y, and Z Coordinates, and then set the position coordinates (**X**, **Y**, **Z**) in the **Details** panel.

  The coordinates can be parameterized.
- Set **Definition of Position** to X and Y Coordinates, Z on Composite Cable Seabed. Then set the Position Coordinates (**X**, **Y**) and a positive **Z Offset** from the Composite Cable Seabed Water Depth at the given X, Y position.
- Set **Definition of Position** to Vertex Selection. Click **Select a Single Vertex** in the **Vertex** field, select a vertex on a structure, and then click **Apply**.

  You can then set an **X Offset**, **Y Offset**, or **Z Offset** from the vertex if needed.

5.7.1.2. Equivalent Mass Properties

Each Dynamic Point must have a **Mass** defined. The Mass value should be realistic, taking into consideration the scale of any other structures in the model; otherwise, the numerical stability of Hydrodynamic Response calculations may be affected. For example, if a ship with a mass of several thousand tons is connected to a Dynamic Point using a Cable (p. 101), the cable experiences a great amount of tension, and the Dynamic Point has a small nominal Mass value of `1.0`; the accelerations experienced by the Dynamic Point may be extremely large and could cause the time domain Aqwa solver to crash. It is therefore better to define a relatively large Mass value to start with and then reduce it after the Hydrodynamic Response calculations run reliably.

By default, the **Displaced Mass** will automatically inherit the Mass value so there is no overall gravity force on the Dynamic Point. You can change the Displaced Mass to any value that is zero or positive. Reset the Displaced Mass to match the Mass value if you want to recover the default inheriting behaviour.

The **Moment of Inertia Definition** allows you to choose how the Dynamic Point **Moment of Inertia** is defined. The following options are available:

- **Large Value (Minimise Rotation)**: by default, the Dynamic Point uses a very large value for the rotational inertia about each global axis so that the rotational motion is minimized.
- **Simple (Isotropic)**: a single value is defined, which is equally applied for the rotational inertia about each global axis.
- **Full (6 x 6 Matrix)**: the individual (including off-diagonal) terms that describe the full moment of inertia matrix, which are **Ixx**, **Ixy**, **Ixz**, **Iyy**, **Iyz** and **Izz**, are individually defined.

5.7.1.3. Hydrodynamic Properties

You can choose to define various hydrodynamic properties for each Dynamic Point. These include Linear Stiffness, Added Mass, Linear Damping and second-order Morison Drag terms. For each type of property, you can select from:

- **None**: by default, no properties are defined.
- **Simple (Isotropic)**: a single translation value and a single rotation value are defined, which are equally applied for every translational and rotational degree of freedom.
- **Full (6 x 6 Matrix)**: a matrix object is added as a child of the Dynamic Point, allowing you to define the full 6 x 6 matrix for the given property. This can be defined by manually entering values or importing a `.CSV`.

5.7.2. Modelling Considerations

Listed below are some points which are helpful to consider when including Dynamic Points in your hydrodynamic model:

- As described in Equivalent Mass Properties (p. 98), the Mass should be defined so that the behaviour of the Dynamic Point is realistic, particularly in the time domain. Unless the overall scale of the hydrodynamic model is small, the Mass of the Dynamic Point should not be small.

One exception to this would be where the Dynamic Point is connected by a Joint to a Part; however, the range of masses over all structures in the model should be less than 4 orders of magnitude (meaning a maximum 10,000x difference between the smallest and largest masses).

- While the physical mass should be represented by the Mass value, which introduces inertia and gravity force to the Dynamic Point, you can also use Added Mass to create inertia (only) in specific freedoms.
- The Aqwa Workbench editor checks that some constraint is applied to the Dynamic Point. For example, you will see a warning when you solve the Hydrodynamic Diffraction analysis if the Dynamic Point has a Mass which is larger than the Displaced Mass, and there is no connection to prevent it from falling freely due to gravity.
- Although Dynamic Points do not have any effect on the results of a Hydrodynamic Diffraction analysis, a change in the properties of a Dynamic Point will cause the Hydrodynamic Diffraction Solution to go out of date. This is because the hydrodynamic database created by the Aqwa solver needs to include the latest Dynamic Point properties before any Hydrodynamic Response calculations are performed. However, the Hydrodynamic Diffraction system should be relatively quick to update, as the radiation-diffraction calculation will not be repeated.

5.8. Line Body Data

When the geometry imported from DesignModeler/Discovery Modeling contains at least one Line Body, a Line Body Data object is automatically added under the Geometry object in the Outline tree. Beneath this object you will find the Beam Sections that have been automatically created to store cross section geometric properties from the imported geometry.

5.8.1. Beam Section

Beam Sections are automatically created to store the geometric properties of imported Line Body cross sections, and also allow you to define the mass and hydrodynamic properties of those Line Bodies.

Each Line Body should be assigned a cross section and an orientation in DesignModeler/Discovery Modeling. In the Aqwa editor, when importing the geometry, each valid cross section is automatically paired with a Beam Section object. Beam Sections are created under the Line Body Data object in the Outline tree.

The **Solver Line Type** indicates what kind of element will be used by Aqwa to represent any Line Bodies which inherit their properties from this Beam Section. Cylindrical sections are represented by tubular (TUBE) elements, and non-cylindrical sections are represented by slender tube (STUB) elements.

The Beam Section **Geometric Properties** contain all of the information that is transferred from DesignModeler/Discovery Modeling for a cross section. This includes the **Cross Section Name** and

Cross Section Type, the dimensions of the cross section, the **Cross Section Area**, and the **Second Moment of Area** (about both local Y and Z axes, for non-cylindrical sections).

Note:

It is not possible to change the geometric properties of a Beam Section in the Aqwa editor. To do this, you must edit the cross section in DesignModeler/Discovery Modeling and re-import the geometry.

Under **Mass Properties** you can define the **Material Density** of the Beam Section. The **Inertia/Unit Length** of the Beam Section (about the local Y and Z axes, for non-cylindrical sections) is automatically calculated as the product of the Material Density and the Second Moment of Area.

The **Hydrodynamic Properties** display the properties of the section that are relevant to the Line Body viscous drag and added mass calculations, and allow you to define corresponding **Transverse Drag Coefficients**, **Axial Drag Coefficient**, and **Added Mass Coefficients**. The **Displaced Area** is used in the added mass calculation. For non-cylindrical sections the **Dimension for Local Y Drag** and **Dimension for Local Z Drag** are used in the viscous drag calculation.

Duplicating Beam Sections

Where multiple Line Bodies have the same cross section but different mass or hydrodynamic properties, a Beam Section can be Duplicated. The duplicate Beam Section will retain the same cross section geometry, but can have different mass or hydrodynamic properties defined. In the Line Body details, the **Beam Section** field can then be used to select which Beam Section the Line Body inherits its properties from.

5.9. Define Connections

The Connections object allows you to create connections between structures or between structures and the environment. The available connections are:

- 5.9.1. Cables
- 5.9.2. Tethers/Risers
- 5.9.3. Connection Data
- 5.9.4. Connection Stiffness
- 5.9.5. Fenders
- 5.9.6. Joints

When the **Connections** object is selected in the Outline tree, and there are one or more Cables, Tethers/Risers, Fenders, or Joints defined, the **Hydrodynamic Response Connections Summary** table will be displayed:

Hydrodynamic Response Connections Summary							
	Activity	Connection	Aqwa Index	Type	Connectivity	First Attachment	Second Attachment
Cable 1	✓	Cable	1	Linear	Fixed Point to Fixed Point (via Pulley)	Anchor 1 (Fixed)	Pulley 1 (Part)
Cable 1	✓	Cable	2	Linear	Fixed Point to Fixed Point (via Pulley)	Pulley 1 (Part)	Pulley 2 (Part 2)
Cable 1	✓	Cable	3	Linear	Fixed Point to Fixed Point (via Pulley)	Pulley 2 (Part 2)	Anchor 2 (Fixed)
Joint 1	✓	Joint	1	Rigid	Fixed Point to Structure	Joint Fixed Point (Fixed)	Joint Attachment (Part)
Fender 1	✓	Fender	5	Fixed	Fender And Contact On Structures	Fender 1 (Part)	Fender 1 (Part 2)
Fender 2	✓	Fender	6	Fixed	Fender And Contact On Structures	Fender 2 (Part)	Fender 2 (Part 2)
Cable 2	✓	Cable	4	Nonlinear Catenary	Fixed Point to Structure	Anchor 3 (Fixed)	Mooring Fairlead (Part 2)
Tether/Riser 1	✓	Tether/Riser	1	Nonlinear	Fixed Point to Structure	Tether Anchor (Fixed)	Tether Attachment (Part 2)

The table summarizes the activity, type, connectivity, and attachment points for each connection included in a Hydrodynamic Response analysis (Connection Stiffness objects are not listed).

The Hydrodynamic Response Connections Summary table also displays the **Aqwa Index** of each connection as it is assigned by the Aqwa solver (the index number is assigned using the groupings: Cables/Fenders, Joints, and Tethers/Risers). This is useful if you are using the Hydrodynamic Pressure Mapping Add-On (p. 273) to transfer Cable, Joint, or Tether loads from a time domain Hydrodynamic Response analysis to a Static Structural analysis because the connections in the structural analysis are identified by their Aqwa Index. This table provides the relation between named connections in the hydrodynamic model and their Aqwa Index in the structural model.

For a linear cable which includes one or more pulleys along its length, each section of the cable is assigned its own index. In the example above, Cable 1 runs from Anchor 1, through Pulleys 1 and 2, to Anchor 2, so it appears three times in the Hydrodynamic Response Connections Summary table.

Note:

The Aqwa solver internally considers Fenders as another type of cable. Aqwa Workbench writes Fenders after Cables in the Aqwa input file; therefore, the Aqwa Index for a Fender starts at max(Aqwa Index for Cables) + 1.

5.9.1. Cables

Aqwa supports four types of cables, each with their own input requirements:

5.9.1.1. Linear Elastic

5.9.1.2. Nonlinear Polynomial

5.9.1.3. Nonlinear Steel Wire

5.9.1.4. Nonlinear Catenary

To add a Cable:

1. Select the **Connections** object in the tree view.
2. Right-click the **Connections** object and select **Insert Connection > Cable**, or click the **Cable** icon in the **Connections** toolbar.
3. Select the **Cable** object in the tree view and set properties.

The **Type** of the cable (**Linear Cable**, **Polynomial Cable**, **Steel Wire Cable**, or **Catenary Cable**) can be set in the details panel.

Cables may either be joined between two structures, between a fixed point and a structure, or between two fixed points via a pulley (only available for Linear Cables); this option can be changed via the **Connectivity** field in the Details panel. A cable cannot have both ends connected to the same structure. For the first two cases an **End Connection Point** must exist at the position of the end of the cable; this can be selected from a dropdown list of existing Connection Points defined on the structures. For the third case and **End Fixed Point** must be selected from a dropdown of existing Fixed Points. When the starting point is on a structure, you can define the start of the cable using **Start Connection Point**, which again is a dropdown list of existing Connection Points defined on the structures. When the starting point is a fixed point, a **Start Fixed Point** dropdown list of defined fixed points is shown.

Linear Elastic and Polynomial cables can be winched by adding a **Cable Winch** (p. 183). All cable types can also have break conditions based on tension or time, defined by adding a **Cable Failure** (p. 187) object.

Set additional options in the Details panel for each cable type as described in the sections below. The extension of the cable and force applied depend on the Connectivity defined (the following information is not applicable to catenary cables):

For a cable attached between a structure and a fixed point

The extension of the cable, at any stage of the analysis, is calculated by subtracting the unstretched length from the distance between the position of the fixed point and the current position of the connection point on the structure.

The direction of this force is given by the vector going from the fixed point to the structure.

For a cable attached between two structures

The extension, at any stage of the analysis, is calculated by subtracting the unstretched length from the distance between the connection points on the two structures at the current position of the respective structures.

The direction of the force on a structure is given by the vector going between the two connection points. The forces on each structure will therefore always be equal and opposite and hence the selection of start and end connection points can be interchanged.

The unstretched length is used to indicate the length at which the mooring line is slack; that is, if the distance between the two attachment points at either end of the mooring line is less than this value, then the tension in the mooring line will be zero. Although unusual, it is quite valid to input this value as zero where the 'cable' is never slack. However, in the special case where both ends of the cable are coincident, the direction of the force exerted by the cable is undefined and is automatically set to zero.

For a cable attached between two fixed points

At any stage of the analysis, the extension is calculated by subtracting the unstretched length from the total distance between each fixed point to the pulley on the connecting structure, at the current position of that structure. When two pulleys are specified, the distance between the pulleys is also included in the total distance.

At each fixed point, the tension in the cable acts in the direction of the vector between the fixed point and the adjacent pulley. The direction of the force on the structure is determined from the resultant force on the pulley due to the cable tensions on each side of the pulley.

Note:

If a cable is not defined properly before running the analysis, the cable will be drawn as a straight red line between the start and end, and an error will be reported in the Message window.

5.9.1.1. Linear Elastic

For linear elastic cables, a **Stiffness** and **Unstretched Length** need to be defined, and up to two pulleys can be defined along the line.

This is a very simple type of cable, simply a tension-only spring, where the tension is proportional to its extension, and the constant of proportionality is termed the stiffness. As the extension may vary during the analysis, the structure(s) to which the cable is attached will experience a force of varying magnitude and direction. The magnitude of this force, which is equal to the cable tension is given by Force = Stiffness x Cable Extension

Note that when the cable is slack, the cable extension is negative and the cable tension is set to zero.

Pulleys

A Pulley has the effect of intersecting the cable and will effectively extend the cable to pass via the pulley position. Adding a second pulley will extend the cable further from the first pulley to the end of the cable, hence the cable will travel from the cable start point to the first pulley then, if it exists, to the second pulley, and then to the cable end point. Pulleys can only be attached to structures, and at least one pulley is required for a cable with a **Connectivity** of **Fixed Point & Fixed Point**.

Along with the pulley position (defined by **Connection Point** which can be selected from the dropdown list of existing connection points), you must enter a **Friction Coefficient** for the pulley. The friction of the pulley is represented by $\frac{T_2}{T_1}$, where T_2 is the larger tension and $T_1$the smaller. $\frac{T_2}{T_1}$ is defined for the situation where the line turns through 180° around the pulley.

For sliding friction cases, a friction factor μ is calculated such that $\frac{T_2}{T_1}=e^{\mu\pi}$. The friction is then varied depending on how far around the pulley the line passes.

$\frac{T_2}{T_1}$ must be in the range $1 \le \frac{T_2}{T_1} \le 2$, with 1 being no friction.

5.9.1.2. Nonlinear Polynomial

For Nonlinear Polynomial cables, enter the polynomial coefficients (**Coefficient A**, **Coefficient B**, **Coefficient C**, **Coefficient D**, **Coefficient E**) and **Unstretched Length** of the cable. The coefficients of the polynomial define the force in the cable as a function of extension, therefore:

$$T=\begin{cases} A\Delta L+B(\Delta L)^2+C(\Delta L)^3+D(\Delta L)^4+E(\Delta L)^5 & \text{if}\Delta L>0 \\ 0 & \text{if}\Delta L\leq 0 \end{cases}$$

where:

A,B,C,D,E = polynomial coefficients.

ΔL = extension of the mooring line.

5.9.1.3. Nonlinear Steel Wire

For Nonlinear Steel Wire cables, enter the **Asymptotic Stiffness** and **Asymptotic Offset** in addition to the **Unstretched Length** of the cable. These constants are physical properties used in defining the tension/extension curve of a steel wire mooring line.

Tension in a steel wire mooring line is given by:

$$T=k\left(e-d\left(\tanh\left(\frac{e}{d}\right)\right)\right)$$

Where:

e = extension of mooring line

k = asymptotic stiffness (constant)

d = asymptotic offset (constant)

The names of the constants k and d arise from the fact that, at large values of extension, $\tanh\left(\frac{e}{d}\right)$ tends to unity and the equation tends to the asymptotic form:

$$T=k(e-d)$$

5.9.1.4. Nonlinear Catenary

Nonlinear Catenary Cables consist of up to 10 Catenary Sections and optionally 9 Catenary Joints between the sections. Catenary Sections and Joints can be defined under the Connection Data object and accessed as required to build up a number of Nonlinear Catenary Cables. For each cable, the **Section Type** field allows you to select the **Catenary Section** via a drop-down list; once chosen, the unstretched **Length** of that section can be entered. If more than one section is used then additional selections for the joint type are displayed to allow the selection of the **Catenary Joint**. The whole cable make-up is summarized in the Catenary Cable Definition Data table.

Each Nonlinear Catenary Cable can, optionally, be analyzed using Cable Dynamics to obtain the dynamic forces in the cable as well as their effect on the motion of structures in the analysis (Set **Use Dynamics** to Defer to Analysis Settings "Use Cable Dynamics" Option, and ensure that the **Use Cable Dynamics** option in your Hydrodynamic Response analysis is set to Yes). You can then enter the **Number of Elements** to use in the analysis of the cable.

If **Use Dynamics** is set to No for a cable whose Connectivity is Fixed Point to Structure, and no Composite Cable **Global Seabed Slope** has been defined in Geometry, the **Seabed Slope** parameter is made available. This value is the seabed slope (in degrees) specific to this mooring line where a positive slope is for the seabed to slope up from the anchor towards the structure Connection Point. If a **Global Seabed Slope** has been defined, the cable-specific **Seabed Slope** field will show "Determined from Global Seabed Slope".

Note:

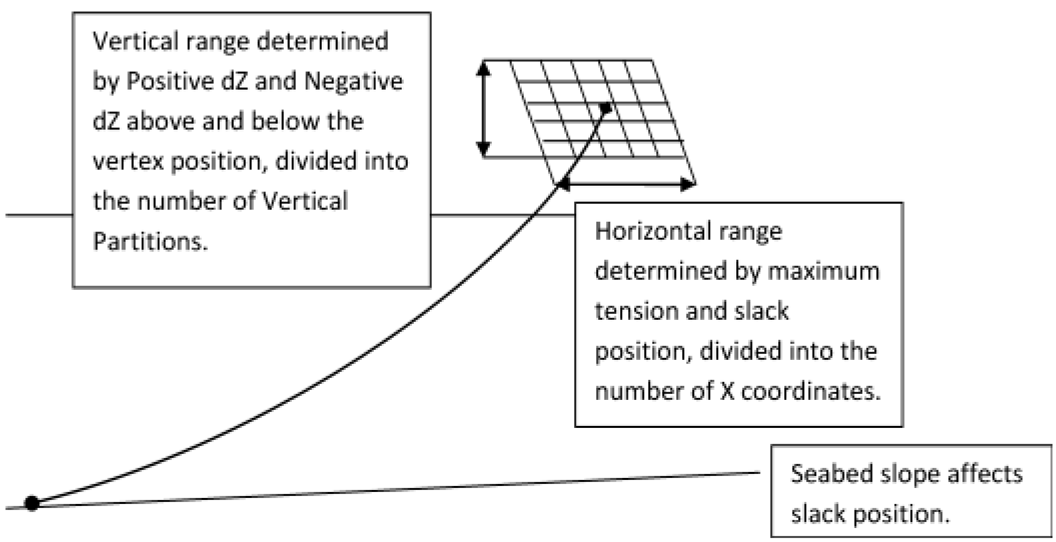
Seabed slope is ignored if Cable Dynamics is being used for the solution.

Once the cable is fully defined, the **Initial Cable Tension at Start** and **Initial Cable Tension at End** are reported in the Details panel.

To avoid the iterative calculation of the mooring forces, the program establishes a database covering all of the expected configurations of the cable.

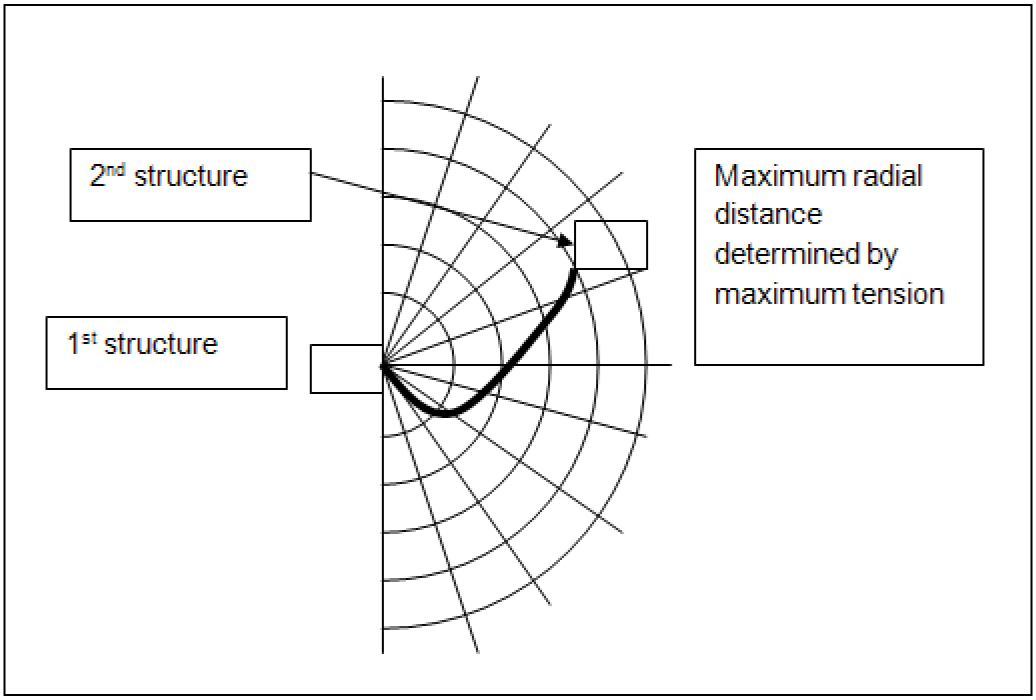
Additional controls enable fine control over the cable. The availability depends upon the cable **Connectivity**.

For cables attached between a fixed point and a structure (**Connectivity** set to **Fixed Point to Structure**), the range of the possible end points of the cable is determined by **Negative dZ Range of Expected Connection Point Vertical Motion** (measured from the lowest anticipated database point to the connection point in the definition position) and **Positive dZ Range of Expected Connection Point Vertical Motion** (measured from the connection point in the definition position to the highest anticipated database point) along with the slack and maximum tension positions (including the effects of the global or cable-specific seabed slope). This area is then divided up to form a database of cable end positions and corresponding tensions which is used in the analysis. It is recommended that the full size of this grid is used, of 600 points, which is formed by the multiplication of **Number of Vertical Partitions** and **Number of X Coordinates**.



For cables attached between two structures (**Connectivity** set to **Structure to Structure**), where the **Seabed Touchdown Expected** option is set to No, the range of the possible end points of the cable is determined by the relative positions of the two structures along with the slack and maximum tension range. The cable is considered to be submerged in water of infinite depth. It is recommended that the full size of the database is used, of 600 points, which is formed by the multiplication of **Number of Vertical Partitions** and **Number of X Coordinates**.

For cables attached between two structures where the **Seabed Touchdown Expected** option is set to No, the database uses a radial coordinate system in which **Number of X Coordinates** becomes the number of radial distances and the **Number of Vertical Partitions** is the number of angular positions.



For cables attached between two structures (**Connectivity** set to **Structure to Structure**), where the **Seabed Touchdown Expected** option is set to Yes, the range of the possible end points at either end of the cable is determined by **Negative dZ Range of Expected Connection Point Vertical Motion** (measured from the lowest anticipated database point to the connection point in the definition position) and **Positive dZ Range of Expected Connection Point Vertical Motion** (measured from the connection point in the definition position to the highest anticipated database point). If **Global Sloped Seabed** is non-zero, you must also specify the **Maximum Radius of Start Connection Point Horizontal Motion** (measured from the connection point in the definition position to the furthest anticipated database point). It is recommended that the full size of the database is used (600 points) which is formed by multiplication of **Number of Vertical Partitions** and **Number of X Coordinates**.

Caution:

Generation of the database required for a **Structure to Structure** cable with **Seabed Touchdown Expected** may increase the computational cost and memory requirement of the analysis significantly. This option should be set to Yes only where there is any chance a cable will touch down.

Note:

For a Structure to Structure cable which is resting on the seabed, the cable position illustrated in the Graphics panel will only account for the seabed effect after a Solve has been performed. For example, in a Stability Analysis, the cable may initially be shown hanging below the level of the seabed, but this is updated once the analysis has been solved.

Tip:

Where the **Global Seabed Slope** is non-zero, it is recommended that the cable **Start Connection Point** is positioned on the structure with the smaller expected range of motion of the two connected structures. Minimizing the **Maximum Radius of Start Connection Point Horizontal Motion** will improve the accuracy of the generated database.

For more information on the different quasi-static databases generated for each cable configuration, see The COMP/ECAT/SSCB Data Records - Composite Catenary Mooring Line in the *Aqwa Reference Manual*.

5.9.2. Tethers/Risers

Aqwa allows for the modeling of installed tethers, such as those used on Tension-Leg Platforms (TLPs), and drilling risers. As those features share some basic properties, they are combined into a single Tether/Riser object.

To add a Tether/Riser:

1. Select the **Connections** object in the tree view.
2. Right-click the **Connections** object and select **Insert Connection > Tether/Riser**, or click the **Tether/Riser** icon in the **Connections** toolbar.
3. Select the **Tether/Riser** object in the tree view and set properties.

Tethers/Risers may only be joined between a fixed point and a structure, therefore the **Connectivity** field in the Details panel is not editable for this type of connection. The **Start Fixed Point** drop-down list of defined fixed points allows you to select a fixed point for the tether/riser, while the **End Connection Point** can be selected from a drop-down list of existing Connection Points defined on the structures.

The Tether/Riser Local Axes (TLA) are defined with their origin at the position of the Start Fixed Point. The TLA Z-axis points from this origin toward the End Connection Point. The TLA X-axis is parallel to the Fixed Reference Axes (FRA) XY plane, and at right-angles to the TLA Z-axis. The TLA Y-axis follows the right-hand rule. When the tether/riser is vertical in the FRA, the TLA will be parallel to the FRA. For more information, see Mass and Stiffness Matrices in the *Aqwa Theory Manual*.

Tethers/Risers can consist of up to 24 Tether/Riser Sections, which can be defined under the Connection Data object and accessed as required to build up a number of Tether/Riser connections. For each tether/riser, the **Section Type** field allows you to select the **Tether/Riser Section** via a drop-down list. Once chosen, the unstretched **Length** of that section can be entered. The whole tether/riser make-up is summarized in the Tether/Riser Definition Data table.

In the Graphical Window, while the tether/riser object is selected in the Outline panel, the tether/riser is drawn in its unstretched condition. Any length from the Start Fixed Point to the End Connection Point which is not covered by the Tether/Riser Section Selection is shown as a yellow dashed line.

When the tether/riser object is not selected in the Outline panel, the tether/riser is drawn in its stretched condition.

Boundary conditions at each end of the tether/riser must be defined using the details under **Tether/Riser Boundary Conditions at Fixed Point** and **on Structure**. For each attachment point you must define an **Axial Stiffness** (in the direction of the Local Tether/Riser X-axis), and a **Rotational Constraint** which may be either Define Stiffnesses or Encastre Condition. Setting **Rotational Constraint** to Define Stiffnesses allows you to set rotational stiffnesses about the Local Tether/Riser Y- and Z-axes, while Encastre Condition prevents any rotation about the Local Tether/Riser Y- and Z-axes.

For a tether/riser which consists of more than one section, it is also possible to define **Intermediate Constraints** along its length. The **Additional Constraint** field contains a drop-down list of the positions between tether/riser sections. Selecting one of these positions allows you to set the **Constraint Type** which must be one of the following:

Rotational, Fixed

No rotations at this position relative to the Fixed Reference Axes (FRA)

Translational, Fixed

No translations at this position relative to the FRA

Rotational, on Structure

No rotations at this position relative to the Local Structure Axes (LSA) of the connected structure

Translational, on Structure

Restricted translations at this position relative to the LSA of the connected structure, where the maximum permissible translation from the definition position is specified by the defined **Gap**

Finally, there are a number of **Tether/Riser Specific Options** that may be optionally defined. Although some of these options are more relevant to the connection when it is considered as a tether or a riser, there is no limitation on the combination of options that may be used.

The **Number of Elements** field allows you to specify the number of finite elements which are used to model the tether/riser, up to a maximum of 250 elements.

The **Tether Group Multiplier** option specifies that a single modeled tether should be considered as a group of tethers, which has the effect of multiplying the tether forces acting on the connected structure by the specified integer factor.

The **Anchor** and **Structure Cap Areas** options can be used to set the areas of the tether/riser that are not subject to external water pressure at the fixed point and connection point, respectively. These values are used in the calculation of the tether/riser effective and wall tensions.

The **Internal Fluid Pressure** and **Density** options can be used to define the properties of the fluid carried by the riser, which are also used in the calculation of the tether/riser effective and wall tensions. If you define a non-zero pressure, you must also define a non-zero density (and vice-versa).

The **Axial Stress Impact Factor** and **Impact Half Life** options can be used to define the tether axial stress peak and decay during an impact. The initial axial stress σ_0 at impact is determined from the product of the impact velocity and the stress impact factor, while the exponential decay of the axial stress σ with time t is determined from the impact half life $t_{0.5}$ as:

$$\sigma(t)=\sigma_0 e^{\frac{-0.69315t}{t_{0.5}}}$$

If you define a non-zero stress impact factor, you must also define a non-zero impact half life (and vice versa).

The **Lower Stop Position** defines the distance of the tether lower stop below the fixed point position. Note that if the lower stop distance is input as zero, the tether can never be free-hanging.

5.9.3. Connection Data

Under the Connections object in the tree, a Connection Data object is automatically added. Under this object, you can insert definitions for various types of Catenary Sections, Catenary Joints, and Tether/Riser Sections that can be used to create Catenary Cables and Tethers/Risers.

5.9.3.1. Catenary Section
5.9.3.2. Catenary Joint
5.9.3.3. Tether/Riser Section

5.9.3.1. Catenary Section

To define the properties of a Catenary Section:

1. Select the **Connection Data** object in the tree view.
2. Right-click the **Connection Data** object in the tree and select **Insert Connection Data > Catenary Section**, or click the **Catenary Section** icon in the **Connection Data** toolbar.
3. Select the **Catenary Section** object in the tree and enter the following information:
    - The mass per unit length (**Mass / Unit Length**) of the section of the composite mooring line.
    - The **Equivalent Cross Sectional Area** of the mooring line. It is often more convenient, especially with wire lines, to specify this parameter so the buoyancy of the line may be calculated and subtracted from the structural weight to give the 'weight in water'. This parameter may also be specified as zero if the mass per unit length is input as the mass of the line LESS the mass of the displaced water per unit length (this does not apply to the cases when cable dynamic analysis is required, for which a non-zero equivalent cross section area must be defined).
    - The stiffness of the line (**Stiffness, EA**), specified in terms of EA, where E is Young's modulus and A is the cross sectional area of the line. The default value is chosen to give a typical value based on the mass/unit length. Clearly this may be in error if the mass per unit length specified includes buoyancy effects.
    - The **Maximum Expected Tension**, which is the highest value of tension that should be used in the database created for this composite mooring line. It is important that this is a realistic value. If a very high value is input the database will cover a very large range of tensions, and the accuracy in the actual working range may be reduced. If a very small value is input the database will only cover a small range of tensions, and constant tension may occur for larger

strain values, that is, no extrapolation is carried out. If cable dynamics is utilized this limiting value is not applied.

- The **Axial Stiffness Coefficients (k1, k2, k3)**. A cable may have nonlinear axial stiffness. The stiffness is calculated using the formula:

$$EA(\varepsilon)=EA(\text{constant})+k_1\varepsilon+k_2\varepsilon^2+k_3\varepsilon^3 \quad \text{for} \quad 0\le\varepsilon\le\varepsilon_{t\max}$$

or

$$EA(\varepsilon)=EA(\varepsilon_{t\max})+\{k_1+2k_2\varepsilon_{t\max}+3k_3\varepsilon_{t\max}^2\}\{\varepsilon-\varepsilon_{t\max}\} \quad \text{for} \quad \varepsilon>\varepsilon_{t\max}$$

where:

$EA(\varepsilon)$ = EA as a function of strain

$EA(\text{const})$ = the stiffness value input above

k_1,k_2,k_3 = Nonlinear axial stiffness coefficients

ε = linear strain $\frac{\sigma_L}{L}$

$\varepsilon_{t\max}$ = strain at $t_{\max}$

$t_{\max}$ = maximum tension specified above

For Cable Dynamics analyses the following optional additional data may be input.

- The **Bending Stiffness** (EI). The Bending Stiffness is specified in terms of EI, where E is Young's modulus and I is the 2nd moment of area of the line. The default value is zero.

- The **Added Mass Coefficient** (Ca). Added mass is calculated by ρ*Ca*A per unit length in which ρ is the water density and A is the equivalent cross section area. Meaning, the added mass is equal to the displaced mass of water multiplied by Ca. For cable dynamic analysis, the equivalent cross section area A must NOT be omitted. The default is 1.0.

- The **Transverse Drag Coefficient** (Cd). Transverse drag force is calculated by 0.5*ρ*Cd*V²*De per unit length where V is the relative transverse velocity. The default is 1.0

- The **Equivalent Diameter** (De) for drag. This allows the drag to be based on a different diameter from the added mass.

- The **Longitudinal Drag Coefficient** (Cx). Inline drag force is calculated by 0.5*ρ*Cx*V²*De per unit length where V is the relative inline velocity. The default is 0.025.

5.9.3.2. Catenary Joint

You can insert either a buoy or a clump weight between catenary cable sections (however, you do not need to specify a joint). Intermediate buoys always have the same buoyancy and do not "know" where the surface is. Therefore they may float above the water surface.

To define the properties of a Catenary Joint:

1. Select the **Connection Data** object in the tree view.

2. Right-click the **Connection Data** object and select **Insert Connection Data > Catenary Buoy** or **Catenary Clump Weight**, or click the **Catenary Buoy** or **Catenary Clump Weight** icon in the **Connection Data** toolbar.

3. Select the **Catenary Joint** object in the tree and enter the following information:

   - **Section Joint Type** should be set to Buoy or Clump Weight based on your menu selection when adding the object.
   - Specify the **Structural Mass** of the buoy or clump weight. This must be smaller than the mass of displaced water for a buoy, or larger for a clump weight. This can be positive, zero or negative.
   - Specify the mass of water displaced (**Displaced Mass of Water**), that is, the buoyancy/gravity. This can be positive, zero or negative.
   - Specify the total (constant) **Added Mass**; that is, not the added mass coefficient. Applicable to Cable Dynamics only.
   - Specify the **Drag Coefficient * Area** (cable dynamics only); drag will be in the direction of the relative velocity of the fluid, VR. The magnitude of the force is given by FD = 0.5 * ρ * CDA * VR * |VR| where CDA = Drag coefficient * projected area.

5.9.3.3. Tether/Riser Section

To define the properties of a Tether/Riser Section:

1. Select the **Connection Data** object in the tree view.

2. Right-click the **Connection Data** object and select **Insert Connection Data > Tether/Riser Section**, or click the **Tether/Riser Section** icon in the **Connection Data** toolbar.

3. Select the **Tether/Riser** object in the tree and enter the following information:

   - The **Density** of the material of which this section is composed.
   - The **Young's Modulus** of the material of which this section is composed.
   - The **Outer Diameter** of this section of the Tether/Riser.
   - The wall **Thickness** of this section of the Tether/Riser.
   - The **Added Mass Coefficient** (C_a). Added mass is calculated as $\rho C_a A$ per unit length, where ρ is the water density and A is the cross-sectional area. That is, the added mass is equal to the displaced mass of water multipled by C_a. The default is 1.0.
   - The **Transverse Drag Coefficient** (C_d). Transverse drag force is calculated as $0.5\rho C_d V^2 D$ per unit length, where V is the relative transverse velocity and D is the outer diameter. The default is 0.75.

5.9.4. Connection Stiffness

The Response Amplitude Operators (RAOs) and Quadratic Transfer Functions (QTFs) calculated in a Hydrodynamic Diffraction analysis assume that each structure is freely floating. The RAOs and QTFs do not account for any Connections (Cables, Joints, Fenders, Tether/Risers) that may have been defined between structures, or between a structure and the ground. However, under the **Connections** object in the Outline tree, you can define one or more **Connection Stiffness** matrices. These linear stiffnesses are included in the RAO/QTF calculations during the Hydrodynamic Diffraction analysis and can be used to model the (linear) effect of, for example, a mooring system.

Connection Stiffness can be defined in one of two ways. You can either manually define the structure connectivity and stiffness values, or import an MFK file which has previously been generated by the Aqwa solver.

Note:

- A Connection Stiffness object applies only to a Hydrodynamic Diffraction analysis.
- The stiffness matrix is defined with respect to the motions of the connected structure COGs.
- If you want to include the effects of Linearized Morison Drag in the analysis, you cannot have two structures connected with a Connection Stiffness matrix.

5.9.4.1. Import MFK File

As part of the Hydrodynamic Response Stability Analysis, the Aqwa solver will create a text-based MFK file which contains the global (linear) stiffness due to Cables, Fenders and Tether/Riser connections, relative to the equilibrium position of each structure. This includes the coupled stiffness terms between structures that are connected by Cables, or in contact through Fenders.

To import an MFK file:

1. Select the **Connections** object in the tree view.
2. Right-click the **Connections** object and the click **Insert Connection > Connection Stiffness > Import MFK File…**

   OR

   In the **Connections** toolbar, click **Connection Stiffness > Import MFK File…**
3. In the **Open** dialog box, select the MFK file to import the data.

Note:

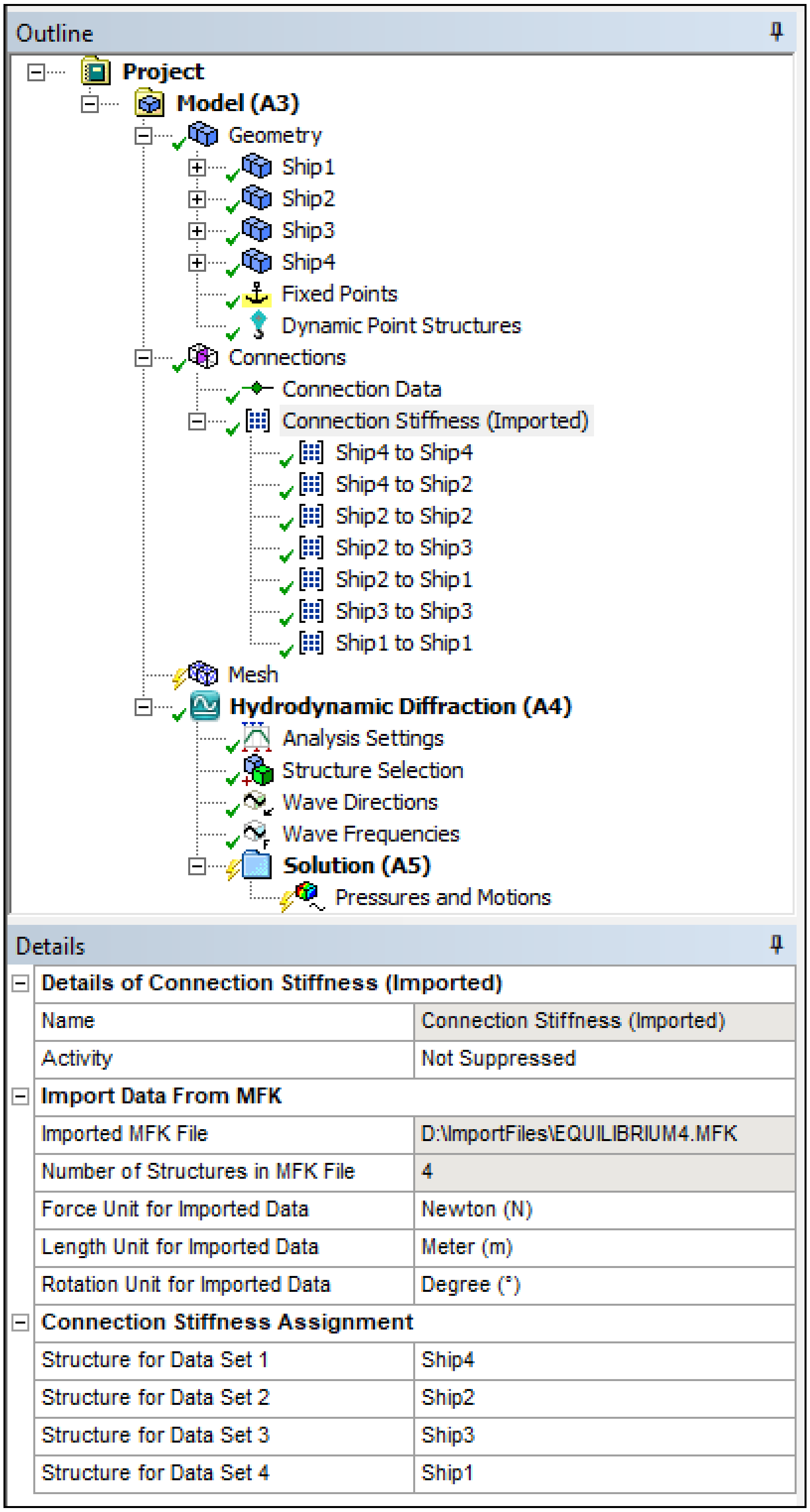
The format of the MFK file is described by comments within the file itself. You can create your own MFK file, or modify an existing file, for importing stiffness data into the Aqwa Workbench editor. The editor requires only the CONFIGURATION and GLOBAL-STIFF data records to be defined.

After an MFK file has been imported, a Connection Stiffness (Imported) object will be added to the Outline tree. Beneath this, one or more Matrix objects are also added, which are populated with the stiffness values contained in the MFK file.

When the Connection Stiffness (Imported) object is selected, the Details panel shows the **Imported MFK File** path and the **Number of Structures in MFK File** for your reference.

The unit system of the data in the MFK file is assumed to match the display unit system of the project with the default rotation unit in radians. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Force Unit for Imported Data**, and **Rotation Unit for Imported Data** to the required unit system.

The structure numbering in the MFK file is assumed to match the order of Parts as they appear in the Outline tree. However, you can use the **Connection Stiffness Assignment** options to re-assign Parts to structure numbers as necessary.



5.9.4.2. Manual Input

This object may be used to input a connection stiffness matrix between structures using tabular input in the Matrix Definition Data window.

To add a Connection Stiffness Matrix:

1. Select the **Connections** object in the tree view.
2. Right-click the **Connections** object and then click **Insert Connection > Connection Stiffness>Manual Input**.

   OR

   In the **Connections** toolbar, click **Connection Stiffness > Manual Input**.
3. Select the **Connection Stiffness** object in the tree and then, in the **Matrix Definition Data** window that appears below the graphics window, enter the matrix coefficients.
4. Set the **Connectivity** and structure information as described below.

The **Connectivity** can be set to Fixity & Structure for a structure connected to a point, or Structure & Structure for two connected structures. Select the first and second (if present) structure in the **Connected Structure A** and **Connected Structure B** fields.

The **Position at Equilibrium** field is available when **Connectivity** is set to Fixity & Structure. With **Position at Equilibrium** set to Geometric Position, the Response Amplitude Operators (RAOs) of the structure are calculated at the structure position as defined by the Geometry. If the **Position at Equilibrium** field is changed to Manual Definition, the **Position X/Y/Z** and **Rotation About X/Y/Z** fields can be used to define an alternative position for the additional stiffness term in the structure RAO calculation. These options are for the purpose of calculating the structure RAOs only – the structure is not moved for the analysis.

Matrix Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 6 columns and 6 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Force Unit for Imported Data**, and **Rotation Unit for Imported Data** to the required unit system.

A connection stiffness matrix between two structures has 12 degrees of freedom. It will normally require four 6×6 submatrices to be defined, as shown below:

$$K=\begin{bmatrix} [k_{1,1}] & [k_{1,2}] \\ [k_{2,1}] & [k_{2,2}] \end{bmatrix}$$

where each $[k_{i,j}]$ is a 6×6 submatrix. Each of these matrices must be defined as described above. The submatrices in the leading diagonal can be defined by using Fixity & Structure or by setting the first and second structures to be the same. Only one of the off-diagonal submatrices must be defined, as it is assumed that the overall matrix is symmetric. Defining both off-diagonal matrices is not permitted.

If you wish to represent multiple connections to the same structure using Connection Stiffness Matrices, the leading diagonal submatrices for that structure (the $\left[k_{1,1}\right]$ terms) must be summed. You cannot have multiple definitions of the same submatrix.

5.9.5. Fenders

Fenders are a type of mooring, which allow you to model material contact between two structures. They are the only way to model contact, so if there are no fenders, structures can pass through each other. From a practical point of view, like other mooring elements fenders are defined as acting between two structures or one structure and a fixed connection point.

Depending on the relative positions of the structure, on the type of Fender (floating, fixed unidirectional, fixed omnidirectional), and on their positions on the structures, they create a varying force acting on the structure and added to the other forces used for computing the structures' motions. These forces are calculated using the properties defined for each fender.

A Contact Plane is defined for each fender; this is the plane which the fender is going to impact. This plane is defined by a point and a vector.

An attachment point is defined for each fender. For fixed fenders, this is the point where the fender is located on the structure. For floating fenders, the attachment point is translated to the mean water surface.

Fixed unidirectional and floating fenders also have a direction of action, represented by a "normal vector". Along with the attachment point given in the fender's definition, it defines a plane which is going to be the second plane pressing on the fender. For floating fenders, the actual attachment point is obtained by translating the original definition point down to the mean water surface, following this plane. It therefore models the ship's side, which is not necessarily vertical. Omnidirectional fixed fenders act in all directions.

The size of the fender must be specified. At each time step, the distance between the fender's attachment and the contact plane is calculated and compared to the fender's size. If the distance is shorter than the size, a force is calculated as a function of the difference following a polynomial law whose coefficients are part of the fender's definition. This force is applied to the structure at the fender's attachment point. The calculation of the distance depends on the type of fenders. Fixed unidirectional and floating fenders have a direction along which the distance is calculated. For the omnidirectional fender, the distance is simply the shortest distance from the contact plane to the attachment point. Fenders can only have compressive forces.

To add a Fender:

1. Select the **Connections** object in the tree view.
2. Right-click the **Connections** object and select **Insert Connection > Fender**, or click the **Fender** icon in the **Connections** toolbar.
3. Select the **Fender** object in the tree and enter the following information:
   - **Connectivity** – Select the type of connectivity for the fender:

- Fender and Contact on Structures
- Fender On Structure, Contact On Fixed Point
- Fender On Fixed Point, Contact On Structure

- **Type** – Fixed or Floating. Note that Floating fenders cannot have a horizontal connection plane. If there is more than a 60 degree angle between the Floating fender and contact plane directions, a warning is generated saying that the forces are in error. But this will not prevent the run from completing.
- **Action** – For a Fixed fender, you can select Omni-Directional, or for either fender type choose the axis (X Direction Only, Y Direction Only, or Z Direction Only) for a uni-directional fender.
- **Fender Connection Point**, **Contact Connection Point** – Allows you to select an existing Connection Point on a structure to define the fender attachment point and contact plane origin. These are present when the Fender/Contact is on a structure.
- **Fender Fixed Point**, **Contact Fixed Point** – Allows you to select from a dropdown list an existing Fixed Point for the Fender/Contact when the connectivity is specified as such.
- **Damping Coefficient** – Material (or structural) damping coefficient β. Damping is modeled as linear material damping, where the damping coefficient is β x the stiffness. Damping is only applied in the direction perpendicular to the contact points.
- **Friction Coefficient** – This is the friction coefficient μ. The friction force is given by $F = \mu R$, where R is the normal reaction.
- **Size** – The fender size.
- Polynomial **Coefficient (A, B, C, D, E)** – The force acting on the structure is $Ax + Bx^2+Cx^3+Dx^4+Ex^5$, where x is the compression applied to the fender.

Fender friction works best in situations where the friction force is smaller than other forces in the same direction. Friction will slow down relative motion between two structures, but is not suitable for keeping them fixed together - there is no "stiction". When the relative velocity changes sign the friction force must also change sign, but to avoid an instantaneous change in force (and therefore an instantaneous change in acceleration) a smoothing function is applied. This means that when the relative velocity is very small the friction force is also small, and the structures can move relative to each other.

Table 5.1: Fender Color Codes

Fender Color	Description
Orange	The fenders are stationary and not in contact.
Red	The fenders are in contact.
Purple	The fenders are deactivated.
Black	The fender compression force has exceeded the maximum permissible value based on the defined fender stiffness coefficients.

When inserting a fender, **Fender Axes** and **Contact Axes** objects are inserted as its children. These axes objects define the orientation of the fender and the contact plane. The orientation can be set using these fields in the Details panel for each axes object:

- **Alignment Method** – Select Global Axes to align the axes with the global axes. You can also set the alignment of the axes using the Vertex Selection or Direction Entry methods.
- **Origin Vertex**, **X Direction Vertex**, **Vertex Defining the XY Plane** – For For an Alignment Method of Vertex Selection, select the **Origin Vertex** of the fender or contact plane, a vertex defining the X direction (**X Direction Vertex**), and a vertex that would, along with the other two vertices, define the XY Plane (**Vertex Defining the XY Plane**). As these vertices must be part of a body in the geometry, you may have to anticipate and include, for instance, a dummy massless line body in the geometry oriented the same way as the fender to be defined. In DesignModeler you can, for example, create such a line body as being perpendicular to the surface of the hull.
- **Rotation about Global Z**, **Rotation about Local Y**, **Rotation about Local X** – For an Alignment Method of Direction Entry, define the alignment using these three rotation fields.

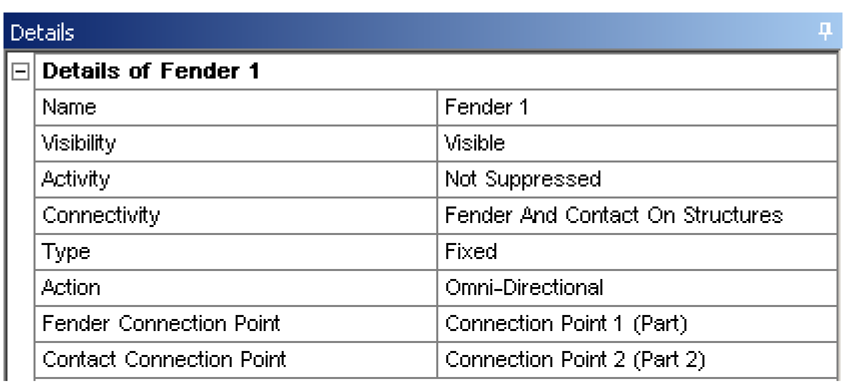
Note:

The Fender Axis definition should point towards the contact surface; meaning, through the fender itself. The Contact Axis definition should point towards the fender attachment surface; meaning, the opposite direction to the Fender Axis definition.

5.9.5.1. Examples of Use of Fenders

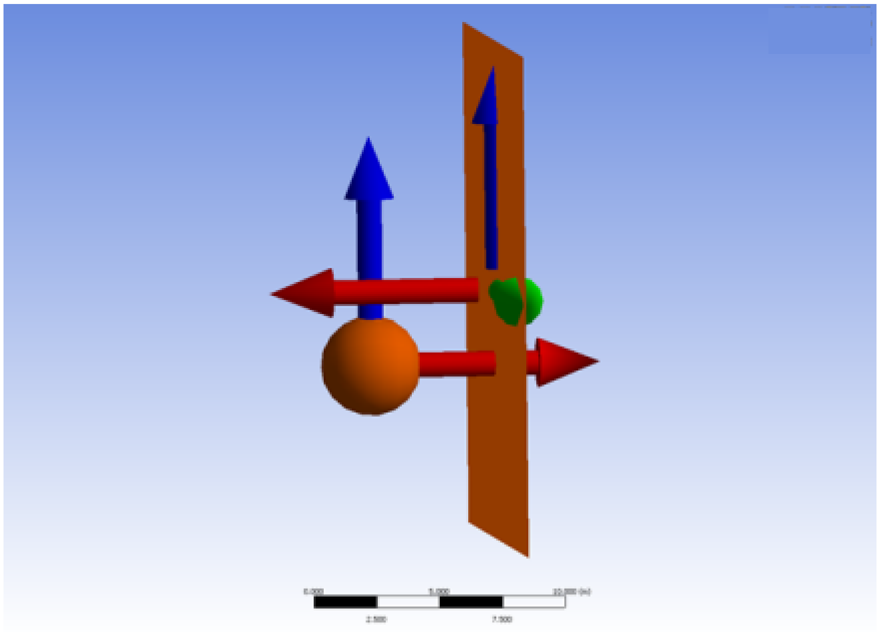
The following images show examples of using omni- and uni-directional fenders with various alignment methods.

Omni-Directional Fender



Details

Details of Fender 1	
Name	Fender 1
Visibility	Visible
Activity	Not Suppressed
Connectivity	Fender And Contact On Structures
Type	Fixed
Action	Omni-Directional
Fender Connection Point	Connection Point 1 (Part)
Contact Connection Point	Connection Point 2 (Part 2)

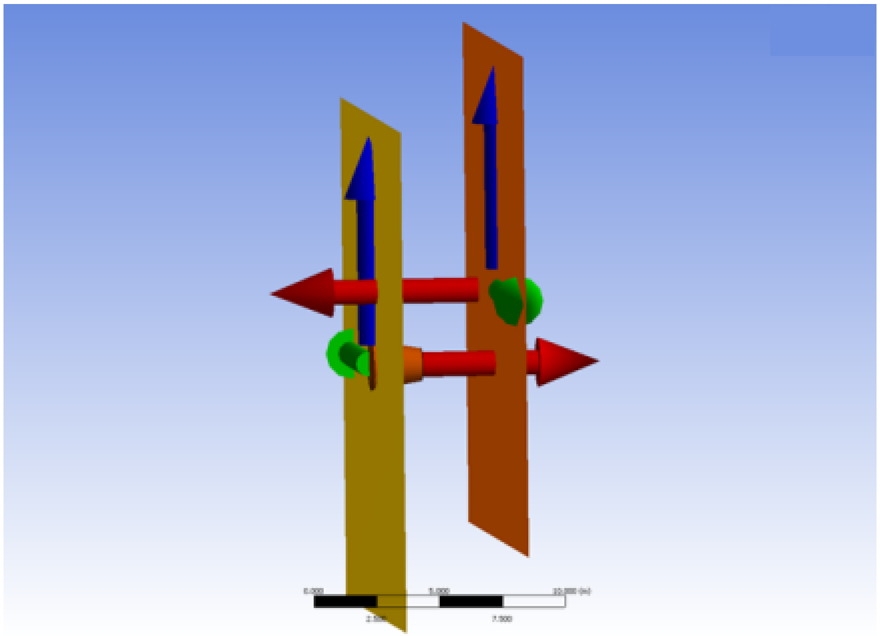


Uni-Directional Fender Aligned to X Axis

Details

Details of Fender 1

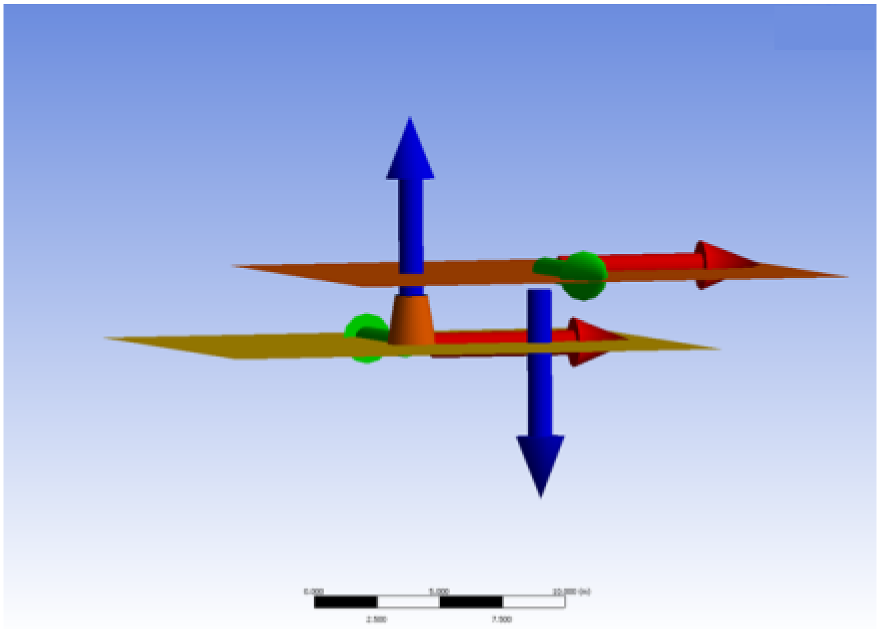
Name	Fender 1
Visibility	Visible
Activity	Not Suppressed
Connectivity	Fender And Contact On Structures
Type	Fixed
Action	X Direction Only
Fender Connection Point	Connection Point 1 (Part)
Contact Connection Point	Connection Point 2 (Part 2)



Uni-Directional Fender Aligned to Z Axis

Details

Details of Fender 1	
Name	Fender 1
Visibility	Visible
Activity	Not Suppressed
Connectivity	Fender And Contact On Structures
Type	Fixed
Action	Z Direction Only
Fender Connection Point	Connection Point 1 (Part)
Contact Connection Point	Connection Point 2 (Part 2)



5.9.6. Joints

A hydrodynamic analysis allows structures to be connected by articulated joints. These do not permit relative translation of the two structures but allow relative rotational movement in a number of ways that can be defined by the user.

A maximum of 99 joints is allowed.

To add a Joint:

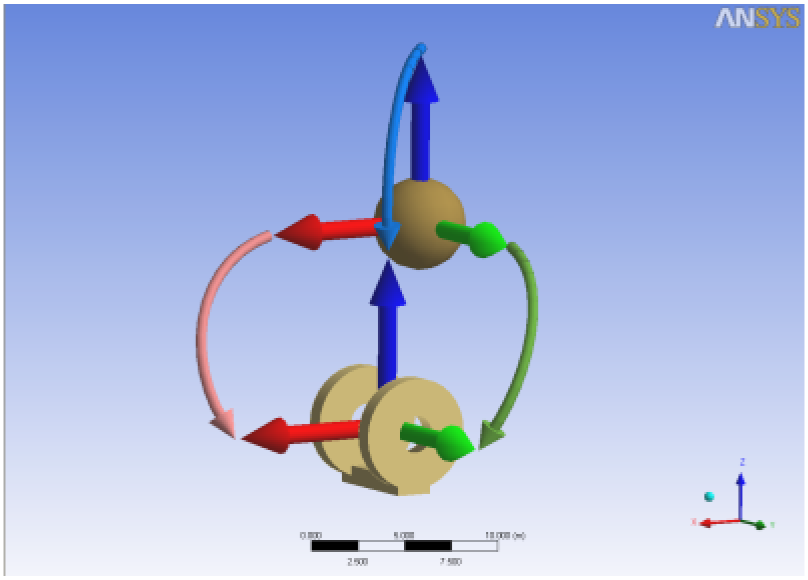
1. Select the **Connections** object in the tree view.
2. Right-click the **Connections** object and select **Insert Connection > Joint**, or click the **Joint** icon in the **Connections** toolbar.
3. Select the **Joint** object in the tree and enter the following information:
   - **Type** – To add a joint, select the type of joint you are adding:

     Note:

     In the following images, the joints are shown with their two ends split apart. Each end is attached to one of the two joint structures (or a structure and a fixed connection point). Each end of the joint has a set of axes attached to it that corresponds to its position with respect to the structure that it is attached to. The curved, colored arrows in the figures below indicate how the two ends will be connected. When the two sets of axes are coincident, the starting position of the structures in the simulation may be different from that in the imported geometry.

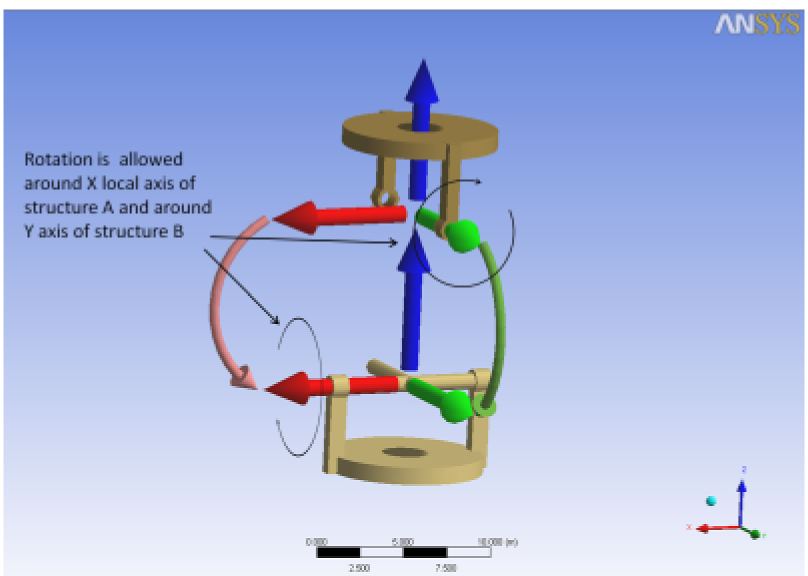
     – Ball and Socket. Free to rotate about all axes.

Figure 5.5: Ball and Socket Joint



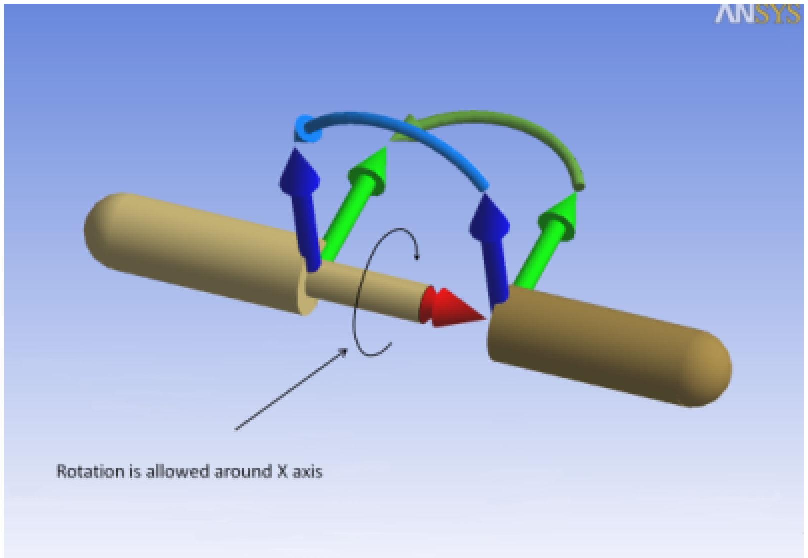
- Universal. Free to rotate about two axes transmitting a moment about the third axis at right angles to the first two.

Figure 5.6: Universal Joint



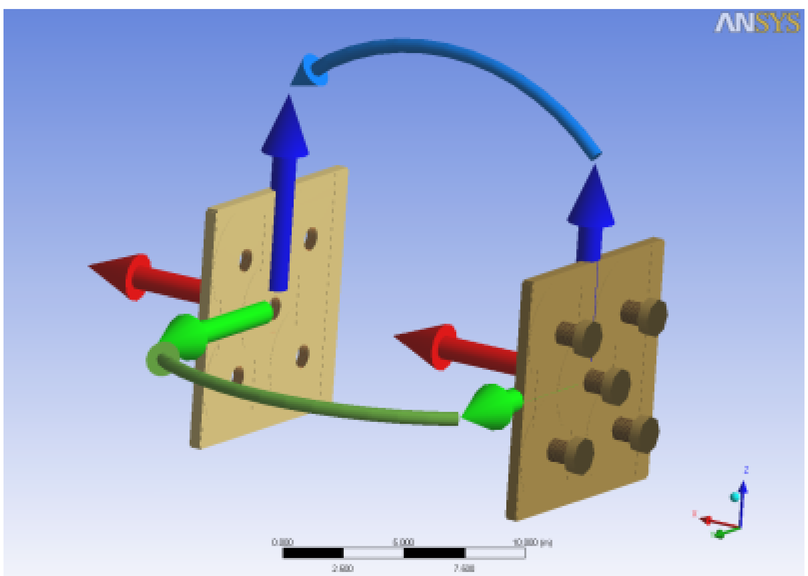
- Hinged. Transmitting a moment about two axes and free to rotate about the third axis at right angles the first two.

Figure 5.7: Hinged Joint



- Rigid. Transmitting a moment about all three axes and not free to rotate at all. This type of constraint enables you to find the reactions between two or more structures. This type of joint rigidly connects the structures together so that the solution of the equations of motion is the same as if one structure was defined.

Figure 5.8: Rigid Joint



In order to see the shape of the connections more easily, you can switch to wireframe mode.

- **Connectivity** – Select the type of connectivity for the joint:
  - Fixed Point and Structure
  - Structure and Structure

- **Fixed Point** – For Connectivity of Fixed Point and Structure, select the fixed point from a dropdown list of existing fixed points.
- **Connection Point on Structure (A/B)** – Allows you to select the connection point on the contact structure(s) from a dropdown of existing connection points.
- **Stiffness About X,Y,Z** – These three fields define the rotational stiffness about the X, Y, and Z axes.
- **Damping About X,Y,Z** – These three fields define the rotational damping about the X, Y, and Z axes.
- **Translation Friction Coefficient** – Friction coefficient for transverse force (k_1).
- **Rotational Coefficient** – Friction coefficient for overturning moment (k_2).
- **Axial Friction Coefficient** – Friction coefficient for axial force (k_3).
- **Constant Friction Moment** – Constant friction moment (k_4).

  The frictional moment is given by:

$$M=\varepsilon\left(k_1\sqrt{\left(F_y^2+F_z^2\right)}+k_2\sqrt{\left(M_y^2+M_z^2\right)}+k_3F_x+k_4\right)$$

  where:

  $\varepsilon=0$ if the relative rotational velocity is less than 0.001 rad/s, 1 otherwise

  k_1,k_2,k_3,k_4 are coefficients. These are not conventional dimensionless friction coefficients, as used in the equation $F=\mu R$. These coefficients are factors to be applied to the appropriate forces to give frictional moments, and they must include effects of the bearing diameter etc. k_1 through k_3 must not be negative. k_1 and k_3 have dimensions of length, and the maximum value allowed is $\frac{0.025g}{g_{SI}}$ where g is the acceleration due to gravity and g_{SI} is the acceleration due to gravity in the [kg, meter, second] unit system. k_2 is non-dimensional and has a maximum value of 0.025.

When inserting a joint, two axis objects are inserted as its children (**Joint Target Axes** and **Joint Contact Axes**). These axes objects define the orientation of the two objects that are connected by the joint. The orientation can be set using these fields in the Details panel for each axes object:

- **Alignment Method** – Select Global Axes to align the axes with the global axes. You can also set the alignment of the axes using the Vertex Selection or Direction Entry methods.
- **Origin Vertex**, **X Direction Vertex**, **Vertex Defining the XY Plane** – For an Alignment Method of Vertex Selection, select the **Origin Vertex** of the axis, a vertex defining the X direction (**X Direction Vertex**), and a vertex that would, along with the other two vertices, define the XY Plane (**Vertex Defining the XY Plane**).

- **Rotation about Global Z**, **Rotation about Local Y** ,**Rotation about Local X** – For an Alignment Method of Direction Entry, define the alignment using these three rotation fields.

> **Note:**
>
> If your joint is connecting the structure to a fixed connection point, it will be indicated by a black sphere. This black sphere is not to be confused with the sphere of a ball and socket joint.

5.9.6.1. Initial Positioning of Jointed Structures

In hydrodynamic analyses, inserting joints between two structures affects the initial position of these structures when the analysis is run.

The program automatically determines the initial positions based on the defined joints so that you do not have to edit the model's initial geometry. If two connection points are not coincident, the program moves one of the structures to align the connection points before running the analysis. The geometry movement will not be shown in the viewer.

You can control which structure will remain in a defined position and which structure is free to move by selecting **Connection Point on Structure A** or **Connection Point on Structure B** in the details dialog for the joint. Fixed connections are shown in black.

The rules for defining fixed and movable structures are as follows:

1. If neither of the two is connected to a fixed connection point, the structure with the **Connection Point on Structure A** option will remain at its original position and the structure with the **Connection Point on Structure B** option will be moved.

2. The **Connection Point on Structure A** or **Connection Point on Structure B** designations will be overridden in some situations. If one of two structures is connected to a fixed connection point or is part of a group of articulated structures that are already connected to a fixed connection point, then that structure will remain in a defined position and the other structure will be moved. If this second structure is also part of a group of articulated structures that are not connected to any fixed connection point, the full group to which this structure is already connected will also be moved. Note that this action is independent of the **Connection Point on Structure A** or **Connection Point on Structure B** designations.

3. If one or both of the structures are part of groups of articulated structures and none of these structures is connected to a fixed connection point, then rule #1 applies and the structure with the **Connection Point on Structure B** designation will be moved, along with its group of structures.

4. If both articulated groups are already connected to fixed connection points, the new joint is then going to close the loop. See rules for Closed Loops (p. 126) below.

5.9.6.2. Moving Structures

Structures are moved in two steps: They are first translated, which causes the joint points to be coincident, and then the structures are rotated so that the two sets of axes attached to the joint object become superimposed. This gives you the freedom to set an initial orientation for the structures at the start of the Time History Analysis. You will not see the structures move in the viewer.

The resulting set of axes defines the local joint axes at the start of the Time History Analysis. In this set of axes, some rotational motion of the structures will be allowed during the Time History Analysis, depending on the type of joint selected:

- **Ball and Socket Joint**: all rotations are allowed.
- **Universal Joint**: rotations are allowed around the X and Y axes of the local axes system.
- **Hinged Joint**: rotations are only allowed around the X axis of the local axes system.
- **Rigid Joint**: no rotation is allowed.

5.9.6.3. Closed Loops

In hydrodynamic analyses, the following conditions and recommendations apply to closed loops in articulated groups of structures:

- The joint positions in the closed loop must be geometrically compatible. It is not possible to connect two structures using two joints if the distance between these joints is different on both structures.
- In some cases, use a single joint of a different type that performs the same function as two simpler joints. For example, two ball and socket joints can be replaced by a single hinged joint or two hinged joints can be replaced by a rigid joint.
- A closed loop should not introduce any redundancy in the way that the degrees of freedom will be locked.

Figure 5.9: A Closed Loop with No Redundant Constraints

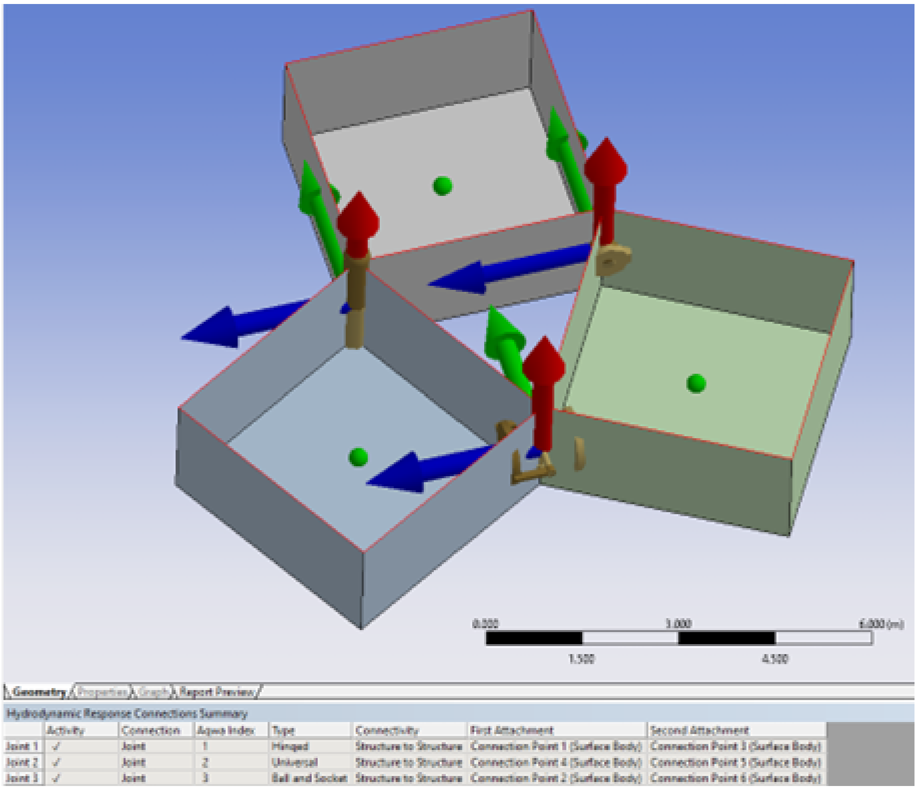


Figure 5.9:**A Closed Loop with No Redundant Constraints** (p. 127) shows a closed loop joining three wireframe structures. Three articulated structures are connected to form a triangle. One of the joints is a vertical hinged type. The second one is a universal type where neither the joint's local x- nor y-axis is parallel to the line passing through the connection points of the second and third joints. The third one is a ball and socket joint because making it the same as the other joints results in redundancy.

Figure 5.10: A Closed Loop with Redundant Constraints

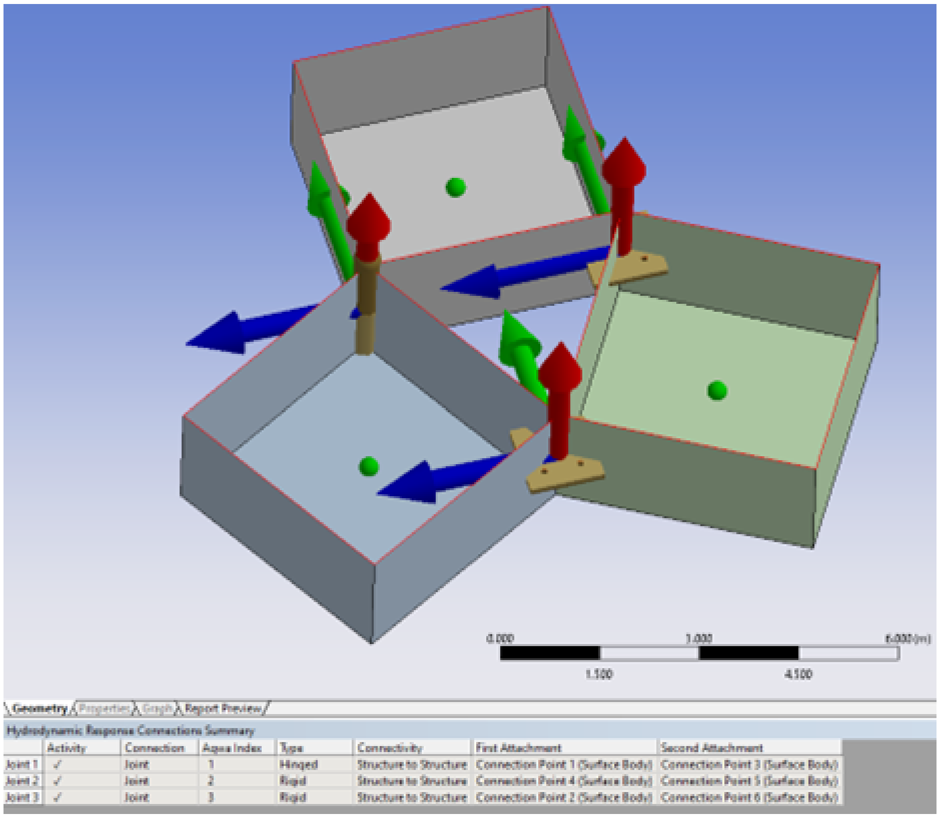


Figure 5.10: A Closed Loop with Redundant Constraints (p. 128) shows a closed loop joining three wireframe structures. The ball and socket and universal joints from Figure 5.9: **A Closed Loop with No Redundant Constraints** (p. 127) have been replaced by the rigid joints which have added five redundant constraints to the problem. This results in the statically indetermined solution of the reaction forces/moments on the joints.

When you close a loop, a warning message is produced to inform you that a loop is being closed which may cause a statically indeterminate solution while solving.

5.9.6.4. Removing a Joint

Joints can be removed from an analysis temporarily or permanently by right-clicking on the **Joint** item in the tree and selecting **Suppress** or **Delete**. The result is that the group of articulated structures that contains the joint is broken into two parts. Closed loops that are suppressed or deleted are opened.

If the starting positions of some structures were modified when the joint was created, they will be moved again using the following rules:

- If the two structures are not part of any group, they will be returned to the positions they held when the geometry was created.
- If more than two structures are part of the group to be broken apart, the program positions both subgroup structures where they would have been if the two subgroups had been originally created with the remaining joints. The suppressed and deleted joints are not considered as part of the new arrangement. The existing joints are applied in chronological order.

5.10. Establish Analysis Settings

The **Analysis Settings** object allows you to specify how the analysis runs and how the results are shown. Different fields appear in the Details view for the object depending on the parent object (Hydrodynamic Diffraction or Hydrodynamic Response).

Analysis options vary depending on the analysis system and the computation type. The following topics are available:

5.10.1. Specifying External Pre- or Post-Solve Operations

5.10.2. Hydrodynamic Diffraction Settings

5.10.3. Hydrodynamic Response Settings

5.10.1. Specifying External Pre- or Post-Solve Operations

The **External Operation before Solving** and **External Operation after Solving** fields can be used to instigate a user-defined operation immediately before or after the hydrodynamic solver runs. The default for each of these fields is None, so that no external operation is performed.

Setting **External Operation before Solving** to Run a Python Script adds a Pre-Solve Python Script object under the Analysis Settings. You can use this object to import an existing Python script by selecting a file in the **Pre-Solve Python Script** field. You can define or edit your Python script directly in the script editor that is provided.

Note:

Modifying the Pre-Solve Python Script will cause an up-to-date Solution to go out of date. If the Pre-Solve Python Script fails to execute, the Solve operation will be terminated before it progresses to the hydrodynamic calculation.

You can also set the **External Operation after Solving** field to Run a Python Script, which will add a Post-Solve Python Script object under the Analysis Settings. You can import an existing Python script by selecting a file in the **Post-Solve Python Script** field. Modifying the Post-Solve Python Script will *not* cause the Solution to go out of date, and the Solution will still be marked as up-to-date at the end of a Solve even if the Post-Solve Python Script fails to execute. However, a dialog box will appear to warn you of such a failure.

When you import a Python script into your hydrodynamic system, the only retained reference to the original file is the path shown in the **Pre-** or **Post-Solve Python Script** field. If you make any subsequent changes in the Aqwa editor's Python script editor, they will not be saved in the original file. If you modify the original file outside of the Aqwa editor, you will need to re-import it for those changes to be captured.

Alternatively, the **External Operation before** or **after Solving** fields can be set to Run a Custom Command. These allow you to specify paths to external programs, in the **External Pre-Solve Command** and **External Post-Solve Command** fields, that should be run immediately before or after the hydrodynamic solver. The paths can be absolute or relative. If the location of the executable is in your %PATH% environment variable, you should not need to specify the full path. You can also enter any Windows command that can be executed from the command line.

When a pre- or post-solve command is specified, the following environment variables are created in the process' environment:

- `AQWA_INPUT_FILE`: points to the .DAT file used by the analysis.
- `AQWA_INPUT_FILE_NOEXT`: same, but without the extension (to access the result files, with extensions `.PLT`, `.RES` etc.).
- `AQWA_WORK_DIR`: the directory where the analysis is run.
- `AQWA_USER_DIR`: the project's user_files subdirectory.
- `AQWA_INSTALL_DIR`: the root of the Ansys installation folder.
- `AQWA_PYTHON_EXE`: the path to the CPython executable shipped with the Ansys installation.

For the post-solve command only, the following variable is also set:

- `AQWA_RUN_COMPLETED`: set to either TRUE or FALSE depending on the outcome of the solve.

5.10.2. Hydrodynamic Diffraction Settings

Hydrodynamic Diffraction analyses can take advantage of **Parallel Processing** with the proper licenses. For more information, see Setting Aqwa Parallel Processing Options (p. 265).

The **Generate Wave Grid Pressures** option specifies whether Aqwa should generate a Cartesian grid of wave pressures around the analyzed structures. This option must be set to Yes if you wish to include results in which the surrounding wave surface is reported (specifically, Pressures and Motions (p. 227) results with **Contour Type** of Wave Surface Elevation or Air Gap).

When **Generate Wave Grid Pressures** is set to Yes the following Wave Grid options are available:

- **Wave Grid Resolution** allows you to double the resolution of the wave surface from 81 x 51 wave grid nodes (Standard) to 161 x 101 nodes (Fine). This is useful to capture the wave shapes of high-frequency wave components, particularly where encounter frequencies are increased due to a vessel’s forward speed.
- **Wave Grid Size Factor** controls how much larger the sea area is than the structure (or structure group, if there are interacting structures present). The default value of 2 will cause the sea area to be twice as long as the structure in the X or Y directions, a ratio of X = 1.6 Y is maintained.

The following options are available to the Hydrodynamic Diffraction analysis system:

5.10.2.1. Common Analysis Options
5.10.2.2. QTF Options
5.10.2.3. Output File Options

5.10.2.1. Common Analysis Options

These analysis settings control how the analysis is performed. In normal circumstances they will not need adjusting. However, if you get modeling warnings then you may wish to turn on the **Ignore Modeling Rule Violations** option.

Note:

Mean wave drift coefficients are always included in the Hydrodynamic Diffraction calculation. If you do not want to include drift forces in a subsequent Hydrodynamic Response analysis, you can achieve this by setting the **Omit Calculation of Drift Forces** to Yes in any Irregular Wave (p. 156) objects that have been added to the Hydrodynamic Response analysis. This option is only available for Irregular Wave Response with Slow Drift calculations.

Ignore Modeling Rule Violations

If Yes, modeling rule violations will not stop the analysis. Most modeling errors generate warnings when this option is set to Yes. Do not use this option unless violations are minor and difficult to correct. For more information on modeling rules, see Mesh Quality Check in the *Aqwa Theory Manual*.

Calculate Extreme Low/High Frequencies

If Yes, in addition to the frequencies defined in **Wave Frequencies**, the solver also calculates solutions at a 'very low' frequency and a 'very high' frequency. These are added to improve the calculation of the full QTF matrix in the Hydrodynamic Diffraction analysis, and to improve the convolution of radiation forces in any subsequent Hydrodynamic Response analyses.

This option **must be turned on** when the model includes active Moonpools. This enables the correct options in the solver file. Failure to do so will result in an error message, and the solve will be stopped.

Include Multi-Directional Wave Interaction

If No, do not account for interactions between different wave directions in the calculation of QTF coefficients. Setting this option to No prevents the **Apply Mean Drift Force with Multi-Directional Wave Interaction** option from being used in any subsequent Hydrodynamic Response analysis.

Near Field Solution

If Program Controlled, the far field solution is used in the calculation of mean drift forces where possible. However, the far field solution only calculates the mean drift force in three horizontal degrees of freedom (surge, sway, and yaw) and is unable to consider the hydrodynamic interaction between structures. For analyses in which the full QTF matrix is to be calculated or which contain multiple interacting structures, the near field solution must therefore be used.

If Yes, specifies that the near field solution should be used.

Linearized Morison Drag

If Yes, computes linearized Morison drag for tube, disc, and slender tube elements. In order for the drag to be computed, after enabling Linearized Morison Drag an Irregular Wave (p. 156) must be added to the Hydrodynamic Diffraction analysis. Only one spectrum can be defined (no Irregular Wave Group). The Irregular Wave must have only a single direction defined (no Cross Swell).

This option will not be available if multiple structures are connected with a Connection Stiffness (p. 112) matrix. If set to Yes and a Connection Stiffness object is then defined between two structures it will show as a yellow field. In addition, the **Analysis Options** item will be marked with the question mark until either: the Connection Stiffness matrix is suppressed or is changed to connect between a point and a structure; or, the option is set to No.

5.10.2.2. QTF Options

The analysis settings control the use of the QTF matrix. For more information, see the *Aqwa Reference Manual*.

Calculate Full QTF Matrix

If Yes, the program calculates the full QTF matrix.

5.10.2.3. Output File Options

These analysis settings control what is written to the Aqwa output text file or to specific additional files. For more information, see the *Aqwa Reference Manual*.

Source Strengths

If Yes, writes the singularity strengths for both the modified and unmodified values to the `*.LIS` file, the modified strengths being a linear combination of the unmodified values. The actual relationship is a function of the number of body symmetries that are used.

Potentials

If Yes, writes the modified and unmodified values of the potential at the diffraction element centers and at the field points to the `*.LIS` file. This information may be used to define the fluid flow field about the body.

Centroid Pressures

If Yes, writes the hydrostatic differential and hydrodynamic fluid pressures at each plate in the model to the `*.LIS` file.

Element Properties

If Yes, writes complete details of each element used in the body modeling to the `*.LIS` file. All important details of the body elements are output together with the resultant properties of the bodies.

ASCII Hydrodynamic Database

If Yes, prints the hydrodynamic database (the `.HYD` file) in a compact ASCII format to a new file with a `.AH1` extension. This option must be set to Yes in order to use the Output Example of Hydrodynamic Database option.

Example of Hydrodynamic Database

If Yes, prints a sample of the `.AH1` file, with annotation to explain the format.

Generate AHD Pressure Output

If Yes, the program runs an additional post-processing step to write real and imaginary pressure components, over all wave frequencies and directions in the Hydrodynamic Diffraction analysis, to a new file with an `.AHD` extension. For analyses that include Morison elements, the disturbed wave kinematics at Line Body mesh nodes are also included.

5.10.3. Hydrodynamic Response Settings

Analysis settings for the Hydrodynamic Response analysis system depend on the **Computation Type** field. This field can be set to **Stability Analysis** for the calculation of static equilibrium starting conditions for a subsequent Hydrodynamic Response analysis, **Time Response Analysis** for a calculation in the time domain, or **Frequency Statistical Analysis** for a calculation in the frequency domain.

By default, a Hydrodynamic Response system connected to a Hydrodynamic Diffraction system has the **Computation Type** of **Stability Analysis**. A Hydrodynamic Response system connected to another Hydrodynamic Response system has a default **Computation Type** of **Time Response Analysis**. See Hydrodynamic Response System (p. 35) for a table of valid system connections.

5.10.3.1. Stability Analysis
5.10.3.2. Time Response Analysis
5.10.3.3. Frequency Statistical Analysis

Parallel Processing

Time response analyses can take advantage of parallel processing for any of the following cases:

- Multiple dynamic cables are defined
- Multiple tether/risers are defined
- One or more quasi-static composite cable(s) that touch down on a sloped seabed are defined
- **Analysis Type** in **Time Response Specific Options** is set to Irregular Wave Response
- Calculation of time domain pressure

Other **Computation Types** of Hydrodynamic Response analysis without quasi-static composite cables that touch down do not make use of parallel processing. For these cases, if **Parallel Processing** is set to **Manual Definition** and anything other than 1 is set for **Number of Requested Cores**, an information message is displayed and only one core is used.

For more information, see Aqwa Parallel Processing Calculation in the *Aqwa Theory Manual*.

Use Cable Dynamics

If Yes, enables cable dynamics to be used (can be set per cable). If Cable Dynamics is disabled, then regardless of the selection in the individual cable, it is not used during the analysis.

5.10.3.1. Stability Analysis

The following analysis settings are specific to a Stability Response analysis.

Output Global Stiffness Matrix

If Yes, output the global stiffness matrix.

Require Convergence for Subsequent Analysis

If Yes, prevent any analysis dependent upon results from the current Stability Response analysis from running if convergence is not achieved.

Maximum Number of Iterations

Defines the maximum number of iterations to perform in the stability analysis. If convergence has not occurred after the maximum number of iterations has been reached, the simulation stops.

Movement Limitations per Iteration Step

Defines the maximum amount a structure can move in the X, Y, Z, RX, RY, and RZ directions in a single time step. When set to Program Controlled, the values for each maximum are 2 m, 2 m, 0.5 m, 0.573°, 0.573°, 1.432° respectively. When set to Manual Definition, the following additional options appear:

Movement Limitations Applied to

Defines the structure to apply manual movement limitations to. When defined, additional **Max Movement/Step** fields appear for the X, Y, Z, RX, RY, RZ directions. You can define up to 5 sets of movement limitations.

Maximum Error in Equilibrium Position

Defines the convergence criteria. When the movement per iteration drops below the values set in this field (in all six freedoms), the solution is considered converged and the simulation stops. When set to Program Controlled, the values for X, Y, Z, RX, RY, and RZ are 0.02 m, 0.02 m, 0.02 m, 0.057°, 0.057°, 0.143° respectively.

5.10.3.1.1. Common Analysis Options

Use Linear Stiffness Matrix to Calculate Hydrostatics

If Yes, uses the linear stiffness matrix and Froude-Krylov forces from the Hydrodynamic Diffraction calculation instead of re-calculating using the individual elements. This normally will reduce the time to run the program substantially.

5.10.3.1.2. Output File Options

These analysis settings control what is written to the Aqwa output text file. For more information, see the *Aqwa Reference Manual*.

Axes System for Joint Reactions

Defines the axis system for the output of joint reaction forces.

Fixed Reference Axes

Joint reactions are output using axes parallel to the global axes.

Local Structure Axes

Joint reactions are output using axes parallel to the local structure axes.

Local Articulation Axes

Joint reactions are output using the individual local joint axes.

Data List

If Yes, include all extended data output in the `.LIS` file.

Element Properties

If Yes, output complete details of each element used in the body modeling to the `*.LIS` file. All important details of the body elements are output together with the resultant properties of the bodies.

Dynamic Cable/Tether Drag

If Yes, the drag force acting on each dynamic cable or tether connection at each iterative step will be output in the `*.LIS` file.

5.10.3.2. Time Response Analysis

The following analysis settings are specific to a Time Response analysis.

Analysis Type

Select one of the analysis types listed. Depending upon the type selected, either a Regular or Irregular Wave object must be defined for the analysis. Available settings are:

- Irregular Wave Response With Slow Drift (Irregular Wave)
- Irregular Wave Response (Irregular Wave)
- Regular Wave Response (Regular Wave)
- Slow Drift Only (Irregular Wave)

Start Time, Duration

Set the **Start Time** and **Duration** for the Time Response simulation. The **Finish Time** is updated automatically. If there is an upstream Hydrodynamic Response system the **Start Time** must be within the range of the **Start Time** and the **Finish Time** of the upstream system.

Output Step

The length of time between outputs to the plotting and text files. The **Output Step** will default to 0.1s, and will always be equal to the nearest whole number of time step intervals. Enter 0 to reset the **Output Step** to the **Time Step** value.

Note:

In order to limit the size of the hydrodynamic output files, the maximum number of output time steps accepted by Aqwa Workbench is 1,000,000. If this is exceeded, the program will set **Output Step** to the minimum permissible for the defined **Time Step**.

Time Step

Set the length of time simulated at each time step. The **Number of Steps** is calculated from the **Time Step** and **Duration**, and is updated automatically.

Starting Position

If a Time Response Analysis is preceded by a Stability Analysis (p. 134), the equilibrium position from the Stability Analysis (p. 134) is used and the (read-only) **Starting Position** is set to **Determined by Upstream System**. This also applies if there is an upstream Time Response Analysis, where the **Starting Position** is determined by the position at the corresponding time in the upstream system.

If a Time Response Analysis is linked directly to a Hydrodynamic Diffraction/Radiation (p. 130) Analysis and **Starting Position** is set to **Program Controlled**, the equilibrium position will be automatically determined before the Time Response Analysis is started and used as the starting position. If **Starting Position** is set to **Based on Geometry**, the current geometry determines the starting position. Note that when the model includes one or more Joints, the **Based on Geometry** option is replaced by **Based on Articulated Positions**.

X- and Y-Position for Wave Surface Elevation Output

Define a Fixed Position at which the Wave Surface Elevation will be available as a result.

5.10.3.2.1. Time Response Pressure Output

When the Time Response **Analysis Type** is set to Irregular Wave Response or Regular Wave Response, the **Time Response Pressure Output** options can be used to set up the generation of instantaneous pressure results for one or more structures in the analysis. The options are:

Output for Structure

Select a structure for which instantaneous time domain pressures will be output; or, in an analysis with multiple structures, you may choose All Structures.

Output Start Time

Set the time at which pressures will start to be recorded. The **Output Start Time** cannot be less than the analysis **Start Time**, and is automatically adjusted to the nearest whole number of **Output Step** intervals.

Output Time Step

Set the length of time between pressure records. The **Output Start Time** is automatically adjusted to the nearest whole number of **Output Step** intervals.

Output Finish Time

Set the time at which pressures will stop being recorded. The **Output Finish Time** cannot be greater than the analysis **Finish Time**, and is automatically adjusted to the nearest whole number of **Output Step** intervals.

Note:

For a workflow which includes time domain Hydrodynamic Response pressure mapping to a Static Structural finite element analysis using the Aqwa Hydrodynamic Pressure Mapping (p. 273) Add-on, the structures and time steps available for pressure mapping in the Static Structural system are defined by the **Time Response Pressure Output** options. Time domain pressure mapping will not be possible while the **Output for Structure** option is set to None.

5.10.3.2.2. Common Analysis Options

These analysis settings control how the Time Response Analysis is performed. In normal circumstances they will not need adjusting. For more information, see the *Aqwa Reference Manual*.

Time Response analyses with the **Use Cable Dynamics** option set to Yes can take advantage of parallel processing with the proper licenses. For more information, see Setting Aqwa Parallel Processing Options (p. 265).

Convolution

If Yes, specifies that convolution method is used in radiation force calculation. This is a more rigorous approach to the radiation force calculation in time domain and will enhance the capability of handling nonlinear response of structures. Convolution requires that a Hydrodynamic Diffraction/Radiation analysis has been performed for at least 5 wave frequencies.

Call Routine "user_force"

If Yes, calls a routine called "user_force" at each stage of the calculation. This routine can be used to add externally calculated forces to the simulation. See External Force Calculation in the *Aqwa Reference Manual* for more information.

Connect to Server for External "user_force" Calculation

If Yes, uses a process running on an external server for the user-force calculation. See External Server for User-Defined Force Calculation in the *Aqwa Reference Manual* for more information.

Calculate Motions Using RAOs Only

If Yes, calculates motions using RAOs only. Note that this option suppresses all motion except that defined by the RAOs. In particular current, wind, drift forces, moorings etc. have no effect on the motions of the structure.

Account for Current Phase Shift

If No, switches off the wave phase shift due to a current speed.

Apply Mean Drift Force with Multi-Directional Wave Interaction

If Yes, utilize the drift coefficients which include interactions between different wave directions that have been calculated in the preceding Hydrodynamic Diffraction analysis. This option is invalid if the **Include Multi-Directional Wave Interaction** option in the preceding Hydrodynamic Diffraction analysis is set to No.

Calculate Wave Drift Damping

If No, stops the automatic calculation of wave drift damping for a floating structure. Note that the wave drift damping calculated by the program is only for the floating structure, damping from risers, etc. is not included.

Include Yaw Wave Drift Damping

If No, suppresses the calculation of wave drift damping for yaw motion. To prevent the calculation of all wave drift damping use the No Automatic Wave Drift Damping Calculation option.

Use Slow Velocity for Hull Drag Calculation

If Yes, uses the slow velocity (drift frequency velocity) for the hull drag calculation, instead of the total velocity (drift frequency velocity + wave frequency velocity) which is the default.

Use Linear Starting Conditions

If Yes, starts a simulation with the motions and velocities derived from the Hydrodynamic Diffraction system results. This can be used to limit the transient at the start of a simulation.

Use Linear Stiffness Matrix to Calculate Hydrostatics

If Yes, uses the linear stiffness matrix and Froude-Krylov forces from the Hydrodynamic Diffraction calculation instead of re-calculating using the individual elements. This normally will reduce the time to run the program substantially.

Include Maneuvering Force

If Yes, adds the maneuvering loads due to the low frequency ship maneuvering motions, as described in Low Frequency Maneuvering Loads in the *Aqwa Theory Manual*, to the total structure forces/moments.

Use Wheeler Stretching

The functionality of this option depends on the context in which it is used.

- If the **Analysis Type** is set to Regular Wave Response: **Use Wheeler Stretching** = Yes turns on Wheeler stretching when calculating the Froude-Krylov force over the instantaneous wave surface.
- If the **Analysis Type** is set to Irregular Wave Response, Wheeler Stretching is always used. The **Use Wheeler Stretching** field can be used to specify:
  - With Linear Wave Theory - calculation will only use linear wave theory.
  - With Second Order Correction - calculation will add a second order correction on wave elevation, pressure, and fluid particle velocity/acceleration in addition to the linear wave theory.

5.10.3.2.3. QTF Options

These analysis settings control the use of the QTF matrix for Time Response Analyses. For more information, see the *Aqwa Reference Manual*.

Use Full QTF Matrix

If Yes, specifies that the full matrix of difference frequency QTFs is to be used when calculating slowly varying drift forces.

Use Sum Frequency QTFs

If Yes, specifies that the full matrix of sum frequency QTFs is to be used when calculating slowly varying drift forces, in addition to using the full matrix of difference frequency QTFs. You must also have Use Full QTF Matrix set to Yes to use this feature.

5.10.3.2.4. Output File Options

These analysis settings control what is written to the Aqwa output text file. For more information, see the *Aqwa Reference Manual*.

Axis System for Joint Reactions

Defines the axis system for the output of joint reaction forces.

Fixed Reference Axes

Joint reactions are output using axes parallel to the global axes.

Local Structure Axes

Joint reactions are output using axes parallel to the local structure axes.

Local Articulation Axes

Joint reactions are output using the individual local joint axes.

Data List

If Yes, include all extended data output in the `*.LIS` file.

Element Properties

If Yes, writes complete details of each element used in the body modeling to the `*.LIS` file. All important details of the body elements are output together with the resultant properties of the bodies.

Dynamic Cable/Tether Drag

If Yes, the drag force acting on each dynamic cable or tether connection at each time step will be output in the `*.LIS` file.

5.10.3.2.5. Output Aqwa FMU Package

When the Hydrodynamic Response Settings (p. 133) is set to Irregular Wave Response or Regular Wave Response, the **Co-simulation FMU Package** options become available to set up the generation of the Aqwa Functional Mock-Up Unit (FMU) package. This Aqwa FMU package contains the information of the current Aqwa model, and it is used for the co-simulation analysis. The inputs for this Aqwa FMU package need to be the displacement, velocity, and acceleration at the center of gravity of each Aqwa structure. The outputs of this Aqwa FMU package are the total forces and moments of hydrostatic, hydrodynamic, and mooring loads at each Aqwa structure's center of gravity. More information about Aqwa co-simulation is elaborated in The Aqwa Co-simulation Add-on (p. 332) section. The following are the available options for the generation of the Aqwa FMU package:

Output FMU Package

Select whether to output an Aqwa FMU package for the current analysis. Select **Yes** if the co-simulation analysis is required. The solver input files of the current analysis and FMI supporting file are contained within the Aqwa FMU package.

FMU Export Version

Select the FMU version for the output Aqwa FMU package file. The selection must be based on the FMU versions supported by the co-simulation platform (p. 344).

Co-simulation Partner's Unit

Select the unit system used by the co-simulation partner, which is planned to be connected to the Aqwa FMU package. The available selections follow the Mechanical application interface unit options and their converting relationships with the Mechanical solver units. During the co-simulation, the unit transformation will be conducted according to the selected units.

Co-simulation Coordinate System Offsets

Select whether there are differences between the coordinate system of the current Aqwa model and the co-simulation partner. If **Yes** is selected, the offsets between the two coordinate systems can be defined by X Offset, Y Offset, Z Offset, Rotational Offset X, Rotational Offset Y, and Rotational Offset Z options. The application of the offsets on the Aqwa coordinate values should be equal to the coordinate values of the co-simulation partner.

5.10.3.3. Frequency Statistical Analysis

The following analysis settings are specific to a Frequency Statistical analysis.

Analysis Type

Select one of the analysis types listed. Available settings are:

- Wave and Drift Frequencies
- Wave Frequencies Only
- Drift Frequencies Only

Direction of Output for RAOs

Allows you to change the direction in which some of the Graphical Results (p. 217) (Motion RAOs, Cable Tension and Fender Force RAOs, and Transfer Functions) are given. When there is more than one direction available, the entries in the menu are the unsuppressed waves contained in the analysis. When **Direction of Output for RAOs** is set to Program Controlled, the output is given in the direction of the spectrum that has the largest significant wave height.

Spectrum Sub-Direction of Output for RAOs

Only editable if **Direction of Output for RAOs** has a Cross Swell defined, or is configured with a **Number of Sub-Spectra** greater than 1 (**Wave Spreading** is not None (Long Crested Waves)).

Axis System for Significant Motions and Nodal Response

Sets the (global or local) axis system in which significant motions and nodal response spectra are reported.

Starting Position

If a Frequency Statistical Analysis is preceded by a Stability Analysis (p. 134), the equilibrium position from the Stability Analysis (p. 134) is used and the (read-only) **Starting Position** is set to Determined by Upstream System.

If a Frequency Statistical Analysis is linked directly to a Hydrodynamic Diffraction/Radiation (p. 130) Analysis and **Starting Position** is set to **Program Controlled**, the equilibrium position is automatically determined before the Frequency Statistical Analysis starts and is used as the starting position. If **Starting Position** is set to **Based on Geometry**, the current geometry determines the starting position. Note that when the model contains one or more Joints, the **Based on Geometry** option is replaced by **Based on Articulated Positions**.

Caution:

The Based on Geometry option should be used with care, as a Frequency Statistical Analysis of any system containing connections or articulations is unlikely to be correct if the tensions and reactions at those connections or articulations are not calculated first. Linking a Frequency Statistical Analysis to a Stability Analysis, or setting **Starting Position** to Program Controlled, ensures that the forces acting on connections and articulations are determined before the Frequency Statistical Analysis is performed.

5.10.3.3.1. Common Analysis Options

These analysis settings control how the Frequency Statistical analysis is performed. In normal circumstances they will not need adjusting. For more information, see the *Aqwa Reference Manual*.

Calculate RAOs with Mooring Lines

If Yes, allows access to the CRAO option in the input file.

Apply Drift Force with Multi-Directional Wave Interaction

If Yes, utilize the drift coefficients which include interactions between different wave directions that have been calculated in the preceding Hydrodynamic Diffraction analysis. This option is invalid if the **Include Multi-Directional Wave Interaction** option in the preceding Hydrodynamic Diffraction analysis is set to No.

Linearized Morison Drag

If Yes, computes linearized Morison drag for tube, disc, and slender tube elements, and for wind and current drag forces.

5.10.3.3.2. QTF Options

The analysis settings control the use of the QTF matrix for Frequency Statistical analyses. For more information, see the *Aqwa Reference Manual*.

Use Full QTF Matrix

If Yes, specifies that the full matrix of difference frequency QTFs is to be used when calculating slowly varying drift forces.

5.10.3.3.3. Output File Options

These analysis settings control what is written to the Aqwa output text file. For more information, see the *Aqwa Reference Manual*.

Axes System for Joint Reactions

Defines the axis system for the output of joint reaction forces.

- **Fixed Reference Axes** - Joint reactions are output using axes parallel to the global axes.
- **Local Structure Axes** - Joint reactions are output using axes parallel to the local structure axes.
- **Local Articulation Axes** - Joint reactions are output using the individual joint axes.

Data List

If Yes, output all extended data output in the `*.LIS` file.

Output Element Properties

If Yes, writes complete details of each element used in the body modeling to the `*.LIS` file. All important details of the body elements are output together with the resultant properties of the bodies.

5.11. Applying Ocean Environment and Forces

The following ocean environment and forces objects can be added to your analysis (or may be included with your analysis by default). Some are restricted to the Hydrodynamic Diffraction analysis and some are restricted to the Hydrodynamic Response analysis.

5.11.1. Structure Selection
5.11.2. Wave Directions
5.11.3. Wave Frequencies
5.11.4. Starting Conditions
5.11.5. Structure Force
5.11.6. Regular Wave
5.11.7. Irregular Wave
5.11.8. Irregular Wave Group
5.11.9. Current
5.11.10. Wind
5.11.11. Cable Winch
5.11.12. Connection Failure
5.11.13. Point Force
5.11.14. Deactivated Freedoms

5.11.1. Structure Selection

The Structure Selection tree object enables the definition of interacting structures, in addition to any structures that you want to exclude for this particular analysis. It also enables the structure order to be changed.

Structures can be selected graphically (use the Ctrl key to select more than one) for inclusion in a group or to be excluded from the analysis. To include structures in a group, click an **Interacting Structure Groups** field. To exclude structures from the analysis, click the **Structures to Exclude** field. Select the structures in the model, then click the **Apply** button. The selected structures will be listed in the field. If a structure is excluded, then it will automatically be removed from any groups and the structure order. A structure may only be included in one interacting structure group.

By default, when Aqwa is first started all parts are considered to be hydrodynamically interacting and are included in the analysis. Note, however, that if a part is made up entirely of non-diffracting elements (for example, completely above the water surface or consisting of Morison elements only), then the part is not considered as interacting and is excluded from the list of Interacting Structure Groups. If a part is subsequently suppressed, it cannot be used in the analysis, and is automatically excluded in the Structure Selection.

Note:

Unsuppressing a structure from the right-click menu on the tree or the Details panel of the structure doesn't affect the structure's status in the Structure Selection. If you suppress a structure and then unsuppress it, you must go to the Structure Selection object and remove it from the Structures to Exclude set in order for it to be included in the analysis. If hydrodynamic interaction is required then this must also be established manually in the Structure Selection.

Once you have defined your groups of interacting structures, you can optionally lock the motions of the structures by applying a **Motion Lock** to the relevant group. The RAOs calculated for the structures in the locked interacting group are then consistent with the RAOs of an equivalent single structure. This is particularly useful for the estimation of splitting forces and moments on a structure in a time domain Hydrodynamic Response calculation. This is achieved in the time domain analysis by including a joint between each of the bodies in the interacting group to output such forces/moments.

If a structure has been removed from the analysis and you want to add it back in, you must reselect the set of **Structures to Exclude**. If you want all structures to be included in the analysis, you may click the field next to **Structures to Exclude** then click **Apply** with no structures selected.

The Aqwa solver requires that a group of interacting structures be consecutively ordered for the analysis. Modifying the interacting groups will automatically update the order of structures in the Aqwa analysis, and this will be reflected in the **Structure Ordering** section of the Details panel.

The **Structure Ordering** will include any Dynamic Points that have been defined in the hydrodynamic model, reflecting the fact that these are considered structures by the Aqwa solver.

Note:

The total number of structures, including Dynamic Points, must be less than or equal to 99. It is possible for all of these structures to interact hydrodynamically (excluding any dynamic points, which are always non-diffracting).

5.11.2. Wave Directions

The Wave Directions tree object enables the definition of a range or single wave direction to use in the Hydrodynamic Diffraction (p. 130) analysis and, optionally, a forward speed at which the structures move in the positive X-direction.

If **Type** is set to Single Direction, Forward Speed, you can enter a structure **Forward Speed** and a **Wave Direction**. In this case, only a single wave direction can be analyzed.

If **Type** is set to Range of Directions, No Forward Speed, waves are automatically created in -180 and +180 directions, and either the **Interval** or the number of intermediate directions (**No of Intermediate Directions**) can be specified. If a direction range is of particular interest, additional ranges or specific directions can be added.

If **Type** is set to Range of Directions, Forward Speed, you can enter a structure **Forward Speed** and perform the analysis over a range of wave directions. Waves are automatically created in -180 and +180 directions, and either the **Interval** or the number of intermediate directions (**No of Intermediate Directions**) can be specified. If a direction range is of particular interest, additional ranges or specific directions can be added.

To add Optional Wave Directions to analyses performed over a range of wave directions, select Single or Range from the **Additional Range** dropdown. For the Single direction, specify the **Start Angle**. For a Range of directions specify the **Start Angle**, **End Angle**, and either the **Interval** or **No of Intermediate Directions**.

Duplicated directions will automatically be removed; although the number of directions does not contribute greatly to the analysis time, there is a limit imposed of a total of 41 directions for any one structure.

Note:

It is not possible to employ symmetry in Aqwa.

5.11.3. Wave Frequencies

The Wave Frequencies tree object enables the definition of a range or single wave frequency to use in the analysis.

By default the frequency **Range** is Program Controlled; this equally spaces the specified **Total Number of Frequencies** between a minimum value based on the water depth (p. 42) (or the default minimum of 0.1 rad/s, whichever is larger) and a maximum based on the mesh size (p. 46). The spacing interval can either be based on constant frequency or constant period increments (**Intervals Based Upon**, which also applies to any additional frequency range definitions).

You can manually define a single frequency or a range of frequencies by selecting Manual Definition for the **Range**. For a single frequency, set **Definition Type** to Single. Then set **Lowest Frequency Definition** to Manual Definition if you wish to edit the **Lowest Frequency** or **Longest Period**.

To specify a range of frequencies, set **Definition Type** to Range. Change **Lowest Frequency Definition** and/or **Highest Frequency Definition** to Manual Definition to edit the **Lowest Frequency** and **Highest Frequency**, or the **Longest Period** and **Shortest Period**, as required.

The **Lowest Frequency** will default to 0.1 rad/s and the **Highest Frequency** will default to that determined by the mesh. The lowest wave frequency permitted is calculated by $0.001 \cdot \max\left(1, \sqrt{\frac{g}{d}}\right)$ rad/s, where d is the water depth and g is the acceleration due to gravity.

The intermediate positions in a range are defined by entering a **Number of Intermediate Values**. The **Interval Frequency** or **Interval Period** will be updated automatically (according to whether **Intervals Based Upon** is set to Frequency or Period).

To specify additional frequencies, set **Additional Range** to Single or Range and enter the Period/Frequency and **Number of Intermediate Values** information. Any duplicate frequencies are automatically removed. The number of frequencies chosen extends the solution time linearly.

The Wave Frequencies table displays the frequencies and associated wavelengths of the waves that are included in the Hydrodynamic Diffraction analysis.

Note:

To perform a subsequent Hydrodynamic Response analysis using Convolution (p. 137), a Hydrodynamic Diffraction/Radiation analysis must be performed with at least 5 wave frequencies.

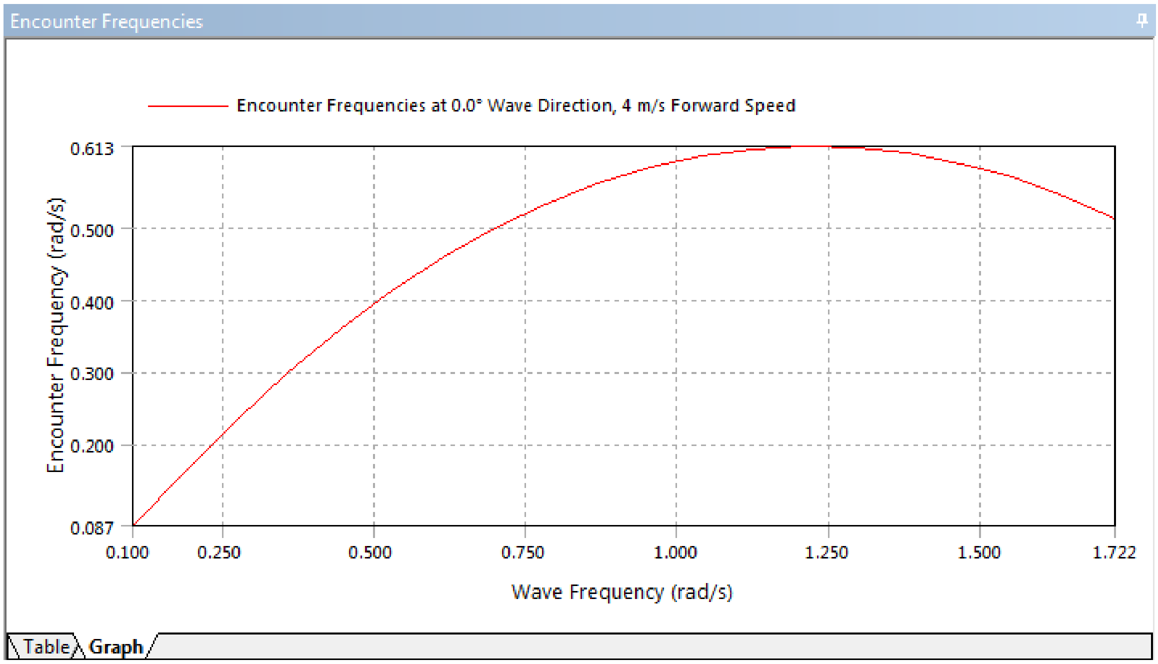
The frequency range must be wide enough to ensure an accurate calculation of the radiation force in the Hydrodynamic Response. For more information, see The FREQ/PERD/HRTZ Data Record - Frequencies/Periods at which the Hydrodynamic Parameters are Computed in the *Aqwa Reference Manual*.

5.11.3.1. Encounter Frequencies for a Single Wave Direction

If the Hydrodynamic Diffraction (p. 130) analysis is to be performed over a single Wave Direction with a Forward Speed defined, a table of the encounter frequencies in the analysis will be displayed in the Table tab of the Encounter Frequencies panel. The number of encounter frequencies will always be equal to the number of defined incident wave frequencies.

Encounter Frequencies		
Wave Frequency (rad/s)	Encounter Frequency (rad/s)	Wavelength (m)
0.1	0.09757	5166.60931
0.23553	0.22421	1110.72128
0.37106	0.34298	447.52977
0.50658	0.45425	240.10319
0.64211	0.55802	149.44429
0.77764	0.65431	101.89285
0.91317	0.7431	73.89236
1.0487	0.82441	56.02756
1.18422	0.89822	43.93728
1.31975	0.96453	35.37661
1.45528	1.02336	29.09429
1.59081	1.07469	24.34812

Clicking on the Graph tab of the Encounter Frequencies panel presents a graphical display of encounter frequencies against the corresponding incident wave frequencies. Negative encounter frequencies (which typically occur at higher incident wave frequencies/larger forward speeds) are permissible.



5.11.3.2. Encounter Frequencies for Multiple Wave Directions

If the Hydrodynamic Diffraction (p. 130) analysis is to be performed over multiple Wave Directions with a Forward Speed defined, additional Encounter Frequencies Options are available. Set **Encounter Frequencies (Target)** as a guide value for the number of encounter frequencies. Following the rules described in The FWDS Data Record - Define Forward Speed, Aqwa will use this target to determine the number of **Encounter Frequencies (Actual)** that will be included in the Hydrodynamic Diffraction (p. 130) analysis. This will always be more than the number of defined incident wave frequencies.

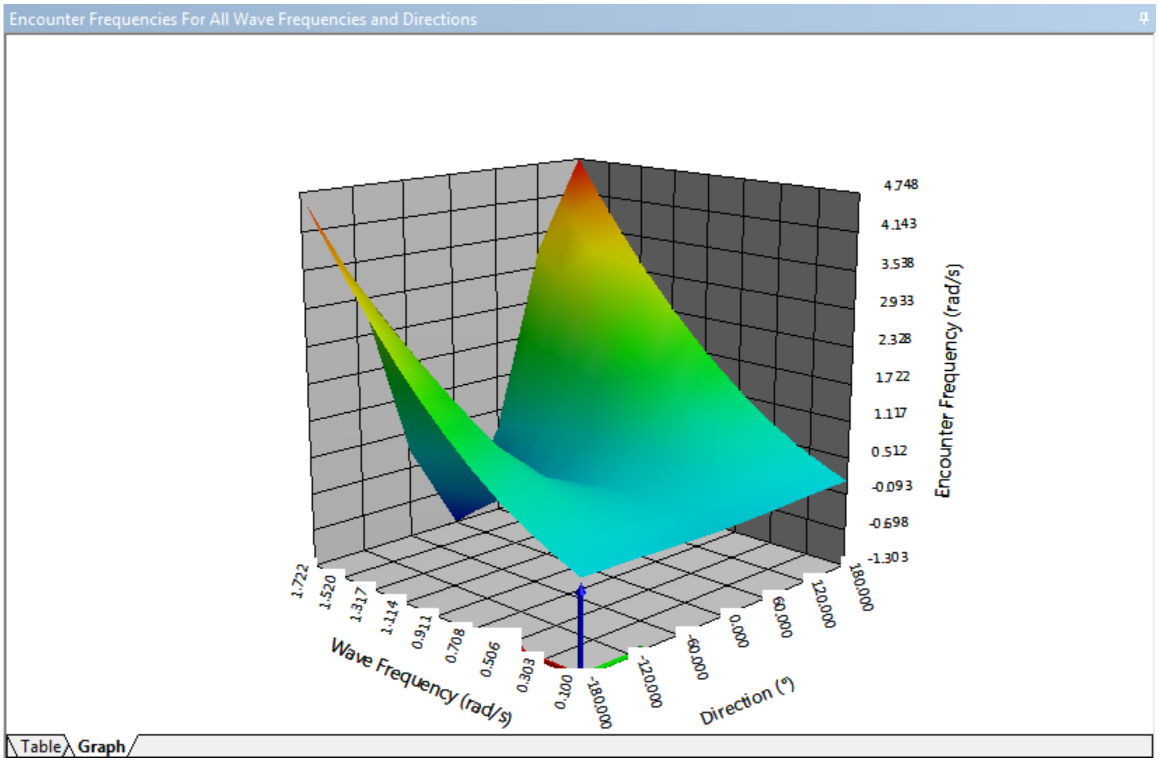
The **Display Encounter Frequencies** option allows you to switch the information that is shown in the Encounter Frequencies panel. When **Display Encounter Frequencies** is set to For All Wave Frequencies and Directions, the Table tab will show the encounter frequencies corresponding to all combinations of incident wave frequencies and directions.

Encounter Frequencies For All Wave Frequencies and Directions

Wave Frequency (rad/s)	-180°	-120°	-60°	0.0°	60°	120°	180°
0.1	0.13249	0.11624	0.08376	0.06751	0.08376	0.11624	0.13249
0.2475	0.33576	0.29163	0.20337	0.15924	0.20337	0.29163	0.33576
0.395	0.56506	0.48003	0.30996	0.22493	0.30996	0.48003	0.56506
0.54249	0.84404	0.69327	0.39172	0.24095	0.39172	0.69327	0.84404
0.68999	1.17553	0.93276	0.44722	0.20446	0.44722	0.93276	1.17553
0.83749	1.55271	1.1951	0.47988	0.12227	0.47988	1.1951	1.55271
0.98499	1.97432	1.47966	0.49032	-4.34319e-3	0.49032	1.47966	1.97432
1.13249	2.4403	1.78639	0.47858	-0.17533	0.47858	1.78639	2.4403
1.27999	2.95065	2.11532	0.44465	-0.39068	0.44465	2.11532	2.95065
1.42748	3.50537	2.46643	0.38854	-0.6504	0.38854	2.46643	3.50537
1.57498	4.10446	2.83972	0.31024	-0.95449	0.31024	2.83972	4.10446
1.72248	4.74791	3.2352	0.20976	-1.30295	0.20976	3.2352	4.74791

Table Graph

The Graph tab will show the same information as a 3-D surface plot:



When **Display Encounter Frequencies** is set to In Hydrodynamic Analysis Only, the Table tab will show the subset of encounter frequencies that Aqwa will choose to include in the hydrodynamic database. The Graph tab will show these encounter frequencies in ascending sequence order.

Note:

Where the number of **Encounter Frequencies (Actual)** is less than the product of the number of incident wave frequencies and the number of defined wave directions, diffraction forces and potentials in the Hydrodynamic Diffraction (p. 130) analysis will be interpolated between the available data in the hydrodynamic database.

Aqwa will try to calculate over the whole range of encounter frequencies (both positive and negative) corresponding to the defined incident wave frequencies, wave directions, and forward speed. However, the minimum absolute encounter frequency is limited to 0.001 rad/s.

5.11.4. Starting Conditions

Depending on the analysis, it is possible to set a starting position and/or a starting velocity for a structure using the Details panel in a Starting Conditions object.

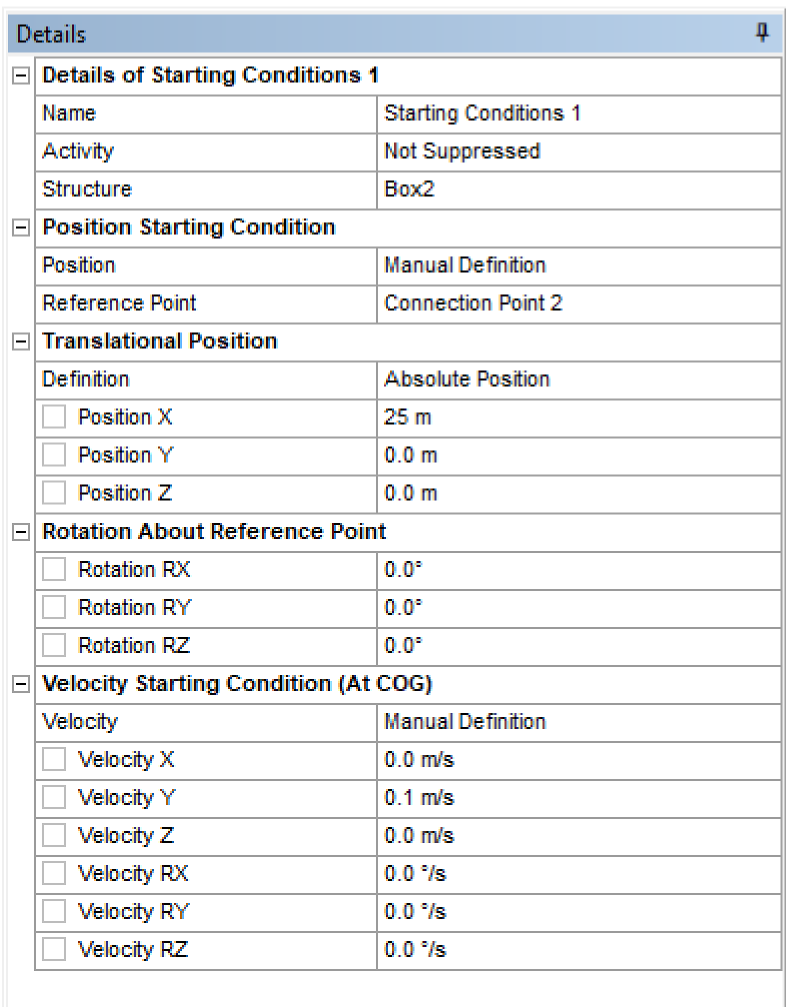
A **Starting Conditions** object can be included in the Outline of an analysis system when the **Computation Type** in the Analysis Settings is set to either **Stability Analysis** or **Time Response Analysis**. In each instance, varying restrictions apply, resulting in hidden fields in the Details panel. These restrictions are described in Setting the Position Starting Conditions (p. 150) and Setting the Velocity Starting Condition (p. 152).

Note:

A Starting Conditions object can not be included in the Outline if the **Computation Type** is set to **Frequency Statistical Analysis**

Starting conditions are attached to a particular structure, so you must select the **Structure** to which the starting conditions should be associated to.

Position and **Velocity** can be set to either **Program Controlled** or **Manual Definition**. When Program Controlled is selected, the conditions will be based on the conditions associated with the geometry at the time the analysis is being solved using the RDEP option in the solver file. When Manual Definition is selected (as shown in the image below), you can define the appropriate values for the analysis' starting positions and/or starting velocity. For more information, refer to: Administration and Calculation Options for the Aqwa Suite in the *Aqwa Reference Manual*

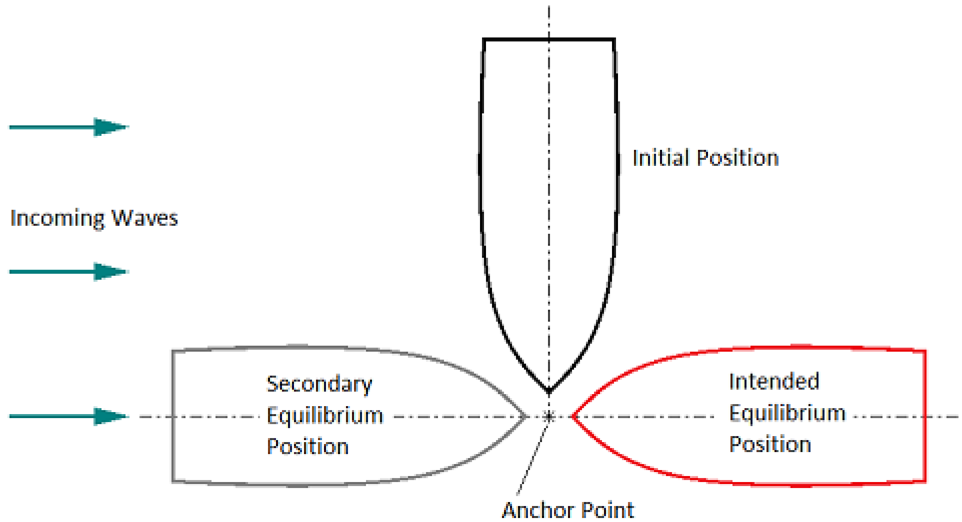


5.11.4.1. Setting the Position Starting Conditions

For a given set of environmental conditions in a Stability Analysis, it is sometimes possible for there to be more than one solution in which the forces acting on the system are equilibrated.

Consider a symmetric vessel, anchored at the bow, in a beam sea: the wave forces on the vessel become more balanced whether the vessel swings towards the direction of the wave, or away from the direction of the wave, as shown in Figure 5.11: Possible Equilibrium Positions (p. 150).

Figure 5.11: Possible Equilibrium Positions



There is usually a preferred state that should be used as the starting condition for a subsequent Frequency Statistical Analysis or Time Response Analysis. In the case above, it is expected that the bow of the vessel points toward the incoming waves. Due to the iterative method used by Aqwa to determine equilibrium position, it is not always possible to predict which equilibrium position the solution will converge towards.

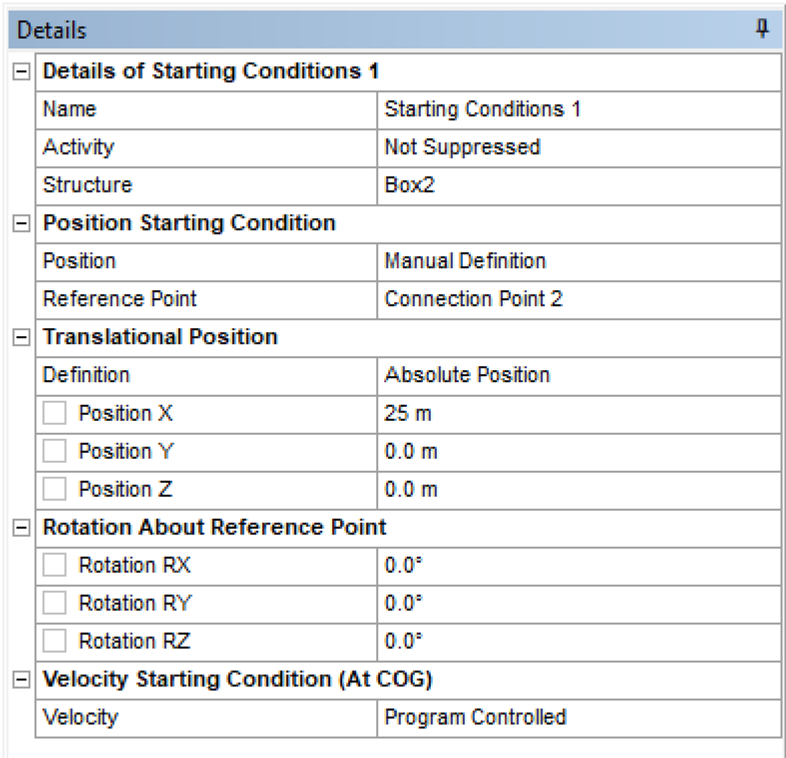
To encourage convergence towards the expected equilibrium position, you can add a **Starting Conditions** object to define the starting position of a structure (or group of articulated structures) for the first iterative step of a Stability Analysis.

In the Details panel of the **Starting Conditions** object, there are position conditions which allow you to set translational or rotational offsets about a selected reference point on a structure. The offsets may be relative to the initial geometric position of the structure (as defined in the original CAD geometry), or relative to an existing Fixed Point; alternatively, the absolute position of the reference point may be set directly.

It is only possible to set a starting position when the **Computation Type** is set to **Stability Analysis**, or when the **Computation Type** is set to **Time Response Analysis** and the **Starting Position** option is set to **Based on Geometry** or **Based on Articulated Positions**

Once a **Structure** has been selected, to which the starting position refers to, the **Reference Point** must be set as a **Connection Point** attached to the selected structure.

Selecting a **Reference Point** automatically populates the **Position X**, **Position Y**, and **Position Z** fields with the absolute position of the selected **Connection Point**.



Details	
Details of Starting Conditions 1	
Name	Starting Conditions 1
Activity	Not Suppressed
Structure	Box2
Position Starting Condition	
Position	Manual Definition
Reference Point	Connection Point 2
Translational Position	
Definition	Absolute Position
Position X	25 m
Position Y	0.0 m
Position Z	0.0 m
Rotation About Reference Point	
Rotation RX	0.0°
Rotation RY	0.0°
Rotation RZ	0.0°
Velocity Starting Condition (At COG)	
Velocity	Program Controlled

The **Definition** field of **Translational Position** can be set to **Absolute Position**, **Relative to Geometric Position**, or **Relative to Fixed Point**.

Modifying the absolute **Position X**, **Y**, and **Z** applies a translation to the position of the selected Reference Point in the Fixed Reference Axes (FRA).

Changing the **Definition** field to **Relative to Geometric Position** displays the **Relative Position X**, **Relative Position Y**, and **Relative Position Z** fields. Varying the **Relative Position X**, **Y**, and **Z** in this mode applies a translation to the position of the selected reference point in an axis system whose origin lies at the original position of the reference point, and whose axes lie parallel to the Fixed Reference Axes (FRA).

Changing the **Definition** field to **Relative to Fixed Point** displays the **Fixed Point** field, which allows you to select an existing **Fixed Point** from the **Geometry**. In this mode, the **Relative Position X**, **Y**, and **Z** are defined in an axis system whose origin lies at the position of the selected **Fixed Point**, and whose axes lie parallel to the Fixed Reference Axes (FRA).

The **Absolute Position** or **Relative Position X**, **Y**, and **Z** are initialized with values described in the following table:

Definition	Initial Values
Absolute Position	Current position of Reference Point in FRA
Relative to Geometric Position	Difference between current position of Reference Point in FRA and original geometric position of Reference Point in FRA
Relative to Fixed Point	Difference between current position of Reference Point in FRA and position of Fixed Point in FRA

The **Rotation About Reference Point** fields **Rotation RX**, **Rotation RY**, and **Rotation RZ** allow you to rotate the selected structure about an axis system centered on the selected reference point, and whose axes lie parallel to the Fixed Reference Axes (FRA).

Details	
Details of Starting Conditions 1	
Name	Starting Conditions 1
Activity	Not Suppressed
Structure	Box2
Position Starting Condition	
Position	Manual Definition
Reference Point	Connection Point 2
Translational Position	
Definition	Relative to Fixed Point
Fixed Point	Fixed Point 3
Relative Position X	25 m
Relative Position Y	-40 m
Relative Position Z	0.0 m
Rotation About Reference Point	
Rotation RX	0.0°
Rotation RY	0.0°
Rotation RZ	0.0°
Velocity Starting Condition (At COG)	
Velocity	Program Controlled

The positions of structures within an articulated group, relative to one another, are not changed by the application of a **Starting Position**. Translations and rotations due to a **Starting Position** are applied as if the articulations within that group are rigid.

A **Starting Position** may only be applied to a structure whose position is not already defined in some way. The **Structure** field is invalid if the structure is:

- Already selected in another **Starting Position** object.
- Connected by a **Joint**, or sequence of **Joints**, to a **Fixed Point**.
- Connected by a **Joint**, or sequence of **Joints**, to a structure which is already selected in another **Starting Position** object.

In the latter two cases, if the initial position of the structure needs to be modified, you should instead re-define the **Alignment Method** of the Joint Axes (see Joints (p. 121)).

5.11.4.2. Setting the Velocity Starting Condition

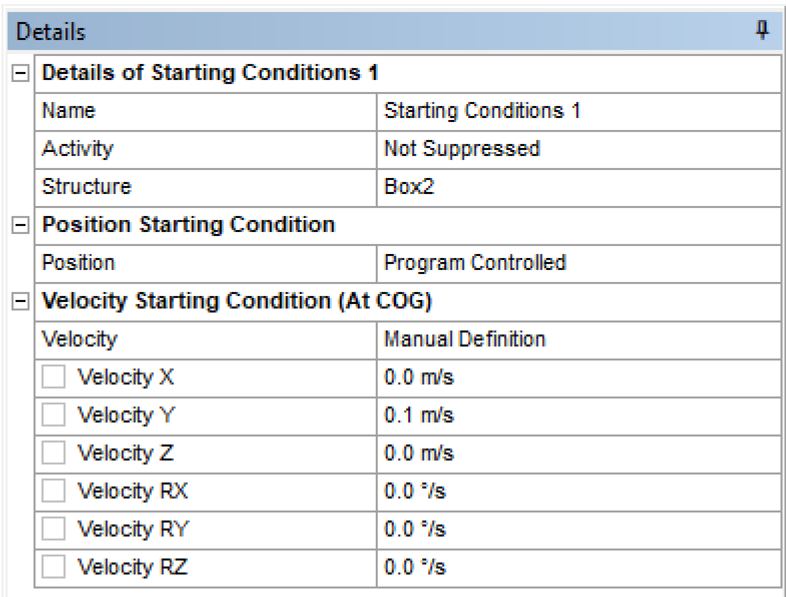
The velocity field is defined for a selected Structure at its **COG location**. Therefore, there is no need to specify a reference point.

The 6 degrees of freedom for the starting velocity can be set only when a **Structure** has been selected, and the **Computation Type** in the Analysis Settings is set to **Time Response Analysis** and the upstream analysis is not of the same type.

Note:

It is not possible to set the starting velocity when **Computation Type** is set to **Stability Analysis**.

These starting conditions correspond to the velocity field at the **Centre of Gravity**.



5.11.5. Structure Force

A time history of structure forces and moments applied at the center of gravity can be defined directly in a table or by importing the data from an existing `.XFT` file.

To add a time history of structure forces:

1. Select the **Hydrodynamic Response** object in the tree view.

2. Right-click the **Hydrodynamic Response** object and select **Insert > Structure Force > Manual Input** if data is to be defined directly in a table; select **Insert > Structure Force > Import XFT File** if data is to be read from a file, and browse to the file location.

   Alternatively, click the **Structure Force** icon in the **Analysis** toolbar and from the drop-down menu select **Manual Input** if the data is to be defined directly in a table; select **Import XFT File** if the data is to be read from a file, and browse to the file location.

Note:

After the data is imported, no record of the file used to import the data is retained.

In the Details panel, the column definition can be associated to the appropriate structures in the model. A row has to be filled in for all structures; additional rows can be added to the end of the table and they will be automatically re-sorted in ascending time order once entry is complete. Position data can also be copied and pasted from an appropriate external source.

Defining the .XFT file

The time defined in the data does not need to match the time step defined in the analysis; the program will interpolate the forces when necessary, using a cubic spline interpolation technique. For this purpose there must be at least 4 entries in the Structure Force Definition Data table, and when modeling periods of constant force, adequate data points must be provided to satisfy the interpolation method. The times included in the Structure Force Definition Data must entirely cover the simulation duration: the first time must not be greater than the Analysis Settings Start Time, and the last time must be greater than or equal to the Finish Time. There is no upper limit on the length of the `.XFT` file.

The unit system of the data in the `.XFT` file is assumed to match the display unit system of the project. However, the imported data unit system can be modified by setting **Force Unit for Imported Data** and **Length Unit for Imported Data** to the required unit system.

It is important to note that the forces defined in the `.XFT` file are in six degrees of freedom for each structure and are in the global axis system.

Comment lines are permitted at any point in the file and must begin with *.

The first line (comments excepted) in the file ought to contain the structures information; this indicates the structure numbers that the columns correspond to. For example, when there are 3 structures, this line could be `structures=1 3 9`. The structure numbers can be determined from the Aqwa Editor by reviewing the Structure Selection in the upstream Hydrodynamic Diffraction analysis.

Before the data defining the time and forces, there must be a line containing only the text, `data_start`.

After the data start, the first column contains time, and subsequent columns (in sets of 6) indicate the forces for specified structures. The forces are ordered as follows, using global directions: X, Y, Z, RX, RY, RZ. The numbers in the `.XFT` file are in a free format and can be separated by spaces.

An example of the data in a file would be:

```
structures=1
data_start
0.0000 4.4800E+05 5.0400E+06 3.1360E+06 5.0400E+07 8.9600E+06 1.5120E+08
0.5000 4.3978E+05 4.9689E+06 3.0784E+06 4.9689E+07 8.7956E+06 1.4907E+08
1.0000 4.1592E+05 4.8207E+06 2.9114E+06 4.8207E+07 8.3183E+06 1.4462E+08
...
```

Structure Force Definition Data can otherwise be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated value (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension
- Contains value separated by commas, tabs, or single spaces (not multiple spaces)

- Contains exactly (1 + 6* number of Force Sets) columns (you must define the **Structure for Force Set** fields before importing)

The unit system of the data in the .CSV file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data** and **Force Unit for Imported Data** to the required unit system.

5.11.6. Regular Wave

The Regular Wave object is used for Time Response analyses. Regular Waves can only be applied when the **Analysis Type** under the Time Response Analysis Setting object is set to Regular Wave Response (rather than the default value of Irregular Wave Response).

To add a Regular Wave:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object and select **Insert > Regular Wave**, or click the **Regular Wave** icon in the **Analysis** toolbar.

Set **Wave Type** to Airy Wave Theory, Stokes 2nd Order Wave Theory, or Stokes 5th Order Wave Theory for the calculation of the Froude-Krylov forces. Airy Wave Theory is not recommended for modeling big waves; Stokes 5th Order Wave Theory is typically suited to shallow water situations. Enter the **Direction** and **Amplitude** of the wave. The direction is shown in the graphics with an arrow (light blue).

The other parameters to define the Regular Wave depend on the **Wave Type**. For Airy or Stokes 2nd Order waves, enter a **Period** or **Frequency** for the wave. The period/frequency in general should be within the range of that in the Hydrodynamic Diffraction analysis.

Note:

Both Frequency and Period are enabled as parameters, but it is only possible to parameterize one of these at a time. For example, if you set Period as a parameter, the Frequency field will not longer be visible; to set Frequency as a parameter, Period must be removed as a parameter first.

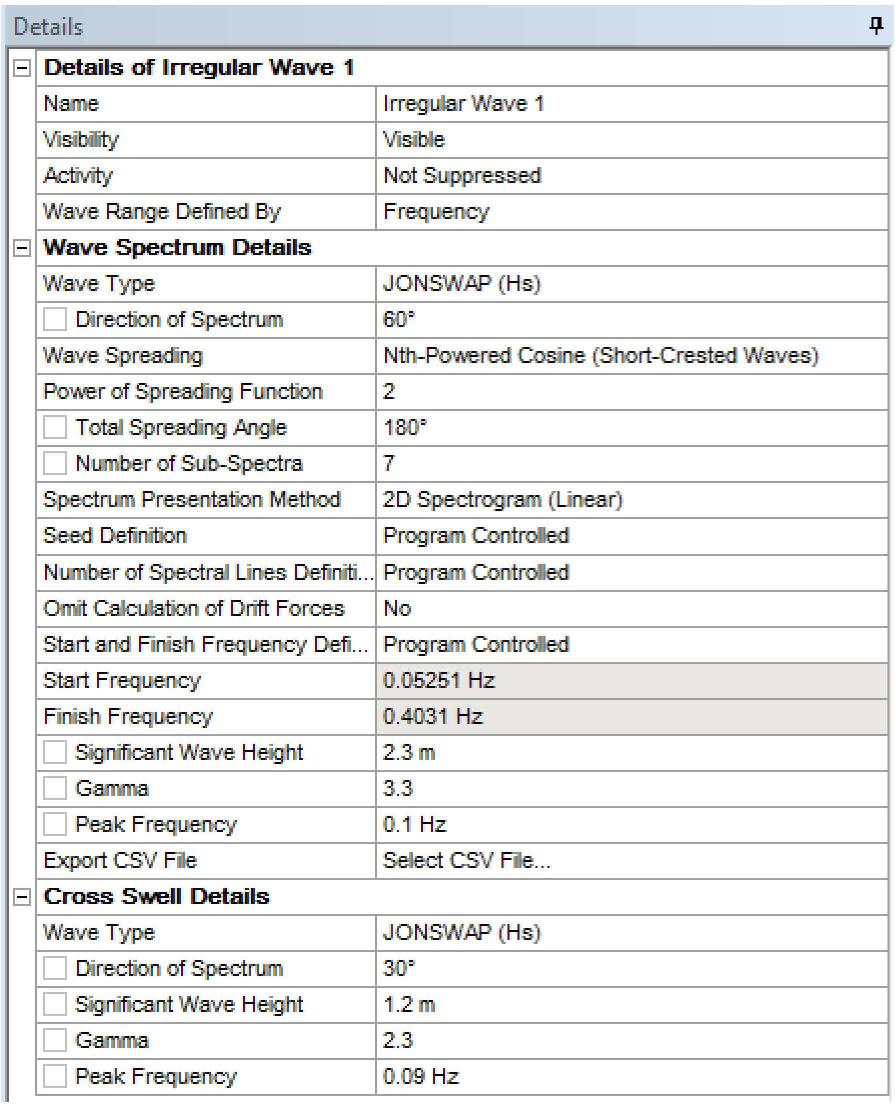
For Stokes 5th Order waves, the wave definition is controlled by 6 parameters: **Period**, **Wavelength**, **Still Water Depth**, **Mean Water Depth**, **Uniform Current**, and **Volume Flux**. These are interdependent, meaning that you can define any 3 parameters at once; the **Define By** option allows you to choose which parameters to set. For example, when **Define By** is selected as **Wavelength, Still Depth, Current**, you must enter values for **Wavelength**, **Still Water Depth** and **Uniform Current**. The **Period**, **Mean Water Depth** and **Volume Flux** values are calculated from your entered data, and displayed (read-only) for your reference. For more information, see Fifth Order Stokes Wave.

The wave ramp is introduced to reduce the transient motion of the structure at the beginning of a time domain analysis. The wave ramp will take effect from t=0.0 to $t = t_w$, during which time a wave ramp factor f ($0.0 < f < 1.0$) will be calculated and then used to multiply the incident wave amplitude. The wave ramp factor f is

$$f = \sin^2\left(\frac{\pi t}{2t_w}\right)$$

It can be seen that at t=0, the factor is 0.0 and at $t=t_w$, the factor is 1.0. If **Ramping Method** is set to Program Controlled, t_w will default to the wave period. Alternatively, change **Ramping Method** to Manual Definition to allow the **Ramping Time** to be set to the desired time t_w, or set **Ramping Method** to No Ramping to turn the wave ramp off.

5.11.7. Irregular Wave



Details

Details of Irregular Wave 1	
Name	Irregular Wave 1
Visibility	Visible
Activity	Not Suppressed
Wave Range Defined By	Frequency
Wave Spectrum Details	
Wave Type	JONSWAP (Hs)
Direction of Spectrum	60°
Wave Spreading	Nth-Powered Cosine (Short-Crested Waves)
Power of Spreading Function	2
Total Spreading Angle	180°
Number of Sub-Spectra	7
Spectrum Presentation Method	2D Spectrogram (Linear)
Seed Definition	Program Controlled
Number of Spectral Lines Definiti...	Program Controlled
Omit Calculation of Drift Forces	No
Start and Finish Frequency Defi...	Program Controlled
Start Frequency	0.05251 Hz
Finish Frequency	0.4031 Hz
Significant Wave Height	2.3 m
Gamma	3.3
Peak Frequency	0.1 Hz
Export CSV File	Select CSV File...
Cross Swell Details	
Wave Type	JONSWAP (Hs)
Direction of Spectrum	30°
Significant Wave Height	1.2 m
Gamma	2.3
Peak Frequency	0.09 Hz

An Irregular Wave can be added to the Hydrodynamic Response system, either individually or as part of an Irregular Wave Group (p. 171). In addition, it can be added to the Hydrodynamic Diffraction system when Linearized Morison Drag (p. 132) is specified for the analysis (however, only a single Irregular Wave can be used in the Hydrodynamic Diffraction system, and no Cross Swell can be specified).

Note:

Although you can add multiple Irregular Wave Groups and individual Irregular Waves to your analysis, you may have only one active wave or wave group; the others must be suppressed.

To add an Irregular Wave:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object and select **Insert** > **Irregular Wave** > ***Wave Type*** , or click the **Irregular Wave** icon in the **Analysis** toolbar and select the wave type from the drop-down menu.

For each type of Irregular Wave the Wave Spectrum Definition Data table displays the **Directions**, **Sub-Weights**, **Frequencies**, and **Spectral Ordinates** that fully define the wave spectrum/sub-spectra. The wave definition in the Wave Spectrum Details section of the Details panel is displayed on the Main Spectrum tab. If you have also defined a secondary spectrum using the Cross Swell Details, the **Frequencies** and **Spectral Ordinates** for this are shown separately on the Cross Swell Spectrum tab. Clicking on the Graph tab of the Wave Spectrum Definition Data panel presents a graphical display of the **Spectral Ordinates** against their corresponding **Frequencies** (and, for multi-directional wave spectra, **Directions**). The Main Spectrum and Graph tabs are empty until the Irregular Wave has been fully defined. The Cross Swell Spectrum tab is disabled if no cross swell is defined. Control of the Spectrum and Graph displays is described in greater detail below.

The following parameters are available in the **Details** panel for all Irregular Wave types.

If the **Analysis Type** in the associated Analysis Settings is set to Irregular Wave Response, a **Ramping Time** may be specified. The wave ramp is described in Regular Wave (p. 155) and introduces the same behavior for Irregular Waves. If **Ramping Method** is set to Program Controlled, t_w defaults to the longest period of any wave component in the defined spectrum. Alternatively, change **Ramping Method** to Manual Definition to allow the **Ramping Time** to be set to the desired time t_w, or set **Ramping Method** to No Ramping to turn the wave ramp off.

Direction of Spectrum sets the direction of waves within a wave spectrum. This option is not available for an Irregular Wave in a Hydrodynamic Diffraction system, or for a User Defined Spectrum (p. 164) where the **Wave Spreading** option is set to Manual Definition (Carpet Spectrum).

Wave Spreading allows you to set a multi-directional wave spectrum. The default setting of None (Long-Crested Waves) produces a unidirectional wave spectrum (all the wave energy acts in a single direction). Setting **Wave Spreading** to Nth-Powered Cosine (Short-Crested Waves) produces a symmetric spread of wave energy over a number of directions. Selecting this option displays the additional parameters **Power of Spreading Function**, **Total Spreading Angle**, and **Number of Sub-Spectra**:

- **Power of Spreading Function** sets the power N to which the cosine of θ is raised, where θ is the angle between the main **Direction of Spectrum** and a sub-spectrum direction. To find the spectral ordinates in a sub-spectrum direction, the spectral ordinates in the main spectrum direction are multiplied by a factor $f(\theta)=\cos(\theta)^N$. Acceptable values for the **Power of Spreading Function** are integer numbers between 2 and 250.

- **Total Spreading Angle** defines the spread range of wave energy in the short-crested waves. This angle may take any value in the range 1° to 180° when the **Power of Spreading Function** is set to 2, but will default to (and cannot be changed from) 180° for any other **Power of Spreading Function** value.

- **Number of Sub-Spectra** sets the number of directions over which the wave energy is spread. For Nth-Powered Cosine wave spreading this value must be an odd number in the range of 7 to 27.

Note:

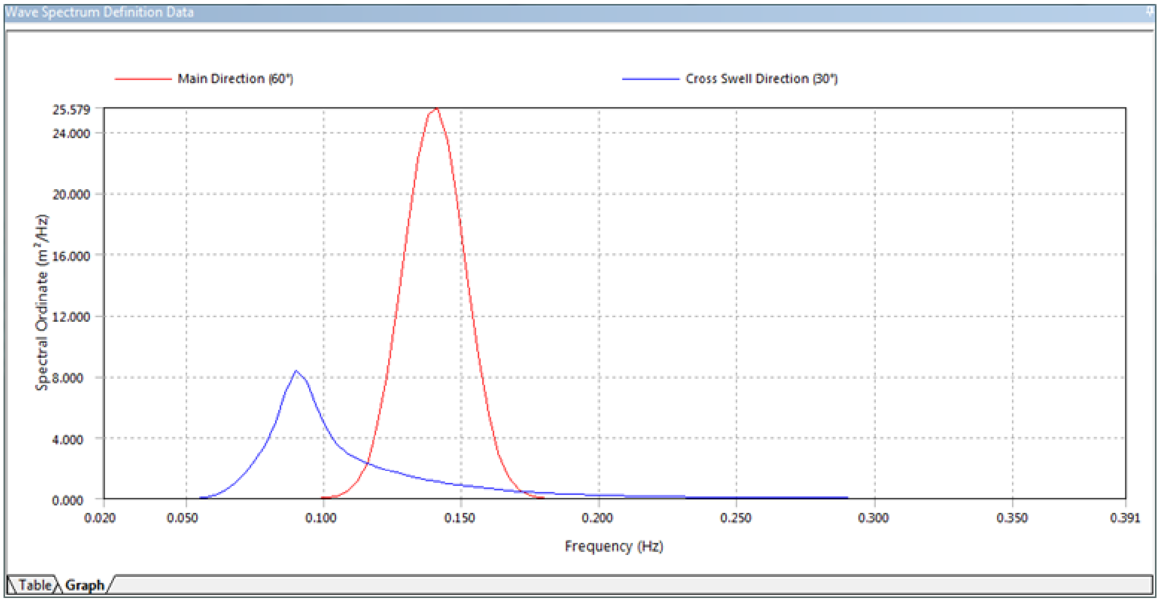
The **Directions** and **Sub-Weights** for an Nth-Powered Cosine wave spread are always determined by the program, and correspond to directions and weighting factors that have been pre-calculated for a Gaussian integration of the $\cos(\theta)^N$ function over the given **Total Spreading Angle** and **Number of Sub-Spectra**. Note also that the factor C from Equation 2.55 in the *Aqwa Theory Manual* is included in the **Sub-Weight** values.

A third option for **Wave Spreading** is Manual Definition (Carpet Spectrum). This option is only available when the **Wave Type** is set to User Defined Spectrum. The definition of, and additional options related to, a Carpet Spectrum are described in User Defined Spectrum (p. 164).

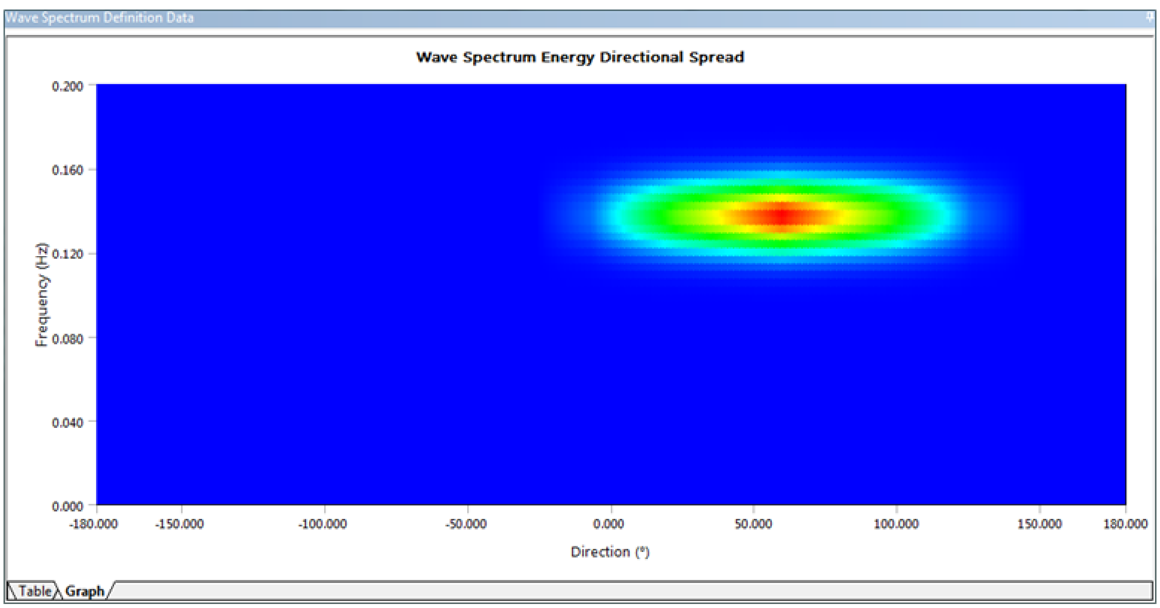
Note:

The **Wave Spreading** parameter is not available when a User Time History spectrum is generated from a Wave Height Time History (`.WHT`) file, as there is no accurate way to combine the Low Frequency Perturbation (LFP) Function (described in User Time History (p. 169)) with an Nth-Powered Cosine spreading function.

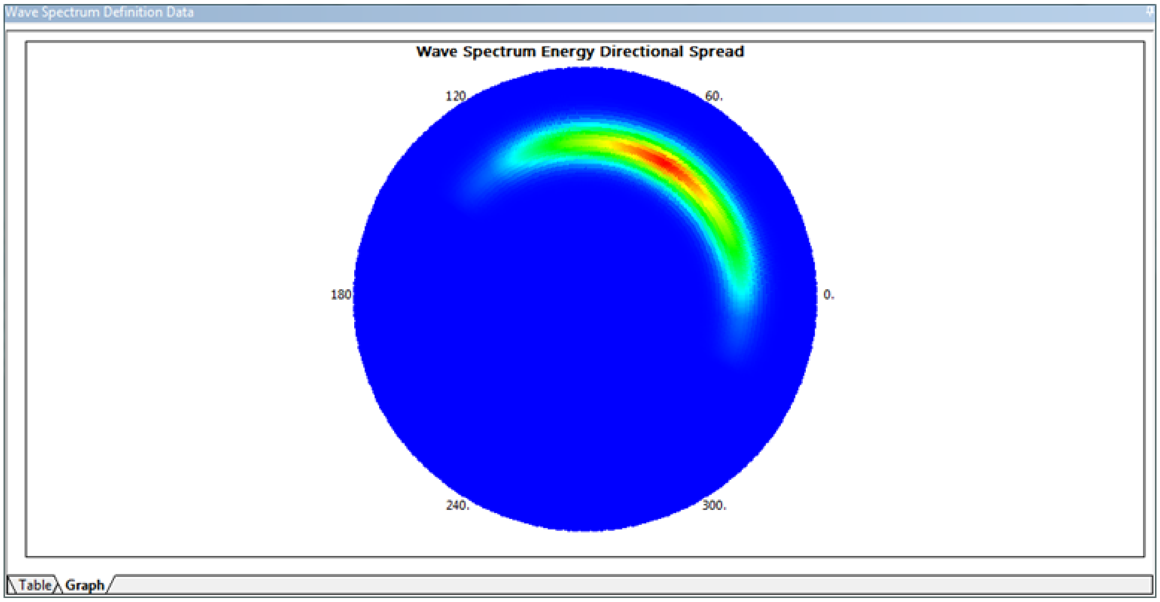
Spectrum Presentation Method allows you to change the display mode for the **Graph** tab of the Wave Spectrum Definition Data panel. For an Irregular Wave with **Wave Spreading** set to Long-Crested Waves, the graph plots the **Spectral Ordinates** against the **Frequencies** calculated or defined for that Irregular Wave (1D Graph option). If you also define a **Cross Swell Spectrum** in a unidirectional wave, the two spectra are plotted on the same axes.



When **Wave Spreading** is set to Nth-Power Cosine (Short-Crested Waves) or Manual Definition (Carpet Spectrum), the default **Spectrum Presentation Method** changes to 2D Spectrogram (Linear) and the **Graph** tab displays a colored contour plot of **Spectral Ordinates** against **Directions** (x-axis) and **Frequencies** (y-axis). Because a **Cross Swell** Spectrum is unidirectional (no spreading is applied to the wave energy), it will not be included in a 2D Spectrogram plot.

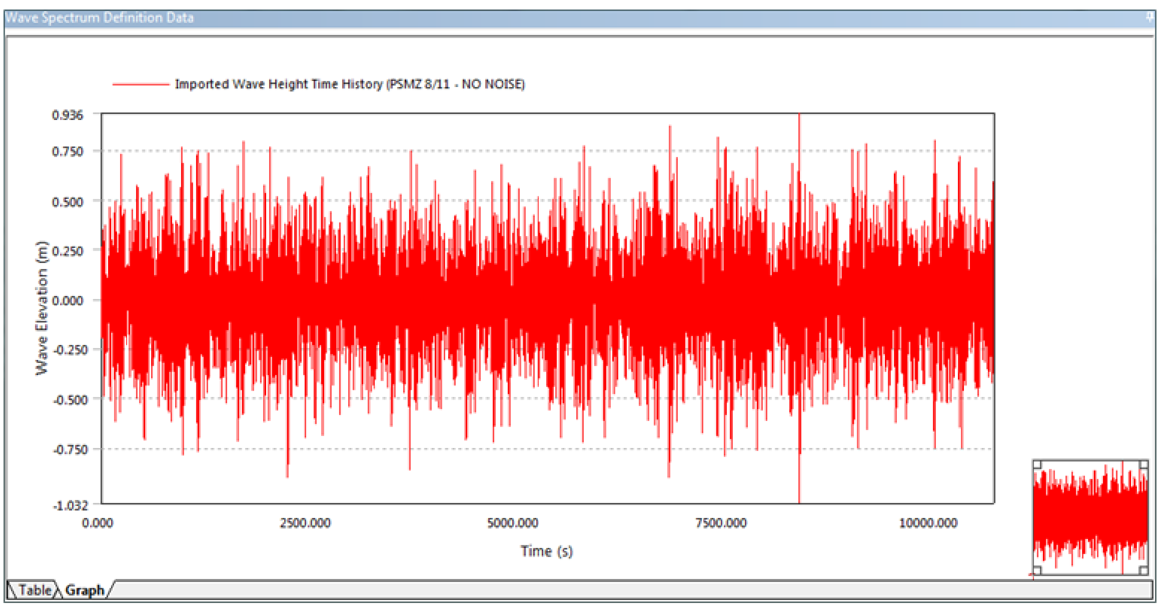


Alternatively you may display this contour plot on polar axes by selecting 2D Spectrogram (Polar), where **Spectral Ordinates** are plotted against **Frequencies** (radius).



While **Wave Spreading** is set to Nth-Power Cosine (Short-Crested Waves) or Manual Definition (Carpet Spectrum), changing **Spectrum Presentation Method** to 1D Graph displays all of the sub-spectra (and the **Cross Swell Spectrum**, if one is included).

For an Imported Time History wave the **Spectrum Presentation Method** has the additional option Wave Height Time History, which plots the wave elevations against the times from the imported `.WHT` file. The zoom box in the lower right corner may be used to view a more limited range of the wave elevation trace.



Seed Definition defines the random seed for a wave spectrum. The options available are Program Controlled or Manual Definition. If you select Manual Definition, you can set the seed value.

Number of Spectral Lines Definition defines the number of individual components used to simulate an Irregular Wave. The options available are Program Controlled or Manual Definition. If you select Manual Definition, you can set the **Minimum Number of Spectral Lines**, up to the maximum of 200.

The actual number of spectral lines used will typically be greater than the **Minimum Number of Spectral Lines** where you have added a **Cross Swell Spectrum** to an Irregular Wave.

Note:

The new user is strongly advised to use the Program Controlled option for **Number of Spectral Lines Definition** . The program will automatically generate the appropriate number of spectral lines for the particular method of analysis, and it is only in unusual circumstances that user input is required.

In the case of a spectrum imported from a Wave Height Time History (`.WHT`) file, the analysis will always use 200 spectral lines and the user will not be allowed to modify this value.

Omit Calculation of Drift Forces disables the second order drift force calculations for the current spectrum. Select Yes to disable the calculations and No to have them applied. This option is only available for Irregular Wave Response with Slow Drift calculations.

The user should, in general, specify a spectral sea state whose range of frequencies is less than the range for which the hydrodynamic parameters for the structure are defined (those specified in the **Wave Frequencies** (p. 145) object). At spectrum frequencies which are outside the range at which the hydrodynamic parameters are defined, the program will automatically extrapolate the values required.

The effects of cross swell are implemented in most of Aqwa. Select the **Cross Swell Spectrum** type in the **Cross Swell Details** section of the **Details** panel. The **Cross Swell Spectrum** can be different from the current **Wave Type**. However the parameters used to define the **Cross Swell Spectrum** are the same as those for the corresponding spectrum **Wave Type**, except that the start and finish frequencies are calculated by the program.

For all wave types except User Time History and User Defined Spectrum, set **Wave Range Defined by** to either Period or Frequency. Next, set **Start and Finish Frequency/Period Definition**. The available options are:

- Program Controlled (default)
- Manual Start Frequency/Period Auto Finish Frequency/Period
- Manual Finish Frequency/Period Auto Start Frequency/Period
- Manual Definition

Start Frequency/Period is the lowest frequency or longest period at which the spectrum is defined and **Finish Frequency/Period** is the highest frequency or shortest period at which the spectrum is defined. Enter values in the fields that are user defined, based on the **Start and Finish Frequency/Period Definition** setting.

For formulated and User Time History spectra, the **Export CSV File** option allows you to output (in a comma-separated value (`.CSV`) file format) the directions, sub-weights, frequencies, and spectral ordinates that define the Irregular Wave.

Other parameters are available in the Details panel based on the wave type defined. The following wave types are available:

5.11.7.1. JONSWAP (Hs or Alpha)
5.11.7.2. Pierson-Moskowitz
5.11.7.3. Ochi-Hubble
5.11.7.4. Bretschneider
5.11.7.5. TMA Spectrum
5.11.7.6. Gaussian
5.11.7.7. User Defined Spectrum
5.11.7.8. User Time History

5.11.7.1. JONSWAP (H_s or Alpha)

The JONSWAP wave spectrum can be used to describe a wave system where there is an imbalance of energy flow (that is, sea not fully developed). This is nearly always the case when there is a high wind speed. It may be considered as having a higher peak spectral value than the Pierson-Moskowitz spectrum but is narrower away from the peak in order to maintain the energy balance.

If you choose JONSWAP (Hs) as the **Wave Type**, **Significant Wave Height** will be used in the calculations rather than the parameter **Alpha**, (which is used when the **Wave Type** is set to JONSWAP (Alpha)).

Parameterization of the classic form of the JONSWAP spectrum (with parameters of fetch and wind speed) was undertaken by Houmb and Overvik (BOSS Trondheim 1976, Vol 1). These empirical parameters (which you must enter) are termed **Gamma** (γ), **Alpha** (α) and **Peak Frequency** (ω_p) (the frequency at which the spectral energy is a maximum). The peak frequency together with empirical parameters termed Gamma and Alpha are used in this formulation. The spectral ordinate (S) at a frequency (ω) is given by

$$S(\omega)=\frac{\alpha g^2\gamma^a}{\omega^5}e^{-\frac{5\omega_p^4}{4\omega^4}}$$

where:

ω_p = the peak frequency in rad/s

γ = the peak enhancement factor

$a=e^{-\frac{(\omega-\omega_p)^2}{2\sigma^2\omega_p^2}}$

$\sigma=\begin{cases}0.07 & \omega\leq\omega_p\\ 0.09 & \omega>\omega_p\end{cases}$

α = a constant that relates to the wind speed and peak frequency of the wave spectrum, which has the following relationship to the significant wave height H_s:

$$\alpha=\frac{\left(\frac{H_s}{4}\right)^2}{\int_0^\infty\frac{g^2\gamma^a}{\omega^5}e^{-\frac{5\omega_p^4}{4\omega^4}}d\omega}$$

5.11.7.2. Pierson-Moskowitz

The Pierson-Moskowitz spectrum is formulated in terms of the two parameters of **Significant Wave Height** (H_s) and average (**Zero Crossing Period**) wave period (T_z). This is considered of more direct use than the classic form, in terms of the single parameter wind speed, or the form involving the peak frequency, where the spectral energy is a maximum. The spectral ordinate (S), at a frequency (ω, in rad/sec), is given by

$$S(\omega)=\frac{1}{2\pi}\frac{H_s^2}{4\pi T_z^4}\left(\frac{2\pi}{\omega}\right)^5 e^{-\frac{1}{\pi T_z^4}\left(\frac{2\pi}{\omega}\right)^4}$$

The average (that is, mean zero-crossing) wave period (**Zero Crossing Period**) and **Significant Wave Height** are the parameters used to describe the Pierson-Moskowitz wave spectrum, the special case for a fully developed sea.

5.11.7.3. Ochi-Hubble

The Ochi-Hubble spectrum is a bimodal spectrum designed to represent both the low- and high-frequency components associated with storm conditions. Each mode is defined by a **Significant Wave Height**, a **Shape Parameter**, and a **Modal Frequency** or **Modal Period**. More information on the formulation may be found here.

5.11.7.4. Bretschneider

The Bretschneider spectrum is a special form of the Ochi-Hubble spectrum, with only a single spectral mode and unit shape parameter. The spectrum is therefore defined only by a **Significant Wave Height** and a **Modal Frequency** or **Modal Period**. More information on the formulation may be found here.

5.11.7.5. TMA Spectrum

The TMA (Texel-MARSEN-ARSLOE) spectrum is a form of the JONSWAP (Alpha) spectrum, modified to account for finite water depth. The spectrum is defined by the parameters **Gamma**, **Alpha**, and **Peak Frequency** or **Peak Period**, and uses the **Water Depth** defined in the Geometry details. More information on the formulation may be found here.

5.11.7.6. Gaussian

The standard Gaussian spectrum is given by

$$S(\omega)=\frac{H_s^2}{16\sigma\sqrt{2\pi}}e^{-\frac{\left(\omega-\omega_p\right)^2}{2\sigma^2}}$$

where:

H_s = significant wave height

ω_p = peak frequency in rad/s

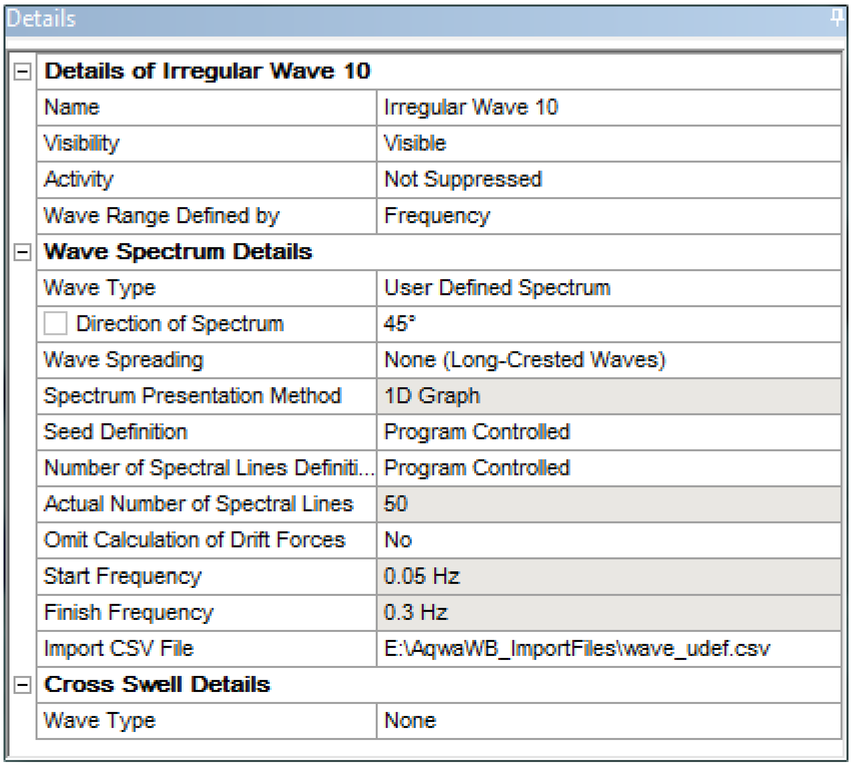
σ = standard deviation

The **Peak Frequency**, **Significant Wave Height**, and **Sigma** (σ) are the parameters used to describe the Gaussian wave spectrum. The value of σ must be at least $0.08\omega_p$.

5.11.7.7. User Defined Spectrum

This option may be used to input any user-defined one- or two-dimensional spectrum. It is normally employed for the input of non-deterministic spectra such as tank spectra, recorded full-scale spectra, or simply where the formulated spectrum is not yet available.

5.11.7.7.1. With Wave Spreading: None (Long-Crested Waves)


Details

Details of Irregular Wave 10	
Name	Irregular Wave 10
Visibility	Visible
Activity	Not Suppressed
Wave Range Defined by	Frequency
Wave Spectrum Details	
Wave Type	User Defined Spectrum
Direction of Spectrum	45°
Wave Spreading	None (Long-Crested Waves)
Spectrum Presentation Method	1D Graph
Seed Definition	Program Controlled
Number of Spectral Lines Definiti...	Program Controlled
Actual Number of Spectral Lines	50
Omit Calculation of Drift Forces	No
Start Frequency	0.05 Hz
Finish Frequency	0.3 Hz
Import CSV File	E:\AqwaWB_ImportFiles\wave_udef.csv
Cross Swell Details	
Wave Type	None

In the Wave Spectrum Definition Data table, enter the frequency (or period, depending on the **Wave Range Defined by** setting) and its corresponding spectral ordinate. The maximum number of frequencies/periods is 200.

The rows of the Wave Spectrum Definition Data table are automatically sorted in ascending order of **Frequency**. Zero or duplicate **Frequency** entries are not valid.

Wave Spectrum Definition Data

Frequency (Hz)	Spectral Ordinate (m^2/Hz)
Direction (°) ➔	45
Sub-Weight ➔	1
0.05	0.0
0.0604	0.1002
0.0708	4.0654
0.0812	17.334
0.0916	29.016
0.102	31.834
0.1125	28.568
0.1229	23.257
0.1333	18.08
0.1437	13.782
0.1541	10.447
0.1645	7.9353
0.175	6.0632
0.1854	4.6701
0.1958	3.6293
0.2062	2.8466
0.2166	2.2532
0.227	1.7994
0.2375	1.4493
0.2479	1.1768
0.2583	0.9628
0.2687	0.7934
0.2791	0.6583
0.2895	0.5497
0.3	0.4618

Table | Graph

If you create an Irregular Wave using one of the formulated spectra available in Aqwa (JONSWAP, Pierson-Moskowitz, or Gaussian) or generate a formulated spectrum from a User Time History, and then change the **Wave Type** to User Defined Spectrum, the spectral ordinates of the formulated spectrum become editable for you to modify as necessary.

Note:

Using a User Time History spectrum as the template for a User Defined Spectrum wave does not retain the Low Frequency Perturbation (LFP) Function of the User Time History wave (described in User Time History (p. 169)). Only the JONSWAP fit of the imported wave elevation spectral density is included.

Wave Spectrum Definition Data can otherwise be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).

- Contains exactly 2 columns, and no more than 200 rows.
- The **Direction** and **Sub-Weight** rows are not included.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Rotation Unit for Imported Data**, and **Frequency Unit for Imported Data** to the required unit system.

5.11.7.7.2. With Wave Spreading: Nth-Powered Cosine (Short-Crested Waves)

When defining a User Defined Spectrum with Nth-Powered Cosine (Short-Crested Waves) wave spreading, only the **Frequency** and main direction **Spectral Ordinate** columns are editable. Modifying one of the main direction spectral ordinates causes the other spectral ordinates at that frequency to be updated automatically.

Wave Spectrum Definition Data

Frequency (Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)
Direction (°) ➔	-40.4197	-21.7378	8.47394	45	81.52606	111.73781	130.4197
Sub-Weight ➔	0.12948	0.27971	0.38183	0.41796	0.38183	0.27971	0.12948
0.05	0.0	0.0	0.0	0.0	0.0	0.0	0.0
0.0604	6.70212e-4	0.01639	0.06787	0.1002	0.06787	0.01639	6.70212e-4
0.0708	0.02675	0.65434	2.70901	4.0654	2.70901	0.65434	0.02675
0.0812	0.11061	2.70538	11.20036	17.334	11.20036	2.70538	0.11061
0.0916	0.18391	4.49832	18.6232	29.016	18.6232	4.49832	0.18391
0.102	0.20212	4.94374	20.46728	31.834	20.46728	4.94374	0.20212
0.1125	0.18185	4.44797	18.41477	28.568	18.41477	4.44797	0.18185
0.1229	0.14836	3.62886	15.02362	23.257	15.02362	3.62886	0.14836
0.1333	0.11547	2.82438	11.69305	18.08	11.69305	2.82438	0.11547
0.1437	0.08807	2.15424	8.91864	13.782	8.91864	2.15424	0.08807
0.1541	0.06678	1.63332	6.76199	10.447	6.76199	1.63332	0.06678
0.1645	0.05073	1.24085	5.13718	7.9353	5.13718	1.24085	0.05073
0.175	0.03875	0.94773	3.92362	6.0632	3.92362	0.94773	0.03875
0.1854	0.02985	0.73002	3.02232	4.6701	3.02232	0.73002	0.02985
0.1958	0.02319	0.5672	2.34822	3.6293	2.34822	0.5672	0.02319
0.2062	0.01818	0.44478	1.84142	2.8466	1.84142	0.44478	0.01818
0.2166	0.01439	0.35199	1.45724	2.2532	1.45724	0.35199	0.01439
0.227	0.01149	0.28107	1.16363	1.7994	1.16363	0.28107	0.01149
0.2375	9.25101e-3	0.22627	0.93679	1.4493	0.93679	0.22627	9.25101e-3
0.2479	7.51124e-3	0.18372	0.76061	1.1768	0.76061	0.18372	7.51124e-3
0.2583	6.14407e-3	0.15028	0.62217	0.9628	0.62217	0.15028	6.14407e-3
0.2687	5.06218e-3	0.12382	0.51261	0.7934	0.51261	0.12382	5.06218e-3
0.2791	4.19949e-3	0.10272	0.42525	0.6583	0.42525	0.10272	4.19949e-3
0.2895	3.50632e-3	0.08576	0.35506	0.5497	0.35506	0.08576	3.50632e-3
0.3	2.94488e-3	0.07203	0.29821	0.4618	0.29821	0.07203	2.94488e-3

Table Graph

The same rules for importing a `.CSV` file apply: the first column of the imported file is inserted into the **Frequency** column, and the second is used for the main direction **Spectral Ordinates**. The spectral ordinates for all other directions are calculated automatically. The **Direction** and **Sub-Weight** should not be included in the `.CSV` file.

5.11.7.7.3. With Wave Spreading: Manual Definition (Carpet Spectrum)

Details

Details of Irregular Wave 1	
Name	Irregular Wave 1
Item ID	27
Visibility	Visible
Activity	Not Suppressed
Wave Range Defined by	Frequency
Wave Spectrum Details	
Wave Type	User Defined Spectrum
Wave Spreading	Manual Definition (Carpet Spectrum)
Number of Sub-Spectra	7
Sub-Spectrum Weighting	Program Controlled (Trapezoidal Integral)
Spectrum Presentation Method	2D Spectrogram (Linear)
Seed Definition	Program Controlled
Number of Spectral Lines Definiti...	Program Controlled
Actual Number of Spectral Lines	50
Omit Calculation of Drift Forces	No
Start Frequency	0.05001 Hz
Finish Frequency	0.30001 Hz
Import CSV File	E:\AqwaWB_ImportFiles\wave_udds.csv

With **Wave Type** set to User Defined Spectrum and **Wave Spreading** set to Manual Definition (Carpet Spectrum), any combination of Directions, Sub-Weights, and User-Defined Wave Spectra may be specified.

With Manual Definition (Carpet Spectrum) wave spreading, the **Number of Sub-Spectra** parameter permits any value in the range of 2 to 41.

Sub-Spectrum Weighting determines the weighting factors for each carpet direction. The available options are:

- Manual Definition, which allows the Sub-Weights to be entered manually (default).
- Program Controlled (Unit Weighting), which assigns a unit weighting factor to each direction.
- Program Controlled (Trapezoidal Weighting), which determines weighting factors W_i for each direction θ_i based on a simple trapezoidal integral, where:

$$W_i = \frac{r_\theta}{2R_\theta} \text{ with } r_\theta = \begin{cases} (\theta_2 - \theta_1) & i=1 \\ (\theta_d - \theta_{d-1}) & i=d \\ (\theta_{i+1} - \theta_{i-1}) & \text{otherwise} \end{cases}$$

  in which d is the **Number of Sub-Spectra** and $R_\theta = \theta_d - \theta_1$.

Wave Spectrum Definition Data

Frequency (Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)	Spectral Ordinate (m^2/Hz)
Direction (°) →	-85.42	-66.738	-36.526	0.0	36.526	66.738	85.42
Sub-Weight →	0.05468	0.1431	0.19532	0.2138	0.19532	0.1431	0.05468
0.05001	0.0	0.0	0.0	0.0	0.0	0.0	0.0
0.06042	6e-4	0.0156	0.0647	0.1002	0.0647	0.0156	6e-4
0.07084	0.0259	0.6341	2.6252	4.0654	2.6252	0.6341	0.0259
0.08125	0.1105	2.7038	11.1938	17.3346	11.1938	2.7038	0.1105
0.09167	0.185	4.5258	18.7372	29.0161	18.7372	4.5258	0.185
0.10208	0.203	4.9654	20.557	31.8342	20.557	4.9654	0.203
0.11251	0.1822	4.4559	18.4478	28.568	18.4478	4.4559	0.1822
0.12292	0.1483	3.6276	15.0183	23.2571	15.0183	3.6276	0.1483
0.13334	0.1153	2.8201	11.6752	18.08	11.6752	2.8201	0.1153
0.14375	0.0879	2.1497	8.8998	13.7821	8.8998	2.1497	0.0879
0.15417	0.0666	1.6296	6.7465	10.4475	6.7465	1.6296	0.0666
0.16458	0.0506	1.2377	5.1242	7.9353	5.1242	1.2377	0.0506
0.17501	0.0387	0.9457	3.9153	6.0632	3.9153	0.9457	0.0387
0.18542	0.0298	0.7284	3.0157	4.6701	3.0157	0.7284	0.0298
0.19584	0.0231	0.5661	2.3436	3.6293	2.3436	0.5661	0.0231
0.20625	0.0182	0.444	1.8382	2.8466	1.8382	0.444	0.0182
0.21667	0.0144	0.3514	1.455	2.2532	1.455	0.3514	0.0144
0.22708	0.0115	0.2807	1.162	1.7994	1.162	0.2807	0.0115
0.23751	9.2e-3	0.2261	0.9359	1.4493	0.9359	0.2261	9.2e-3
0.24792	7.5e-3	0.1835	0.7599	1.1768	0.7599	0.1835	7.5e-3
0.25834	6.1e-3	0.1502	0.6217	0.9628	0.6217	0.1502	6.1e-3
0.26875	5.1e-3	0.1238	0.5124	0.7934	0.5124	0.1238	5.1e-3
0.27917	4.2e-3	0.1027	0.4251	0.6583	0.4251	0.1027	4.2e-3
0.28958	3.5e-3	0.0857	0.355	0.5497	0.355	0.0857	3.5e-3
0.30001	2.9e-3	0.072	0.2982	0.4618	0.2982	0.072	2.9e-3

Table / Graph

For a User Defined Spectrum with Manual Definition (Carpet Spectrum) wave spreading, the Wave Spectrum Definition Data table maybe populated using the method described in With Wave Spreading: None (Long-Crested Waves) (p. 164): create a User Defined Spectrum with **Wave Spreading** set to None (Long Crested Waves), turn this into a multidirectional spectrum by changing **Wave Spreading** to Nth-Powered Cosine (Short Crested Waves), and then make all fields of the Wave Spectrum Definition Data table editable by changing **Wave Spreading** to Manual Definition (Carpet Spectrum).

Wave Spectrum Definition Data can otherwise be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated value (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly N+1 columns, where N is the **Number of Sub-Spectra**, and no more than 200 rows.
- The **Direction** and **Sub-Weight** rows are included.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data**, **Rotation Unit for Imported Data**, and **Frequency Unit for Imported Data** to the required unit system.

The **Direction** and **Sub-Weight** rows must be included in the `.CSV` file, even if the sub-weights are Program Controlled (where any values in the **Sub-Weight** row of the `.CSV` file are ignored). There must also be (arbitrary) entries for the 'Direction' and 'Sub-Weight' header cells.

An example of such a `.CSV` file, as used to create the table shown above, follows:

```
dir,-85.42,-66.738,-36.526,0,36.526,66.738,85.42
wt,0,0,0,0,0,0,0
0.050006,0,0,0,0,0,0,0
```

```
0.060415,0.0006,0.0156,0.0647,0.1002,0.0647,0.0156,0.0006
ETC.
```

5.11.7.8. User Time History

A time history series of wave elevations may be imported into your Hydrodynamic Response analysis, in order to reproduce model test wave conditions as accurately as possible. Select **Insert > Irregular Wave > Import WHT File...**, or select **Irregular Wave > Import WHT File...** from the Analysis toolbar, and browse to the file location.

The **Imported WHT File** field shows the file location on the disk. If the imported file contains the optional **NAME** field, the value is displayed in the **Imported WHT Name** field.

When a time domain analysis is carried out, the wave elevation time-history will be reproduced exactly, within the frequency range of the fitted spectrum and subject to the limitations of roundoff error. This is achieved by multiplying each of the spectral wave components by a different Low Frequency Perturbation (LFP) Function; that is:

Wave elevation = $\sum_{j=1}^{N} a_j \cos\left(-\omega_j t + k_j x + \alpha_j\right) F_j(t)$

where:

N = number of spectral lines (set to 200)

j = wave component number

t = time

ω_j = frequency (as normally output by the Aqwa solver)

α_j = phase (as normally output by the Aqwa solver)

a_j = amplitude

k_j = wave number

$F_j\left(t\right)$ = LFP function

Note that no spurious low frequency waves are generated by the above method. For any wave component, the minimum frequency present in the wave elevation is $\omega_j - d_w$, where d_w is the highest frequency present in the LFP function. Note also that there is no frequency overlap for each wave component. Each LFP function can be considered as a frequency spreading function over a limited set of contiguous frequency bands. In this case each wave component has a different energy (as opposed to the standard Aqwa wave components which have equal energy).

Import of the time series will also generate a user-defined spectrum, using a Fast Fourier Transform, whose frequency range is based on a JONSWAP fit of the wave elevation spectral density. If Computation Type is Stability Analysis or Frequency Statistical Analysis in a Hydrodynamic Response system, this spectrum will be used in the same way as a normal user-defined spectrum. As the phases of the spectral wave components are allocated randomly, the input wave elevation time history will not be reproduced.

Note:

Current is ignored when calculating the phase wave forces on the structure and the wave kinematics for Morison elements.

The `*.WHT` file is an ASCII file with the wave elevation data in free format with 2 values per line. The first value is the time and the second value is the wave elevation.

The following 2 statements are required in the file:

```
DEPTH=value
G=value
```

The DEPTH value should match the Water Depth specified in the Geometry Details or else a warning will be generated. The G value is compared to the Gravity specified in the Geometry Details (p. 42) to determine the units used. If no G value is defined, you have the option to set the unit system for the imported data manually, using the **Length Unit for Imported Data** and **Angle Unit for Imported Data** controls under **Wave Spectrum Details**.

The following optional data can also be input. If any are omitted, the relevant value defaults to zero.

```
DIRECTION=value(degrees)
X_REF=value
Y_REF=value
NAME=Spectrum Name
CURRENT_SPEED=value
CURRENT_DIRECTION=value(degrees)
```

Note:

- The X_REF and Y_REF values are used in the calculation of the phase of the wave and are the position where the wave elevation was measured. For example (in SI units), if the DIRECTION of the wave is zero degrees (that is, along the positive X-axis in the global coordinate system) then values of X_REF/Y_REF of 100.0/0.0 will indicate that the wave elevation was measured 100 metres downstream of the 0,0 wave reference point. Omission of these data will default the reference point to 0,0. Meaning, the wave elevation will be calculated using the origin of the global coordinate system as the point at which the wave elevation will be reproduced. X_REF, Y_REF and DIRECTION values appear in the Details panel for the Wave object.

- The Spectrum Name will be used for graphs and tables where appropriate throughout the program, and is appended to the Wave object name in the tree.

- CURRENT SPEED and CURRENT DIRECTION are needed for calculation of the wavelengths of the wave components used to reproduce the wave elevation. If present they must match the corresponding values in the unsuppressed Current object in the tree (if there is no Current object, a warning will be generated.) If omitted, it is assumed that there is no current and a warning will be issued.

- The duration of the time history in the file should be at least 7200s. This duration is necessary in order to give sufficient resolution of low frequency resonant responses. If the file contains less data than this, the data will be extended automatically up to 7200s, using a process of mirroring and copying.
- The maximum number of timesteps in the .WHT file is 150000.
- Comments (starting with * in Column 1) may be added anywhere in the file.

An example of a `*.WHT` file is shown below.

```
 ------------------------------------
* This is an example of a *.wht file
*
DEPTH=30.0
G=9.81
DIRECTION=0.0
X_REF=100.0
Y_REF=0.0
NAME=EXAMPLE
CURRENT_SPEED=0.6
CURRENT_DIRECTION=90
* TIME WAVE HT
*     s      m
0.0000 -1.088
0.2366 -1.188
0.4732 -1.268
0.7098 -1.351
0.9464 -1.427
1.1830 -1.471
1.4196 -1.494
1.6562 -1.476
1.8928 -1.406
2.1294 -1.293
2.3660 -1.149
2.6026 -0.966
ETC.
------------------------------------
```

5.11.8. Irregular Wave Group

The Irregular Wave Group allows you to include multiple Irregular Waves in your analysis.

To add an Irregular Wave Group:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object and select **Insert > Irregular Wave Group**, or click the **Irregular Wave Group** icon in the Analysis toolbar.
3. Click the **Irregular Wave Group** object in the tree view.
4. Right-click the **Irregular Wave Group** object and select **Insert > Irregular Wave >** ***`Wave Type`***, or click the **Irregular Wave** icon in the **Analysis** toolbar, and select the wave type from the drop-down menu.
5. Define the options of each individual wave spectra in the group.

To view a summary of the waves under an Irregular Wave Group, select the wave group object in the tree.

Irregular Waves Summary

	Irregular Wave 1	Irregular Wave 2	Irregular Wave 3
Activity	Not Suppressed	Not Suppressed	Not Suppressed
Wave Type	JONSWAP (Hs)	Imported Time History	Pierson-Moskowitz
Direction of Spectrum	0.0°	10°	30°
Wave Spreading	Nth-Powered Cosine (Short-Crested Waves)	-	None (Long-Crested Waves)
Power of Spreading Function	2	-	-
Total Spreading Angle	180°	-	-
Number of Sub-Spectra	7	-	-
Significant Wave Height	2.4 m	-	1.8 m
Gamma	3.3	-	-
Peak Frequency	0.09 Hz	-	-
Cross Swell Wave Type	None	-	JONSWAP (Alpha)
Zero Crossing Period	-	-	12 s
Cross Swell Direction	-	-	-60°
Cross Swell Alpha	-	-	0.008
Cross Swell Gamma	-	-	3.1
Cross Swell Peak Frequency	-	-	0.11 Hz

Table Graph

If the **Analysis Type** in the associated Analysis Settings is set to Irregular Wave Response, a **Ramping Time** may be specified. The wave ramp is described in Regular Wave (p. 155) and introduces the same behavior for Irregular Wave Groups. If **Ramping Method** is set to Program Controlled, t_w defaults to the longest period of any wave component in all the wave spectra defined in the Irregular Wave Group. Alternatively, change **Ramping Method** to Manual Definition to allow the **Ramping Time** to be set to the desired time t_w, or set **Ramping Method** to No Ramping to turn the wave ramp off.

Note:

- Although you can add multiple Irregular Wave Groups and individual irregular waves, you must have only one active in order for the analysis to solve; the others must be suppressed.
- There are limits on the total number of Irregular Waves you can insert in a group. The total number of spectra is 41; an irregular-wave of formulated type (Pierson-Moskowitz, JONSWAP) counts for 1 (or 2 if you add Cross-Swell), imported wave time history counts for 1 (but you can only have a maximum of 5 imported history), and user defined counts for 1 as well.

5.11.9. Current

A Current can be added to the Hydrodynamic Response system to supplement the wave forces being applied to the structure. Current loading will be applied to tubular beams and slender tube elements using the velocities at the depth of the element. If cable dynamics is used for the definition of a composite catenary cable, current loading will also be applied along the length of the cable using the variable current velocity with depth.

Current loading is also applied at the center of gravity of a vessel (as long as Current Force Coefficients have been provided) and will utilize the current velocity at a specified **Current Calculation Position**, as defined in the Part details. Alternatively, where Morison Hull Drag Coefficients have been defined, current loads in a Hydrodynamic Response analysis are calculated using the current velocity at the depth of the structure center of gravity.

Note:

Although you can add multiple Current objects to your analysis, you may only have one active Current; the others must be suppressed.

To add a Current:

1. Select the **Hydrodynamic Response** object in the tree view.

2. Right-click the **Hydrodynamic Response** object and select **Insert** > **Current** > ***`Current Type`*** , or click the **Current** icon in the **Analysis** toolbar and select the current type from the drop-down menu.

3. Click the **Current** object in the tree and enter the current data.

Note:

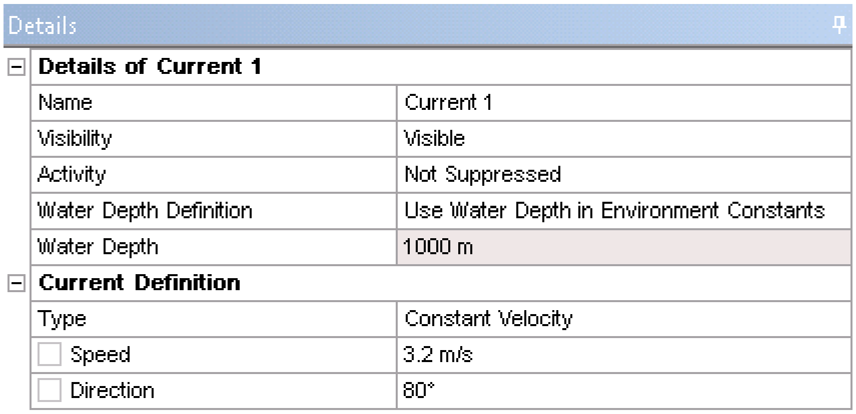
If an Irregular Wave is defined by importing a User Time History of wave elevations from a WHT file, it may be necessary to define the current profile with negative velocities: the WHT file must contain a positive constant current in order for the waves to be calculated correctly. The total current velocity will be the sum of the value in the WHT file and the values defined here.

For all types of Current it is necessary to define a **Water Depth** to which the current profile extends. By default, the **Water Depth Definition** is set to Use Water Depth in Environment Constants, and the **Water Depth** field will reflect the value defined under Details of Geometry; or, where a Composite Cable Seabed is defined, the **Water Depth Definition** may be set to Use Composite Cable Seabed Water Depth at Reference Point. Alternatively, the **Water Depth Definition** can be set to Manual Definition and the total **Water Depth** for the current profile can be entered directly.

Use the **Type** option to select the type of current that you would like to define. The following current types are available:

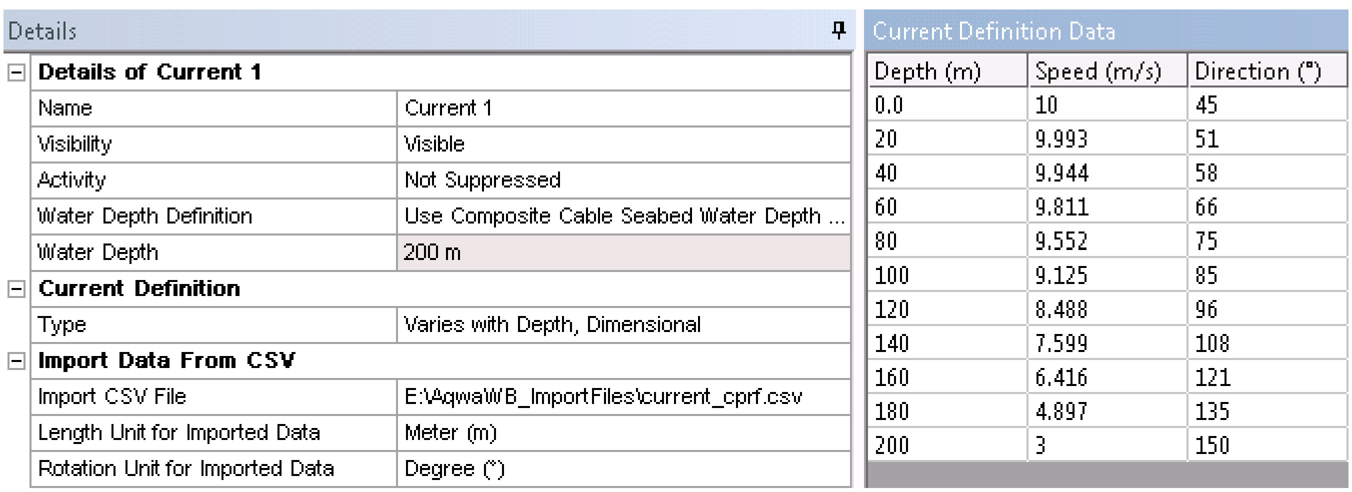
- Constant Velocity (p. 174)
- Varies with Depth, Dimensional (p. 174)
- Varies with Depth, Non-Dimensional (p. 175)
- Formulated (p. 176)

5.11.9.1. Constant Velocity



A Constant Velocity current does not vary with depth, and requires only its **Speed** and **Direction** to be defined. Both of these properties can be parameterized as part of a design point study.

5.11.9.2. Varies with Depth, Dimensional



In a more realistic description of the simulated environment the current definition should account for the variation of speed and direction with depth. When the current **Type** is set to Varies with Depth, Dimensional, this data can be entered directly into the Current Definition Data table.

Current depth is measured from the sea surface. The rows of the Current Definition Data table are automatically sorted in order of **Depth** from the sea surface to the sea bed.

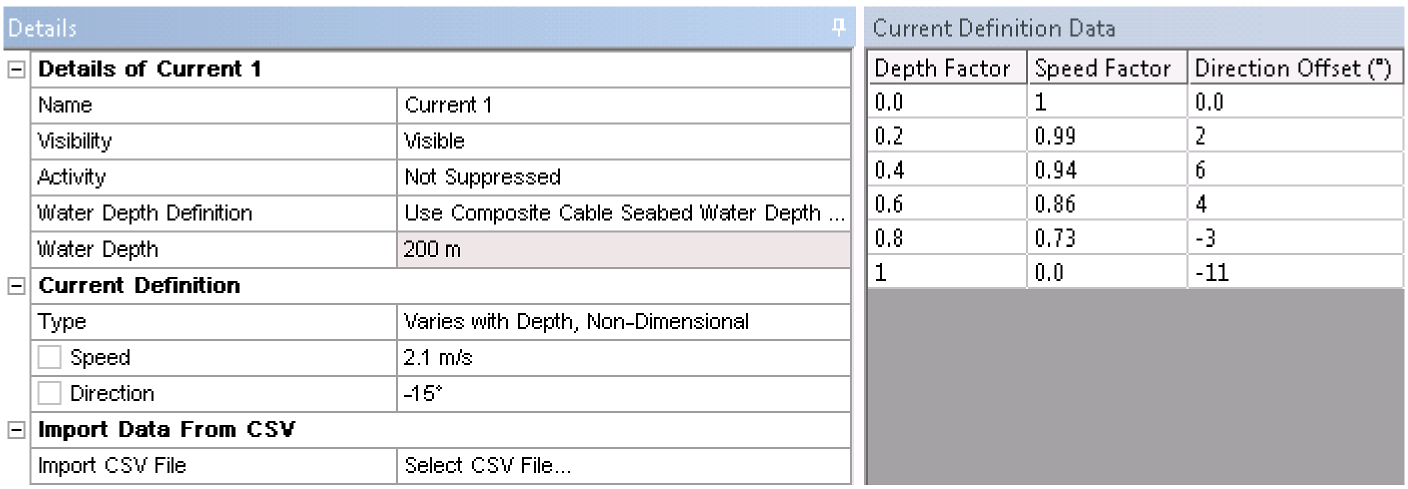
Duplicate **Depth** entries, negative entries, or entries greater than the **Water Depth** are not valid. A maximum of 25 rows may be added. If a complete definition of current with depth is not defined between the water surface and the defined **Water Depth**, constant values will be assumed based upon the lowest and highest defined data.

Current Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the .CSV extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 3 columns, and no more than 25 rows.

The unit system of the data in the .CSV file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data** and **Rotation Unit for Imported Data** to the required unit system.

5.11.9.3. Varies with Depth, Non-Dimensional



When the current **Type** is set to Varies with Depth, Non-Dimensional, data can be entered as factors and offsets into the Current Definition Data table. **Depth Factors** are multiplied by the **Water Depth**; **Speed Factors** are multiplied by the defined current **Speed**; and **Direction Offsets** (which may be negative) are added to the defined current **Direction**. Current **Speed** and **Direction** can both be parameterized as part of a design point study.

Duplicate **Depth Factor** entries, negative entries, or entries greater than 1.0 are not valid. A maximum of 25 rows may be added. If a complete definition of current with depth is not defined between the water surface and the defined **Water Depth**, constant values will be assumed based upon the lowest and highest defined data.

Current Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a .CSV file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the .CSV extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 3 columns, and no more than 25 rows.

The unit system of the data in the .CSV file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Rotation Unit for Imported Data** to the required unit system.

5.11.9.4. Formulated

Details

Details of Current 1	
Name	Current 1
Visibility	Visible
Activity	Not Suppressed
Water Depth Definition	Manual Definition
Water Depth	80 m
Current Definition	
Type	Formulated
Current Model	
Tidal Speed	3.2 m/s
Tidal Direction	25°
Tidal Variation Inverse Exponent	7
Mean Wind Speed	25 m/s
Mean Wind Direction	-65°
Reference Depth	50 m
Wind Speed Factor	0.03
Export CSV File	Select CSV File...

Current Definition Data

Depth (m)	Speed (m/s)	Direction (°)
0.0	3.28672	11.80939
18.28578	3.12002	16.22983
32.8653	2.97819	20.04916
44.38697	2.85186	23.30821
50	2.78162	25
53.40509	2.73415	25
60.39046	2.61769	25
65.74022	2.50123	25
69.78684	2.38477	25
72.80636	2.2683	25
75.02592	2.15184	25
76.63056	2.03538	25
77.76933	1.91892	25
78.5609	1.80246	25
79.09836	1.68599	25
79.4537	1.56953	25
79.68155	1.45307	25
79.82255	1.33661	25
79.90626	1.22015	25
79.95355	1.10368	25
79.97872	0.98722	25
79.99116	0.87076	25
79.99677	0.7543	25
79.999	0.63784	25
80	0.0	25

In cases where it is impractical to measure current velocities at depth, you can use a Formulated current.

To represent a current generated by the tide, you can define a **Tidal Speed** at the water surface, a **Tidal Direction** at the water surface, and a **Tidal Variation Inverse Exponent**. The variation of current velocity V_t with increasing depth z due to a tide is determined as

$$V_t(z)=V_{t0}\left(\frac{d-z}{d}\right)^{\frac{1}{m}}$$

Where V_{t0} is the tidal speed at the water surface and d is the total water depth. By default the **Tidal Variation Inverse Exponent** will be set to 7, which yields a one-seventh power law profile (the classic approximation for a fully-developed turbulent flow over a smooth solid boundary).

To represent a current generated by the wind, you can define a **Mean Wind Speed** and **Mean Wind Direction** (as measured over 1 hour at a height of 10 meters), a **Reference Depth**, and a **Wind Speed Factor**. The variation of current velocity V_w with increasing depth z due to a wind is determined as

$$V_w(z)=\begin{cases} V_{w0}\left(\frac{h_0-z}{h_0}\right) & \text{where } z<h_0 \\ 0.0 & \text{otherwise} \end{cases}$$

Where h_0 is the reference depth, and V_{w0} is the wind-generated current speed at the water surface defined by

$$V_{w0}=kU_0$$

In which k is the wind speed factor and U_0 is the mean wind speed measured over 1 hour at a height of 10 meters. The **Reference Depth** has a default value of 50 meters, and the **Wind Speed Factor** should typically lie within the range of 0.015 - 0.03.

Tide and wind-generated currents can be combined by defining parameters for both. The total current velocity V with increasing depth z due to tide and wind is then

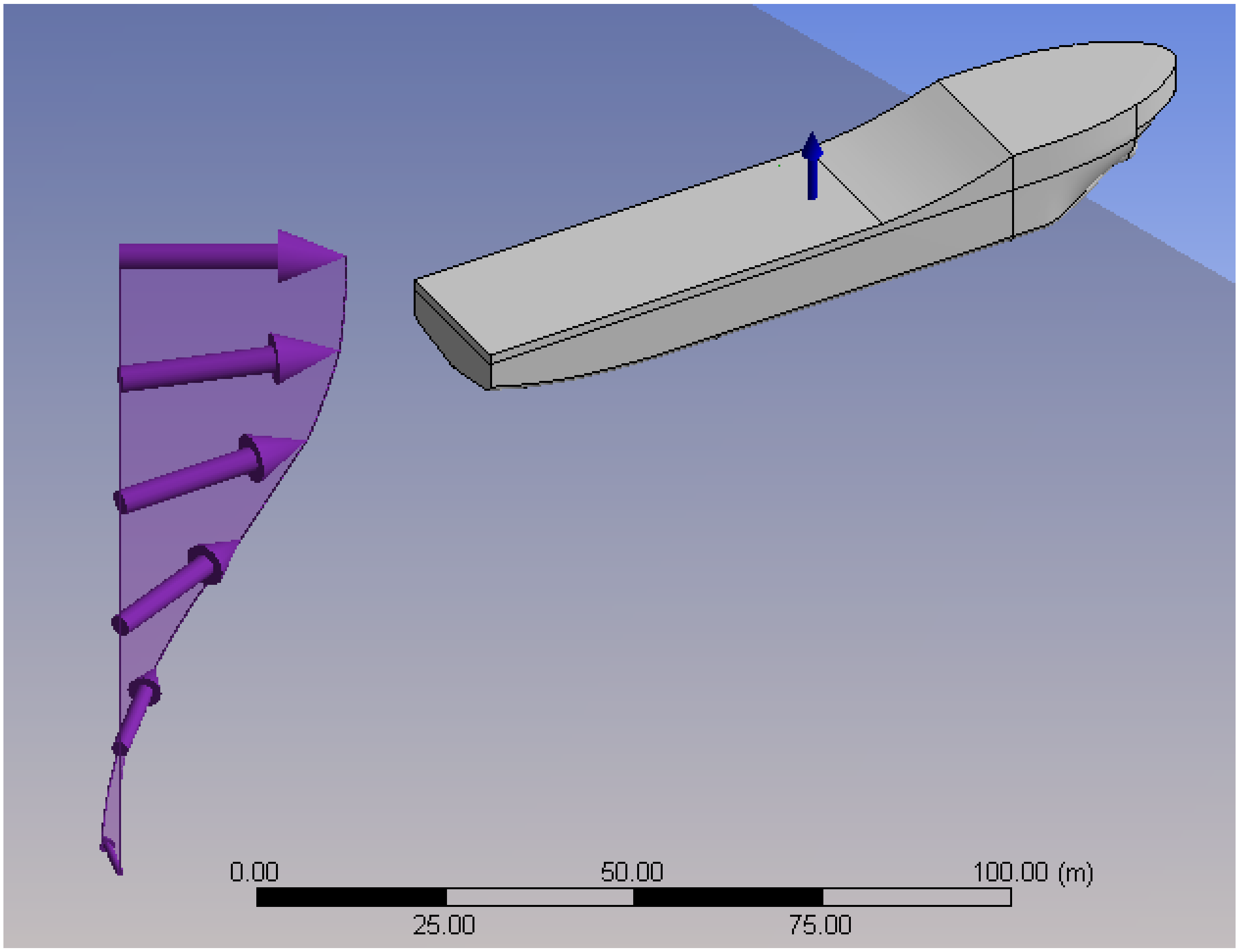
$$V\left(z\right)=V_t\left(z\right)+V_w\left(z\right)$$

All properties of a Formulated current may be parameterized as part of a design point study; however, at least one of **Tidal Speed** and **Mean Wind Speed** must always be non-zero.

The Current Definition Data table will display the current profile which is generated from the input data. Use the **Export CSV File** option if you would like to save this data in the `.CSV` file format.

5.11.9.5. Graphical Representation

For each current depth, an arrow is drawn to represent the current direction and speed; these will be purple if valid, or yellow if invalid. For a Constant Velocity current, a single arrow is shown at the water surface. The arrows are scaled with speed, while the overall current profile is illustrated by the purple shaded region.



5.11.10. Wind

To supplement the wave forces being applied to the structure, wind forces can be modeled. Wind can be modeled using a number of spectrum options (including user defined), user time dependent data, or constant velocity. The wind is shown in the graphic by an arrow at the reference height (or at ht = 0, if not applicable), scaled with speed.

To add a Wind:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object and select **Insert > Wind >** ***`Wind Type`*** , or click the **Wind** icon in the **Analysis** toolbar and select the wind type from the drop-down menu.

   Available wind types are:

   - Constant Velocity
   - Ochi and Shin Spectrum
   - API Standard Spectrum
   - NPD Standard Spectrum

- ISO Standard Spectrum
- User Defined Spectrum

You can also define Time Dependent Velocity wind data. For more information, see Time Dependent Velocity (p. 181).

The wind type of Constant Velocity is permitted for any wave type. Enter the **Speed** and **Direction** of the constant wind in the Wind Spectral Definition section of the Details panel.

Details of Wind 1	
Name	Wind 1
Visibility	Visible
Activity	Not Suppressed
Wind Spectral Definition	
Spectrum	None (Constant)
Speed	1 m/s
Direction	0.0°

The wind types (Ochi & Shin, API, NPD, ISO, and User Defined) are only permitted for analysis types that permit irregular waves. The following options are shown in the Details panel for all of the spectra, except User Defined (multiple sets of these options are defined for Time Dependent Velocity (p. 181)). Enter the **Reference Height** at which the wind speed is measured, the **Speed** of the wind at the reference height, and the **Direction** of the wind spectrum.

Wind Spectral Definition	
Spectrum	Ochi and Shin
Seed Definition	Program Controlled
Reference Height	10 m
Speed	1 m/s
Direction	0.0°

Seed Definition defines the random seed for a wind spectrum. The options available are Program Controlled or Manual Definition. If you select Manual Definition you can set the seed value.

For a **User Defined** spectrum, in addition to the Reference Height, Speed, and Direction, you need to enter the following in the Details panel:

- **Frequency Coefficient** - c_f
- **Speed Coefficient** - surface drag coefficient or roughness - c_s
- **Normalized Turbulence Intensity** - $I(Z)$

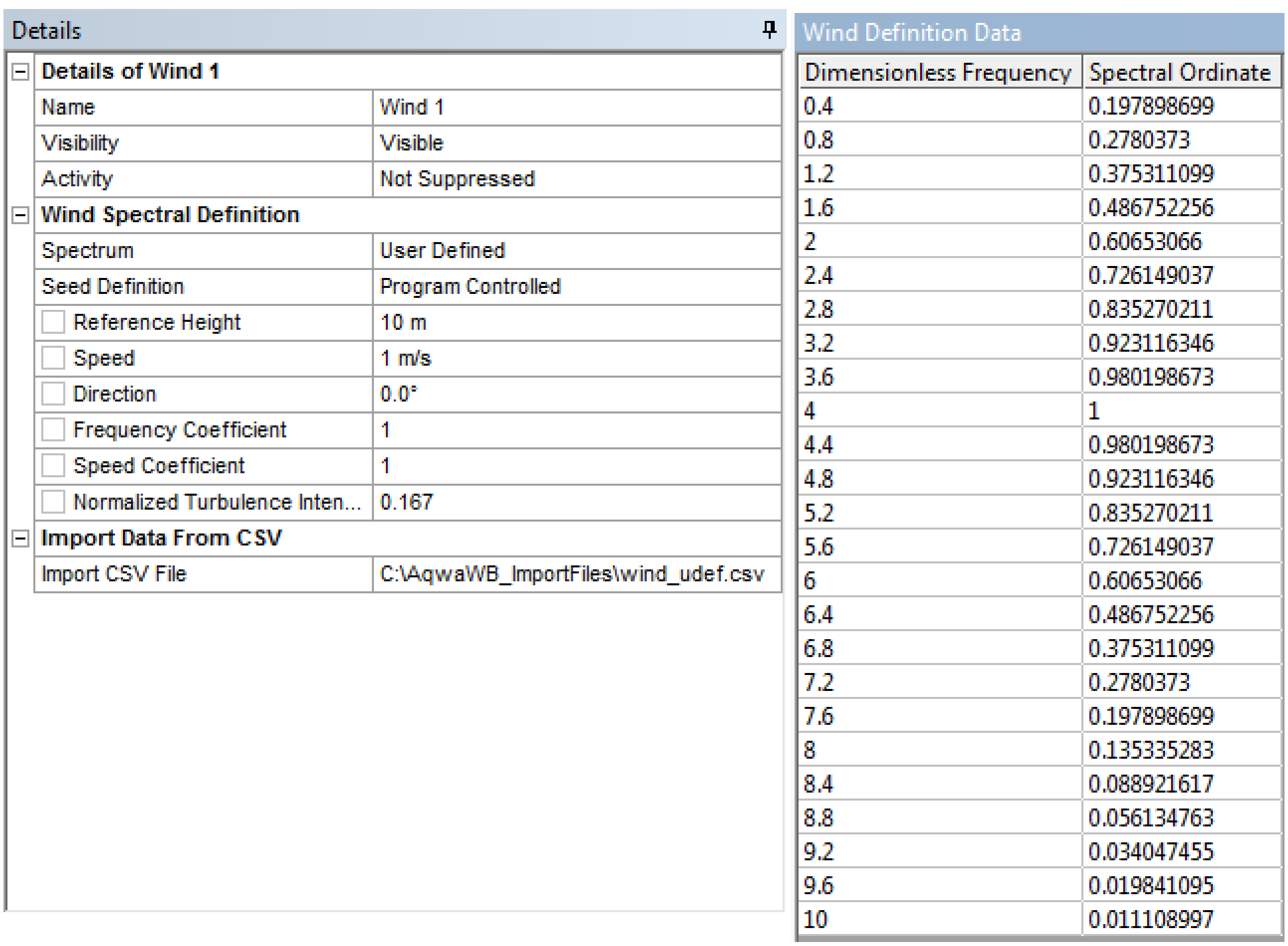
In the Wind Definition Data panel, enter a **Dimensionless Frequency** $\tilde{f}$ and its associated **Spectral Ordinate** $S(\tilde{f})$ in each row of the table. For user-defined spectra, the dimensionless frequency is defined as $\tilde{f}=c_f f \frac{Z}{U_z}$, where f is the frequency in hertz. The wind speed spectral density is defined as $S(f)=\frac{(I(z)U_z)^2}{f} c_s S(\tilde{f})$. Up to 200 rows may be entered.

The rows of the Wind Definition Data table are automatically sorted in ascending order of **Dimensionless Frequency**. Duplicate **Dimensionless Frequency** entries are not valid.

Wind Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 2 columns, and no more than 200 rows.

The unit system of the data in the `.CSV` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting the **Length Unit for Imported Data** and **Rotation Unit for Imported Data** to the required unit system.



Details

Details of Wind 1	
Name	Wind 1
Visibility	Visible
Activity	Not Suppressed
Wind Spectral Definition	
Spectrum	User Defined
Seed Definition	Program Controlled
Reference Height	10 m
Speed	1 m/s
Direction	0.0°
Frequency Coefficient	1
Speed Coefficient	1
Normalized Turbulence Inten...	0.167
Import Data From CSV	
Import CSV File	C:\AqwaWB_ImportFiles\wind_udef.csv

Wind Definition Data

Dimensionless Frequency	Spectral Ordinate
0.4	0.197898699
0.8	0.2780373
1.2	0.375311099
1.6	0.486752256
2	0.60653066
2.4	0.726149037
2.8	0.835270211
3.2	0.923116346
3.6	0.980198673
4	1
4.4	0.980198673
4.8	0.923116346
5.2	0.835270211
5.6	0.726149037
6	0.60653066
6.4	0.486752256
6.8	0.375311099
7.2	0.2780373
7.6	0.197898699
8	0.135335283
8.4	0.088921617
8.8	0.056134763
9.2	0.034047455
9.6	0.019841095
10	0.011108997

For more in-depth description of the available wind spectra, refer to the following sections in the *Aqwa Theory Manual*.

- Ochi and Shin Wind Spectrum
- API Wind Spectrum
- NPD Wind Spectrum
- ISO Wind Spectrum
- User Defined Wind Spectrum

5.11.10.1. Time Dependent Velocity

A time history of wind velocity can be defined directly in a table or by importing the information from an existing `.WVT` file.

To add a Time Dependent Wind Velocity:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object. To define the data directly in a table, select **Insert > Wind > Time Dependent Velocity > Manual Input**. To read data from a file, select **Insert > Wind > Time Dependent Velocity > Import WVT File**

   Alternatively, click the **Wind** icon in the **Analysis** toolbar. To define the data directly, select **Manual Input** from the drop-down menu. To read the data from a file select **Import WVT File**.

Note:

After the data is imported, no record of the file used to import the data is retained.

For Manual Input, enter in each row of the table in the Data panel the **Time**, and the **Velocity** and **Direction** of the wind at that time. The number of rows is not limited.

Details	
Details of Wind 1	
Name	Wind 1
Visibility	Visible
Activity	Not Suppressed
Wind Spectral Definition	
Spectrum	None (Time Dependent)
Steady-State Definition Uses	Initial Value
Speed	16.68 m/s
Direction	96.73°
Import Data From CSV	
Import CSV File	C:\AqwaWB_ImportFiles\wind_wvt.csv
Length Unit for Imported Data	Meter (m)
Rotation Unit for Imported Data	Degree (°)

Wind Definition Data

Time (s)	Velocity (m/s)	Direction (°)
0	16.68251141	96.73004565
5	13.79478242	85.17912966
10	6.037900322	54.15160129
15	5.133833817	50.53533527
20	7.994042331	61.97616933
25	15.07513922	90.30055688
30	17.71811307	100.8724523
35	17.25026525	99.00106101
40	16.01735829	94.06943317
45	18.79708503	105.1883401
50	19.22430819	106.8972328
55	20.26431788	111.0572715
60	22.27897687	119.1159075
65	27.4034642	139.6138568
70	25.45494955	131.8197982
75	17.39092793	99.56371173
80	5.112104906	50.44841962
85	4.037569109	46.15027644
90	8.748639188	64.99455675
95	17.49860653	99.99442611
100	17.80120141	101.2048056
105	14.98483751	89.93935004
110	12.27278258	79.0911303
115	13.75632701	85.02530804
120	14.95022756	89.80091025

The rows of the Wind Definition Data table are automatically sorted in ascending order of **Time**. Duplicate **Time** entries are not valid, and the **Time** in the last row of the table must equal or exceed the **Finish Time** defined in the Analysis Settings object.

Wind Definition Data can be entered manually, can be copied and pasted from an external source (for example, an Excel spreadsheet), or can be imported from a comma-separated values (`.CSV`) file using the **Import CSV File** option. For the **Import CSV File** option, the file must meet the following requirements:

- Has the `.CSV` extension.
- Contains values separated by commas, tabs, or single spaces (not multiple spaces).
- Contains exactly 3 columns.

The time defined in the table rows does not need to match the time steps defined in the analysis; the program will interpolate the wind speed and direction when necessary, using a cubic spline interpolation technique. When modeling periods of constant wind velocity, adequate data points must be provided to satisfy the interpolation method.

Defining the .WVT File

Comment lines beginning with * can be placed at any point in the file.

Before the data defining the time and wind velocities/directions, there must be a line containing only the text, "data_start".

After the data start, for each line the first column contains time (in s), the second column is wind speed, and the third column is wind direction (blowing towards). The numbers in the `.WVT` file are in a free format and can be separated by spaces.

The unit system of the data in the `.WVT` file is assumed to match the display unit system of the project. However, the imported data can be modified by setting **Length Unit of Imported Data** and **Angle Unit for Imported Data** to the required units.

There is no limit on the length of the `.WVT` file.

The following is an example of data from a `.WVT` file:

```
data_start
0.0000 8.0000 90.0000
0.5000 7.8532 88.7310
1.0000 7.4271 86.0840
...
```

Time Dependent Velocity in a Stability Analysis

A time dependent wind velocity can be added to a Time Response Analysis or a Stability Analysis. The option to include a Time Dependent Velocity in a Stability Analysis is provided so that a time dependent Wind object can be propagated into downstream Time Response Analysis systems (see Propagate (p. 258)).

In a Stability Analysis you must a provide a **Steady-State Definition**, which is used as a constant value for that analysis. This can be either:

- **Initial Value**, taken from the first row of the Wind Definition Data table
- **Manual Definition**, which may be entered directly into the **Speed** and **Direction** fields

5.11.11. Cable Winch

Two types of winch can be applied to a cable,

- a winch that can wind in or pay out as required to maintain a constant tension in the cable
- a winch that can wind in or pay out at a given rate to lengthen or shorten the line

> **Note:**
>
> In a Hydrodynamic Response System where the **Computation Type** is set to Stability Analysis (p. 134) or Frequency Statistical Analysis (p. 140), only the first of these options is available.

To add a Cable Winch:

1. Select the **Hydrodynamic Response** object in the tree view.

2. Right-click the **Hydrodynamic Response** object and select **Insert > Cable Winch**, or click the **Cable Winch** icon in the **Analysis** toolbar.

3. Select the **Cable Winch** object in the tree and configure it.

5.11.11.1. Winch that Maintains Constant Tension

Cable winches that maintain constant tension can only be used with Linear Elastic or Nonlinear Polynomial cables.

In the case of a constant tension winch, friction coefficients during the winding in (F_w) or paying out (F_p) process are required; hence the tension in the winch mooring line when winding-in is given by $T_w = T_s \left(1 - F_w\right)$ where T_s is the winch tension specified. When paying-out, the winch tension is given by $T_p = T_s \left(1 + F_p\right)$. For example, if the tension specified is 1000 tonnes and F_w and F_p are 0.3 and 0.1 respectively, then the tensions will be 700 and 1100 tonnes respectively.

The initial tension is undefined. The default initial tension is the winding-in tension; that is, 700 tonnes in the example above. The winding-in friction coefficient should be specified as negative if the paying-out value of tension is required as the initial tension.

The tension will be varied according to whether the range (distance between the anchor and vessel attachment point) is increasing or decreasing. If the range is less than the initial length specified the line becomes slack and the tension is zero.

To set up the constant tension Cable Winch object, click on the object. In the Details panel:

- Select **Maintains Constant Tension** from the **Cable Type** dropdown list
- Select the cable to associate with the winch from the **Cable Selection** dropdown
- Enter the value of the **Winding In Friction**
- Enter the value of the **Paying Out Friction**
- Enter the desired **Winch Tension** value

5.11.11.2. Winch that Changes Cable Length

Cable Winches that change the cable length can only be used with Linear Elastic (without Pulleys). In the case when the winch changes the cable length, it can be made to start at the given speed after the start time, and pauses if the maximum tension is exceeded. The winch will stop paying out or winding in when the final length is reached. An additional length, L_a, can be entered to affect the initial stiffness of the cable and the maximum stiffness ($K_{\mathrm{max}} = \frac{AE}{L_a}$).

The initial stiffness of the line, K_0 is used, which is equal to $\frac{EA}{L + L_a}$ for the line (E = Young's modulus, A = cross sectional area). The stiffness of the line during the winch action will change as the length changes; that is, the stiffness during the simulation, assuming that the length varies from L to the final length, L_f, will be $\frac{EA}{L + L_a}$ to $\frac{EA}{L_f + L_a}$; where $EA = K_0 L$.

The speed entered is positive for paying out and negative for winding in. In mathematical terms the speed is $\frac{dL}{dT}$, where L is the unstretched length of line and t is time. For lines which have significant strain (ε) (this precludes steel which yields at about 0.001), you may wish to consider the speed of the drum in terms of stretched length.

When winding in, the line wound onto the drum will have the same tension as the free line itself at any particular time. This means that in order to wind a length of unstretched line the effective speed must be increased by a factor of $(1+\varepsilon)$. This is done automatically. If you want to simulate a stretched line speed for winding in, then the speed specified should be input with a speed reduction factor of $\frac{1}{(1+\varepsilon)}$, where ε is the average strain.

When paying out, the adjustment of speed is not straightforward. The elastic energy of the line on the drum will depend on exactly how the line was wound on the drum originally. This 'energy' stored on the drum is unknown and is assumed to be zero. That is, the line on the drum when paying out is assumed to be unstretched. The effective winch speed, (effectively with only 1 side of the line stretched) is $\left(1+\frac{\varepsilon}{2}\right)$. If you want to simulate a stretched line speed for paying out, then the speed specified should be input with a reduction factor of $\frac{1}{\left(1+\frac{\varepsilon}{2}\right)}$, where ε is the average strain.

To set up the changing cable length Cable Winch object, click on the object. In the Details panel:

- Select **Changes Cable Length** from the **Cable Type** dropdown list
- Select the cable to associate with the winch from the **Cable Selection** dropdown
- Enter the value of the **Start Time**
- Enter the cable's **Final Length** at which winching will stop
- Enter the value of the **Speed** of the winch
- Enter the **Max Tension** at which winching will pause
- Enter the **Additional Length** which affects the initial stiffness of the cable

Note:

- The exact length of the line at any time will depend on the previous motions which have been encountered by the structure connected to this line. This in turn means that the length of the line has 'memory'. The implication of this is that in situations where initial or specific positions are used in Aqwa, the line length cannot be determined and will be assumed to be the initial length. An example of this is the hot start. A warning message to this effect will be issued in these cases.
- The resolution of switching on and off the drum winch can only be the same as the time step. This means that the winch drum can only be switched at the beginning or the end of a time step and NOT in the middle. In order to conserve energy/momentum in the equations of motion, the length of the line can only be changed in steps of (time step)*speed of the winch. The tension resolution will therefore be stiffness*time step*speed. Large stiffnesses or drum speed should therefore be specified with appropriately small timesteps.

- If the unstretched length of the cable is less than 0.1m, it will be defaulted to 0.1m.

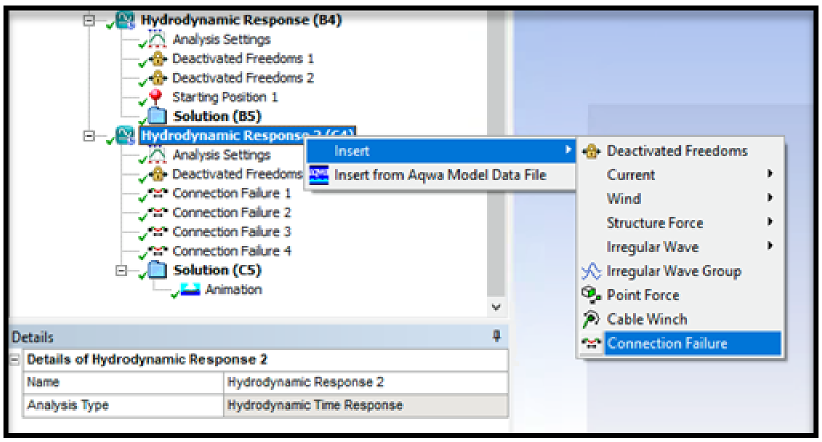
5.11.12. Connection Failure

Connections such as Cables (p. 101) and Fenders (p. 116) can be selected to fail for purposes that suit your simulation objectives.

To add a Connection Failure:

1. Right-click the **Hydrodynamic Response** object in the tree view.

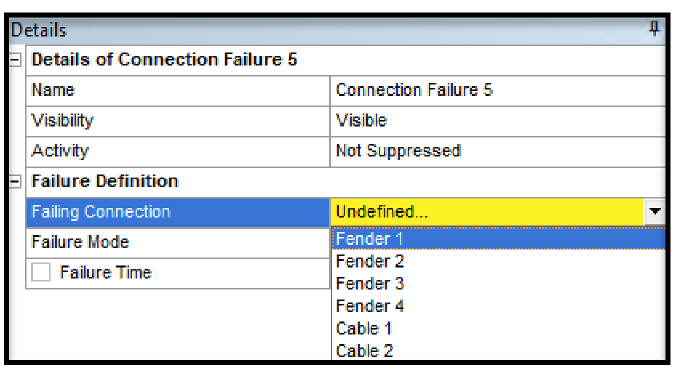
2. Select **Insert**>**Connection Failure**.



In the **Failing Connection** drop-down menu in the Details panel, select either **Cables** or **Fenders** for your analysis.

Note:

Only the **Failure Time** option is visible when **Fenders** is selected. The **Failure Mode**, **Failure Tension at Cable Start**, and **Failure Tension at Cable End** options are all hidden, as they only apply to Cables.



5.11.12.1. Cable Failure

A cable can be defined to fail when a given tension is reached and/or after a specified period of time. Note that the tension in a cable will be the same at the start or the end of the cable unless Nonlinear Catenary cables are used.

To add a Cable Failure:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object and select **Insert > Connection Failure**.
3. Select the Cable within the **Failing Connection** drop-down menu and configure the settings in the Details Panel.

The behavior of the Cable Failure depends on the **Computation Type** of the parent Hydrodynamic Response system. If you are performing a Stability Analysis (p. 134) or a Frequency Statistical Analysis (p. 140), the selected cable is considered as broken throughout the calculation.

If you are performing a Time Response Analysis (p. 135), select one of the following for **Failure Mode**, and enter the parameters for that failure mode:

- At given time – enter the analysis **Failure Time** at which the cable fails.
- After given time if tension at Cable Start is exceeded – enter the analysis **Failure Time** after which the cable fails if the cable start tension exceeds **Failure Tension at Cable Start**.
- After given time if tension at Cable End is exceeded – enter the analysis **Failure Time** after which the cable fails if the cable end tension exceeds **Failure Tension at Cable End**.
- After given time if tension at Cable Start or End is exceeded – enter the analysis **Failure Time** after which the cable fails if the cable start tension exceeds **Failure Tension at Cable Start** or the cable end tension exceeds **Failure Tension at Cable End**.
- If tension at Cable Start is exceeded – enter the **Failure Tension at Cable Start** at which the cable fails.
- If tension at Cable End is exceeded – enter the **Failure Tension at Cable End** at which the cable fails.
- If tension at Cable Start or End is exceeded – enter the **Failure Tension at Cable Start** and the **Failure Tension at Cable End** at which the cable fails.

5.11.12.2. Fender Failure

A single fender or a group of fenders can be set to fail after a specified period of time. In Aqwa, fenders are used to model contact. Fender Failure can model the kinetic contact between two bodies. This is particularly useful when performing a launch analysis.

To add a Fender Failure:

1. Right-click **Hydrodynamic Response** and select **Insert**>**Connection Failure**.
2. Select the Fender within the **Failing Connection** drop-down menu.

3. Set a time (in seconds) in **Failure Time** for the fender to fail.

   To set failure times for additional Fenders, right-click **Connection Failure 1** in the Outline tree, select **Duplicate**, and set the time.

Table 5.2: Fender Color Codes

Fender Color	Description
Orange	The fenders are stationary and not in contact.
Red	The fenders are in contact.
Purple	The fenders are deactivated.
Black	The fender compression force has exceeded the maximum permissible value based on the defined fender stiffness coefficients.

Applying Connection Failure in a Launch Analysis

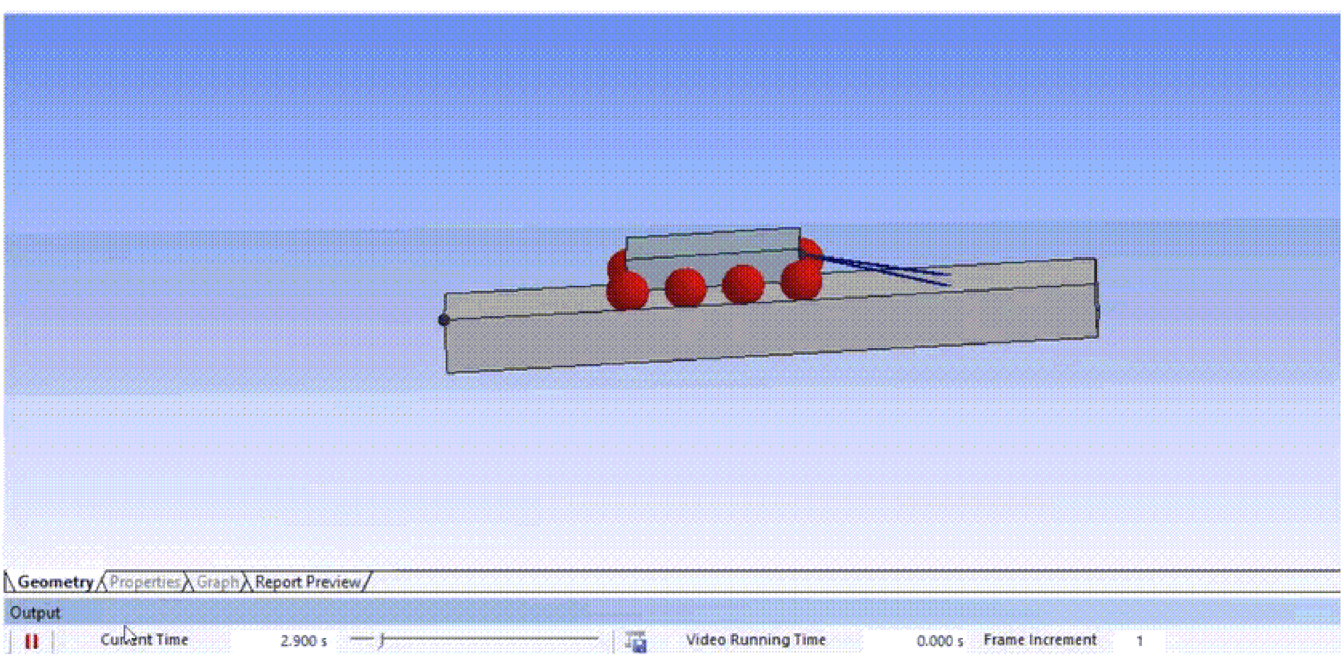
An example of the application of Connection Failure is a launch analysis. In this application, models of a barge and load block are used. The load is usually placed directly on the barge.

This workflow simulates the contact between the two models, which is depicted by the fenders, as well as the moment the fenders are deactivated. Cables are also included in this demonstration. This results in the load sliding off and dropping into the water body.

To view a video tutorial for this application, see Connection Failure in Aqwa Workbench.

Note:

It is recommended to run the simulation without any Connection Failure objects first, and then play the Animation to see when each fender should be turned off.



To perform a launch analysis with fenders:

1. Create your Barge and Load models.
2. Right-click **Barge** in the Outline tree and select **Add**>**Connection Point** to add a Connection Point to a corner of the Barge, which will define the Contact Plane for the Fenders.
3. Right-click **Load** in the Outline tree and select **Add**>**Connection Point** to add Connection Points to the corners of the Load, which will define the position of the fenders
4. Under **Connections** in the Outline tree, click each Fender to set the **Fender Connection Point**, **Connection Point on Contact Plane**, **Size**, and **Coefficients** (for fender stiffness) in the Details panel.
5. Click **Contact Axes** and **Fender Axes** under **Connections** in the Details panel to orientate the Contact Plane and the Fender, respectively.
6. For the time-domain calculation, right-click **Hydrodynamic Response** and select **Insert**>**Connection Failure**.
7. Select the desired Fender within the **Failing Connection** drop-down menu, and set a time (in seconds) in **Failure Time** for the Fender to fail.

   To set failure times for additional Fenders, right-click **Connection Failure 1** in the Outline tree, select **Duplicate**, and set the time.
8. Click **Solution**>**Solve** to solve the calculation.
9. Click **Animation** and select Play in the Output panel to review the results.

5.11.13. Point Force

A force of constant magnitude may be applied to any point on a structure by the addition of a Point Force object.

To add a Point Force:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object and select **Insert > Point Force**, or click the **Point Force** icon in the **Analysis** toolbar.

Point Forces can be applied to Stability Analyses, Frequency Statistical Analyses, and Time Response Analyses. A Point Force which has been added to a Stability Analysis can be Propagated to any downstream Frequency Statistical or Time Response Analyses.

Under the Point Force **Force Definition**, select **Application Position** from the drop-down menu of structure Connection Points. The **Behavior** of the Point Force can be set to either:

- Direction is Constant: the position at which the force is applied moves with the structure, but the direction that the force acts in remains constant in the Fixed Reference Axes (FRA). This can be used to model the force of the wind acting on a vessel's superstructure, for example.

- Direction Moves with Structure: both the position and direction of the force move with the structure (the direction that the force acts in remains constant in the Local Structure Axes (LSA)). This can be used to model a vessel's thruster, for example.

Under **Application Definition**, use the **Defined By** option to specify whether the Point Force is defined by:

- Components: use the **Component X**, **Component Y**, and **Component Z** fields to set the individual force components in the Fixed Reference Axes (FRA).
- Magnitude and Direction: specify the absolute **Magnitude** of the Point Force, and select a Fixed Point or Connection Point from the **Direction Vertex** drop-down menu to define the direction in which the force will initially act.

5.11.14. Deactivated Freedoms

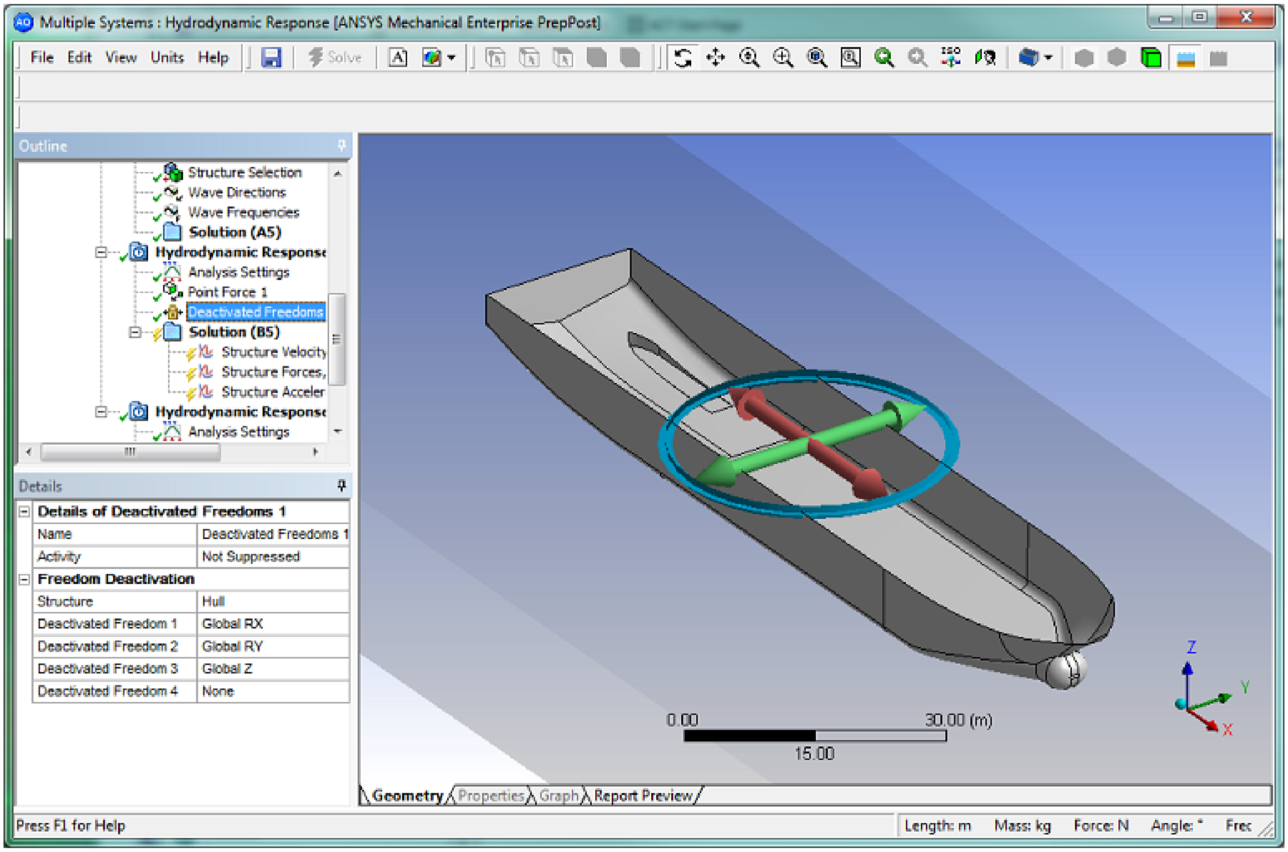
The motions of a structure can be restricted by the deactivation of one or more of its degrees of freedom.

To add a Deactivated Freedoms:

1. Select the **Hydrodynamic Response** object in the tree view.
2. Right-click the **Hydrodynamic Response** object and select **Insert > Deactivated Freedoms**, or click the **Deactivated Freedoms** icon in the **Analysis** toolbar.

Deactivated Freedoms objects can be added to Stability Analyses and Time Response Analyses. A **Deactivated Freedoms** object which has been added to a Stability Analysis can be propagated to any downstream Time Response Analyses.

Under **Freedom Deactivation**, select a **Structure** from the drop-down menu of structures included in the analysis. Then select up to six **Deactivated Freedoms**. A graphical representation of the unrestricted freedoms will be displayed at the structure's center of gravity. In the example below, the motion of the selected structure has been restricted to the horizontal (XY) plane.



Note:

If all translational degrees of freedom are to be deactivated, it is preferrable to use a Joint connection to restrict the motion of the structure.

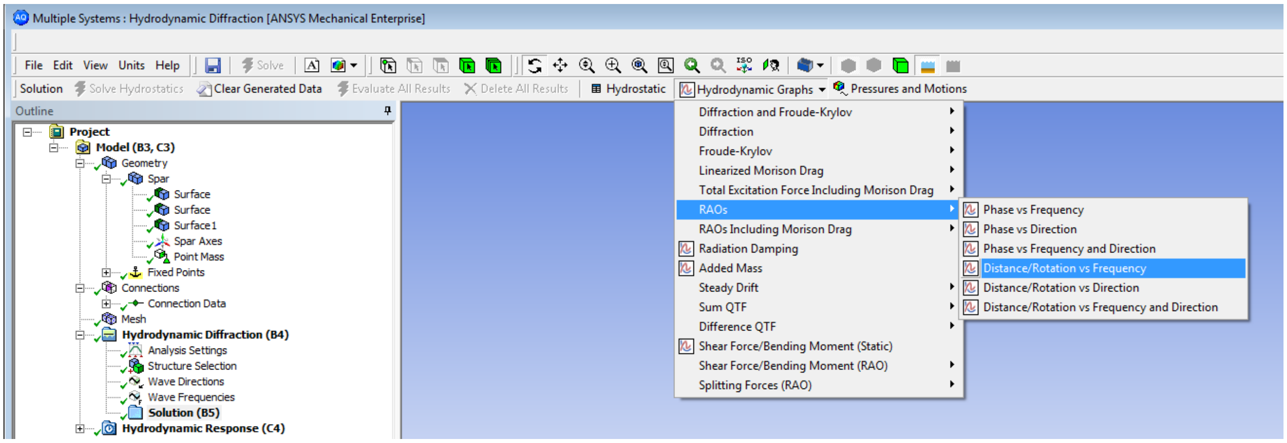
5.12. Solution

Results can be added when the **Solution** object is selected in the tree; this can be either before or after an analysis is performed.

To add a Results object:

1. Select the **Solution** object in the tree view.

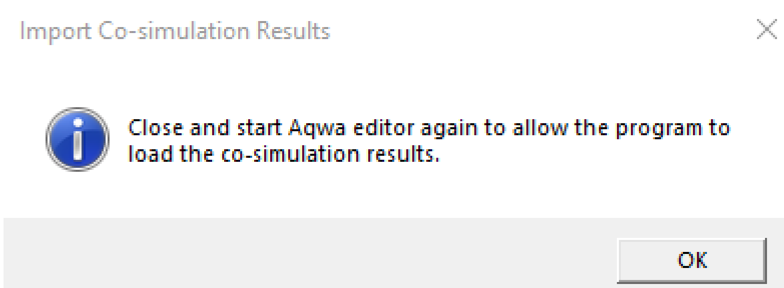
2. Right-click the **Solution** object and select **Insert Result >** ***`Result Type`*** , or click the required result type in the **Solution** toolbar. Some icons have dropdown menus to refine the selection.



When you select a Hydrodynamic Graph result object under Solution, the graph will display in the Graph tab of the upper right pane of the window, and the data for the graph will display in the pane below the graph. Various fields will appear in the Details panel, depending on the type of graph and the axes that are displayed.

If the input to an analysis is changed, then the results objects will indicate that they are out of date via the yellow lightening bolt icon, and any existing results will become unavailable. Note that the solution files stored in the project files directory will remain unless **Clear Generated Data** is selected. Changing the **Axes Selection** for any graph will cause that result object to require an update, which can be done from the right-click menu.

For certain types of Hydrodynamic Response analyses, the results can also be read from the external co-simulation result files by selecting **Import Co-simulation Results**. However, the co-simulation result files must be derived from solving the Aqwa-FMU package generated by the selected Hydrodynamic Response analysis and the model or analysis settings must not be modified since the generation of the Aqwa-FMU package. Once **Import Co-simulation Results** is selected, the program will ask for the location of the Aqwa co-simulation input `.DAT` file in the co-simulation result file folder. All the corresponding Aqwa co-simulation result files will be copied into the solver file directory of the selected Hydrodynamic Response analysis. The time settings of the Hydrodynamic Response analysis will be altered to the co-simulation time settings. After the Aqwa Workbench editor has been closed and re-opened as suggested in the message below, the co-simulation results can be evaluated in Solution.



In relation to Moonpools, you can only produce and access the Hydrodynamic Diffraction results such as the hydrostatic information, hydrodynamic graphs, and pressure display when a Geometry model contains active Moonpool objects.

The following types of results are available:

- 5.12.1. Hydrostatic Results
- 5.12.2. Hydrodynamic Graphical Results

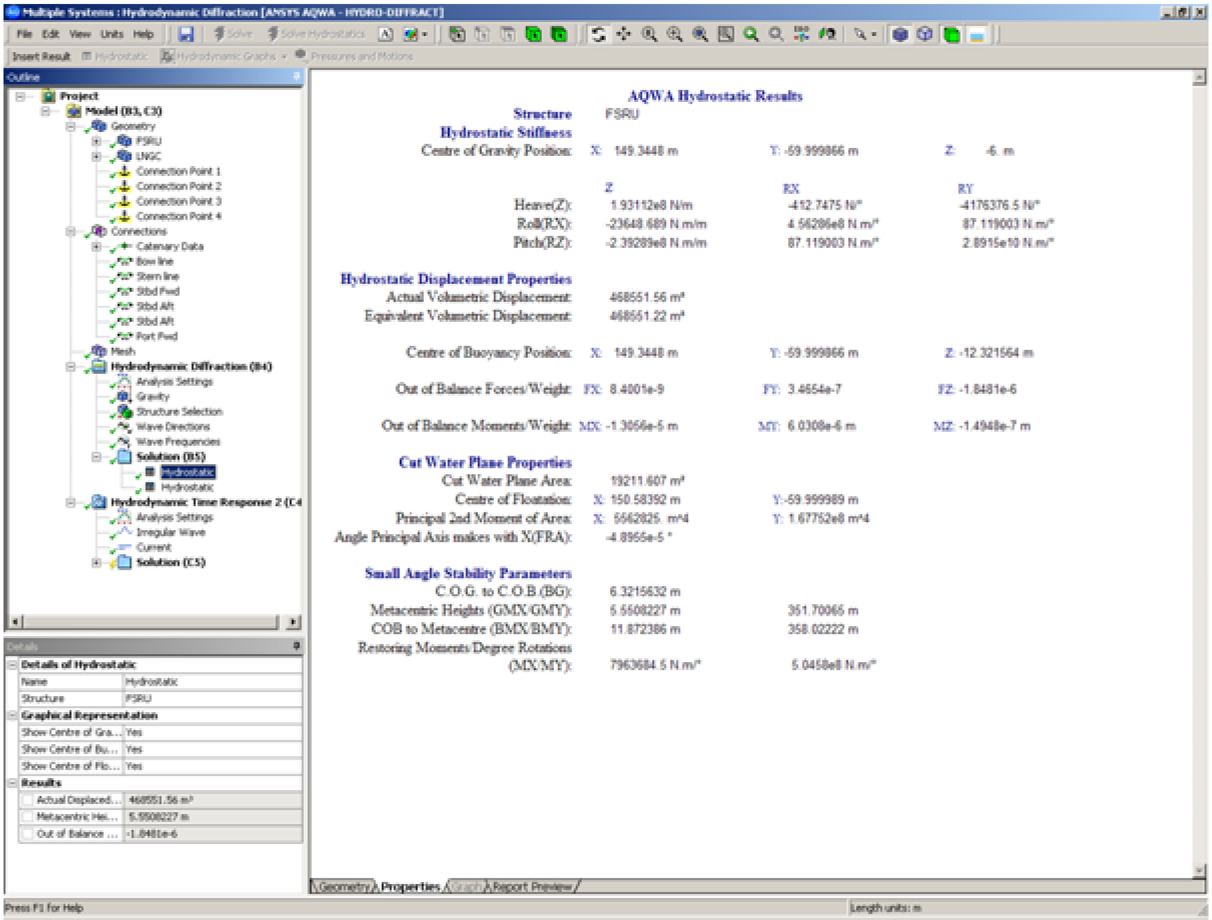
5.12.3. Frequency Domain Tabular Results
5.12.4. Pressures and Motions Results
5.12.5. Internal Tank Pressures Results
5.12.6. Animation Results
5.12.7. Dynamic Natural Modes
5.12.8. Time Domain Statistical Results
5.12.9. Time Domain Pressures Results

5.12.1. Hydrostatic Results

The Hydrostatic results tree item enables you to view the centers of buoyancy, flotation and gravity in the graphical view once a hydrostatic or hydrodynamic solve has been performed.

Detailed hydrostatic results are available by selecting the **Properties** tab at the bottom of the graphical view.

In the Details panel, select the **Structure** whose results you want to display. You can add as many Hydrostatic results objects as you like to display the results of different structures. For each result object, you can suppress or enable the display of the **Center of Gravity**, **Center of Buoyancy**, and **Center of Flotation** information from the Details panel. Any number of Hydrostatic results objects can be added to the Solution.

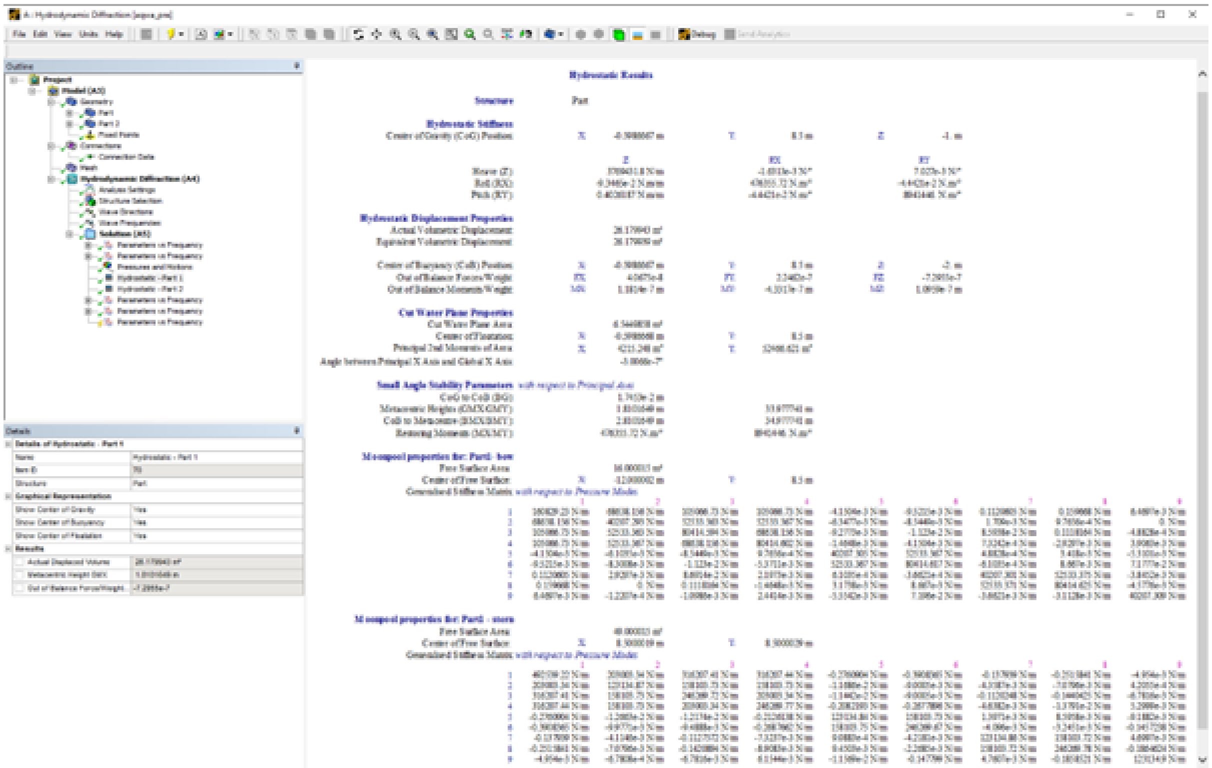


Note:

For best graphical display of hydrostatic results, it is recommended that you change the View options to enable Wireframe and turn off Select Tree Related Items.

When the selected structure includes Moonpools, the hydrostatic Moonpools properties will also be displayed. They are grouped per structures and all Moonpool properties are listed back-to-back:

- The **centre point** coordinate and the area of the free surface
- The **general stiffness matrix** for each pressure mode (see Equation 4.119)



When generating the report, the hydrostatic stiffness matrix may be wrapped if the size of the matrix exceeds the size of an A4 sheet.

5.12.2. Hydrodynamic Graphical Results

The following analysis types can output graphical results:

- 5.12.2.1. Diffraction
- 5.12.2.2. Time Response
- 5.12.2.3. Frequency Domain
- 5.12.2.4. Stability Response

Hydrodynamic graphs allow you to plot your results using either a line, surface, or contour surface plot.

There are three different types of graphs that can be displayed: 2D lines can be used to plot a parameter against one axis (for example, Structure Position vs Time), while 3D surfaces and contour surface plots can be used in a Hydrodynamic Diffraction analysis to plot a parameter against two axes (for example, RAOs vs Frequency and Direction). The chart type and axes will be based on the selection that you make from the right-click menu when you add the object to the Solution.

When you create a 2D line graph object,a child input object is also created to define the properties of that line; you can select it and edit its properties in the Details pane. You can right-click and Add Line on the parent graph object to add more lines (up to 20 per graph), or you can right-click and Duplicate on the child input object to quickly add multiple lines with similar properties. For 3D surface and contour surface plots, the result properties are defined directly in the Details pane for that graph.

For graph results under a Hydrodynamic Response analysis, the child input object includes a **Result Source** selection. Where there are multiple Hydrodynamic Response systems of the same **Computation Type** appearing in the Outline tree, you may select as a Result Source any of the systems whose Computation Type matches that of the current analysis. This allows results from similar analyses – for example, structure position time histories under different environments – to be compared on the same graph.

After inserting a new graph or changing input in an existing graph, the results need to be evaluated. The update results option is available via the context (right-click) menu. When you select an up-to-date graphical object in the tree, the graph will be displayed in a Graph tab in the main window pane, and the data for the graph will be displayed in the Tabulated Results Data panel below the graph. Where you have multiple child inputs for a 2D line graph, clicking on an input will cause the graph to display only the line corresponding to that input.

A Maximum and Minimum value for each plotted parameter will be shown in the Details panel of the graph object, and will also be highlighted in the data table (minimum is blue and maximum is red). Multiple entries are highlighted in the table if they have the same value. The positions of the Minimum and Maximum points are also given in the Details panel. For a Stability Analysis, Initial and Final parameter values are shown instead – simulation parameters should not be considered to have any physical meaning at non-converged iteration steps.

The Maximum, Minimum, Position of Maximum, Position of Minimum, Initial and Final fields can be set as output design point parameters by clicking the check box next to each field. See Parameters (p. 269) for more information. Once you have parameterized an output field, you will not be able to modify or delete the associated line input – you must de-parameterize the output first.

Note:

Output values can only be parameterized when their **Result Source** is defined as the current analysis system.

The lower right corner of the graph window contains a control that allows you to zoom and pan on the chart by selecting the corners of the control to zoom and the center to pan.

Note:

Aqwa always produces results in global coordinate systems. Traditionally, the ship is defined with its longitudinal axis in the global X direction. However, if you are aware of the results orientation, this does not have to be respected in DesignModeler.

Figure 5.12: Line Graph

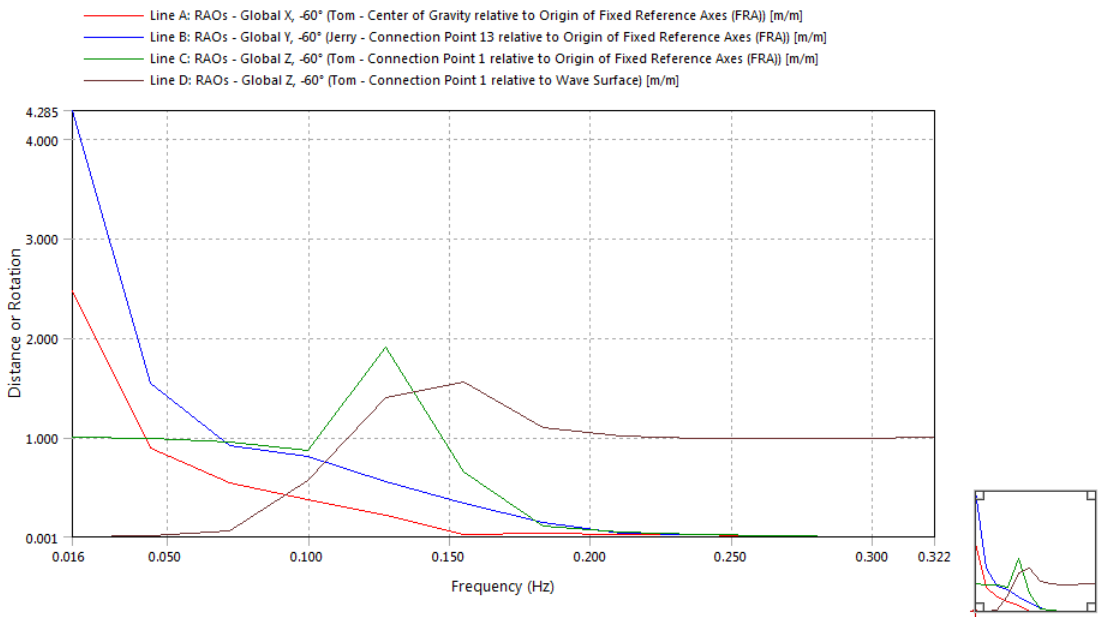


Figure 5.13: Surface Graph

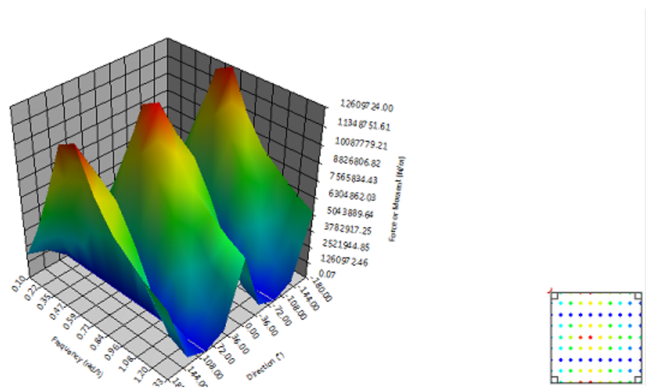
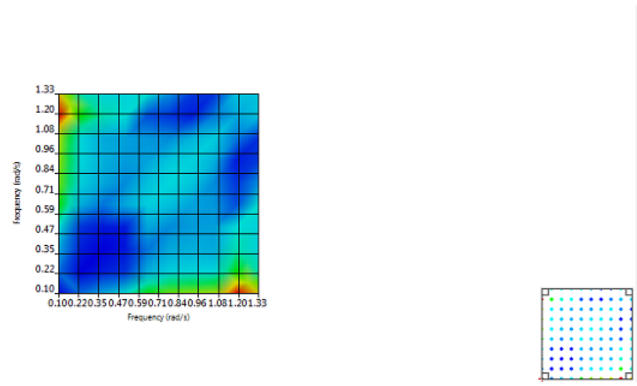


Figure 5.14: Surface Contour Graph



Note:

Hydrodynamic Diffraction is the only analysis supporting geometries containing **active Moonpool** objects. Time Response, Frequency Domain, and Stability Response **will not run** under this model configuration.

5.12.2.1. Diffraction

The following graph types for the Hydrodynamic Diffractions Solution object are available from the Hydrodynamic Graphs menu on the Insert Results toolbar.

5.12.2.1.1. Diffraction, Froude-Krylov, Diffraction + Froude-Krylov, Linearized Morison Drag, and Total Exciting Force Including Morison Drag

5.12.2.1.2. Response Amplitude Operators (RAOs) and RAOs with Linearized Morison Drag

5.12.2.1.3. Radiation Damping & Added Mass

5.12.2.1.4. Steady Drift

5.12.2.1.5. Sum QTF and Difference QTF

5.12.2.1.6. Splitting Forces (RAO)

5.12.2.1.7. Bending Moment and Shear Force

5.12.2.1.1. Diffraction, Froude-Krylov, Diffraction + Froude-Krylov, Linearized Morison Drag, and Total Exciting Force Including Morison Drag

Result Description:

2D or 3D graphs to illustrate how these forces, moments, or the corresponding phase angle change with direction, frequency or both direction and frequency. Total Exciting Force Including Morison Drag is the sum of the already existing forces (FK+Diff) and the additional Linearized Morison Drag (p. 132).

Plot availability:

- Line graph presentation: Phase Angle plotted against either Direction or Frequency
- Line graph presentation: Force/Moment plotted against either Direction or Frequency

- Surface graph presentation: Phase Angle plotted against Direction + Frequency
- Surface graph presentation: Force/Moment plotted against Direction + Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the **Component** input to select which force component to plot, either X, Y, Z for forces or RX, RY, RZ for moments.

  When the selected Part includes active Moonpools, the **Component** is enhanced with additional degrees of freedom in the form of a list of Moonpool pressure modes associated with each active Moonpool under the selected Part. See the note on pressure modes modification and impact on results' validity under Pressure Modes (p. 71).
- When the plot is performed against Direction, then the required frequency can be chosen using the **Frequency** input.
- When the plot is performed against Frequency, then the required direction can be chosen using the **Direction** input.

Line Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the first Frequency or Direction at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

Surface Graph Input:

Use the **Component** input to select which force component to plot, either X, Y, Z for forces or RX, RY, RZ for moments.

Surface Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the Frequency and Direction at which they occur (**Abscissa (X) Position of Minimum**, **Abscissa (X) Position of Maximum** and **Ordinate (Y) Position of Minimum**, **Ordinate (Y) Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.1.2. Response Amplitude Operators (RAOs) and RAOs with Linearized Morison Drag

Result Description:

2D or 3D graphs to illustrate how the amplitude and phase of the structure response change with wave direction, frequency or both direction and frequency. You can view RAOs and RAOs taking into account the linearized Morison drag effects, if Linearized Morison Drag (p. 132) is specified.

Plot availability:

- Line graph presentation: Phase Angle plotted against either Direction or Frequency

- Line graph presentation: Distance/Rotation plotted against either Direction or Frequency
- Surface graph presentation: Phase Angle plotted against Direction + Frequency
- Surface graph presentation: Distance/Rotation plotted against Direction + Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the **Component** input to select which degree of freedom to plot, either X, Y, Z for translation or RX, RY, RZ for rotation.

  When the selected Part includes active Moonpools, the **Component** is enhanced with additional degrees of freedom in the form of a list of Moonpool pressure modes associated to each active Moonpool under the selected Part. See the note on pressure modes modification and impact on results' validity under Pressure Modes (p. 71).
- When the plot is performed against Direction, then the required frequency can be chosen using the **Frequency** input.
- When the plot is performed against Frequency, then the required direction can be chosen using the **Direction** input.
- For a translation RAO, the Reference Point at which the RAO is reported can be the Center of Gravity of the selected **Structure**, or can be any Connection Point on that structure.
- A translation RAO is typically reported for **Motion Relative To** the Origin of the Fixed Reference Axes (FRA). However, when the **Component** is set to Global Z and the **Reference Point** is a Connection Point, the RAO may instead be reported relative to the undiffracted Wave Surface.

Note:

For Hydrodynamic Diffraction RAO results, the Connection Point **Include in Results** option is not required for that Connection Point to be used as a Reference Point.

Line Graph Output:

Maximum Value and **Minimum Value** of Angle or Distance/Rotation and the first Frequency or Direction at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

Surface Graph Input:

Use the **Component** input to select which degree of freedom to plot, either X, Y, Z for translation or RX, RY, RZ for rotation. For translational degrees of freedom, use the **Reference Point** and **Motion Relative To** inputs to set the position and frame of reference for the reported RAOs.

Surface Graph Output:

Maximum Value and **Minimum Value** of Angle or Distance/Rotation and the Frequency and Direction at which they occur (**Abscissa (X) Position of Minimum**, **Abscissa (X) Position of**

Maximum and **Ordinate (Y) Position of Minimum**, **Ordinate (Y) Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.1.3. Radiation Damping & Added Mass

Result Description:

2D graphs to illustrate how the Radiation Damping or Added Mass varies with frequency.

Plot Availability:

- Line Graph Presentation: Radiation Damping or Added Mass plotted against Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

Use the **Subtype** and **Component** inputs to select which value from the Radiation Damping or Added Mass matrix to plot, X, Y, Z for linear components or RX, RY, RZ for rotational components.

When the selected Part includes active Moonpools, the **Component** and the **Subtype** are appended with additional degrees of freedom in the form of a list of pressure modes associated to each active Moonpool under the selected Part. See the note on pressure modes modification and impact on results' validity under Pressure Modes (p. 71).

This image shows how the lists of **Subtype** and **Component** have been appended with the Moonpool pressure modes:

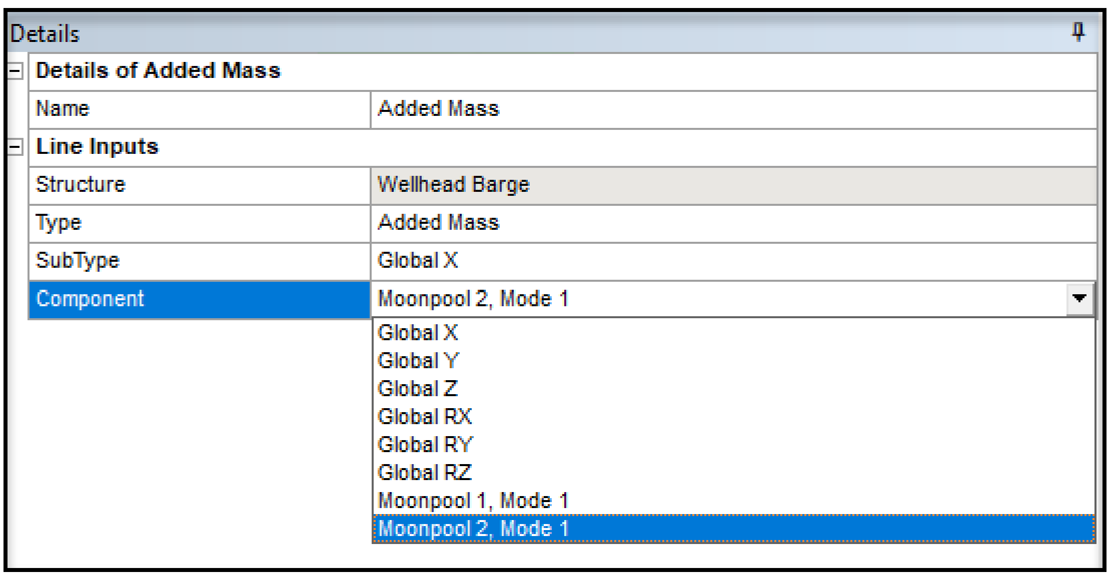


Figure 4.7: Coefficient Locations in Added Mass and Hydrodynamic Damping Matrices details which coefficient is being queried or selected in the Added Mass coefficients matrix for a given structure selection.

The same applies for the radiation damping coefficient matrix.

The combination between the 6 positional degrees of freedom (X,Y,Z, RX, RY, RZ) and the Moonpool pressure modes will result in the following unit system. The **SubType** and **Component** selection will give the following type of units:

Component SubType▼	Translation (X,Y,Z)	Rotational (RX, RY, RZ)	Moonpool Pressure Mode
Translation (X,Y,Z)	Tran-Tran	Rot-Tran	Tran-Tran
Rotational (RX, RY, RZ)	Tran-Rot	Rot-Rot	Tran-Rot
Moonpool Pressure Mode	Tran-Tran	Rot-Tran	Tran-Tran

Line Graph Output:

Maximum Value and **Minimum Value** of Radiation Damping and the first Frequency at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.1.4. Steady Drift

Result Description:

2D or 3D graphs to illustrate the mean drift forces/moments applied on the structure and how they change with direction, frequency, or both direction and frequency.

Plot availability:

- Line graph presentation: Force/Moment plotted against either Direction or Frequency
- Surface graph presentation: Force/Moment plotted against Direction + Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the **Component** input to select which force component to plot, either X, Y, or Z for forces or RX, RY, or RZ for moments.
- When the plot is performed against Direction, then the required frequency can be chosen using the **Frequency** input.
- When the plot is performed against frequency, then the required direction can be chosen using the **Direction** input.

Line Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the first Frequency or Direction at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

Surface Graph Input:

Use the **Component** input to select which force component to plot, either X, Y, or Z for forces or RX, RY, or RZ for moments.

Surface Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the Frequency and Direction at which they occur (**Abscissa (X) Position of Minimum**, **Abscissa (X) Position of Maximum** and **Ordinate (Y) Position of Minimum**, **Ordinate (Y) Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.1.5. Sum QTF and Difference QTF

Result Description:

3D graphs to illustrate how these quadratic transfer function (QTF) matrix (which is a force or moment) results change with frequency, for a given direction.

Plot availability:

- Surface graph presentation: Phase Angle plotted against Frequency + Frequency
- Surface graph presentation: Force/Moment plotted against Frequency + Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Surface Graph Input:

- Use the **Component** input to select which force component to plot, either X, Y, Z for forces or RX, RY, RZ for moments.
- The required **Direction** can be chosen.
- When Force/Moment plots are performed, the **SubType** field is shown to enable the selection of the full **Amplitude** component, or just the **Real** or **Imaginary** components.

Surface Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the Frequency and Direction at which they occur (**Abscissa (X) Position of Minimum**, **Abscissa (X) Position of Maximum** and **Ordinate (Y) Position of Minimum**, **Ordinate (Y) Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.1.6. Splitting Forces (RAO)

Result Description:

2D or 3D graphs to illustrate how the Splitting Forces (RAO) vary with frequency, direction, or both frequency and direction.

Splitting Forces (RAO) calculations depend upon the masses being accurately defined in the geometry section and are only available for structures that are not included in an Interacting Structure Group.

> **Note:**
>
> Splitting Forces (RAO) results are not available for structures containing only a single mass (from Point Mass or Line Body definitions).

Plot availability:

- Line graph presentation: Force/Moment vs. Frequency or Direction
- Line graph presentation: Phase Angle vs. Frequency or Direction
- Surface graph presentation: Force/Moment vs. Frequency + Direction
- Surface graph presentation: Phase Angle vs. Frequency + Direction

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the **Component** input to select which force component to plot, either X, Y, or Z for forces or RX, RY, or RZ for moments.
- When the plot is performed against Direction, then the required frequency can be chosen using the **Frequency** input.
- When the plot is performed against Frequency, then the required direction can be chosen using the **Direction** input.
- When Force/Moment plots are performed, the **SubType** field is shown to enable the selection of the full **Amplitude** component, or just the **Real** or **Imaginary** components.
- When Force/Moment plots are performed, use **Bounding Box Min Coordinate [X Y Z]** to enter the coordinates of one corner of a bounding box and **Bounding Box Max Coordinate [X Y Z]** to enter the coordinates of the opposite corner of the box. Then enter a **Calculation Coordinate [X Y Z]** to enter the coordinates of the point about which the moments are calculated. Values entered should be separated by spaces.

Line Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the first Frequency or Direction at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

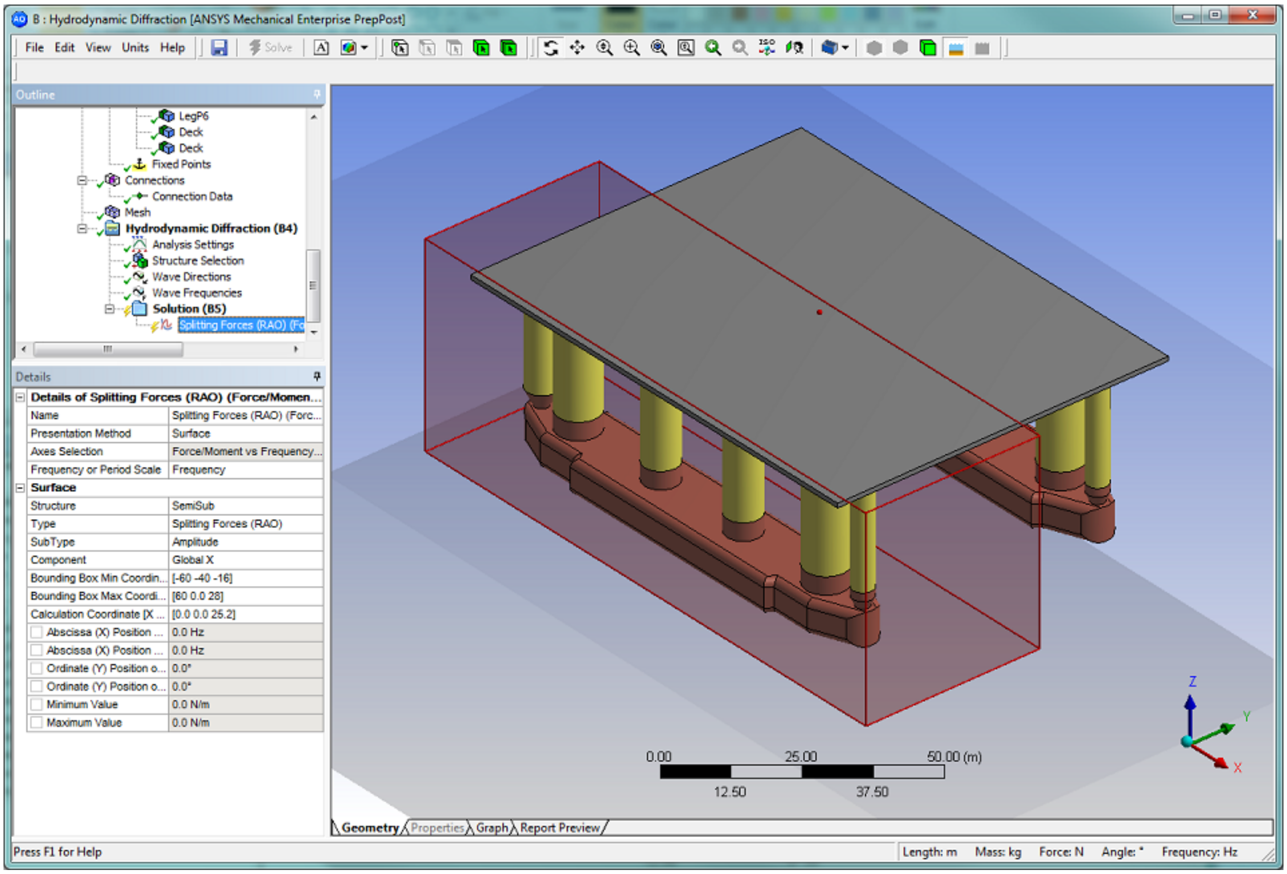
Surface Graph Input:

- Use the **Component** input to select which force component to plot, either X, Y, Z for Forces or RX, RY, RZ for moments.
- When Force/Moment plots are performed, the **SubType** field is shown to enable the selection of the full **Amplitude** component, or just the **Real** or **Imaginary** components.
- When Force/Moment plots are performed, use **Bounding Box Min Coordinate [X Y Z]** to enter the coordinates of one corner of a bounding box and **Bounding Box Max Coordinate [X Y Z]** to enter the coordinates of the opposite corner of the box. Then enter a **Calculation Coordinate [X Y Z]** to enter the coordinates of the point about which the moments are calculated. Values entered should be separated by spaces.

Surface Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the Frequency and Direction at which they occur (**Abscissa (X) Position of Minimum**, **Abscissa (X) Position of Maximum** and **Ordinate (Y) Position of Minimum**, **Ordinate (Y) Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

When defining the **Bounding Box Min**, **Max Coordinates [X Y Z]**, and **Calculation Coordinate [X Y Z]** for the Splitting Forces (RAO) calculation, while the result object is not yet evaluated, graphical representations of the bounding box and calculation position are shown in the Geometry tab.



5.12.2.1.7. Bending Moment and Shear Force

Static Bending Moments and Shear Forces can be calculated by the SF/BM (Static) result, and are always plotted along the neutral axis of the vessel. Alternatively, response amplitude operator (RAO) based results can be determined for each wave direction, frequency, and position along the neutral axis. Depending upon the combination of requirements, these will be shown on both 2D and 3D graphs appropriately. When graphs are plotted against Position, the Frequency or Direction can be explicitly selected or they can be enveloped so that the critical result can be identified easily. When graphs are not plotted against Position, then the calculation Position can be specified, and if appropriate, the Frequency or Direction can be explicitly defined (in this situation they cannot be enveloped).

Bending Moment and Shear Force calculations depend upon the masses being accurately defined in the geometry section and will only be permitted if there is only a single structure in the analysis.

Note:

If you define only a single Point Mass on a structure, and then request Bending Moment and Shear Force results, the mass is assumed to be distributed over the length of the Part in the selected Neutral Axis direction.

If you define multiple Point Masses, or a Distributed Mass, the mass definitions are split into a number of sections such that the masses are evenly distributed along the length of the Part in the selected Neutral Axis direction (while maintaining the overall CoG position of the Part). Masses from Line Bodies are distributed over the extents of each Line Body, and are then superimposed on the mass distribution.

Result Description:

2D or 3D graphs to illustrate the shear forces/bending moments applied on the structure and how they change with direction, frequency, position along the neutral axis, or combinations of direction/frequency/position.

Plot availability:

- Line graph presentation (static): Force/Moment plotted against Position
- Line graph presentation (RAO): Force/Moment plotted against either Position, Direction, or Frequency
- Line graph presentation (RAO): Phase Angle plotted against either Position, Direction, or Frequency
- Surface graph presentation (RAO): Force/Moment plotted against Direction + Frequency, Direction + Position, or Frequency + Position
- Surface graph presentation (RAO): Phase Angle plotted against Direction + Frequency, Direction + Position, or Frequency + Position

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the **Component** input to select which force component to plot, either X, Y, or Z for forces or RX, RY, or RZ for moments.
- When the plot is performed against Direction (RAO), then the required frequency can be chosen using the **Frequency** input and the **Position** can be entered.
- When the plot is performed against Frequency (RAO), then the required direction can be chosen using the **Direction** input, and the **Position** can be entered.

- When the plot is performed against Position (RAO), then the required direction can be chosen using the **Direction** input, and the required frequency can be chosen using the **Frequency** input.

Line Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the first Frequency, Direction, or Position at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

Surface Graph Input:

Use the **Component** input to select which force component to plot, either X, Y, or Z for forces or RX, RY, or RZ for moments.

Surface Graph Output:

Maximum Value and **Minimum Value** of Angle or Force/Moment and the Frequency and Direction or Frequency and Position or Position and Direction at which they occur (**Abscissa (X) Position of Minimum**, **Abscissa (X) Position of Maximum** and **Ordinate (Y) Position of Minimum**, **Ordinate (Y) Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2. Time Response

Under the Solution object of a Hydrodynamic Response system, a number of graphs will be available to track the behavior of a number of parameters over time. The graphs are all line graphs, and the following categories of graphs are available:

5.12.2.2.1. Wave Surface Elevation
5.12.2.2.2. Structure Position
5.12.2.2.3. Structure Velocity
5.12.2.2.4. Structure Acceleration
5.12.2.2.5. Structure Forces
5.12.2.2.6. Fender Forces
5.12.2.2.7. Fender Motions
5.12.2.2.8. Joint Forces
5.12.2.2.9. Cable Forces
5.12.2.2.10. Tether/Riser Motions
5.12.2.2.11. Tether/Riser Tensions
5.12.2.2.12. Tether/Riser Shear Force/Bending Moment
5.12.2.2.13. Tether/Riser Stresses
5.12.2.2.14. Time Step Error

Note:

For Time Response results, the table of data points is hidden by default. Use the **Show Tabulated Results Data** option to show the data points in tabular form.

5.12.2.2.1. Wave Surface Elevation

2D graph to illustrate the elevation of the wave surface over time at a selected position.

Plot Availability:

- Line graph presentation: Distance plotted against Time

Line Graph Input:

Use the **Reference Point** input to select the position at which the Wave Surface Elevation is reported. This may either be at the fixed position specified in the Time Response Analysis Settings (p. 135) or at the (moving) position of any Connection Point that has its **Include in Results** option to set to yes before the analysis is solved.

Line Graph Output:

Maximum Value and **Minimum Value** of Wave Surface Elevation and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by clicking the adjacent check box.

5.12.2.2.2. Structure Position

Result Description:

2D graphs to illustrate the following response subtypes of the Structure Position during the analysis:

- Actual Response
- Low Frequency – The low frequency subtype values are obtained by filtering the actual response with a filter which has a cut-off frequency of one third of the frequency of the $(10\% + 1)^{th}$ spectral line; that is, with N spectral lines, $n = 0.1N + 1$, $\omega_{cutoff} = \omega_n/3$
- Wave Frequency – the wave frequency response is that which remains when the low frequency response is subtracted from the actual response
- RAO Based – The RAO based motions are those that are calculated using only the RAOs, ignoring the effects of connections (unless these are included as additional matrices in the diffraction analysis), using the applied wave

These values are available in all translational and rotational component directions, and are calculated at the center of gravity or (translation components only) at a Connection Point on the selected structure.

Plot availability:

- Line graph presentation: Distance/Rotation plotted against Time

Line Graph Input:

Use the **Component** input to select which Position component to plot: distance from a defined reference point, X, Y, or Z for translation or RX, RY, or RZ for rotation.

For a translation, the **Reference Point** at which the result is reported can be the Center of Gravity of the selected **Structure**, or can be any Connection Point on that structure.

The **Motion Relative To** input can be used to set the frame of reference for the reported translation. This may be either the Origin of the Fixed Reference Axes (FRA), the Center of Gravity of another structure, a Connection Point on another structure, or a Fixed Point. When the **Component** is set to Global Z and the **Reference Point** is a Connection Point, the result may also be reported relative to the Wave Surface. For the latter, the selected Connection Point must have its **Include in Results** option set to Yes before the analysis is solved, and the result **SubType** must be set to Actual Response.

Line Graph Output:

Maximum Value and **Minimum Value** of Structure Position and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.3. Structure Velocity

Result Description:

2D graphs to illustrate the following response subtypes of the Structure Velocity during the analysis:

- Actual Response
- Low Frequency – The low frequency subtype values are obtained by filtering the actual response with a filter which has a cut-off frequency of one third of the frequency of the $(10\% + 1)^{th}$ spectral line; that is, with N spectral lines, $n = 0.1N + 1$, $\omega_{cutoff} = \omega_n/3$
- Wave Frequency – the wave frequency response is that which remains when the low frequency response is subtracted from the actual response
- RAO Based – The RAO based motions are those that are calculated using only the RAOs, ignoring the effects of connections (unless these are included as additional matrices in the diffraction analysis), using the applied wave

These values are available in all translational and rotational component directions, and are calculated at the center of gravity or (translation components only) at a Connection Point on the selected structure.

Plot availability:

- Line graph presentation: Velocity plotted against Time

Line Graph Input:

Use the **Component** input to select which Velocity component to plot, either X, Y, or Z for translational velocity or RX, RY, or RZ for rotational velocity.

For a translational velocity, the **Reference Point** at which the result is reported can be the Center of Gravity of the selected **Structure**, or can be any Connection Point on that structure.

The **Motion Relative To** input can be used to set the frame of reference for the reported translational velocity. This may be either the Origin of the Fixed Reference Axes (FRA), the Center of Gravity of another structure, or a Connection Point on another structure. When the **Component** is set to Global Z and the **Reference Point** is a Connection Point, the result may also be reported relative to the Wave Surface. For the latter, the selected Connection Point must have its **Include in Results** option set to Yes before the analysis is solved, and the result **SubType** must be set to Actual Response.

Line Graph Output:

Maximum Value and **Minimum Value** of Structure Velocity and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.4. Structure Acceleration

Result Description:

2D graphs to illustrate the following response subtypes of the Structure Acceleration during the analysis:

- Actual Response
- Low Frequency – The low frequency subtype values are obtained by filtering the actual response with a filter which has a cut-off frequency of one third of the frequency of the $(10\% + 1)^{th}$ spectral line; that is, with N spectral lines, $n = 0.1N + 1$, $\omega_{cutoff} = \omega_n/3$
- Wave Frequency – the wave frequency response is that which remains when the low frequency response is subtracted from the actual response
- RAO Based – The RAO based motions are those that are calculated using only the RAOs, ignoring the effects of connections (unless these are included as additional matrices in the diffraction analysis), using the applied wave

These values are available in all translational and rotational component directions, and are calculated at the center of gravity or (translation components only) at a Connection Point on the selected structure.

Plot availability:

- Line graph presentation: Acceleration plotted against Time

Line Graph Input:

Use the **Component** input to select which Acceleration component to plot, either X, Y, or Z for translational acceleration or RX, RY, or RZ for rotational acceleration.

For a translational acceleration, the **Reference Point** at which the result is reported can be the Center of Gravity of the selected **Structure**, or can be any Connection Point on that structure.

The **Motion Relative To** input can be used to set the frame of reference for the reported translational acceleration. This may be either the Origin of the Fixed Reference Axes (FRA), the Center of Gravity of another structure, or a Connection Point on another structure. When the **Component** is set to Global Z and the **Reference Point** is a Connection Point, the result may also be reported relative to the Wave Surface. For the latter, the selected Connection Point must have its **Include in Results** option set to Yes before the analysis is solved, and the result **SubType** must be set to Actual Response.

Line Graph Output:

Maximum Value and **Minimum Value** of Structure Acceleration and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.5. Structure Forces

Result Description:

2D graphs to illustrate the following response subtypes of the Structure Forces during the analysis:

- Total Force/Moment
- Mooring Sum Only
- Gyroscopic Only
- Diffraction Only
- Linear Damping Only
- Morison Drag Only
- Drift Only
- Froude-Krylov Only
- Gravitation Only
- Current Drag Only
- Wave Inertia Only
- Hydrostatic Only
- Wind Only
- Slam Only
- Point Only
- Yaw Drag Only
- Slender Body Only
- Radiation Only
- Fluid Momentum Only

- Fluid Gyroscopic Only
- Externally Applied Only
- Linear Wave Drift Damping Only
- Maneuvering Force Only

The sum of forces for each individual subtype will be shown in the graph. If all of the forces are taken in combination, they will form the result available by the Total Force/Moment subtype. In the case where the force is not included in the analysis the subtype will not be available for selection.

> **Note:**
>
> When including drift effects, the Froude-Krylov Only excludes those calculated on diffracting panels; instead, these are included under the Diffraction Forces.

These values are calculated at the center of gravity and are available in all translational and rotational component directions.

Plot availability:

- Line graph presentation: Force/Moment plotted against Time

Line Graph Input:

Use the **Component** input to select which Structure Force/Moment component to plot, either X, Y, or Z for force components or RX, RY, or RZ for rotational components.

Line Graph Output:

Maximum Value and **Minimum Value** of the Structure Force and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.6. Fender Forces

Result Description:

2D graphs to illustrate the fender forces during the analysis. You must select the **Structure** and **Connection** to that structure (connected Fender) for which you want to display the results.

Plot availability:

- Line graph presentation: Force plotted against Time

Line Graph Input:

Use the **Component** input to select which Fender Force component to plot; either X, Y, or Z force components, or one of the following: Total Force, Elastic Force, Damping Force, or Friction Force.

Line Graph Output:

Maximum Value and **Minimum Value** of the Fender Force and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.7. Fender Motions

Result Description:

2D graphs to illustrate the fender motions during the analysis. You must select the **Structure** and **Connection** to that structure (connected Fender) for which you want to display the results.

Plot Availability:

- Line graph presentation: Distance plotted against Time

Line Graph Input:

Use the **Component** input to select which Fender Motion to plot; either Compression, Horizontal Movement, or Upward (Z) Movement.

Line Graph Output:

Maximum Value and **Minimum Value** of the Fender Motion and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.8. Joint Forces

Result Description:

2D graphs to illustrate the joint forces during the analysis. You must select the **Structure** and **Connection** to that structure (connected Joint) for which you want to display the results.

Plot availability:

- Line graph presentation: Force/Moment plotted against Time

Line Graph Input:

Use the **Component** input to select which Joint Force component to plot; either X, Y, or Z force components, or RX, RY, or RZ rotational components.

Line Graph Output:

Maximum Value and **Minimum Value** of the Joint Force and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.9. Cable Forces

The force components are available for each cable. For multi-segment catenary cables, section tension is also available.

Result Description:

2D graphs to illustrate cable tension during the analysis.

- Whole Cable Forces
- Cable Section Tension

Plot availability:

- Line graph presentation: Force or Tension plotted against Time

Line Graph Input:

- For Whole Cable Forces, use the **Connection** field to select which cable's characteristics to plot. Use the **Component** input to select which Cable Force component to plot; either X, Y, or Z force components, or Anchor Uplift, Tension, or Laid Length.
- For Cable Section Tension, use the **Connection** field to select which catenary cable's characteristics to plot. Use the **Cable Section** field to select the Catenary Section for which you want to plot section tension.

Line Graph Output:

Maximum Value and **Minimum Value** of Cable Force/Tension and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.10. Tether/Riser Motions

Result Description:

2D graphs to illustrate the variation in the following results of a tether/riser node during the analysis:

- Displacement
- Velocity
- Acceleration

You must select the **Structure** and **Connection** to that structure (connected tether/riser) for which you want to display the results.

Tether/Riser motion results are output in the Tether/Riser Local Axes (TLA), which are described in Tethers/Risers (p. 107). As the TLA points from the Start Fixed Point to the End Connection Point of the Tether/Riser, where the End Connection Point will be moving with its associated Structure, the motions of the first and last nodes will always be zero.

Plot Availability:

- Line graph presentation: Result plotted against Time

Line Graph Input:

Use the **Tether/Riser Node** input to select the node for which results will be displayed. Use the **Component** input to select which tether/riser nodal result component to plot: either X or Y translational components, or RX or RY rotational components.

Line Graph Output:

Maximum Value and **Minimum Value** of the tether/riser nodal result and the times at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.11. Tether/Riser Tensions

Result Description:

2D graphs to illustrate the following subtypes of the Tether/Riser Tensions during the analysis:

- Connection Point Tension
- Section Tension
- Nodal Tension
- Wall Tension

You must select the **Structure** and **Connection** to that structure (connected tether/riser) for which you want to display the results.

Plot availability:

- Line graph presentation: Force plotted against Time

Line Graph Input:

The input required depends on the selected graph subtype.

For Connection Point Tension, no other input in required; the graph will display the Tether/Riser tensions at the structure Connection Point.

For Section Tension, use the **Tether Section** input to select the section for which results will be displayed.

For Nodal Tension, use the **Tether/Riser Node** input to select the node for which results will be displayed. The definition elevation of the selected node is shown for reference.

For Wall Tension, use the **Tether/Riser Element** input to select an element along the tether, and the **Tether/Riser Element End** input to select the end of that element for which results will be displayed. The definition elevation of the selected element end is shown for reference.

Line Graph Output:

Maximum Value and **Minimum Value** of tether/riser tensions and the times at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.12. Tether/Riser Shear Force/Bending Moment

Result Description:

2D graphs to illustrate the following subtypes of the Tether/Riser Shear Force/Bending Moment during the analysis:

- Element Shear Force/Bending Moment
- Connection Point Shear Force/Bending Moment

You must select the **Structure** and **Connection** to that structure (connected tether/riser) for which you want to display the results.

Plot availability:

- Line graph presentation: Force/Moment plotted against Time

Line Graph Input:

If the selected graph subtype is Element Shear Force/Bending Moment, use the **Tether/Riser Element** input to select an element along the tether, and the **Tether/Riser Element End** input to select the end of that element for which results will be displayed. The definition elevation of the selected element end is shown for reference.

Regardless of the selected graph subtype, you should use the **Component** input to select which Shear Force/Bending Moment component to plot: Local Tether/Riser X or Y for shear force components, or Local Tether/Riser RX or RY for bending moment components.

Line Graph Output:

Maximum Value and **Minimum Value** of tether/riser shear forces/bending moments and the times at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.13. Tether/Riser Stresses

Result Description:

2D graphs to illustrate the following subtypes of the Tether/Riser Stresses during the analysis:

- Shear Stress
- Maximum Bending + Axial Stress
- Minimum Bending + Axial Stress
- Von Mises Stress
- Y Bending Stress

You must select the **Structure** and **Connection** to that structure (connected tether/riser) for which you want to display the results.

Plot availability:

- Line graph presentation: Stress plotted against Time

Line Graph Input:

Use the **Tether/Riser Element** input to select an element along the tether, and the **Tether/Riser Element End** input to select the end of that element for which results will be displayed. The definition elevation of the selected element end is shown for reference.

Line Graph Output:

Maximum Value and **Minimum Value** of tether/riser stresses and the times at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.2.14. Time Step Error

The movement error is graphed for each time step. The program outputs the movement error at each time step in each degree of freedom which is related to the chosen time step. These errors can always be reduced by shortening the time step.

Result Description:

2D graphs to illustrate the movement error per step during the analysis.

Plot availability:

- Line graph presentation: Movement Error plotted against Time

Line Graph Input:

None.

Line Graph Output:

Maximum Value and **Minimum Value** of Movement Error and the Time at which they occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.3. Frequency Domain

A number of line graphs may be added under the Solution object of a Frequency Domain Analysis system to illustrate the variation of different parameters with frequency. The following topics are available:

5.12.2.3.1. Wave Spectra
5.12.2.3.2. Structure Response
5.12.2.3.3. Connection Response

5.12.2.3.1. Wave Spectra

Result Description:

2-D graphs to illustrate an input wave energy spectrum, over the range of frequencies assumed or defined in an active Irregular Wave object, from the present analysis. Note that a Wave Spectrum may be plotted on the same axes as a Structure Position Response Spectrum by changing the plot **Type** of an additional Line to Position Response Spectra.

Plot Availability:

- Line graph presentation: Spectral Density plotted against Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the Wave Spectrum Input to select an Irregular Wave spectrum to plot.
- If the selected Irregular Wave contains more than one sub-spectrum and/or Cross Swell Spectrum, the **Spectrum Sub-Direction** input should be used to select a specific sub-spectrum for plotting.

Line Graph Output:

Maximum Value and **Minimum Value** of Spectral Density, and the Frequencies at which these occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.3.2. Structure Response

Result Description:

2-D graphs to illustrate various structural force and motion responses over the range of frequencies used in the present Frequency Statistical Analysis. The direction used for output of Motion RAOs and Transfer Functions is that specified in **Spectrum Sub-Direction of Output for RAOs** under **Analysis Settings**. Note that a Position Response Spectrum may be plotted on the same axes as a Wave Spectrum by changing the plot **Type** of an additional Line to Wave Spectra, and a Force/Moment Response Spectrum may be plotted on the same axes as a Joint Reaction Spectrum by changing the plot **Type** of an additional Line to Joint Reaction Spectra.

Plot Availability:

- Line graph presentation: Response Amplitude Operator (RAO) plotted against Frequency
- Line graph presentation: Force/Moment Response Spectrum plotted against Frequency
- Line graph presentation: Position Response Spectrum plotted against Frequency
- Line graph presentation: Velocity Response Spectrum plotted against Frequency
- Line graph presentation: Acceleration Response Spectrum plotted against Frequency
- Line graph presentation: Transfer Function plotted against Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the **Structure** input to select a structure from the present analysis.

- Use the **Component** input to select which component to plot: either X, Y, Z for forces/translations or RX, RY, RZ for moments/rotations.
- For translational Motion RAOs and translational Position/Velocity/Acceleration Response Spectra, the **Reference Point** at which the result is reported can be the Center of Gravity of the selected Structure, or can be any Connection Point on that structure. For the latter, the selected Connection Point must have it **Include in Results** option set to Yes before the analysis is solved.
- Translational Motion RAOs and translational Position/Velocity/Acceleration Response Spectra are typically reported for **Motion Relative To** the Origin of the Fixed Reference Axes (FRA). However, when the **Component** is set to Global Z and the **Reference Point** is a Connection Point, the translational RAO/Response Spectrum may instead be be reported relative to the Wave Surface. Alternatively, if the **Reference Point** is a Connection Point and the **Axis System for Significant Motions and Nodal Response** option under **Analysis Settings** is set to Local Structure Axes, **Motion Relative To** can be set to the Origin of the Local Structure Axes (LSA).

Line Graph Output:

Maximum Value and **Minimum Value** of the selected Structure Response variable, and the Frequencies at which these occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selected the adjacent check box.

5.12.2.3.3. Connection Response

Result Description:

2-D graphs to illustrate forces and reactions in cables, fenders, and joints over the range of frequencies used in the present Frequency Statistical Analysis. The direction used for output of Cable Tension RAOs and Fender Force RAOs is that specified in **Spectrum Sub-Direction of Output for RAOs** under **Analysis Settings**.

Plot Availability:

- Line graph presentation: Cable Tension Response Amplitude Operator (RAO) plotted against Frequency
- Line graph presentation: Fender Force Response Amplitude Operator (RAO) plotted against Frequency
- Line graph presentation: Joint Reaction Spectrum plotted against Frequency

The **Frequency Scale** can be modified to be **Period Scale** if required.

Line Graph Input:

- Use the **Structure** input to select a structure from the present analysis.
- Use the **Connection** input to select a valid connection for that structure.
- For Joint Reaction Spectra, use the **Component** input to select which component to plot: either X, Y, Z for forces, or RX, RY, RZ for moments.

Line Graph Output:

Maximum Value and **Minimum Value** of the selected Connection Response variable, and the Frequencies at which these occur (**Abscissa Position of Minimum**, **Abscissa Position of Maximum**). These values can be parameterized by selecting the adjacent check box.

5.12.2.4. Stability Response

Under the Solution object of a Stability Response analysis, a number of graphs will be available to track the behavior of a number of parameters as they vary with each iteration step. The graphs are all line graphs, and the following categories are available:

5.12.2.4.1. Structure Position
5.12.2.4.2. Structure Forces
5.12.2.4.3. Fender Forces
5.12.2.4.4. Fender Motions
5.12.2.4.5. Joint Forces
5.12.2.4.6. Cable Forces
5.12.2.4.7. Tether/Riser Tensions
5.12.2.4.8. Tether/Riser Shear Force/Bending Moment
5.12.2.4.9. Iteration Step Error

5.12.2.4.1. Structure Position

Result Description:

2D graphs to illustrate the Structure Position during the analysis.

These values are available in all translational and rotational component directions, and are calculated at the center of gravity or (translation components only) at a Connection Point on the selected structure.

Plot availability:

- Line graph presentation: Distance/Rotation plotted against Iteration Step

Line Graph Input:

Use the **Component** input to select which Position component to plot, either X, Y, or Z for translation or RX, RY, or RZ for rotation.

For a translation, the **Reference Point** at which the result is reported can be the Center of Gravity of the selected **Structure**, or can be any Connection Point on that structure.

The **Motion Relative To** input can be used to set the frame of reference for the reported translation. This may be either the Origin of the Fixed Reference Axes (FRA), the Center of Gravity of another structure, a Connection Point on another structure, or a Fixed Point.

Note:

For Hydrodynamic Equilibrium Results, the Connection Point **Include in Results** option is not required for that Connection Point to be used as a **Reference Point**.

Line Graph Output:

Initial Value and **Final Value** of Structure Position. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.2. Structure Forces

Result Description:

2D graphs to illustrate the following response subtypes of the Structure Forces during the analysis:

- All
- Mooring Sum Only
- Articulation Sum Only
- Morison Drag Only
- Drift Only
- Gravitational Only
- Current Drag Only
- Hydrostatic Only
- Wind Only
- Point Only
- Slender Body Only
- Externally Applied Only

The sum of forces for each individual subtype will be shown in the graph. If all of the forces are taken in combination, they will form the result available by the All subtype. In the case where the force is not included in the analysis (for example, Drift Forces, when drift is excluded) the subtype will produce zero values.

Note:

When including drift effects, the Froude-Krylov Only excludes those calculated on diffracting panels; instead, these are included under the Diffraction Forces.

These values are calculated at the center of gravity and are available in all translational and rotational component directions.

Plot availability:

- Line graph presentation: Force/Moment plotted against Iteration Step

Line Graph Input:

Use the **Component** input to select which Structure Force/Moment component to plot, either X, Y, or Z for force components or RX, RY, or RZ for rotational components.

Line Graph Output:

Initial Value and **Final Value** of the Structure Force. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.3. Fender Forces

Result Description:

2D graphs to illustrate the fender forces during the analysis. You must select the **Structure** and **Connection** to that structure (connected Fender) for which you want to display the results.

Plot availability:

- Line graph presentation: Force plotted against Iteration Step

Line Graph Input:

Use the **Component** input to select which Fender Force component to plot; either X, Y, or Z force components, or one of the following: Total Force, Elastic Force, Damping Force, or Friction Force.

Line Graph Output:

Initial Value and **Final Value** of the Fender Force. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.4. Fender Motions

Result Description:

2D graphs to illustrate the fender motions during the analysis. You must select the **Structure** and **Connection** to that structure (connected Fender) for which you want to display the results.

Plot Availability:

- Line graph presentation: Distance plotted against Iteration Step

Line Graph Input:

Use the **Component** input to select which Fender Force component to plot; either Compression, Horizontal Movement, or Upward (Z) Movement.

Line Graph Output:

Initial Value and **Final Value** of the Fender Motion. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.5. Joint Forces

Result Description:

2D graphs to illustrate the joint forces during the analysis. You must select the **Structure** and **Connection** to that structure (connected Joint) for which you want to display the results.

Plot availability:

- Line graph presentation: Force/Moment plotted against Iteration Step

Line Graph Input:

Use the **Component** input to select which Joint Force component to plot; either X, Y, or Z force components, or RX, RY, or RZ rotational components.

Line Graph Output:

Initial Value and **Final Value** of the Joint Force. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.6. Cable Forces

The force components are available for each cable. For catenary cables sections, section tension is also available.

Result Description:

2D graphs to illustrate cable tension during the analysis.

- Whole Cable Forces
- Cable Section Tension

Plot availability:

- Line graph presentation: Force or Tension plotted against Iteration Step

Line Graph Input:

- For Whole Cable Forces, use the **Connection** field to select which cable's characteristics to plot. Use the **Component** input to select which Cable Force component to plot; either X, Y, or Z force components, or Anchor Uplift, Tension, or Laid Length.
- For Cable Section Tension, use the **Connection** field to select which catenary cable's characteristics to plot. Use the **Cable Section** field to select the Catenary Section for which you want to plot section tension.

Line Graph Output:

Initial Value and **Final Value** of Cable Force/Tension. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.7. Tether/Riser Tensions

Result Description:

2D graphs to illustrate the following subtypes of the Tether/Riser Tensions during the analysis:

- Connection Point Tension
- Section Tension

You must select the **Structure** and **Connection** to that structure (connected tether/riser) for which you want to display the results.

Plot availability:

- Line graph presentation: Force plotted against Iteration Step

Line Graph Input:

The input required depends on the selected graph subtype.

For Connection Point Tension, no other input in required; the graph will display the Tether/Riser tensions at the structure Connection Point.

For Section Tension, use the **Tether Section** input to select the section for which results will be displayed.

Line Graph Output:

Initial Value and **Final Value** of Tether/Riser tensions. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.8. Tether/Riser Shear Force/Bending Moment

Result Description:

2D graphs to illustrate the Tether/Riser Shear Force/Bending Moment at the structure Connection Point during the analysis.

You must select the **Structure** and **Connection** to that structure (connected tether/riser) for which you want to display the results.

Plot availability:

- Line graph presentation: Force/Moment plotted against Iteration Step

Line Graph Input:

Use the **Component** input to select which Connection Point Shear Force/Bending Moment component to plot: Local Tether/Riser X or Y for shear force components, or Local Tether/Riser RX or RY for bending moment components.

Line Graph Output:

Initial Value and **Final Value** of Tether/Riser shear forces/bending moments. These values can be parameterized by selecting the adjacent check box.

5.12.2.4.9. Iteration Step Error

The ratio of movement at each iterative step to the maximum allowable error is graphed for each iteration step. These errors can always be reduced by decreasing the Movement Limitation or increasing the Maximum Error in Equilibrium Position.

Result Description:

2D graphs to illustrate the relative error per step during the analysis.

Plot availability:

- Line graph presentation: Relative Error plotted against Iteration Step

Line Graph Input:

None.

Line Graph Output:

Initial Value and **Final Value** of Relative Error. These values can be parameterized by selecting the adjacent check box.

5.12.3. Frequency Domain Tabular Results

Frequency Domain Statistics allows you to view text statistics in the graphical view for a Frequency Statistical Analysis (p. 140).

Statistics are available by clicking the **Properties** tab at the bottom of the graphical view.

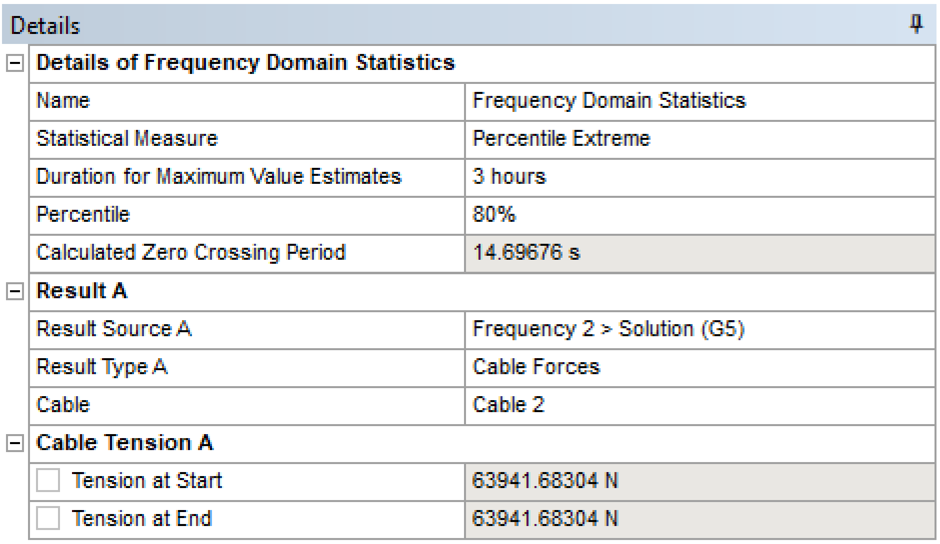
In the Details panel, select the **Statistical Measure** that you would like to display. By default, each statistic shown will be a Significant Value (Amplitude); alternatively, each value can be scaled as a Probable Maximum, a Median Maximum, an Expected Extreme, or a Percentile Extreme. The scaling requires a **Duration for Maximum Value Estimates** to be defined, and when you choose to display each statistic as a Percentile Extreme, the **Percentile** must also be specified. The **Calculated Zero Crossing Period** is automatically determined from the Irregular Wave spectra defined for the analysis. See Significant Value Calculation in the *Aqwa Theory Manual* for more information.

Note:

- Statistical values are calculated under the assumption that the corresponding response spectra are Rayleigh-distributed, and that the zero crossing period of the response is similar to the zero crossing period of the defined Irregular Wave spectrum.
- The zero crossing period calculated by Aqwa may not match the zero crossing period defined within an Irregular Wave, because the wave spectrum may not exactly fit a Rayleigh distribution.

Next, select a **Result Type** in one of the Result A-E fields. You can select from **Motions**, **Joint Reactions**, **Cable Forces**, or **Fender Forces**.

Each of the Result A-E sets includes a **Result Source** selection. Where there are multiple Frequency Statistical Analysis systems in the Outline tree, you may select any as a Result Source for the current Result Set. This allows results from similar Frequency Statistical Analyses – for example, structure significant motions under different environments – to be compared in the same table of statistics.



When a result type is selected, the Result object will expand with further information in the Details panel.

For a **Motions** result, select an active **Structure** from the current model and a **Reference Point** on the structure where the results should be reported.

If the Reference Point is a Connection Point, you may use the **Motion Relative To** field to specify whether the motions are relative to the Wave Surface. Otherwise, the motions will be reported in the axis system defined by the **Axis System for Motion Statistics and Nodal Response** option in Analysis Settings. In addition to the conventional motion properties, the statistical values of the translational effective acceleration are also reported (see 27.4. The ALLM Data Record - All Motions). If the data is not available in the Aqwa database (due to the resumption of a previous Aqwa Workbench project or removing the ACEF option), **N/A** will be displayed both in the table in the graphical window and the Property View of the current solution in the Aqwa Editor. In the former case, users need to solve the solution again to access these values.

For **Joint Reactions**, **Cable Forces**, and **Fender Forces**, you should specify a **Joint**, **Cable**, or **Fender** from the current model, respectively.

Results also appear in the Properties view, and are included in the text summary of the report.

Note:

You can include any of the results in your design points by clicking the check box next to the result in the Details panel. However, output values can only be parameterized when their **Result Source** is defined as the current analysis system.

Caution:

When the selected **Result Type** is Motions, and the selected **Reference Point** is a Connection Point, the **Include in Results** option for that Connection Point must be set to Yes before the analysis is performed. Connection Points for which the **Include in Results** option is set to No are not available for use as a result **Reference Point** in a Frequency Domain analysis.

5.12.4. Pressures and Motions Results

The Pressures and Motions results object enables the visualization and display of a number of results generated from Aqwa once a hydrodynamic solve has been performed. Any number of Pressures and Motions results may be added.

Note:

If the associated Analysis Settings (p. 130) option **Generate Wave Grid Pressures** is set to No, the wave surface, and any results related to it, are not available in the Pressures and Motions result.

You must define the structures (Parts (p. 54)) that are shown in the result. The **Structure Selection** field provides a dropdown list that allows you to select any of the Interacting Structure Groups (p. 143), or structures that do not belong to an interacting group, in the Hydrodynamic Diffraction analysis. An individual structure must have at least one diffracting panel element to be a valid selection. Pressures and Motions results are not relevant for structures consisting only of Morison elements.

The **Frequency** (p. 145) and **Direction** (p. 144) options allow you to select a Frequency and Direction from those defined in the Hydrodynamic Diffraction Wave Frequencies and Wave Directions objects. If you have only a single Frequency or Direction defined, the corresponding field becomes read-only, and is automatically updated if the single Frequency or Direction is modified.

The **Incident Wave Amplitude** can be modified to provide results that are factored from the unit amplitude wave that is the default; extreme modification may extend results beyond the capabilities of a linear analysis, in which case inaccurate results can be produced.

The **Result Type** can be defined as:

- Phase Angle, where an equivalent phase position of the incident wave component can be selected (or is shown in the graph as the time as a proportion of wave period if a sequence is chosen)

- Amplitude

- Maximum value of the selected result
- Minimum value of the selected result

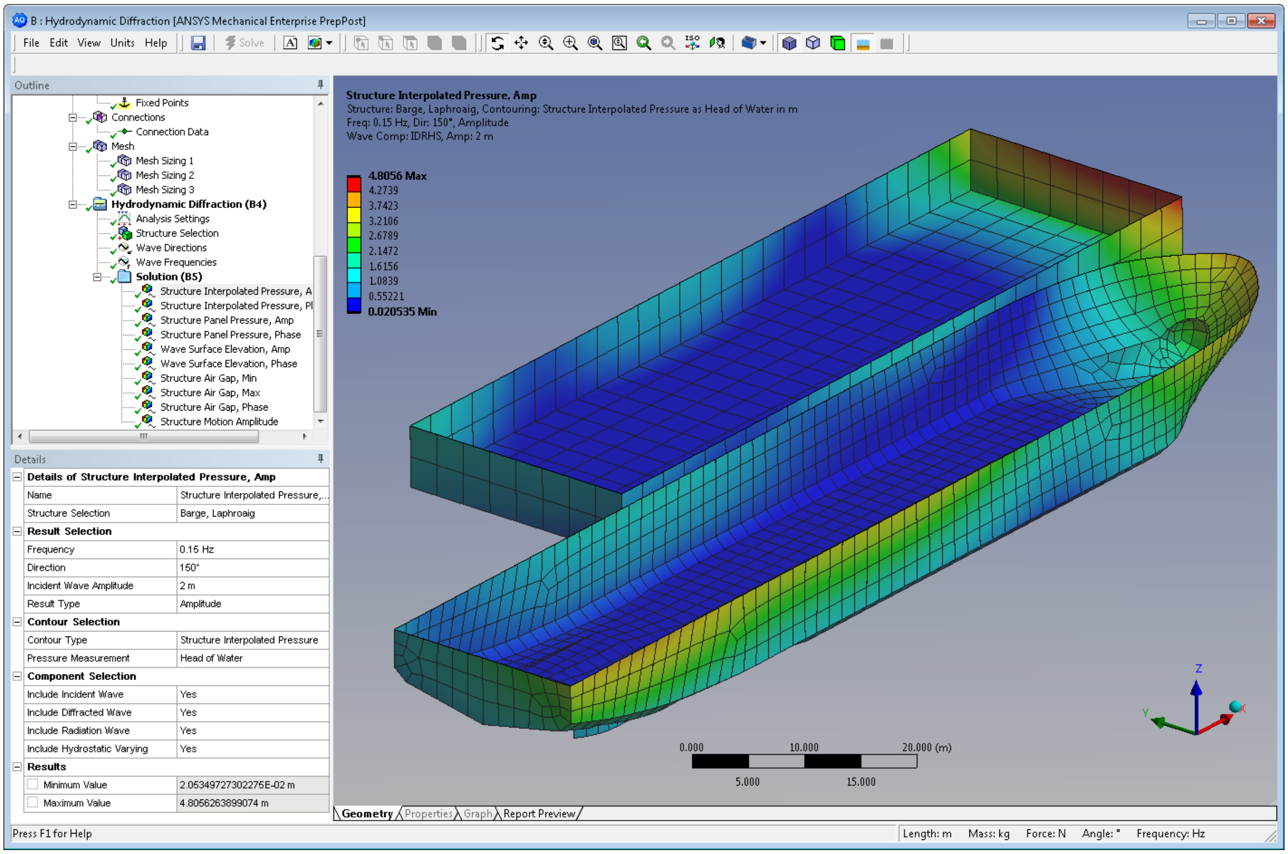
When Phase Angle is selected, **Wave Position (Phase)** can be set to 0° (t/T = 0) or 90° (t/T = 0.25), or you can enter a specified phase position (in rotation units), or select a sequence of results for animation. If a sequence of results is selected, then the **Number of Steps** can be selected.

If you select a sequence of results for animation, after you evaluate the Pressures and Motions results (right-click the object and select Evaluate All Results), controls appear that allow you to start, pause, and stop the animation. If a particular result is required from the animated set, click the graph to display it. Other controls on the graph allow you to change the time over which the animation occurs and create an `.avi` animation file.

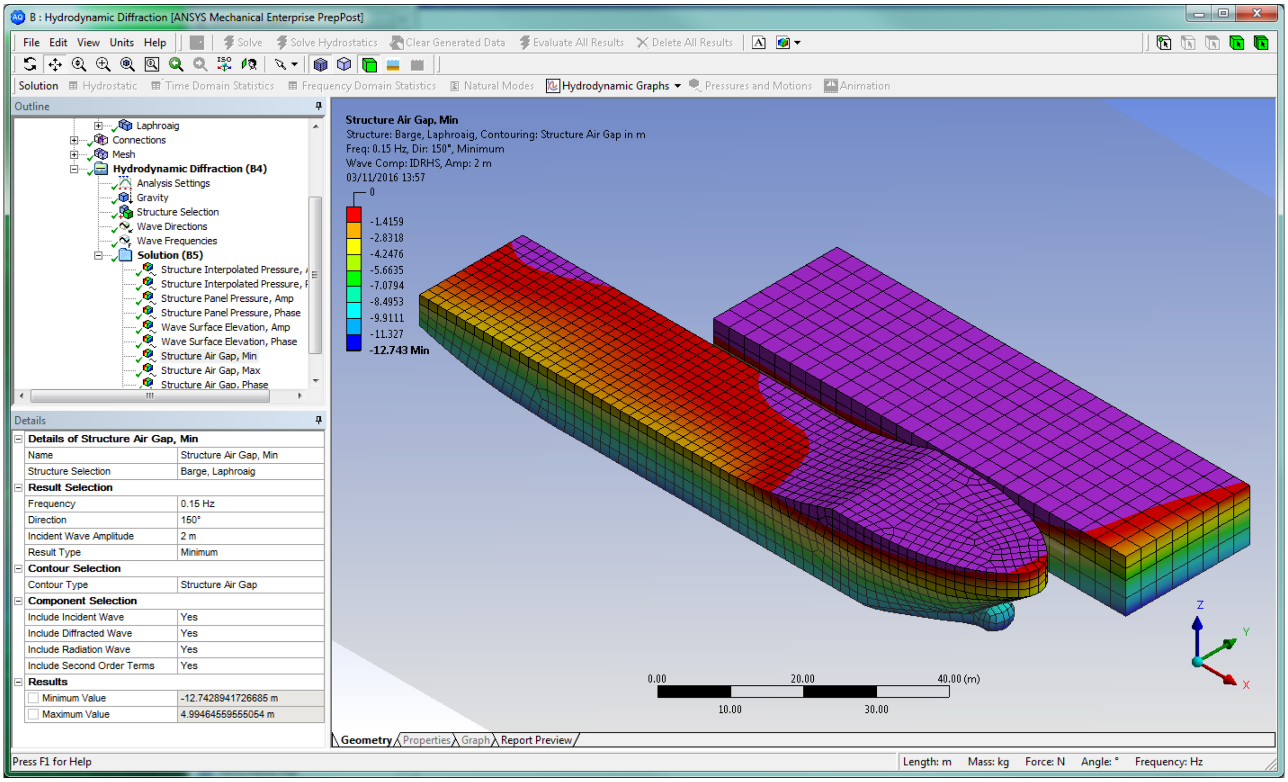
Under the Contour Selection section of the Details panel, the **Contour Type** field allows you to select which results are plotted over the structure or water surface mesh. This field can be defined as:

- Structure Interpolated Pressure
- Structure Panel Pressure
- Wave Surface Elevation (requires **Generate Wave Grid Pressures** to be set to Yes)
- Structure Air Gap (requires **Generate Wave Grid Pressures** to be set to Yes)
- Structure Motion Amplitude

Structure Interpolated Pressure displays element pressures which are interpolated from the nodal pressures calculated by Aqwa. This produces a smoothly-varying contour plot. Conversely, Structure Panel Pressure shows the pressures at the element centroids, and can be directly compared with results in Mechanical once a load transfer procedure has been completed (see The Hydrodynamic Pressure Add-on (p. 273) for more information). When plotting Structure Interpolated Pressure or Structure Panel Pressure, you can use the **Pressure Measurement** option to change the reported results between Head of Water and Force/Area.



Structure Air Gap plots the vertical distance between the sea surface and any point on the structure, with negative values at a given position indicating that this position is immersed. The contour legend can be adjusted to display a solid color above or below a given value, as shown in the following figure. Air Gaps can only be displayed for a **Result Type** of Minimum, Maximum, or Phase Angle.

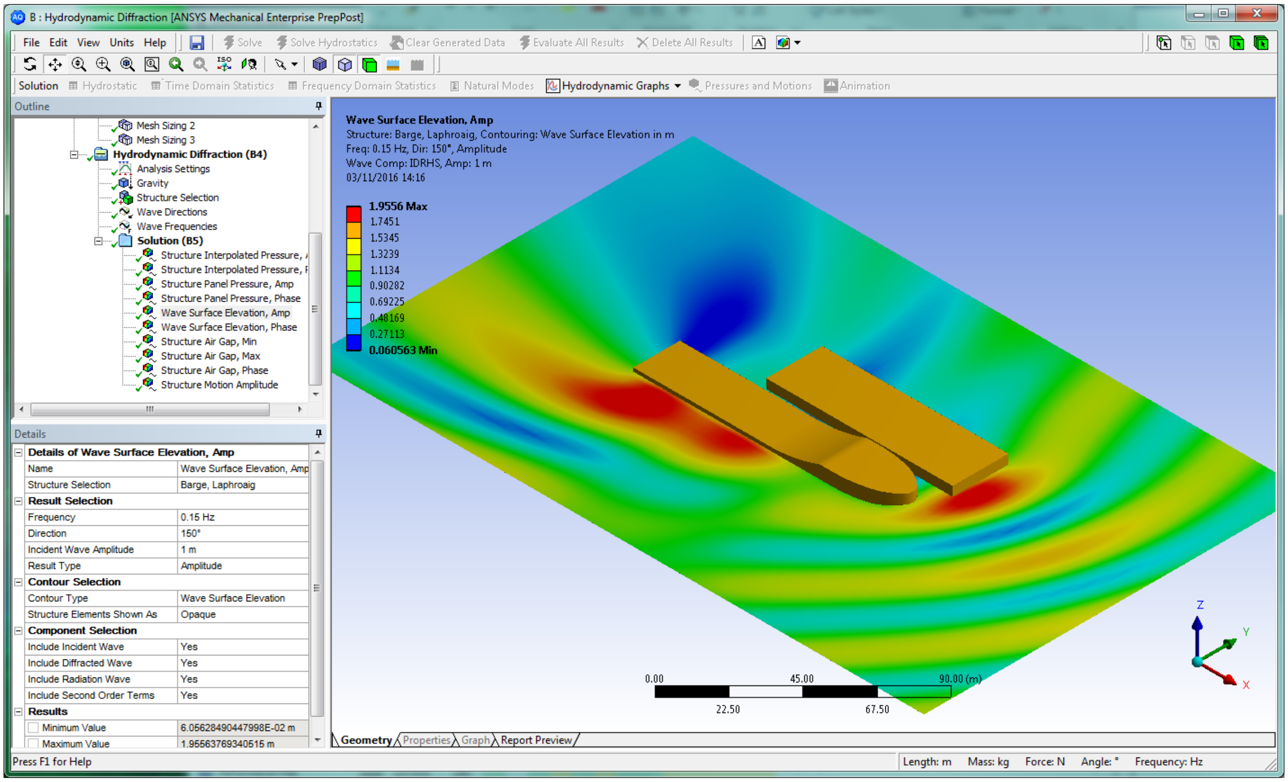


Structure Motion Amplitude displays the displacement or any position on the structure, as interpolated from the nodal displacements calculated by Aqwa. Motion Amplitudes can only be displayed for a Result Type of Phase Angle.

When plotting Structure Interpolated Pressure, Structure Panel Pressure, Structure Air Gap, or Structure Motion Amplitude, and the **Result Type** is set to **Phase Angle**, the position of the wave surface is plotted by default. To hide the waves, the **Water Elements Shown As** option can be changed from **Opaque** to **Dimmed**.

When plotting Structure Interpolated Pressure or Structure Panel Pressure, the non-diffracting panel elements are shown with zero pressure values by default. To hide the non-diffracting panels, the **Non-Diffracting Elements Shown As** option can be changed from **Opaque** to **Dimmed**.

Wave Surface Elevation plots the vertical position of the wave surface. In the figure below, plotting the Amplitude of the Wave Surface Elevation highlights the shielding effect of the structures in the lee of the oncoming wave; Mesh Element Outcomes Off is set to provide a clearer image.



When plotting Wave Surface Elevation, the structure mesh is hidden by default. To show the structure, the **Structure Elements Shown As** option can be changed from **Dimmed** to **Opaque**.

When plotting Structure Interpolated Pressure, Structure Panel Pressure, Wave Surface Elevation, or Structure Air Gap, where the **Result Type** is set to **Phase Angle** and the **Display RAO-Based Motions** option is set to **Yes**, the RAO-based motion of the structure at the defined phase angle (or sequence of phase angles) is included in the graphical display. While this option is turned on, the wave mesh extends throughout the structure; however, the results that are presented within dry areas are not accurate. Setting the **Display RAO-Based Motions** option to **No** fixes the structure in its geometric definition position, and hides any wave mesh that lies inside the structure, except for any Moonpool free surface elements.

Note:

The **Display RAO-Based Motions** option is automatically set to **No**, and cannot be changed, when the Pressures and Motions Structure Selection includes structures with Moonpools.

When plotting Structure Motion Amplitude, the RAO-based motion of the structure is always displayed and the wave surface is always hidden.

Any combination of wave components can be included in the reported results (exceptions are described in the following notes). To display a particular component, set its corresponding field in the Component Selection section of the Details panel to **Yes**:

- **Include Incident Wave**
- **Include Diffracted Wave**

- **Include Radiation Wave**
- **Include Hydrostatic Varying**
- **Include Second Order Terms**

Note:

The Hydrostatic Varying component is only applicable for Structure Interpolated Pressure and Structure Panel Pressure results.

Second Order Terms are only applicable for Wave Surface Elevation and Structure Air Gap results, and cannot be included if a Forward Speed has been defined in Wave Directions. For Structure Air Gap results, **Second Order Terms** refers only to the wave surface; the structure motion is always linear in this context.

When plotting Structure Air Gap or Structure Motion Amplitude results, the Incident, Diffracted, and Radiation terms are always included.

The wave mesh size can be altered via the Analysis Settings (p. 129), **Wave Grid Size Factor** setting.

The **Minimum Value** and **Maximum Value** of the results are shown and can be parameterized.

Once the Pressures and Motions result has been evaluated, you can export data into a comma-separated values (`.CSV`) file using the **Export to CSV File** option. For results of Structure Interpolated Pressure, Structure Panel Pressure, Wave Surface Elevation, and Structure Air Gap, when the **Result Type** is set to Phase Angle and the **Wave Position (Phase)** is set to Sequence, you can also set **Export Phase Angle** to define the wave position for which results will be exported.

The content of the exported file depends on the **Contour Type**, as described below:

Structure Interpolated Pressure

The `.CSV` file contains a row for each panel node on the selected structure(s), and has the following columns:

- Part name
- Surface Body name
- Structure node position (X, Y, Z coordinates)
- Total nodal pressure

Structure Panel Pressure

The `.CSV` file contains a row for each panel element on the selected structure(s), and has the following columns:

- Part name
- Surface Body name
- Element centroid position (X, Y, Z coordinates)

- Unit normal vector (X, Y, Z components) for the element orientation
- Element area
- Total element pressure

Wave Surface Elevation

The .CSV file contains a row for each water surface node on the selected structure(s), and has the following columns:

- Part name
- Water node position (X, Y, Z coordinates)
- Total wave surface elevation

Structure Air Gap

The .CSV file contains a row for each panel node on the selected structure(s), and has the following columns:

- Part name
- Surface Body name
- Structure node position (X, Y, Z coordinates)
- Total air gap to wave surface

Structure Motion Amplitude

The .CSV file contains a row for each panel node on the selected structure(s), and has the following columns:

- Part name
- Surface Body name
- Structure node position (X, Y, Z coordinates)
- Total nodal motion, real part (X, Y, Z components)
- Total nodal motion, imaginary part (X, Y, Z components)

5.12.5. Internal Tank Pressures Results

The Internal Tank Pressures results object enables the visualization and display of a number of results generated from Aqwa once a hydrodynamic solve has been performed. Any number of Internal Tank Pressures results may be added.

You must define the Internal Tanks (p. 64) that are included in the result. The **Internal Tank Selection** field provides a dropdown list that allows you to select any of the Internal Tanks in the Geometry, or you can select a Part for which all of the associated Internal Tanks will be displayed.

The **Frequency** and **Direction** options allow you to select from those defined in the Hydrodynamic Diffraction Wave Frequencies (p. 145) and Wave Directions (p. 144) objects. If you have only a single Wave Frequency or Direction defined in the Hydrodynamic Diffraction analysis, the corresponding field in the Internal Tank Pressures result becomes read-only, and is automatically updated if the single Wave Frequency or Direction is modified.

The **Incident Wave Amplitude** can be modified to provide results that are factored from the unit amplitude wave that is the default; extreme modification may extend results beyond the capabilities of a linear analysis, in which case inaccurate results can be produced.

The **Result Type** can be defined as:

- Phase Angle, where an equivalent phase position of the incident wave component can be selected (or is shown in the graph as the time as a proportion of wave period if a sequence is chosen)
- Maximum value of the selected result
- Minimum value of the selected result

When Phase Angle is selected, **Wave Position (Phase)** can be set to 0° (t/T = 0) or 90° (t/T = 0.25), or you can enter a specified phase position (in rotation units), or select a sequence of results for animation. If a sequence of results is selected, then the **Number of Steps** can be selected.

If you select a sequence of results for animation, after you evaluate the Internal Tank Pressures results (right-click the object and select Evaluate All Results), controls appear that allow you to start, pause, and stop the animation. If a particular result is required from the animated set, click the graph to display it. Other controls on the graph allow you to change the time over which the animation occurs and create an `.avi` animation file.

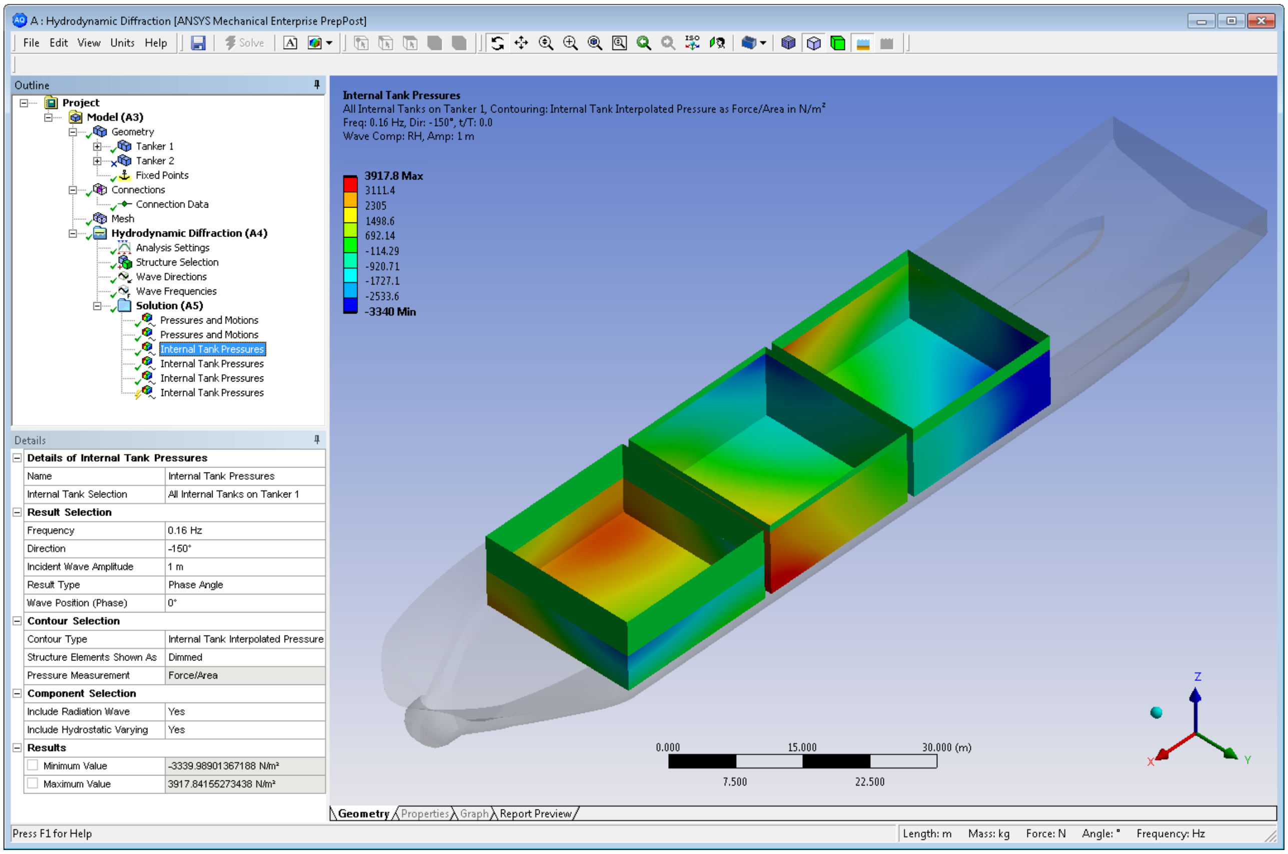
Under the **Contour Selection** section of the Details panel, the **Contour Type** field allows you to select which results are plotted over the internal tank surface mesh. This field can be defined as:

- Internal Tank Interpolated Pressure
- Internal Tank Panel Pressure

Internal Tank Interpolated Pressure displays element pressures which are interpolated from the nodal pressures calculated by Aqwa. This produces a smoothly-varying contour plot. Conversely, Internal Tank Panel Pressure shows the pressures at the element centroids. You can use the **Pressure Measurement** option to change the reported results between Head of Internal Fluid and Force/Area.

Note:

The Pressure Measurement option is restricted to Force/Area when results are displayed for multiple Internal Tanks with different internal fluid densities.



The parts of the structure that do not constitute the Internal Tank(s) selected for display are shown with a high transparency, by default. To modify this, you can change the **Structure Elements Shown As** option from Dimmed to Opaque.

Any combination of radiated or hydrostatic-varying internal fluid pressure components can be included in the reported results. To display a particular component, set its corresponding field in the **Component Selection** section of the Details panel to Yes:

- **Include Radiation Wave**
- **Include Hydrostatic Varying**

The **Minimum Value** and **Maximum Value** of the results are shown and can be parameterized.

Once the Internal Tank Pressures result has been evaluated, you can export data into a comma-separated values (`.CSV`) file using the **Export to CSV File** option. The content of the exported file depends on the **Contour Type**, as described below:

Internal Tank Interpolated Pressure

The `.CSV` file contains a row for each panel node on the selected internal tank(s), and has the following columns:

- Part name
- Surface Body name
- Structure node position (X, Y, Z coordinates)

- Hydrostatic pressure, real and imaginary parts
- Radiation pressure, real and imaginary parts

Internal Tank Panel Pressure

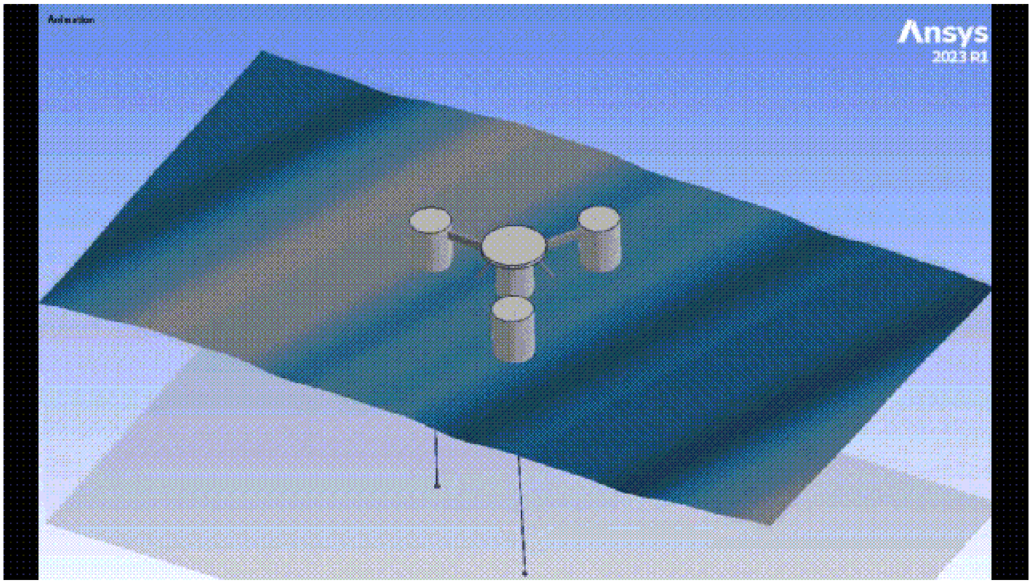
The .CSV file contains a row for each panel element on the selected internal tank(s), and has the following columns:

- Part name
- Surface Body name
- Element centroid position (X, Y, Z coordinates)
- Unit normal vector (X, Y, Z components) for the element orientation
- Element area
- Hydrostatic pressure, real and imaginary parts
- Radiation pressure, real and imaginary parts

5.12.6. Animation Results

Structural motions may be animated either in time for a Time Response Analysis or by iteration step for a Stability Analysis. The Animation results object allows you to view an animation of the motion of the parts in your project over the entire analysis period. This feature is particularly useful for observing the interaction between marine structures and the motion of waves. You can insert the Animation object into the tree before or after running the analysis. Only one of these results can be added to an analysis.

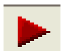
The controls described in Animation Controls (p. 237) allow you to start and pause the animation, change the portion of the animation that plays, and create an AVI movie file. For Equilibrium Motion Results, each iteration is depicted as 1 second in the control panel.



5.12.6.1. Animation Controls

Below is a table enlisting the available controls for animations in Aqwa Workbench and their function:

Table 5.3: Animation Controls

Control	Description
	Play the animation from the current time
	Pause the animation
Current Time (a slider to the right of the field can be used instead)	Enter the time at which to start the animation for viewing or video export
	Start the video export
Frame Increment	Play or export every Nth animation frame

To export a video of all or part of an animation set **Current Time**. Then click the video export icon to begin the recording. A dialog box will appear asking you to provide a file name and browse to the location where you want to save the file. The **Video Running Time** text box displays the length of the video currently being recorded. The video export button changes to a red dot to indicate that recording is in progress. You can click the red dot icon to stop recording. You can pause or resume recording by clicking the play and pause animation button on the left. The recording will automatically end when the animation is complete.

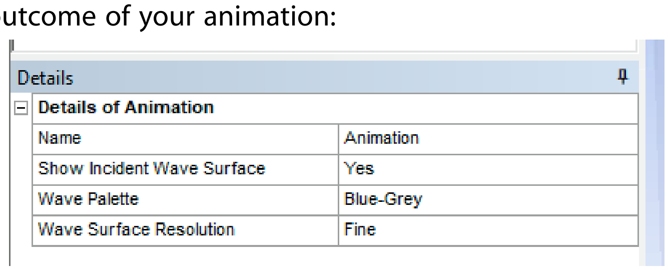
Note:

When entering time into the animation toolbar fields, you must press the **Enter** key before leaving the field in order to set the value.

The animation must be stopped in order to export a video. If you try to export a video while the animation is playing on the screen, nothing will happen.

5.12.6.2. Animation Settings

There is a collection of settings available in the Details panel, which enables you to determine the outcome of your animation:



Name

This enables you to modify the name of your animation. This helps to keep track of animation objects in the Outline tree.

Show Incident Wave Surface

This setting enables you to include or remove the entire water body, together with the motion of waves, in your animation by selecting either **Yes** or **No**.

Wave Palette

This setting enables you to change the gradient color scheme of the waves surrounding the marine structure, based on your preference. The available options include:

- **Blue-Grey**
- **Blue-Green**
- **Green-Grey**
- **Dark Blue**

Wave Surface Resolution

This setting enables you to change the resolution of the animation. The available options include:

- **Fine**
- **Medium**
- **Coarse**

Fine or **Medium** is suitable for high-speed computers. If your computer's speed is limited, you may have to change your resolution from **Fine** or **Medium** to **Coarse**. This should help the animation run smoother in situations where the motion appears to lag.

Note:

If you add a new animation to a different Time-Domain Hydrodynamic Response calculation, the **Wave Palette** defaults to **Blue-Grey** and the **Wave Surface Resolution** defaults to **Medium**. However, you can create alternative default settings by following this path: **Edit** > **Options** > **Aqwa** > **Animation**.

5.12.7. Dynamic Natural Modes

The natural frequencies of a system of one or more structures can be found using the **Natural Modes** result object.

When a **Natural Modes** result is added to a Hydrodynamic Diffraction analysis, natural modes are determined for individual freely-floating structures or groups of hydrodynamically-interacting structures in their positions as defined in the geometry editor. The effects of Connections (Cables, Joints, Tethers, Fenders and Connection Stiffness matrices) are not included, and any Motion Locks applied to hydrodynamic interaction groups are ignored.

When a **Natural Modes** result is added to a Hydrodynamic Response stability analysis, the natural modes are determined for the entire system of structures, including their connections and the effects of any defined environment (Wind, Current, drift due to Irregular Waves, etc). Structures are moved into their equilibrium positions, as calculated during the stability analysis, before the natural modes calculation is performed.

Note:

If you are resuming a Workbench project created in previous releases, where the Hydrodynamic Diffraction Solution is in a solved state and where the Hydrodynamic Diffraction analysis did **not** include Shear Forces/Bending Moments or Linearized Morison Drag, you will need to Clear Generated Data before you can add a Natural Modes result to the Hydrodynamic Diffraction Solution.

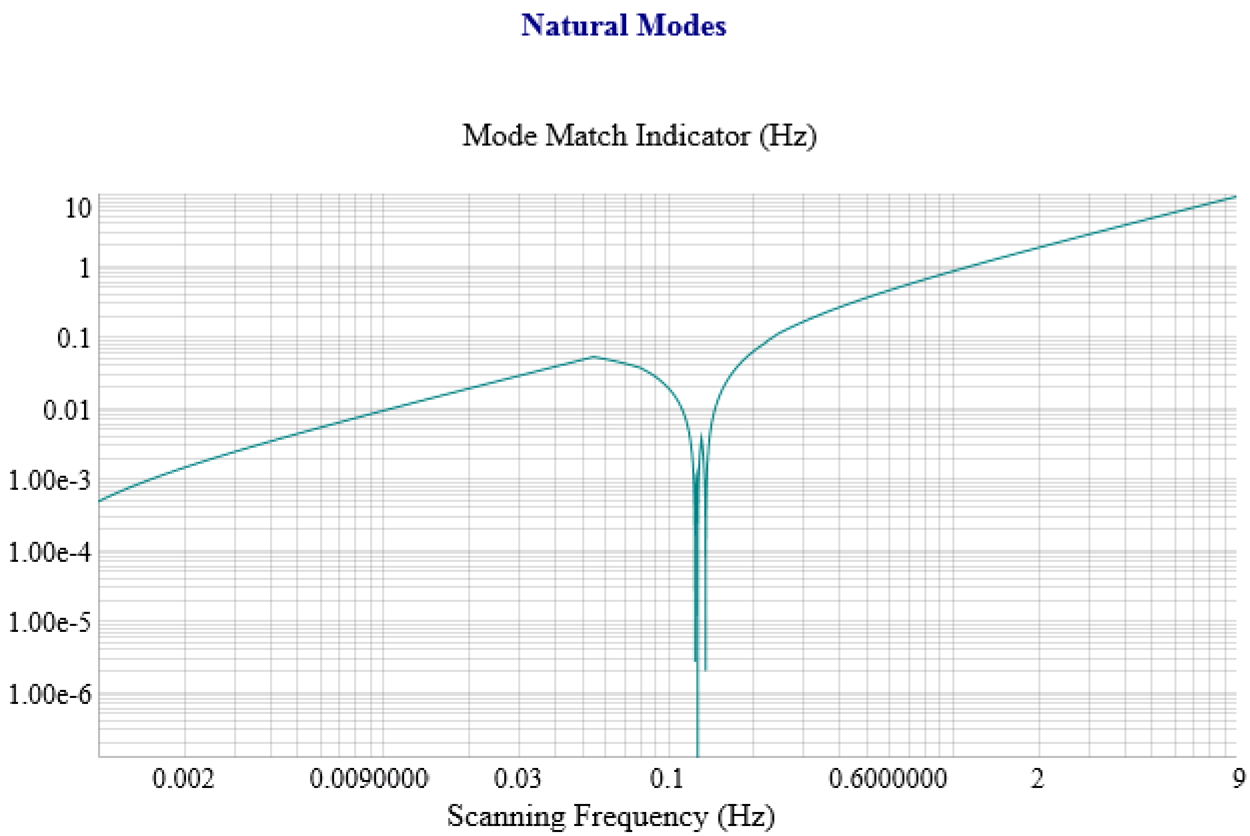
Figure 5.15: Natural Modes Result Object

Details of Natural Modes	
Name	Natural Modes
Use Advanced Options	No

As Natural Modes depend on frequency-dependent Added Mass and Damping, a direct calculation of the eigenvalues and the eigenmodes of a single matrix is not possible. Instead, the following method is used:

Aqwa scans a range of input frequencies and solves the eigenvalue problem based on Added Mass and Damping values for the current scanning frequency, which yields a list of pairs of eigenmodes and their corresponding frequency. Aqwa discards the modes in which the frequency does not match the scanning frequency and retains the matching modes as actual physical modes.

Figure 5.16: Mode Match Indicator



In order to provide you with an immediate view on the quality of the search, the results listed under the **Monitor** tab of the output window contain a graph of Mode Match Indicator vs. Scanning Frequency. This indicator, calculated for each scanning frequency, is the difference between the scanning frequency and the closest frequency found when solving the eigenvalue problem using Added Mass and Damping values associated with the scanning frequency. A match is obtained where this indicator reaches a local minimum below a certain tolerance. This tolerance is adaptive, complex, and depends (among other things) on the local resolution of the scan.

The local resolution of the scan is also adaptive, and based on the value of the match indicator calculated at the previous scanning point. It is limited by the **Maximum Relative Precision** parameter in the Details panel when **Use Advanced Options** is on.

The start and end frequency of the range may be controlled using the **Use Advanced Options** field.

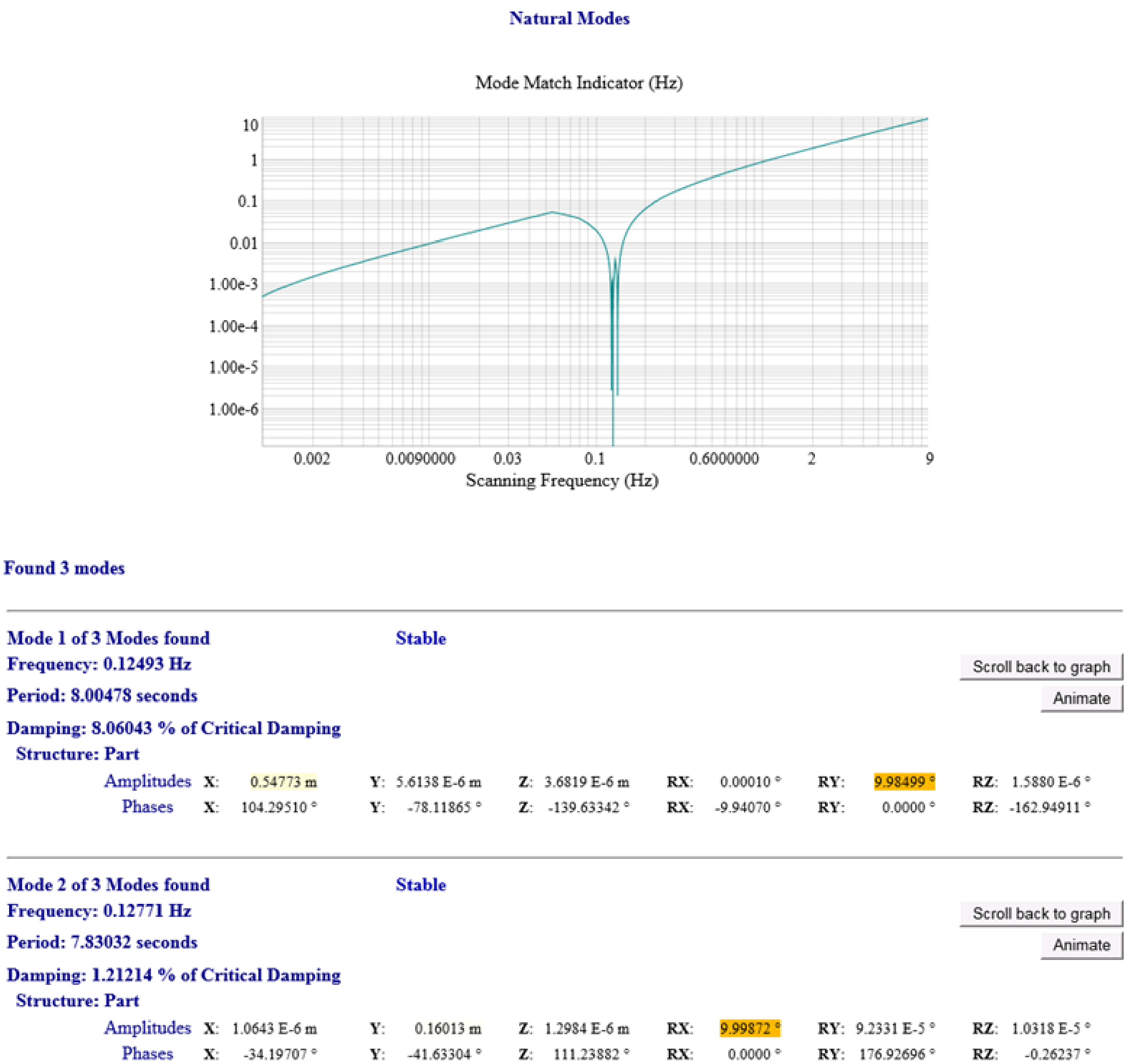
Figure 5.17: Natural Modes Advanced Options

Details of Natural Modes	
Name	Natural Modes
Use Advanced Options	Yes
Use Linearized Stiffness	No
Start Frequency for Scanning	0.001 Hz
End Frequency for Scanning	10 Hz
Maximum Relative Precision f...	1e-4

Underneath the Mode Match Indicator graph, the list of the retained Natural Modes is displayed.

The values for the amplitudes are highlighted with a color depending on the value, allowing for easier visual identification of the leading degree of freedom. Next to the mode, an **Animate** button allows you to visualize the motions of the structures corresponding to the natural mode in the geometry window. The **Scroll Back to Graph** button allows you to navigate back to the top of the page easily. By clicking the dot moving along the curve displayed on the graph, you can easily navigate to the mode with frequency closest to the clicked point's abscissa. You can zoom the graph by selecting a portion of the graph with click and drag. You can zoom out by double-clicking the left mouse button.

Figure 5.18: Mode List with Animation



5.12.8. Time Domain Statistical Results

Time Domain Statistics allow you to view statistics and probability distributions for a Time Response Analysis (p. 135).

By default, statistics are calculated over the Full Time Series (all the steps) in the time response analysis. Changing the **Statistics Calculated Over** setting to Cropped Time Series allows you to specify

Cropping from Start of Time Series and **Cropping from End of Time Series** times. The calculated statistics will then be restricted to this time frame.

In the Details panel, select a **Result Type** in one of the Result A-E fields. You can select from any quantity which has been calculated in the time response analysis, such as Wave Surface Elevation, Structure Motions and Forces, Connection Forces, Tether/Riser Properties, or Time Step Error.

Each of the Result A-E sets includes a **Result Source** selection. Where there are multiple Time Response Analysis systems in the Outline tree, you may select any as a Result Source for the current Result Set. This allows results from similar Time Response Analyses – for example, cable tension statistics under different environments – to be compared in the same table of probability distributions.

Details	
Details of Time Domain Statistics	
Name	Time Domain Statistics
Statistics Calculated Over	Cropped Time Series
Cropping from Start of Time Series	30 s
Cropping from End of Time Series	0.0 s
Statistics Shown As	Probability Density Function
Result A	
Result Source A	Time 1 > Solution (C5)
Result Type A	Structure Position
Structure	Part
SubType	Actual Response
Component	Global Z
Reference Point	Fairlead 3 (Part)
Motion Relative To	Origin of Fixed Reference Axes (FRA)
Statistics Parameters A	
Mean Value	0.01766 m
Standard Deviation	1.19315 m
Minimum Value	-3.67777 m
Maximum Value	3.12958 m
Mean of Lowest-Third Peaks	-2.08375 m
Mean of Highest-Third Peaks	1.95513 m

When a result type is selected, the Result object expands with further information in the Details panel.

Note:

You can include any of the results in your design points by clicking the check box next to the result in the Details panel. However, output values can only be parameterized when their **Result Source** is defined as the current analysis system.

If you wish to calculate statistics for the translation motion of the structure, the **Reference Point** at which the result is reported can be the Center of Gravity of the selected **Structure**, or any Connection Point on that structure.

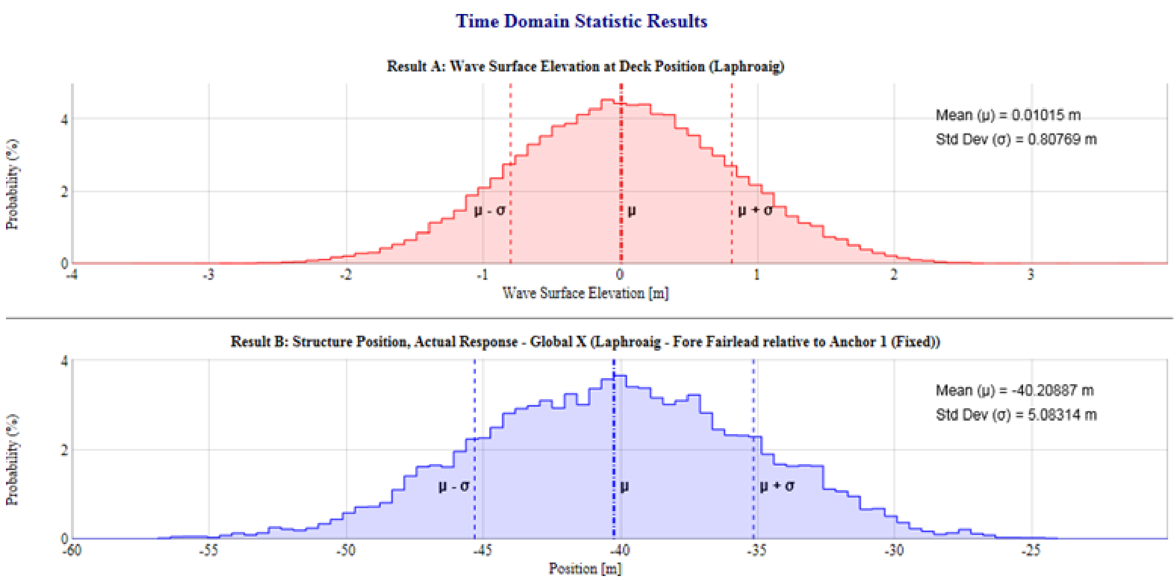
The **Motion Relative To** input can be used to set the frame of reference for the reported translation motion. This may either be the Origin of the Fixed Reference Axes (FRA), the Center of Gravity of another structure, a Connection Point on another structure, or a Fixed Point.

Note:

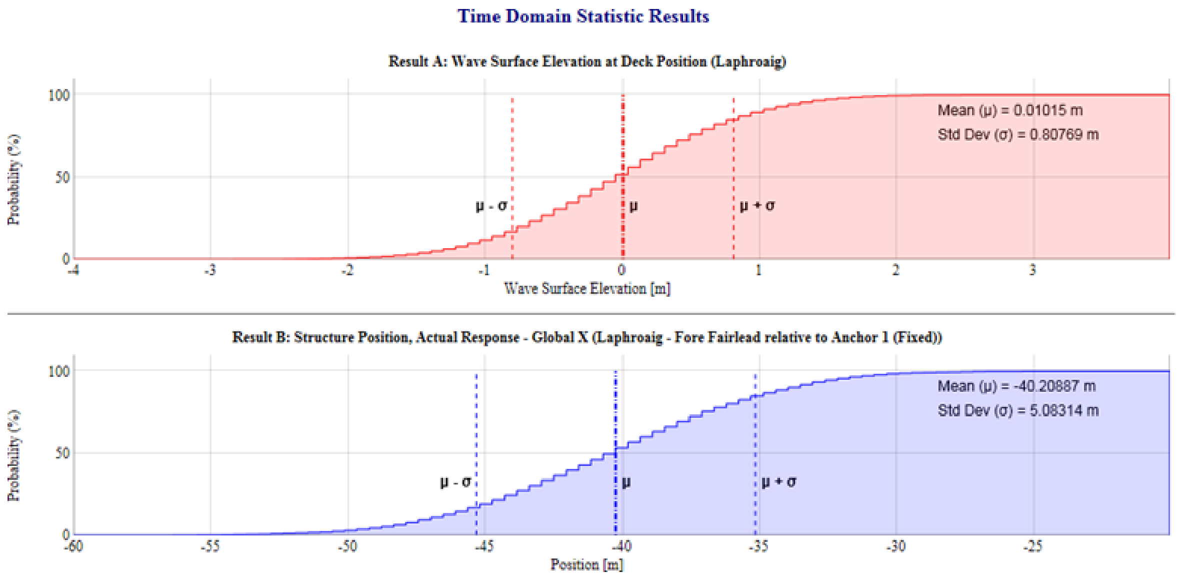
When the **Type** is set to Structure Position, the **SubType** is set to Actual Response, the **Component** is set to Global Z, and the **Reference Point** is a Connection Point, the result may also be reported relative to the Wave Surface.

The selected Connection Point must have its **Include in Results** option set to Yes before the analysis is solved.

Probability Density Functions are available by clicking the **Properties** tab at the bottom of the graphical view. The distributions are shown as histograms, where the number of bins for each histogram is automatically determined as a function of the number of time steps. Mean and standard deviations are plotted as dot-dashed and dashed lines, respectively.



You can also use the **Statistics Shown As** setting to optionally plot these histograms as Cumulative Distribution Functions.



5.12.9. Time Domain Pressures Results

The **Time Domain Pressures** results object enables the display of the instantaneous internal tank and external hull surface pressures generated by Aqwa, during a time domain response analysis in which the Hydrodynamic Response Analysis Settings option **Analysis Type** is set to Irregular Wave Response or Regular Wave Response. Any number of Time Domain Pressures results may be added.

The **Time Domain Pressures** result shows the pressures at the element centroids, and can be directly compared with results in Mechanical once a load transfer procedure has been completed (see The Hydrodynamic Pressure Mapping Add-On (p. 273) for more information).

When there is more than one structure in the model and **Output for Structure** is set to **All Structures** in the Analysis Settings, the **Structure** dropdown list allows you to select the structure(s) that should be shown in the Time Domain Pressures result. The **Structure** field will be read-only if there is only one structure for which time domain pressures are being calculated.

The **Display Pressures At** option can be defined as:

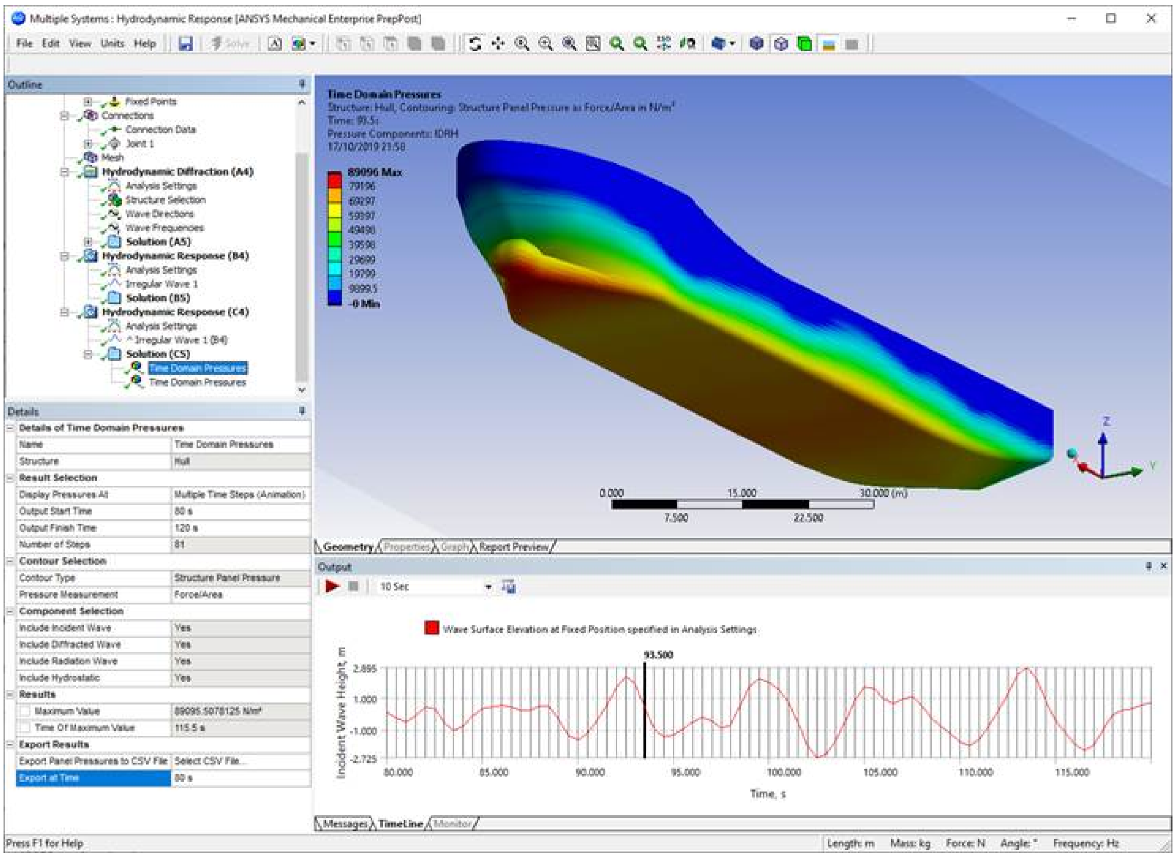
- Single Time Step - select a single **Output Time** at which pressures will be displayed.
- Multiple Time Steps (Animation) – select an **Output Start Time** and **Output Finish Time** to define the start and finish times for an animation of time domain structure motions and panel pressures. The **Number of Steps** field is automatically updated to indicate the number of frames that will be displayed in the animation.

Note:

The **Output Time**, **Output Start Time**, and **Output Finish Time** fields are automatically adjusted to be within the range of the Hydrodynamic Response Analysis Settings **Output Start Time** and **Output Finish Time**, and are rounded to the nearest whole number of **Output Step** intervals.

When the **Display Pressure At** option is set to Multiple Time Steps (Animation), and once you have evaluated the Time Domain Pressures results (right-click the object and select Evaluate All Results), controls appear that allow you to start, pause, and stop the animation. You can click the wave elevation timeline to display results at a particular instant. Other controls allow you to change the speed of the animation, and to create a `.avi` animation file from the current frames.

Under the **Contour Selection** section of the Details panel, the read-only **Contour Type** field indicates that Structure Panel Pressure will be displayed. You can use the **Pressure Measurement** option to change the reported results between Head of Water and Force/Area. You may choose to set Mesh Element Outlines Off from the graphical toolbar to provide a clearer image.



The combination of wave components included in the reported results is indicated in the **Component Selection** section of the Details panel. **Incident**, **Diffracted**, **Radiation**, and **Hydrostatic** components are always included in the Time Domain Pressures result.

The **Maximum Value** of the results are shown, and where **Display Pressures At** is set to Multiple Time Steps (Animation), the **Time of Maximum Value** is also provided. Both of these fields can be parameterized.

Once the Time Domain Pressures result has been evaluated, you can export pressure data into a comma-separated values (`.CSV`) file using the **Export Panel Pressures to CSV File** option. From the range of times defined in the Result Selection details, set **Export at Time** to define the time at which panel pressures will be output; **Export to Time** will be read-only if **Display Pressures At** is set to Single Time Step. The **Export to Time** field is automatically adjusted to the nearest whole number of **Output Step** intervals. The `.CSV` file contains a row for each panel element on the selected structure(s), and has the following columns:

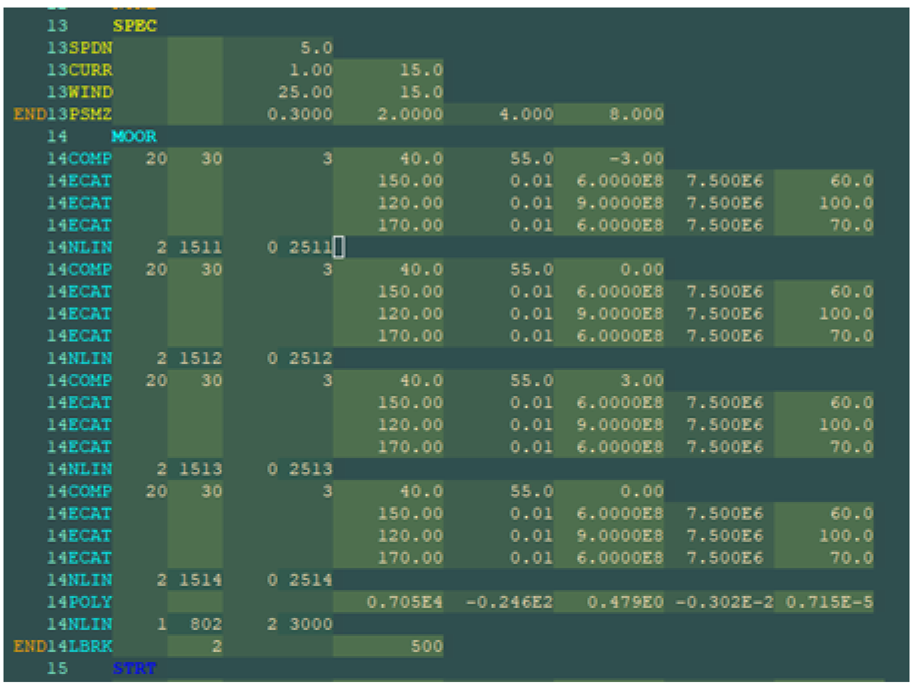
- Part name

- Surface Body name
- Element number
- Instantaneous position (X, Y, Z) of the element centroid
- Instantaneous unit normal vector (X, Y, Z) for the element orientation
- Element area
- Instantaneous total pressure (sum incident, diffracted, radiation and hydrostatic components)

5.13. Aqwa Model Data Import

The Hydrodynamics systems in Workbench support the import of data from Aqwa models defined using the Aqwa solver input format contained in files with the `.DAT` extension. The syntax for this format is described in the *Aqwa Reference Manual*.

An Aqwa `.DAT` file is divided into a number of sections, called *data categories*, which contain the information needed by the solver to perform an analysis. The Workbench Aqwa editor is capable of reading some of that information in order to create the corresponding objects in the Workbench project's Outline tree.


```
    13     SPEC
    13SPDN                5.0
    13CURR               1.00      15.0
    13WIND              25.00      15.0
 END13PSMZ             0.3000    2.0000     4.000     8.000
    14     MOOR
    14COMP   20   30        3      40.0      55.0     -3.00
    14ECAT                       150.00      0.01 6.0000E8  7.500E6      60.0
    14ECAT                       120.00      0.01 9.0000E8  7.500E6     100.0
    14ECAT                       170.00      0.01 6.0000E8  7.500E6      70.0
    14NLIN    2 1511      0 2511
    14COMP   20   30        3      40.0      55.0      0.00
    14ECAT                       150.00      0.01 6.0000E8  7.500E6      60.0
    14ECAT                       120.00      0.01 9.0000E8  7.500E6     100.0
    14ECAT                       170.00      0.01 6.0000E8  7.500E6      70.0
    14NLIN    2 1512      0 2512
    14COMP   20   30        3      40.0      55.0      3.00
    14ECAT                       150.00      0.01 6.0000E8  7.500E6      60.0
    14ECAT                       120.00      0.01 9.0000E8  7.500E6     100.0
    14ECAT                       170.00      0.01 6.0000E8  7.500E6      70.0
    14NLIN    2 1513      0 2513
    14COMP   20   30        3      40.0      55.0      0.00
    14ECAT                       150.00      0.01 6.0000E8  7.500E6      60.0
    14ECAT                       120.00      0.01 9.0000E8  7.500E6     100.0
    14ECAT                       170.00      0.01 6.0000E8  7.500E6      70.0
    14NLIN    2 1514      0 2514
    14POLY                      0.705E4 -0.246E2  0.479E0 -0.302E-2 0.715E-5
    14NLIN    1  802      2 3000
 END14LBRK         2                500
    15     STRT
```

To import data from an Aqwa solver input file:

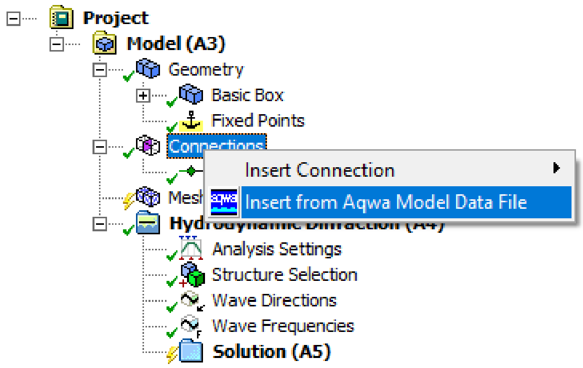
1. Create a new Workbench project and add a Hydrodynamic Diffraction system, and any linked Hydrodynamic Response systems as required, to the Project Schematic.

2. Attach a geometry, or a dummy geometry, to the system and open the Aqwa editor.

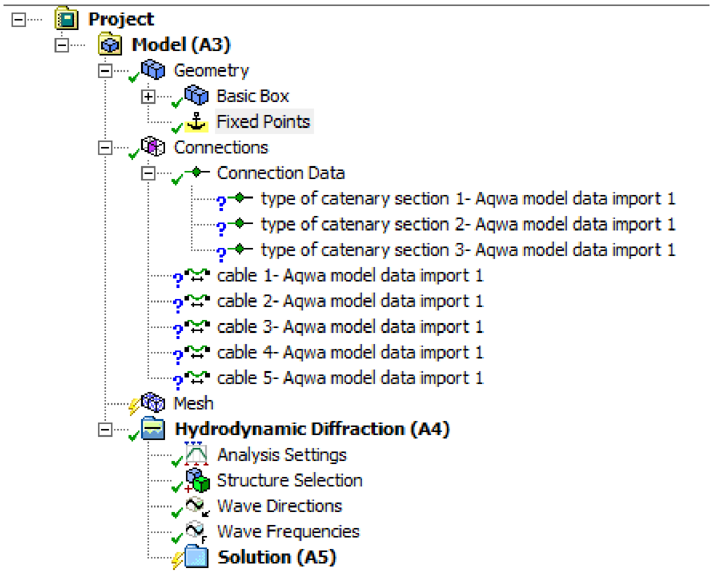
Note:

To import the Aqwa solver input file, a dummy geometry is required if there is no geometry in the project. This dummy geometry may be suppressed if not required in the subsequent analysis.

3. In the Outline tree, select the branch under which you want to add the objects corresponding to the information you want to import from the file. Right-click the branch and select Insert from Aqwa Model Data File.



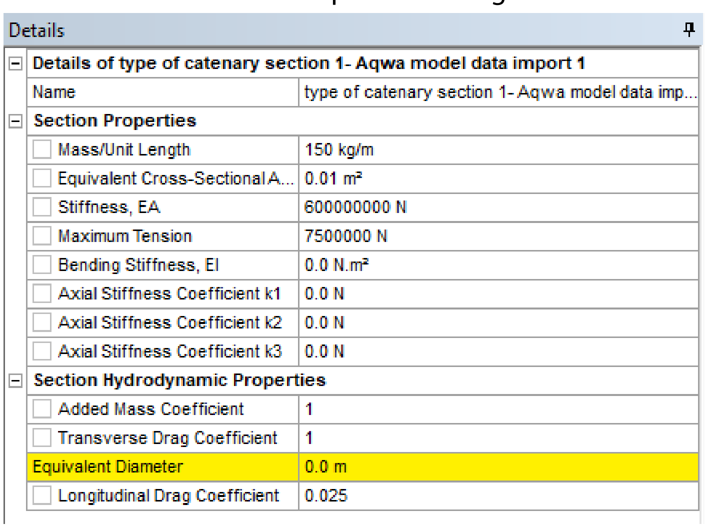
Browse to the file containing the information you want to import and select it. The program then loads the input file, analyzes its contents, and selects the part of the contents which are relevant in the context of the selected tree item. In this example the tree object is Connections, which can import Cables, Joints, and Fenders. It then creates objects in the tree that correspond to the data in the file.



Note:

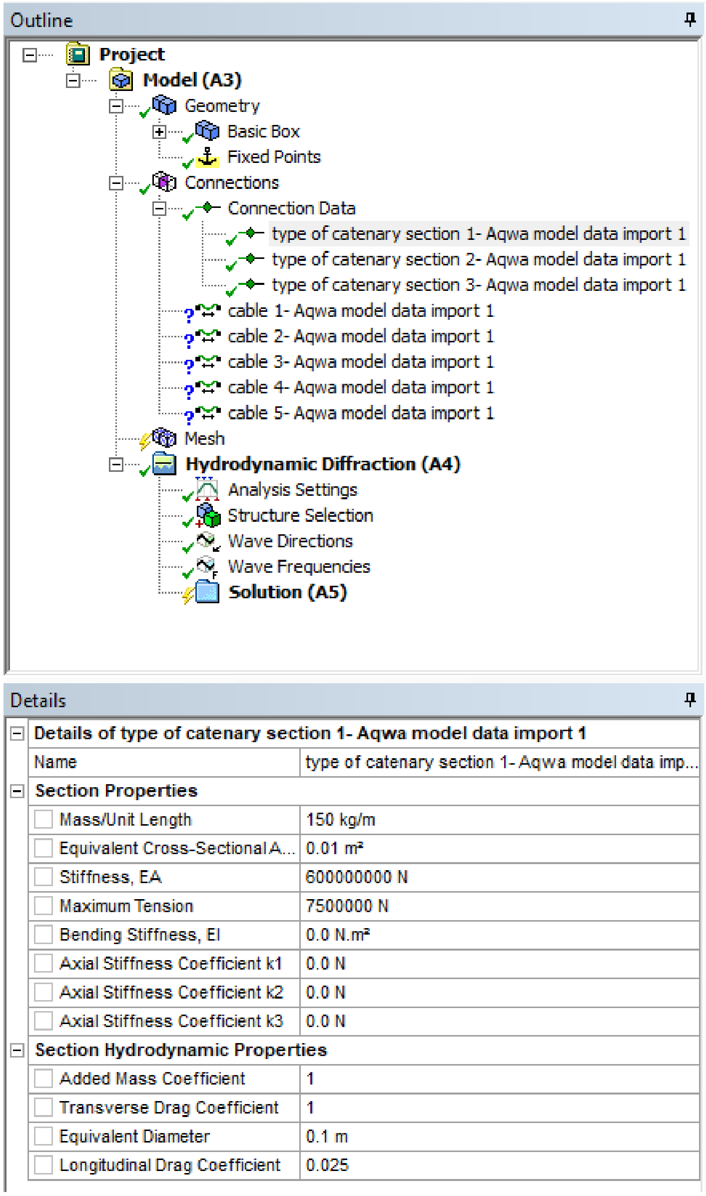
The Aqwa solver input file does not specify units. Be sure that the Hydrodynamic Solver Unit System selected in the Details of your Project are appropriate for the model you are importing.

4. In the example given of **Insert from Aqwa Model Data File** in the **Connections** object, the created items are shown as incomplete. Looking at their details shows that some information is missing:

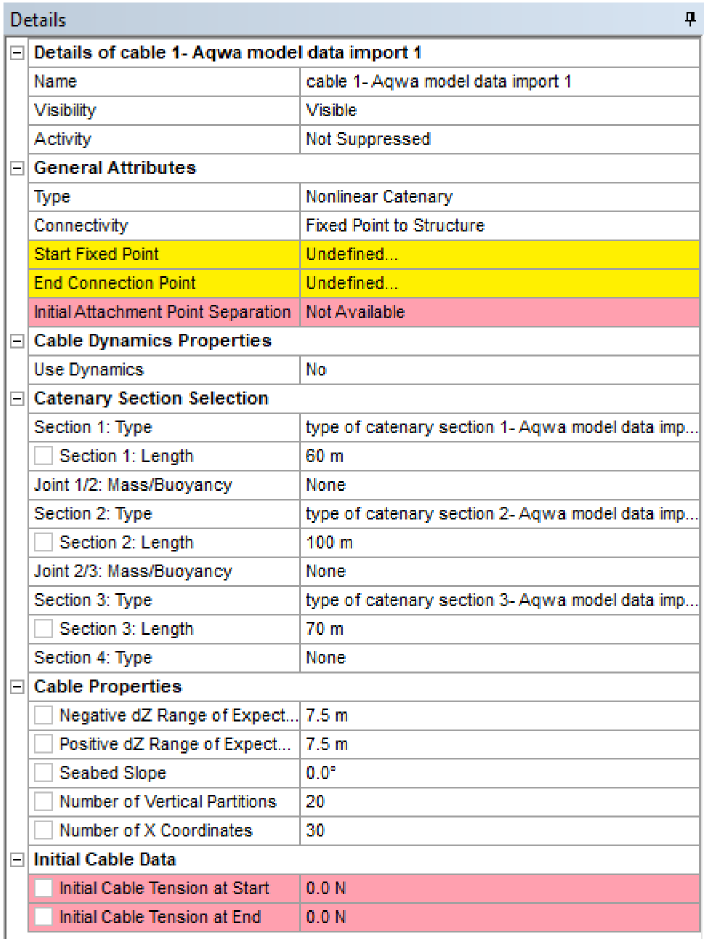


The information was missing from the input file used.

5. Once you have added the missing information to the Catenary Sections, the objects show as up-to-date.



The cables are still showing as incomplete. Looking at their details shows that the connection points are undefined.



The connection points are undefined because the import was initiated from the **Connections** object in the Outline tree, which does not include the **Geometry** branch where the attachment points have to be defined. Selecting a particular tree item for initiating the import operation allows you to select which part of that information you want to include or to exclude from the process. The table below summarizes the extent of the import depending on the tree item selected at the start of the process.

OutlineTree Object Selected when Starting The Import process	Information Read from the Input File	Objects Added to the Tree
Project/Model	All Items in the column below.	All items as listed in the column below, with the following differences: Cable/Fender/Joint/Force definitions, **with** the corresponding Connection Points on structures and Fixed Points, as well as full catenary definitions (type of catenary, buoys, clump weights). *If there are multiple hydrodynamic systems in your project, objects created (imported) under Project or Model are duplicated for each analysis as the*

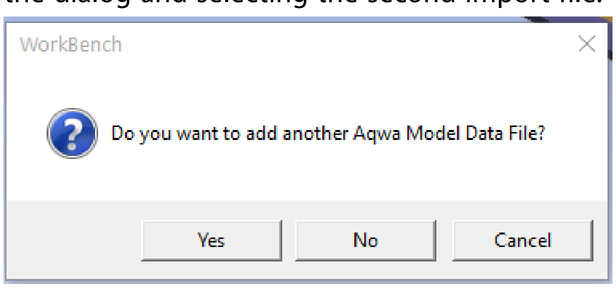
		program has no way of deciding to which specific analysis they should belong.
Geometry	Nodes, elements and structures.	Parts, Bodies, Point Masses, Point Buoyancies, Wind Force Coefficients, Current Force Coefficients.
Connections	Cable, Catenary, Fender, and Joint definitions.	Cable/Fender/Joint, **without** the corresponding Connection Points on structures and Fixed Points, and without fender and joint axis directions.
Connection Data	Catenary definitions.	Catenary definitions (type of Catenary, Buoy, Clump Weight).
Analysis	Winch; Cable Failure; Point Force of constant magnitude and direction; Current, Wind, and Wave definitions (either regular or irregular); Deactivated Freedoms.	Cable Failure. Cable Winch. Point Force of constant magnitude and direction. Current, Wind, and Wave objects. Deactivated Freedoms.

Note:

- New parts created through importing Geometry information cannot be shared with any other Ansys system by using a link between the Geometry cells on the schematic. Any non-Aqwa downstream system added on the Workbench schematic will be unable to see them. Also, their mesh is fixed and cannot be modified as it corresponds always to the panels defined in the original input file.

- If the imported Current and Wind Force Coefficients matrices are defined as symmetrical in the Aqwa data file, the corresponding table entries will be filled out in the Workbench system objects.

- Importing the information at the Project level implies that the input files contains all the corresponding data categories. This is possible if the imported Aqwa model data file covers the analysis stages 1 to 5. Since the Aqwa solver allows the analysis stages to be split over several files (for instance a first file containing the definitions for the analysis stages 1 to 3, and a second file covering stages 4 to 5), you may either merge these multiple files into one single file in order to import all of the information at once, or import one file for the analysis stages 1 to 3 and add another file for the analysis stages 4 to 5 by clicking **Yes** in the dialog and selecting the second import file.



Chapter 6: Aqwa Workbench Interface

Hydrodynamic analysis systems are available to place on the Workbench Project Schematic from the Toolbox. After configuring the connections to/from the Hydrodynamic analysis systems and/or any inputs to the systems, you will open the Aqwa Editor to perform the analysis and view the results of the Hydrodynamic analysis system. This chapter discusses how to place and configure your analysis system on the Project Schematic and describes the User Interface of the Aqwa Editor. The topics covered are:

6.1. The Aqwa Editor User Interface

6.2. Editing Objects

6.3. Displayed Positions and Features

6.4. Setting Aqwa Application Options

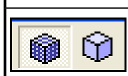
6.1. The Aqwa Editor User Interface

Hydrodynamic analysis systems open a Workbench based editor that is a graphical interface for Aqwa analyses. Using this editor you can create an element based model from geometry defined in Design Modeler format, apply Aqwa specific input, and view results. The layout is similar to other Workbench based applications (for example, the Mechanical application) and is composed of a number of objects in a tree based layout.

Hydrodynamic analysis systems follow the conventions used in other Ansys Workbench products where it is appropriate for the Aqwa Editor. The user interface has a number of key areas, but is tree driven. Along with the tree and its details pane are toolbars and graphical / text display windows.

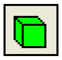
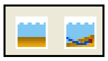
The toolbar uses a number of standard Workbench icons along with a number of specialist additions.

Table 6.1: Standard Workbench Toolbars

Symbols	Purpose
	View manipulation
	Selection control
	File shortcuts
	Show/hide element boundaries
	Add a Comment (p. 270) to the selected tree object
	Add a Figure (p. 272) or Image (p. 271) to the selected tree object

Symbols	Purpose
	Control the output of the Report Preview (p. 269) results

Table 6.2: Aqwa Editor Toolbars

Symbols	Purpose
	Turn on/off highlighting of objects selected in the tree
	Show/hide the sea surface and (left) diffracting or (right) composite cable seabed
	Solver shortcuts

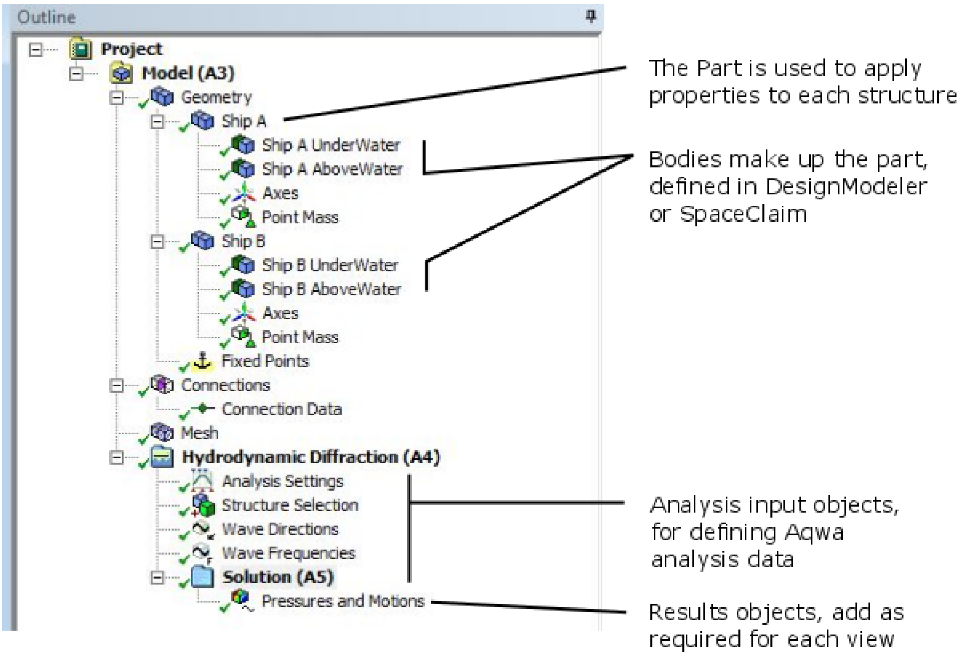
The tree and the details are where the objects that are used to define the modelling requirements are organized. Depending upon the selection in the tree, a detail pane, normally located below the tree will show the details of the selected object.

The style of each editable detail depends on the data: in some places you type a value into a field, while in others you select an entry from a drop-down menu, or choose a file location on disk. Where a floating-point value can be typed into a field, you can alternatively write a simple expression, which should start with an equals character **=**. For example, instead of **`1.4142135623731`**, you could write **`=2^0.5`**. Permitted operators are shown in the table below.

Operation	Example entry	Value after evaluation
Add x and y	=4.6+3.2	7.8
Subtract y from x	=4-0.165	3.835
Multiply x and y	=5.4*2.2	11.88
Divide x by y	=6.3/1.6	3.9375
Raise x to the power of y	=2.3^1.2	2.716898…
Multiply x by 10 to the power of y	=2.3e6	2300000
Raise natural number e to the power of x	=exp(2.2)	9.025013…
Natural logarithm of x	=log(2.2)	0.788457…
Trigonometric functions sin, cos, tan	=sin(1.57)	0.999999…
Inverse trigonometric functions asin, acos, atan	=asin(1)	1.570796…
Hyperbolic functions sinh, cosh, tanh	=cosh(0.6)	1.185465…

Expressions can include multiple brackets (and). The value of pi is not included, but can be calculated as '=2*acos(0)'. Note that the typed expression is not retained after it has been evaluated by the editor.

Symbols are shown next to each object in the tree to indicate its state.



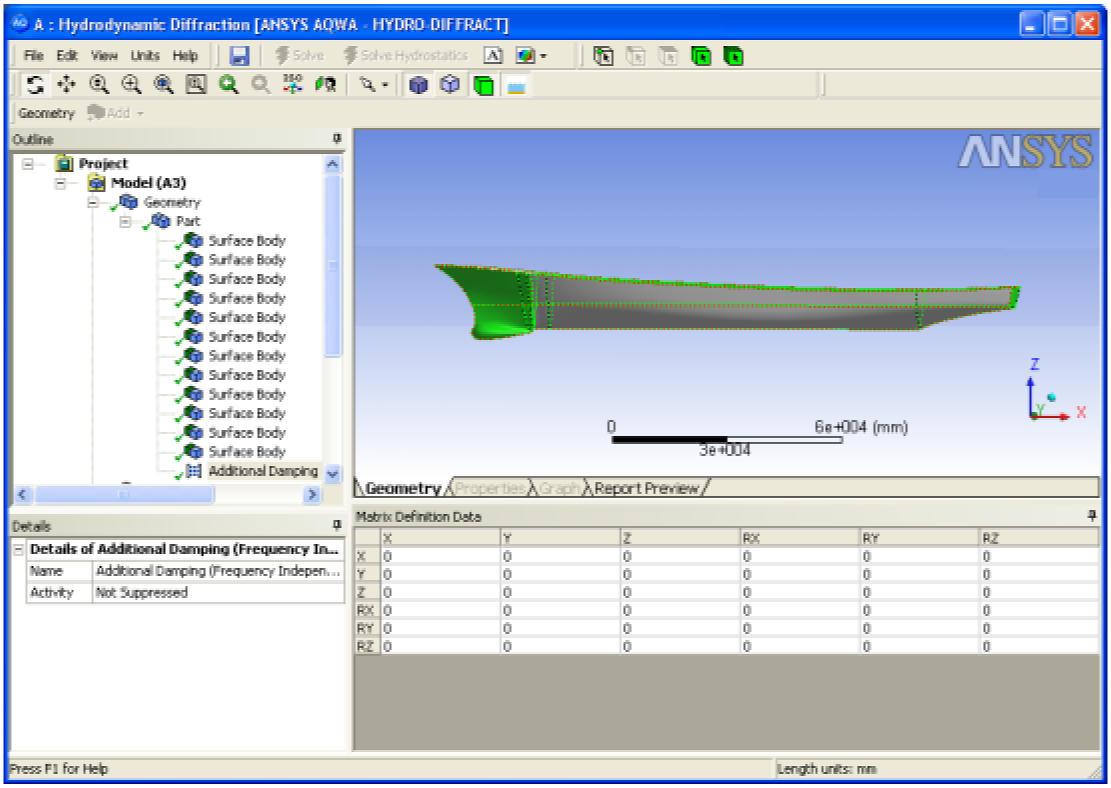
Each object in the tree has properties that are used to control the analysis and view the results. For some objects, the right mouse button can be used to insert additional objects under that object, such as masses or result views. Alternatively, the dynamic toolbar can be used to insert additional objects.

If you are an existing Aqwa user, then you may find the section Information for Existing Aqwa Users (p. 349) a useful comparison of capabilities and definition methodology.

There are either one or two other windows shown, depending upon the state of the analysis or tree item that is selected:

- The main window which generally shows the model (either geometry or mesh views) in the Geometry tab, Hydrodynamic results in the Graph tab, a report of your analysis in the Report Preview tab, and Hydrostatics results in the Properties tab where you can view text based results.

- The secondary window which is used for displaying messages, controlling the results, and entering data. This will appear automatically, depending upon the tree item selected. Messages are also displayed in the Messages view of the Project Schematic.



6.2. Editing Objects

Right-clicking on an object in the tree view, or selecting an object and clicking **Edit** in the menu toolbar, allows you to perform a number of editing operations on that object:

- 6.2.1. Copy and Paste
- 6.2.2. Duplicate
- 6.2.3. Propagate
- 6.2.4. Break Link
- 6.2.5. Delete

The editing operations that are available depend on the type of object selected.

Table 6.3: Availability of Editing Operations

	Geometric Features	Connections	Environment Features	Results Objects
Copy and Paste [a]	√		√	√ [b]
Duplicate [a] [c]	√ [d]	√		√
Propagate [a] [c]			√	
Break Link			√ [e]	
Delete	√	√	√	√

[a] An object must be up-to-date (shown with a green check mark in the tree view) before the Copy, Duplicate and Propagate operations can be used.

[b] Only between analysis systems of the same Computation Type, or for graphical results between a Stability Analysis and a Time Response Analysis.

[c] Duplicate and Propagate use a separate clipboard than Copy. If you copy one object and then duplicate or propagate another, the data of the first object is still available to be pasted.

[d] Only on an object of which more than one per structure is permitted.

[e] On an object that has been created by a Propagate operation.

6.2.1. Copy and Paste

Copy and paste can be used to copy an object from one structure or analysis into another structure or analysis. They are used to create a copy under a different parent object in the tree.

Some object properties are not copied. For example, the object **Name** is unique and is generated automatically when the object is pasted.

A paste operation can only be performed where the copied object could normally be added or inserted. For example, a copied Connection Point can be pasted into a structure or into the Geometry, but a copied Point Mass can only be pasted into a structure.

The following are some examples where copy and paste can be useful:

- A model contains two identical vessels. Wind and Current Force Coefficients are defined for the first vessel. Rather than entering the same data again, these objects can be copied and pasted into the second vessel.
- A Joint is to be created between two structures A and B. Once its position has been defined, the Connection Point on structure A can be copied and pasted into structure B.

Note:

Copying an object creates an unlinked copy of that object on the clipboard. This means that once it has been pasted, changes to the original object are not reflected in the original.

The data on the clipboard may still be pasted even after the original object has been deleted.

Changing the unit system after a Copy operation does not have any effect on the copied data on the clipboard. For example, copying a Connection Point with X of 100 m and then changing the length unit from meters to feet will result in a subsequently pasted Connection Point with X of 100 feet.

6.2.2. Duplicate

Duplicating an object creates a copy of that object under the same parent object in the tree.

Some object properties are not duplicated: the Connection Points of a Cable, for example, are unique, and will be undefined in the duplicate object.

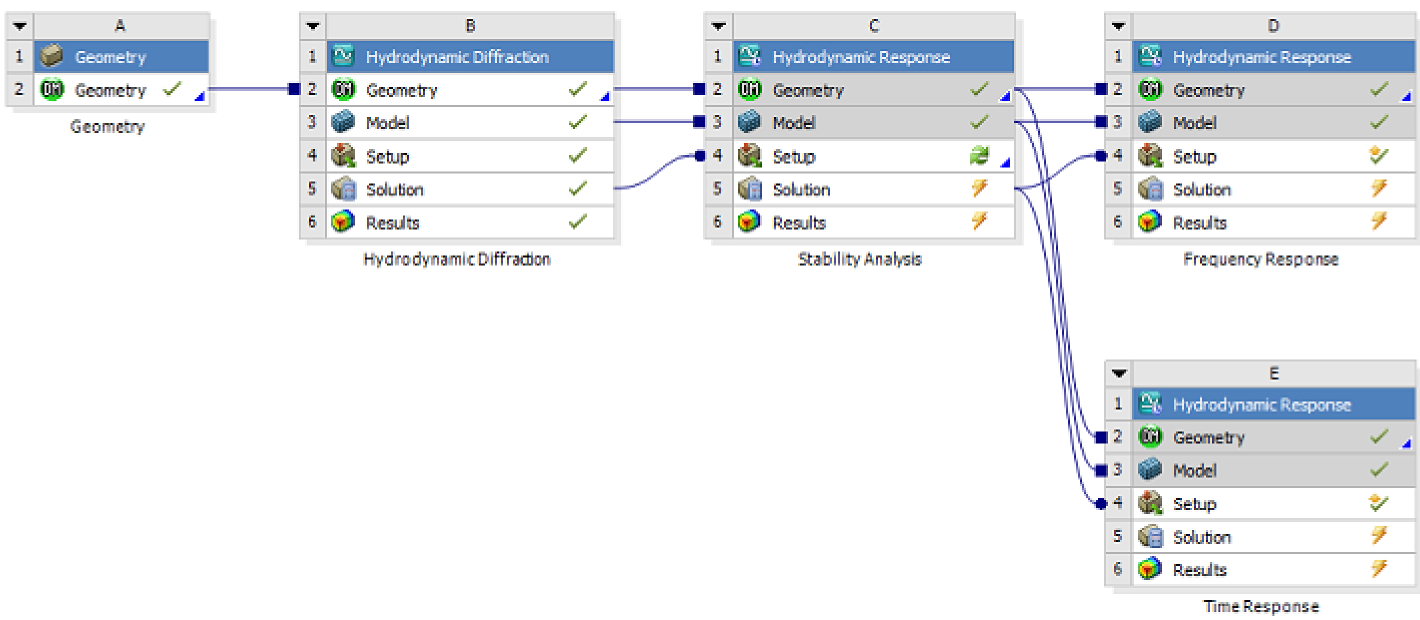
The following are some examples where duplicate can be useful:

- A row of four Connection Points are required along the edge of a pier structure. The **Position Co-ordinates** of the Connection Points vary only in X. Once the first Connection Point has been created, it is duplicated three times - only the X position will need to be redefined in the duplicates.
- A Nonlinear Catenary Cable consisting of three Catenary Sections is to be modeled. The material properties of each Section are similar, differing only in **Mass/Unit Length** and **Stiffness, EA**. Once the first Catenary Section has been defined, it is duplicated twice. The **Mass/Unit Length** and **Stiffness, EA** are then re-defined in each duplicate as necessary.
- A moored turret is connected to four Fixed Points by identical Nonlinear Catenary Cables. Each Cable is composed of three Catenary Sections, and each Section is connected by a Catenary Buoy. Once the first Cable has been fully defined, it is duplicated three times - only the **Start Fixed Point** and **End Connection Point** will need to be defined in the duplicates.

6.2.3. Propagate

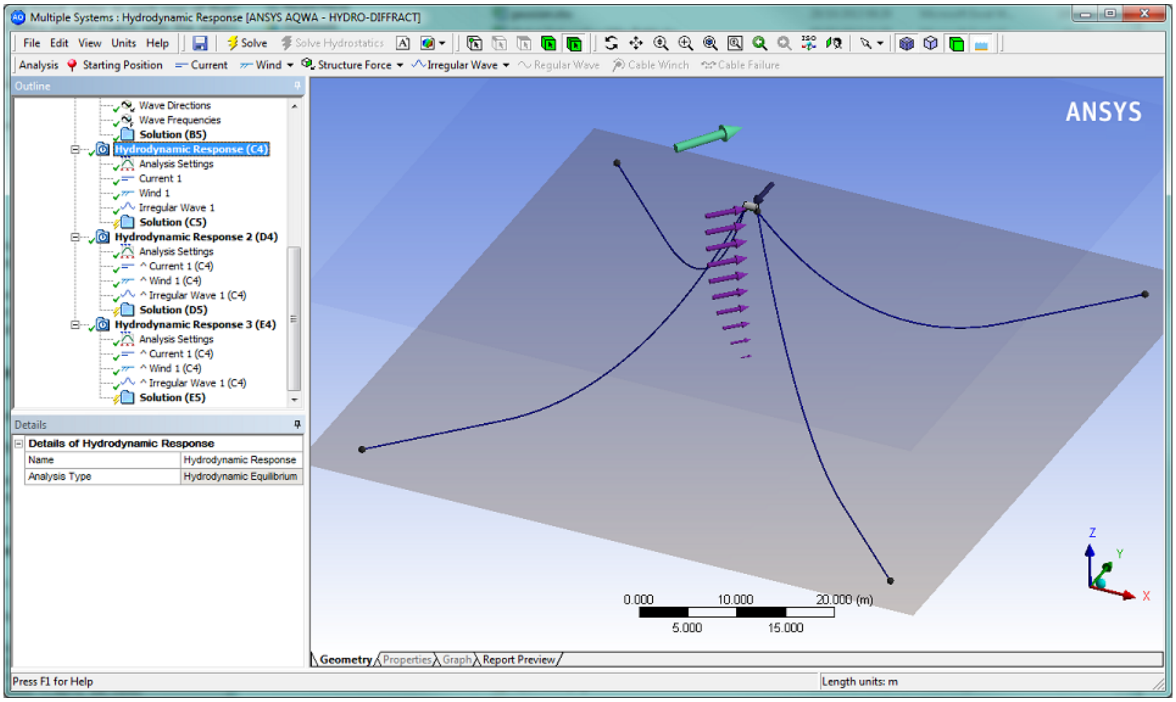
Propagating an object creates a linked copy of that object in every downstream analysis system. This operation is useful where an environment object in a Stability Analysis system should be re-used in subsequent Frequency Statistical Analysis and Time Response Analysis systems. Figure 6.1: Project Schematic of Linked Hydrodynamic Response Systems (p. 258) shows a schematic for such a case.

Figure 6.1: Project Schematic of Linked Hydrodynamic Response Systems



Linked objects created by a propagate operation cannot be edited. The **Name** of the linked object is prefixed with a ^ symbol to indicate that it is linked to an upstream environment object. Figure 6.2: Project Outline with Linked Environment Objects (p. 259) shows an example of environment objects that have been propagated into two downstream systems.

Figure 6.2: Project Outline with Linked Environment Objects



Changes to a property in the original object are reflected in the linked copies, so that the environment remains consistent throughout the different analyses. This also applies to Parameters (see Parameters (p. 269)); parameterizing a property in the original object effectively parameterizes the same property in any linked copies.

6.2.4. Break Link

The Break Link operation turns a linked copy into an unlinked copy. It is only available on linked copies of environment objects that were created using a propagate operation. Breaking the link to the original allows the properties of the copy to be modified, and subsequent changes to the original object will not be reflected in the copy.

Note:

Careful consideration should be given before the link between environment objects is broken. Typically, results from a Stability Analysis should only be used as input conditions for a subsequent Frequency Statistical Analysis or Time Response Analysis if the environments are consistent.

Once a link between objects has been broken it cannot be reinstated, and links can only be created by a Propagate operation.

6.2.5. Delete

Deleting an object removes it from the tree. If that object is an environment object that has been propagated, any linked objects in downstream systems are also deleted.

Pasted and duplicated objects, as well as environment objects that have been created by a Propagate operation but have subsequently had their link to the original object broken, are not deleted when the original object is deleted.

6.3. Displayed Positions and Features

In the Graphical Window, the displayed positions of the structures, as well as the features included in the display, depend on the current selection in the tree view outline.

The table below summarizes which positions and features are shown for each selected object, where:

- Geometric features are Fixed/Connection Points, Point Masses/Buoyancies, Discs, and Structure Axes.
- Connections are Cables, Joints, Fenders, Catenary Joints and Joint/Fender Axes.
- Environment Features are Currents, Winds, Regular/Irregular Waves, Structure Forces, and Cable Winches.

Table 6.4: Structure Positions and Features Displayed According to Tree Selection

Selection	Structure Positions	Displayed Features
Project Model Connections	Connected positions (accounting for articulations between structures)	• Included structures • Geometric features • Connections
Geometry	Geometric positions (as defined in the original CAD geometry)	• Included structures • Geometric features
Mesh		The generated mesh (if up-to-date)
HD Analysis Type or Solution		• Included structures • Analysis-specific features (for example, Wave Directions)
HD Structure Selection		All structures (including those structures that may have been excluded in Structure Selection)
HR (Stability) Analysis Type or Solution	• Unsolved: Connected positions, including Starting Positions • Solved, converged: Equilibrium positions	• Included structures • Geometric features • Connections • Environment features

<table>
<tr><th>Selection</th><th>Structure Positions</th><th>Displayed Features</th></tr>
<tr><td></td><td>Solved, unconverged: Final iteration positions</td><td></td></tr>
<tr><td>HR (Stability) Setup Objects</td><td>Connected positions, including Starting Positions</td><td>Included structures
Geometric features
Connections
Object-specific features (for example, Wind Direction)</td></tr>
<tr><td>HR (Frequency/Time) Analysis Type or Solution</td><td rowspan="2">Upstream unsolved Stability Analysis: Initial positions from upstream analysis
Upstream solved Stability Analysis: Final positions from upstream analysis
Stand-alone, unsolved: Connected positions
Stand-alone, solved: Equilibrium or final iteration positions</td><td>Included structures
Geometric features
Connections
Environment features</td></tr>
<tr><td>HR (Frequency/Time) Setup Objects</td><td>Included structures
Geometric features
Connections
Object-specific features (for example, Wind Direction)</td></tr>
</table>

The following sequence of figures are provided to illustrate the structure positions and features displayed during the Hydrodynamic Diffraction/Radiation and Hydrodynamic Response analysis of two connected vessels. These include:

- Figure 6.3: Geometric Features Defined (p. 262): After geometric features have been defined.
- Figure 6.4: Connections Defined (p. 263): After connections and articulations have been defined.
- Figure 6.5: Setting Wave Directions (p. 263): Setting wave directions.
- Figure 6.6: Stability Analysis Starting Conditions (p. 264): Starting conditions before a Stability Analysis calculation.

- Figure 6.7: Stability Analysis Final Conditions (p. 264): Final conditions after a Stability Analysis calculation.

Note that for illustration purposes:

- A hinge articulation is defined between the vessels, with an initial rotation of 15° about the hinge axis.
- A Current profile varying with depth, as well as an Irregular Wave, are added to the environment.
- A Starting Position has been applied to translate and rotate the moored vessel about the mooring Connection Point, so that the starting conditions of the Stability Analysis are closer to the expected final positions.

Figure 6.3: Geometric Features Defined

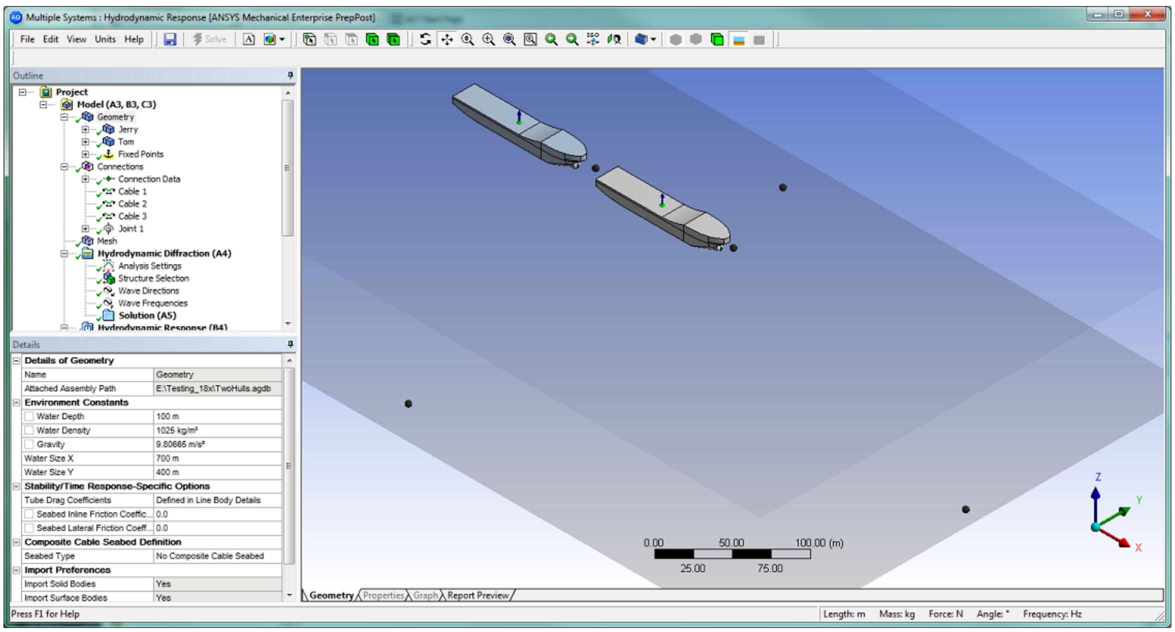


Figure 6.4: Connections Defined

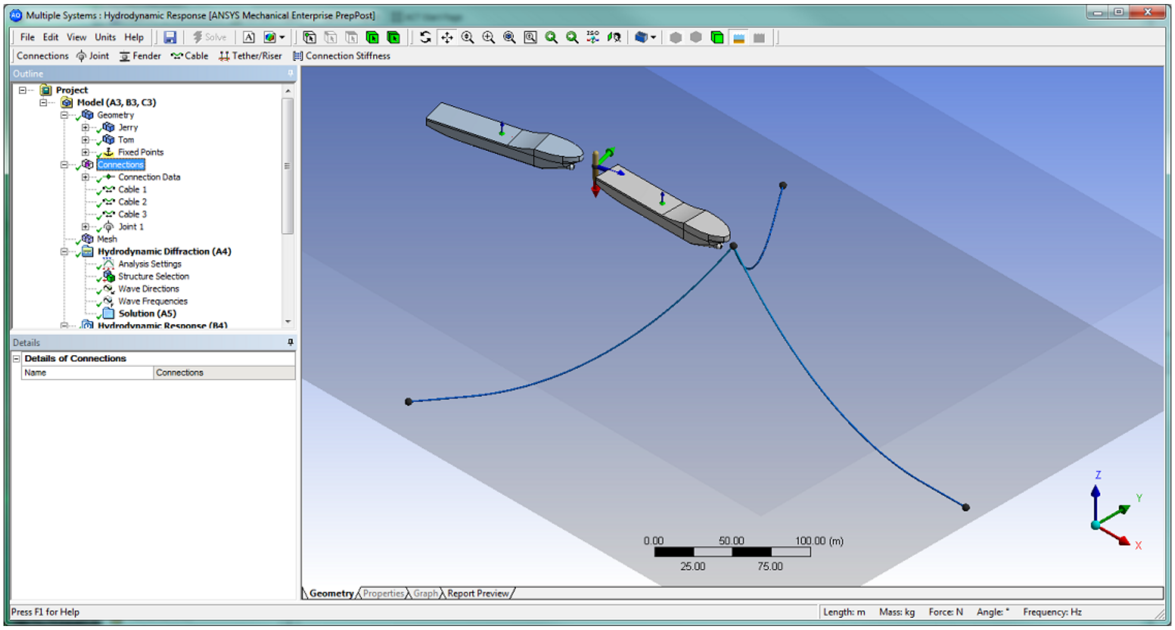


Figure 6.5: Setting Wave Directions

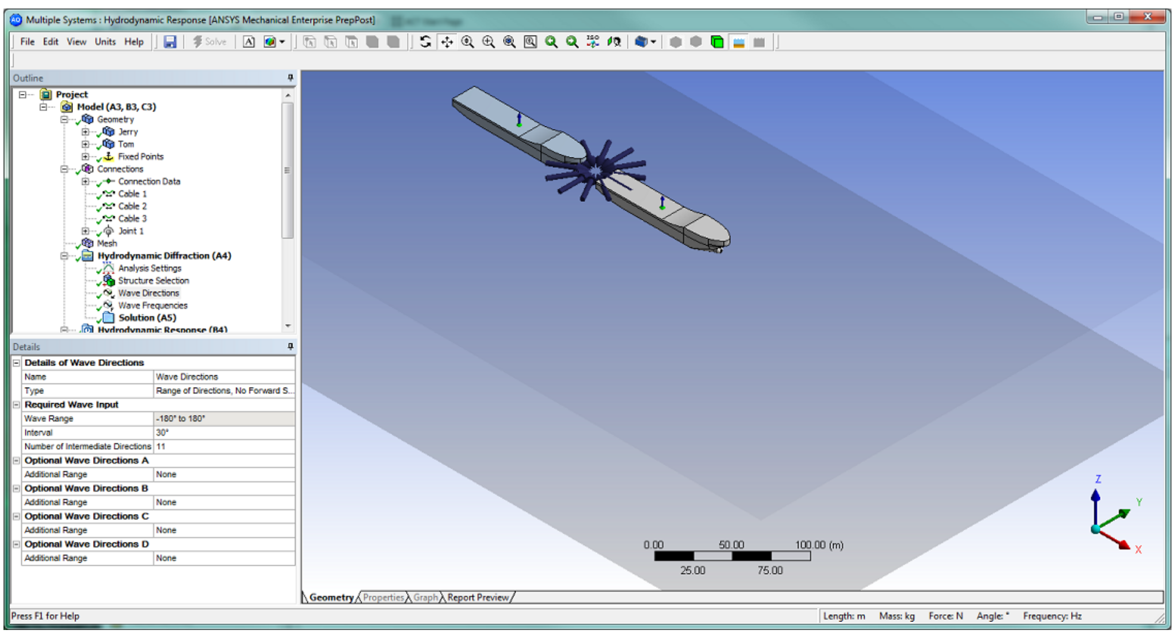


Figure 6.6: Stability Analysis Starting Conditions

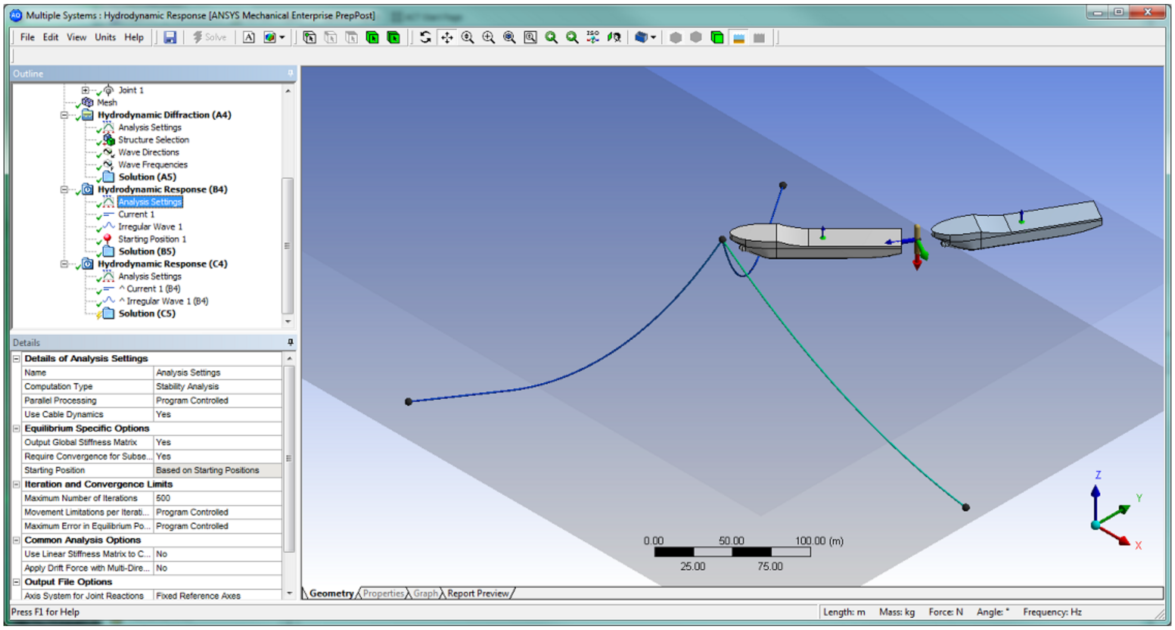
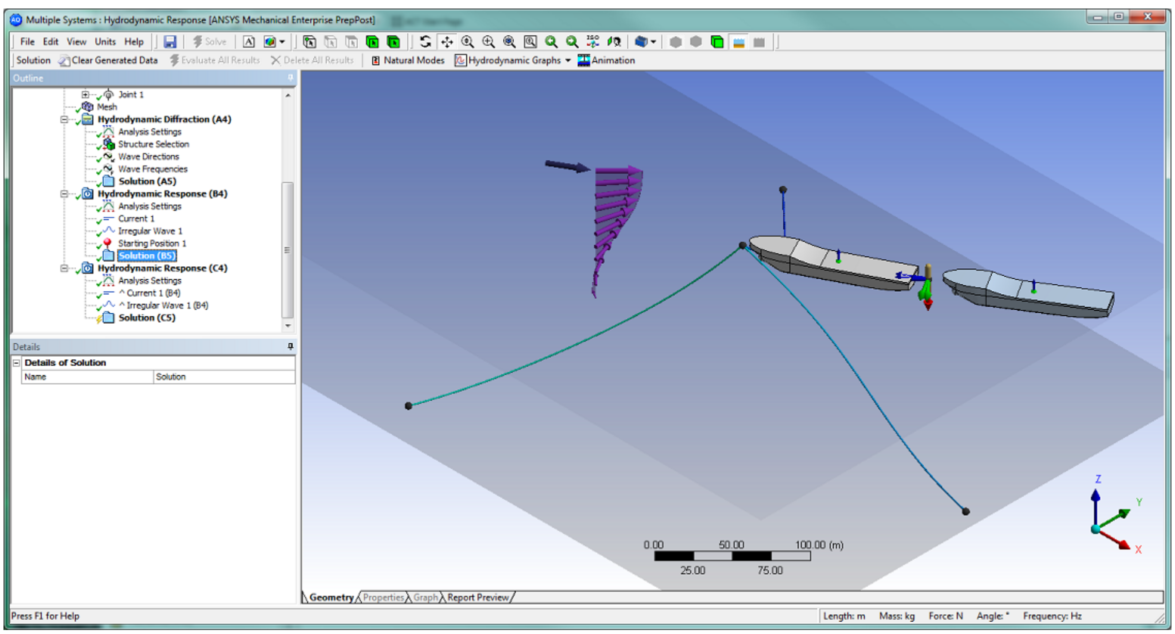


Figure 6.7: Stability Analysis Final Conditions



6.4. Setting Aqwa Application Options

Once you have created your Hydrodynamics analysis system and attached a geometry, you can open the Aqwa Editor and set some options.

1. Select **Edit** > **Options** from the Aqwa Editor window.

2. Expand the **Aqwa** item in the tree.

3. Select the **Analysis** entry. This panel allows you to set the following **Solver** options:

- **Aqwa Executable Location** - location of the executable file for the Aqwa solver. *You should not have to change this from the default setting.*
- **Severity to Force Showing of Messages** - level of error messages that will cause reporting to the Workbench Message pane.
- **Maximum Program Controlled Number of Cores** - default number of parallel processing cores to use when the Analysis Settings Parallel Processing option is set to Program Controlled. For more information on how this value is used, see Setting Aqwa Parallel Processing Options (p. 265).

4. Select the **Report** entry. This panel allows you to customize the following report generation options:

   - **Figure Options** - whether figures sould be included in the report, and the default figure size.
   - **Graphics Options** - the default image size and resolution.
   - **Table Options** - which tables to include in the report, and the maximum number of rows and columns in any table. If the table size is greater than the maximum *N* rows/columns defined here, the table in the report will contain (approximately) equally-spaced entries up to *N* rows/columns.
   - **Customization** - specify the location of a folder for your own report-generating scripts. The folder contents should match that of `installation_directory\aisol\aqwa\AQWAPages\report\`.

5. When you are finished, click **OK**. Click **Cancel** to discard your changes.

Note:

Option settings within a particular language are independent of option settings in another language. If you change any options from their default settings, then start a new Workbench session in a different language, the changes you made in the original language session are not reflected in the new session. You are advised to make the same option changes in the new language session.

6.4.1. Setting Aqwa Parallel Processing Options

Aqwa can take advantage of multi-core machines to improve computational speed during the solution stages. For more information, see Aqwa Parallel Processing Calculation in the *Aqwa Theory Manual*. There are several ways to determine and specify the number of cores to be used in a given analysis.

When the Aqwa Editor is opened for the first time after installation, it automatically detects the number of processing cores that exist on the node on which it is run. The number is set as the default value for the **Maximum Program Controlled Number of Cores** option, which can be found in the **Edit > Options > Aqwa > Analysis** menu.

Note:

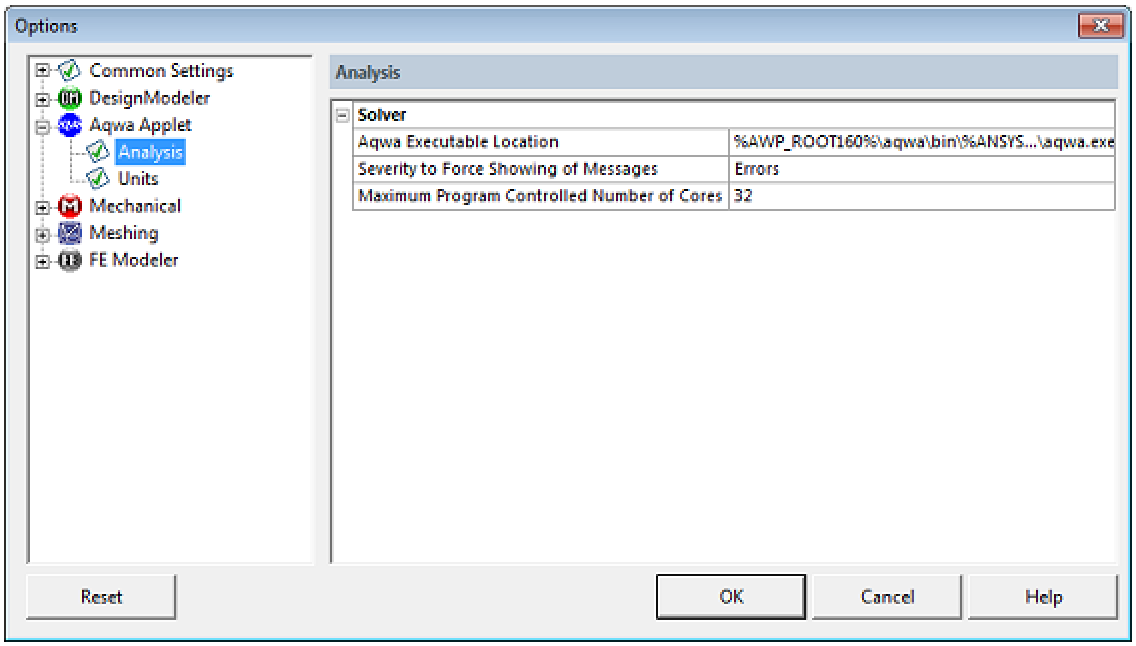
The **Maximum Program Controlled Number of Cores** is set to the maximum number of physical cores on your machine. You cannot request more than this number of cores.

If you manually set **Maximum Number of Program Controlled Cores** to 0, or a number greater than the total number of cores in the node, the Aqwa Editor will reset the value of **Maximum Number of Program Controlled Cores** to the default value.

Note:

The **Number of Processors** option, found under **Edit > Options > Common Settings > Parallel Processing**, is not used by the Aqwa Editor at any stage.

If you click **Reset** in the **Edit > Options** window, the **Maximum Number of Program Controlled Cores** will be set to 0 and the behavior described above will apply.



Within each analysis system, under **Analysis Settings**, there is a **Parallel Processing** drop-down menu. This menu has three options: **Serial**, **Manual Definition**, and **Program Controlled** (default).

Setting **Parallel Processing** to **Program Controlled** causes the Aqwa Editor to automatically determine how many cores to request during the solve process. The number of requested cores varies depending on the **Computation Type**, and on whether the analysis includes dynamic cables and/or tethers, etc. See Aqwa Parallel Processing Calculation in the *Aqwa Theory Manual* for a table of actual number of cores used in different Aqwa analysis cases.

Setting **Parallel Processing** to **Serial** causes the Aqwa Editor to request a single core for that analysis, regardless of **Computation Type**.

Setting **Parallel Processing** to **Manual Definition** shows an additional **Requested Number of Cores** option. This field accepts any value greater than 0 and less than or equal to the number of physical cores in the node. Setting a value outside of this range will result in an error and will prevent the solve process from executing. Setting a value that is higher than the value that would be used if

Parallel Processing were set to **Program Controlled** will result in an informational message in the log.

Note:

Changing Parallel Processing or Requested Number of Cores does not invalidate an up-to-date solution.

The **Maximum Program Controlled Number of Cores** option is a global setting for the machine, it is not saved within a project file or reset between projects. The **Parallel Processing** and **Requested Number of Cores** options are project specific and are saved in the project file.

Although the number of available HPC licenses may change due to the activity of other users within your organization, the Aqwa Editor itself does not perform the licensing check. The Aqwa solver performs the check when you execute the solve process. The number of cores used in the computation may be less than the number requested (automatically or manually), depending on the number of available licenses.

Chapter 7: Aqwa Common Features

There are a number of features common to all of the Aqwa analysis systems. The following are discussed in this chapter:

7.1. Generating Reports

7.2. Parameters

7.3. Comments, Images, and Figures

7.1. Generating Reports

You can click on the **Report Preview** tab in the main window pane to generate a summary of all of the objects in your Outline. Once started, the report generation process must run to completion. Avoid clicking anywhere in the Workbench window while the report is generating because it will stop the report and may cause an error.

The Details information for each object appears as tables in the report. Figures and images appear as specified in the Outline. Charts that appear in the outline are also included.

When you are viewing the Report Preview tab, several related buttons will appear in your toolbar. Use the **Print** button to print a copy of the report. The **Publish** button will save the report in one of three formats:

- .mht, a web page archive format that can be opened by Internet Explorer 5.5 or later
- .html file with graphics in a sub directory
- .html file with graphics in the same directory

Click **Publish** then select the format in which you want to save from the **Save as type** drop down list.

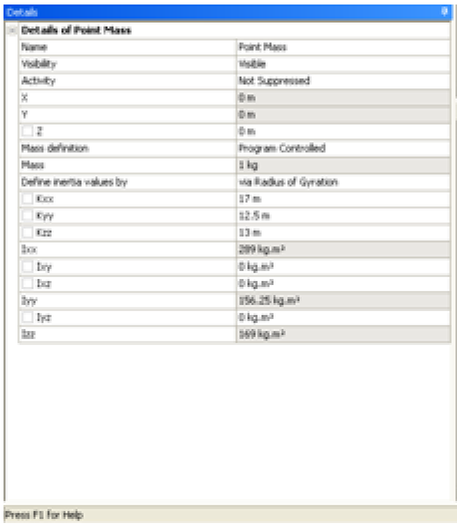
Use the **Send To** button to send the report to an email recipient (.mht file format only), Word, or PowerPoint. The document is opened in Word as any published HTML file would be. Only the images contained in the report are imported in PowerPoint; no other information is imported.

See Report Preview for more information on generating reports.

7.2. Parameters

Certain quantities in your Hydrodynamic analysis can be exposed to the Workbench environment as parameters. A parameter may be exposed by selecting the checkbox next to its Detail field, which shows a 'P' when selected. For an input parameter, the value set in the field at the moment when the quantity is parameterized becomes the value for that quantity in the first (**Current**) design point. The field is then read-only, and any changes to the value must be made through the design point table. Parameters can be used in post-processing to conduct optimization analysis and "what-if" scenarios. See Working with Parameters and Design Points for more information about working with parameters.

Figure 7.1: Example of Parameters in the Details View



When an item in the project tree is suppressed or deleted, any parameters and design point values associated with it are removed from the design point table.

If you have parameterized a property which then goes out of context due to changes made to another property, the parameter and its design point values are permanently removed from the design point table. For example, if you have parameterized the **Significant Wave Height** of an **Irregular Wave** with **Wave Type** of **JONSWAP (Hs)**, and then change the **Wave Type** to **JONSWAP (Alpha)**, the **Significant Wave Height** parameter is removed.

7.3. Comments, Images, and Figures

You can insert Comment objects, Image objects, and Figure objects under various parent objects in the tree to add text or graphical information that pertains specifically to those parent objects. These objects can all be deleted from their parent objects by right clicking on them and selecting **Delete**.

7.3.1. Comments
7.3.2. Images
7.3.3. Figures

7.3.1. Comments

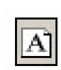
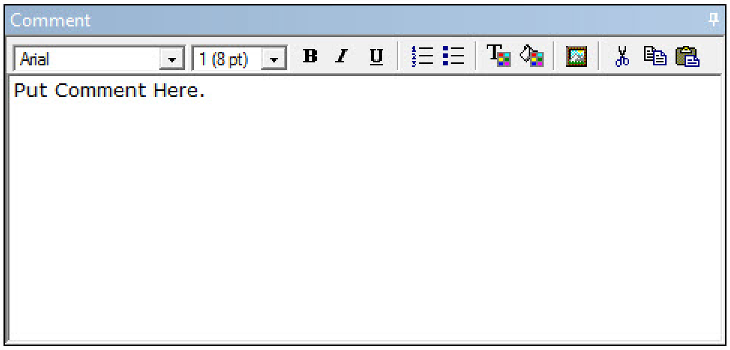
You can insert a Comment object in the tree by clicking on the **Comment** button in the toolbar . A window pane will open below the main graphics pane displaying a text box where you can enter the comment. Any time you click on the Comment object in the tree, this comment edit pane will open. You can enter text, do some basic formatting, change the background and foreground colors, and add pictures to the comment. In the Details view, you can enter an author name in the **Name** field. Comments will appear in a generated Report under the parent object in the tree.

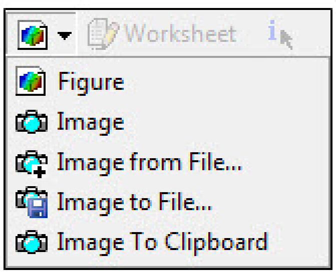
Figure 7.2: Comment Edit Pane



7.3.2. Images

You can insert an Image object in the tree or save the current Model view to a file by clicking on the **New Figure or Image** button  in the toolbar.

Figure 7.3: New Figure or Image Menu



From the drop down menu, you can select:

- **Image** to insert an Image object in the tree containing a snapshot of the Model's current view in the Geometry window.
- **Image from File...** to insert an image from a file as an Image object in the tree. Browse to an existing .bmp, .jpg, or .png file to insert the image.
- **Image to File...** to save the Model's current view. Browse to the desired location, and chose the file format: .png, .bmp, .jpg, .tif, or .eps.

In the Details view, you can enter an annotation for the image in the **Text** field. Image objects will appear in a generated Report under the parent object in the tree.

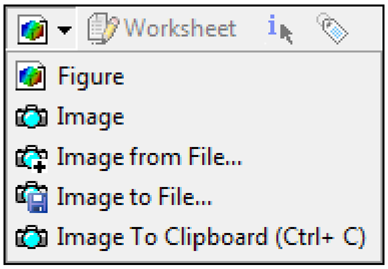
Note:

If you are running on Windows 7, you must be using a Basic Theme for your desktop in order to capture an image. If you are using an Aero Theme, a warning dialog box will appear when inserting the Image or Image to File objects, and the image will not be properly created.

7.3.3. Figures

The Figure object allows you to capture a graphic displayed for a particular object in the Geometry window. A Figure object can be further manipulated (rotated, for example), unlike an Image object which is a static screen shot of the current model view, or an imported static figure. You can insert a Figure object in the tree by clicking on the **New Figure or Image** button in the toolbar. From the drop down menu, select **Figure**.

Figure 7.4: New Figure or Image Menu



In the Details view, you can enter a **Name** for the Figure object. Figures will appear in a generated Report under the parent object in the tree.

Figures allow you to capture result contours, mesh previews, graphs, etc. for later display in the Report. Any object that displays 3D graphics may contain figures. View settings maintained by a figure include:

- camera settings
- result toolbar settings
- legend configuration

A figure's view settings are fully independent from the global view settings. Figures always display the data of their parent object. For example, following a geometry Update and Solve, a result and its figures display different information but reuse the existing view and graphics options.

Chapter 8: Hydrodynamic Add-ons

Ansys provides Add-ons to provide additional functionality and system interoperability for the Mechanical Application.

The following Add-ons are provided for hydrodynamic analyses:

8.1. The Hydrodynamic Pressure Add-on

8.2. The Offshore Add-on

8.3. The Aqwa Co-simulation Add-on

8.1. The Hydrodynamic Pressure Add-on

The external surface and internal tank pressures, Morison loads, and structural accelerations calculated in a Hydrodynamic Diffraction or time domain Hydrodynamic Response analysis can be transferred to panel and line elements in a Static Structural analysis, using the Hydrodynamic Pressure Add-on, through a link on the Workbench Project Schematic page. This removes the need to create, manipulate, and run files external to Workbench.

Note:

Moonpool pressures can also be transferred, but from Hydrodynamic Diffraction to Static Structural only.

When mapping from a Hydrodynamic Diffraction system, multiple wave phase angles can be analyzed in a single Static Structural calculation, to provide a clear picture of finite element results due to hydrodynamic loading over the whole wave cycle. Similarly, for a Hydrodynamic Diffraction analysis including multiple Wave Frequencies or Wave Directions, pressures and loads can be transferred for some or all of those frequency/direction pairs in a single Static Structural analysis.

For pressure mapping from a time domain Hydrodynamic Response system, external surface and internal tank pressures, as well as Morison loads, can be transferred for: a single time step; a range of time steps; or for all time steps for which time domain pressures have been calculated. In addition, cable forces, joint forces/moments and tether forces/moments calculated in the hydrodynamic model can be applied to the structural model.

8.1.1. Introduction

The Hydrodynamic Pressure Add-on is designed to facilitate the transfer of hydrodynamic pressures and Morison loads calculated in a Hydrodynamic Diffraction or time domain Hydrodynamic Response system to a Static Structural finite element analysis.

When mapping from a Hydrodynamic Diffraction system the hydrodynamic pressure terms are linear (first order), and include incident (Froude-Krylov), diffracted, radiated and hydrostatic-varying components. Hydrostatic pressure may be added if required. The wave direction and frequency pairs are selected from those included in the Hydrodynamic Diffraction calculation; an incident wave amplitude is also specified. The load mapping can be carried out either at a single specified wave phase angle, over multiple phase angles covering the whole of the wave cycle, or for real and imaginary components (in linear cases). Similarly, the mapping may be performed either for a single frequency/direction pair or, where the Hydrodynamic Diffraction system includes multiple Wave Frequencies or Wave Directions, over some or all of those frequency/direction pairs. The summed pressure terms are mapped onto triangular and quadrilateral panel elements in the Static Structural mesh, either by interpolation, or by direct evaluation of the diffracting panel source strengths at the Static Structural mesh node positions.

The Hydrodynamic Pressure Add-on also allows the Hydrodynamic Diffraction first order (Froude-Krylov, added mass) and second order (viscous drag) forces acting on Line Bodies (element types BEAM188 and PIPE288) to be included in a Static Structural analysis. The effect of ocean current may be accounted for in the drag calculation.

For pressure mapping from a time domain Hydrodynamic Response system, the hydrodynamic pressure terms are linear for the diffracted and radiated components and nonlinear for the incident (Froude-Krylov) and hydrostatic components. These latter terms are estimated under the instantaneous wave surface in the time domain calculation. The first order (Froude-Krylov, added mass) and second order (viscous drag) forces on Line Bodies, as well as the forces and moments on Morison disc elements, can also be mapped from the hydrodynamic model. The pressure mapping can be carried out at a single specified time step, over a range of time steps, or for all time steps included in the time domain Hydrodynamic Response pressure output. The total pressure terms are mapped by interpolation on to triangular and quadrilateral panel elements in the Static Structural mesh.

The Hydrodynamic Pressure Add-on allows the instantaneous cable forces, joint forces/moments and tether forces/moments, calculated in a time domain Hydrodynamic Response system, to be mapped on to selected locations of the structural model.

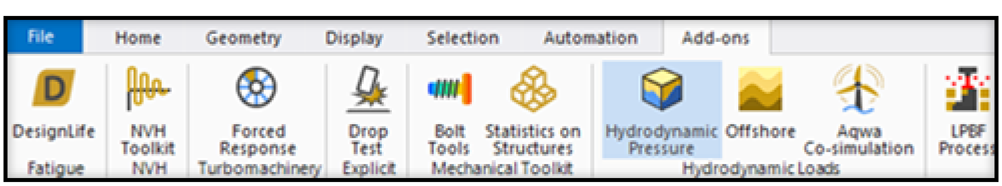
For both Hydrodynamic Diffraction and time domain Hydrodynamic Response pressure mapping, the resultant translational and rotational accelerations about the structure center of gravity are determined for the balance of inertial loads in the Static Structural analysis. Standard Earth Gravity is added to the translational accelerations where hydrostatic pressure is included in the mapping.

8.1.2. Loading the Hydrodynamic Pressure Add-on

The Hydrodynamic Pressure Add-on is automatically loaded on the creation of a Hydrodynamic Diffraction system or a Hydrodynamic Response system.

To make its capabilities available where the Add-on has been unloaded in a previous session, click the **Hydrodynamic Pressure** icon in the **Add-ons** Ribbon. The icon will be highlighted in blue, indicating that the Add-on is loaded.

Figure 8.1: Hydrodynamic Pressure Add-on Showing Loaded Status



8.1.3. Setting up your Workflow in Workbench

The Solution cell of a Hydrodynamic Diffraction or Hydrodynamic Response system can be connected to the Setup cell of a Static Structural system. Typical workflows are shown in Figure 8.2: Typical Workflow for Hydrodynamic Diffraction Pressure Mapping (p. 275) and Figure 8.3: Typical Workflow for Time Domain Hydrodynamic Response Pressure Mapping (p. 276). It is not necessary to have different Geometry sources in each system, but for the typical use case the Static Structural model will include internal and/or other structural components that are not required for the Hydrodynamic Diffraction or time domain Hydrodynamic Response calculation.

Once the link has been created between the Hydrodynamic Diffraction/time domain Hydrodynamic Response and Static Structural systems, it is always necessary to right-click the Hydrodynamic Solution cell and select Update. This operation manages the transfer of data files between the analyses.

Note:

When some hydrodynamic data has already been imported into the Static Structural system, and the Hydrodynamic Solution is subsequently updated, the Static Structural Setup cell must be refreshed. In Mechanical, it is essential to click the Hydrodynamic Pressure object after this Refresh operation to utilize the new hydrodynamic data.

Figure 8.2: Typical Workflow for Hydrodynamic Diffraction Pressure Mapping

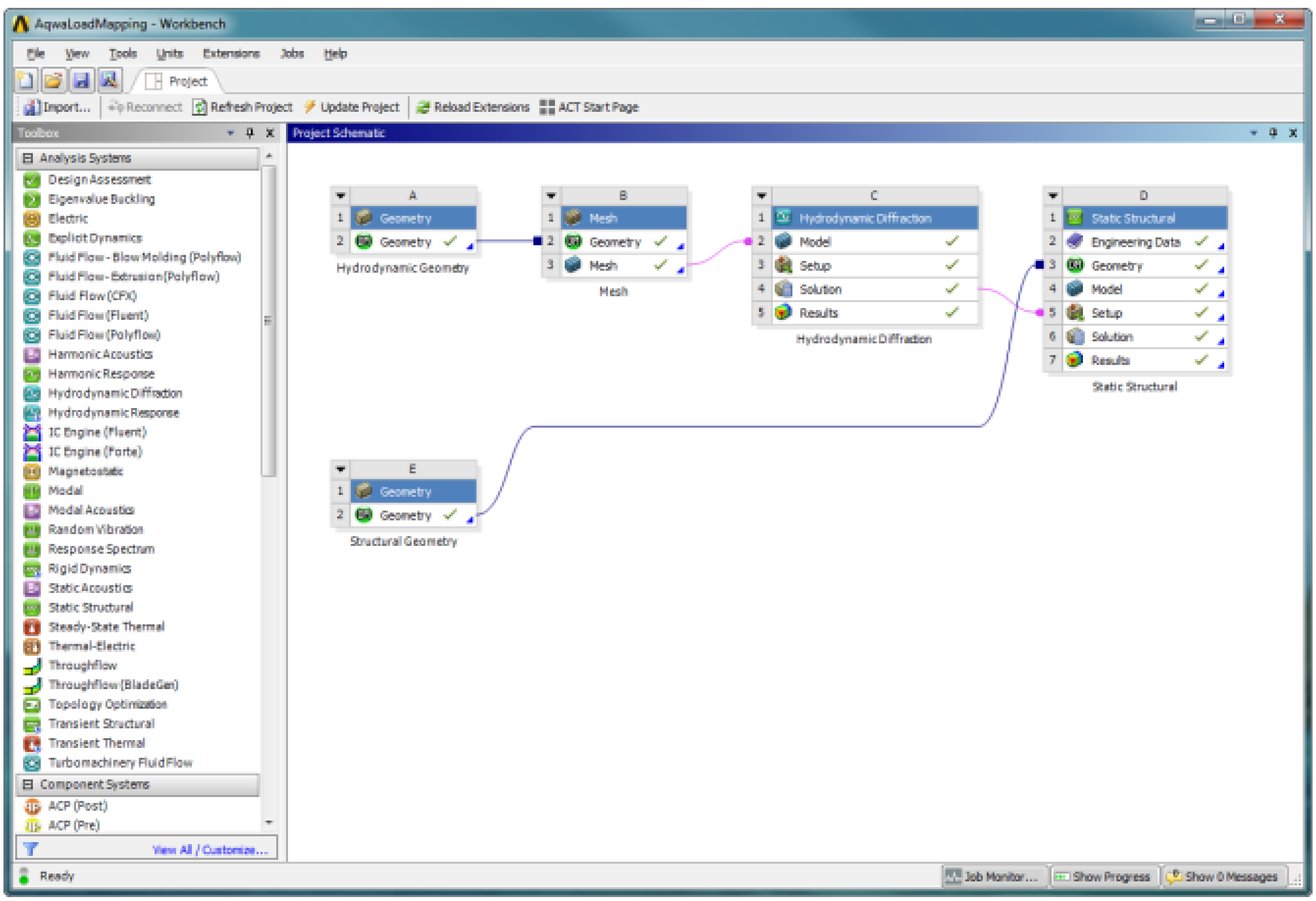
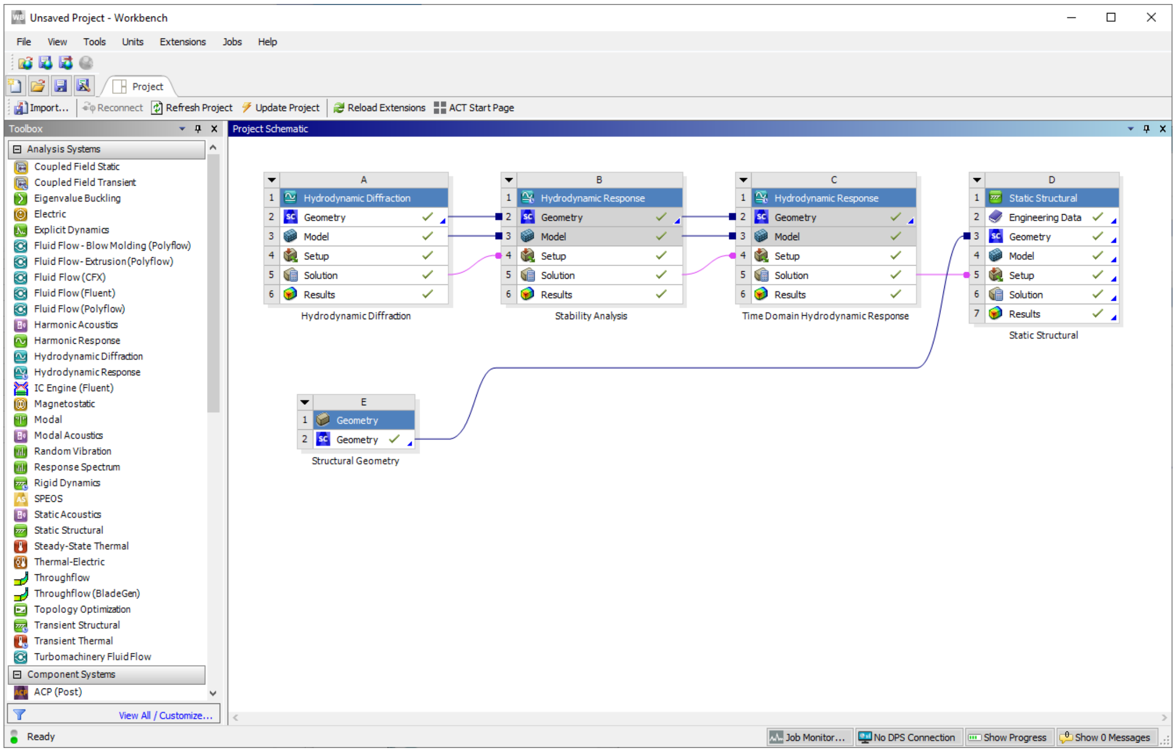


Figure 8.3: Typical Workflow for Time Domain Hydrodynamic Response Pressure Mapping



8.1.4. Importing Hydrodynamic Diffraction Pressures and Loads to Mechanical

Opening the Static Structural system in the Mechanical editor, you will find an additional tab that gives you access to the Hydrodynamic Pressure Add-on.

Clicking the 

icon adds a Hydrodynamic Pressure object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Hydrodynamic Pressure** from the context menu.

Note:

It is only possible to add one Hydrodynamic Pressure object to a Static Structural analysis.

8.1.4.1. Configuring the Hydrodynamic Pressure Object

The Details of Hydrodynamic Pressure panel provides options that allow you to configure the Hydrodynamic Pressure object. These are described below.

Figure 8.4: Details of Hydrodynamic Pressure

Details of "Hydrodynamic Pressure"	
Definition	
Activity	Unsuppressed
Structures in Hydrodynamic Model	3
Load Configuration	
Wave Case Definition	Single Wave Case
Wave Direction	150.0°
Wave Frequency/Period	0.25 Hz / 4.0 sec
Incident Wave Amplitude	1 m
Wave Position	Specified
Specified Phase Angle	0 °
Mapping Configuration	
Model Type for Mapping	External Surfaces Only
Pressure Mapping	Direct
Include Incident Wave	Yes
Include Diffracted Wave	Yes
Include Radiated Wave	Yes
Include Hydrostatic Pressure	No
Include Hydrostatic Varying	Yes
Include Second Order Terms	No
Axis Transformation	
Static Structural Position	Matches Hydrodynamic Diffraction Analysis
Imported Pressures	
Minimum Pressure	-50042.034655 Pa
Maximum Pressure	43792.947952 Pa
Advanced Options	
Perform Mass/Inertia Check	Yes, Selected Bodies Only
Write APDL Commands	Into Separate Files

The **Activity** field allows you to set the suppression state of the Hydrodynamic Pressure object. Set to Suppressed if you want to exclude the object from the analysis.

The **Structures in Hydrodynamic Model** field displays the number of structures included in the pressure database from the upstream Hydrodynamic Diffraction system. It is possible to transfer pressures for one, several, or all of these structures. One Hydrodynamic Structure (p. 285) object is automatically added to the analysis when you create the Hydrodynamic Pressure object, but you can create additional Hydrodynamic Structure objects up to the number of structures in the hydrodynamic model.

8.1.4.1.1. Load Configuration

The **Wave Case Definition** option allows you to analyze either single or multiple wave conditions within a single Static Structural analysis.

When **Wave Case Definition** is set to Single Wave Case, the **Wave Direction** and **Wave Frequency/Period** fields are used to select a frequency/direction pair at which the hydrodynamic pressures acting on the structure will be evaluated. If a forward speed has been defined in the Hydrodynamic Diffraction analysis, the **Encounter Frequency** for this frequency/direction pair is also shown.

Setting **Wave Case Definition** to Single Frequency, All Directions, and selecting a single **Wave Frequency/Period** value, causes the Static Structural analysis to be performed for all wave directions at that wave frequency. Similarly, a **Wave Case Definition** of Single Direction, All Frequencies allows you to analyze over all wave frequencies for a given **Wave Direction**.

To perform the Static Structural analysis over all frequency/direction pairs in the Hydrodynamic Diffraction analysis, set **Wave Case Definition** to All Frequencies, All Directions.

For all wave case configurations, an **Incident Wave Amplitude** should be specified.

The **Wave Position** allows you to choose between single or multiple wave phase angles in the Static Structural analysis. When this option is set to Specified, use the **Specified Phase Angle** field to define the (single) wave phase angle. When **Wave Position** is set to Sequence, the Static Structural analysis will be carried out over multiple phase angles spaced equally over the wave cycle (from 0° to 360°); use the **Number of Phase Increments** field to set the number of steps. Alternatively, setting **Wave Position** to 0° (Real) and 90° (Imaginary) causes the analysis to be carried out over only these two phase angles. This last option is only applicable to linear cases (no beam loads). The phase angle is with respect to the COG of the first structure in each hydrodynamic interaction structure group.

Where multiple phase angles and/or multiple frequency/direction pairs are to be analyzed, the Static Structural Analysis Setting **Number of Steps** field will be updated automatically. For example: for a Hydrodynamic Diffraction analysis containing 20 wave frequencies and 13 wave directions, with the **Wave Case Definition** set to All Frequencies, All Directions and **Number of Phase Increments** set to 12, the total **Number of Steps** in the Static Structural analysis will be 20 x 13 x 12 = 3120.

The additional options **Display for Wave Case** and **Display Pressures/Accelerations At** can be used to set the wave case and wave phase angle at which the mapped pressures (in the graphical window) and output values (in the Details panel, described in Output: Imported Pressures and Loads (p. 279)) are shown.

When **Model Type for Mapping** includes Line Bodies, a constant or profiled current can be included by creating an Ocean Current (p. 315) object in the Static Structural analysis and selecting that object from the **Include Current** drop-down menu.

8.1.4.1.2. Mapping Configuration

Model Type for Mapping indicates the types of loads that will be mapped from the Hydrodynamic Diffraction analysis to the Static Structural analysis. Generally, it is possible to map pressures onto surfaces (Surface Bodies and the outer faces of Solid Bodies) and distributed loads onto beams or pipes (Line Bodies).

The meshes in the Static Structural and Hydrodynamic Diffraction systems are generally not coincident, which necessitates some approach to map the hydrodynamic pressures onto the structural mesh. The **Pressure Mapping** option can be set to Interpolated or Direct.

Using the interpolated method, hydrodynamic pressures at each node of the selected surfaces in the structural mesh are interpolated from the element-centered pressures calculated in the Hydrodynamic Diffraction analysis. This method can be used for models that include Line Bodies in the load mapping selection. However, the accuracy of the method is inherently limited by the interpolation of the pressure data and is dependent on the resolution of the mesh in the Hydrodynamic Diffraction analysis. Furthermore, the interpolated method cannot be used for mapping onto the outer surfaces of Solid Bodies, or where the Hydrodynamic Diffraction analysis contains more than one structure.

Where the direct method is used, hydrodynamic pressure components at the position of each selected node are evaluated directly from the diffracting panel source strengths calculated in the Hydrodynamic Diffraction analysis. The direct method permits mapping for multiple hydrodynamic structures, as well as mapping onto the outer surfaces of Solid Bodies. This method avoids the loss of accuracy due to interpolation, but it cannot be used to map loads onto Line Bodies.

Internal Tank Pressures always use an interpolation method for mapping between the hydrodynamic and structural models. Moonpool Pressures always use a direct mapping method. These are independent of the **Pressure Mapping** setting described above, and are also valid for mapping from multiple hydrodynamic structures.

Regardless of the selected method, the mapped hydrodynamic loads will always include incident, diffracted and radiated components, as indicated by the **Include Incident**/**Diffracted**/**Radiated Wave** fields. Hydrostatic pressure can be optionally added by setting **Include Hydrostatic Pressure** to Yes. The hydrostatic-varying pressure component is always included when the interpolated method is used but is optional for the direct method. Second order pressure terms on diffracting panels are not calculated, but Line Body loads do include a second order viscous drag component.

8.1.4.1.3. Axis Transformation

Although you can share one Geometry source between the Hydrodynamic Diffraction and Static Structural systems, in the typical use case, the Static Structural model will include internal and/or other structural components that are not required for the Hydrodynamic Diffraction calculation. Depending on the context, the geometries in the Static Structural and Hydrodynamic Diffraction systems may therefore employ different axis systems. To account for this, it is possible to define an axis transformation from the Static Structural analysis to the Hydrodynamic Diffraction analysis.

Where the axis systems are consistent between the two analyses, the **Static Structural Position** may be set as Matches Hydrodynamic Diffraction Analysis. Otherwise, this option should be set to Differs from Hydrodynamic Diffraction Analysis, and **Structure Position/Rotation Offsets** should be defined for the translational and rotational freedoms. Offsets are defined relative to the Hydrodynamic Diffraction global axis system (FRA).

8.1.4.1.4. Output: Imported Pressures and Loads

When **Model Type for Mapping** includes External Surfaces, the **Minimum** and **Maximum Pressures** on diffracting panel elements are reported. When Model Type includes Internal Tanks, the **Minimum** and **Maximum Internal Tank Pressures** are provided. When Model Type includes Moonpools, the **Minimum** and **Maximum Moonpool Pressures** are stated. When Model Type includes Line Bodies, the **Minimum** and **Maximum Beam Loads** (as Force per unit Length) are also shown. These quantities are evaluated over all of the surfaces and line bodies included for mapping (that is, across all structures); the Hydrodynamic Structure (p. 285) object shows maxima and minima per structure.

Where applicable, values are displayed for the wave case and phase angle selected in **Display for Wave Case** and **Display Pressures/Accelerations At Phase Angle**, respectively.

8.1.4.1.5. Advanced Options

The Hydrodynamic Pressure object accesses the material properties of each Body so that it can estimate the structural mass, inertia, and center of gravity (CoG) position, and compare these to the corresponding properties from the hydrodynamic model. Generally, the force and moment reactions on any defined boundary conditions should be relatively small in the Static Structural system, and checking this helps ensure the applied loads and accelerations are well-balanced.

The **Perform Mass/Inertia Check** setting can be used to adjust the behavior of the Hydrodynamic Pressure object in this respect. This setting has 4 options:

- **Yes, Entire Assembly**: The mass/inertia/CoG of the entire assembly, including all Bodies in the Geometry, will be calculated and compared to the corresponding properties of the selected structure in the upstream hydrodynamic analysis. This option is invalid if there is more than one Hydrodynamic Structure in the analysis.

- **Yes, Selected Bodies Only**: The mass/inertia/CoG will be calculated and compared only for the specific Bodies that are included in the **Structural Acceleration Application** scope for each Hydrodynamic Structure.

- **No, Use Hydrodynamic Model CoG Position**: The mass/inertia/CoG will not be calculated. The accelerations included by the Hydrodynamic Pressure object will be applied about the CoG position of the selected structure(s) in the upstream hydrodynamic analysis (accounting for any defined Axis Transformation (p. 279)). These accelerations will be applied to the specific Bodies that are included in the **Structural Acceleration Application** scope for each Hydrodynamic Structure. You must ensure that the mass properties are consistent between the hydrodynamic and structural models.

- **No, Exclude Accelerations**: The mass/inertia/CoG will not be calculated, and the Hydrodynamic Pressure object will not apply any accelerations in the structural analysis. You must ensure that the mass properties are consistent between the hydrodynamic and structural models.

In some cases, a Body's material properties may be inaccessible. For example, where Ansys Composite PrepPost (ACP) has been used to define a composite material, or where the structural analysis employs a material with no density value defined. For these cases, the **Perform Mass/Inertia Check** option must be set to **No, Use Hydrodynamic Model CoG Position** or **No, Exclude Accelerations**. The force/moment reactions on any defined boundary conditions must be checked carefully.

8.1.4.2. Generating Hydrodynamic Pressures and Inertial Loads

Once the Hydrodynamic Pressure object has been configured as required, and at least one Hydrodynamic Structure (p. 285) has been defined, right-click the Hydrodynamic Pressure object and select Generate to start the load transfer process. This involves two main steps:

- Using the Ansys solver to determine the structural mass properties, which are compared to the mass properties from the Hydrodynamic Diffraction calculation;

- Reading the output of the Hydrodynamic Diffraction calculation to determine hydrodynamic pressures at the structural mesh nodes and calculate translational and rotational accelerations to balance inertial loads.

Note:

The Hydrodynamic Pressure Add-on will take into account any difference in the unit systems that are employed in the Hydrodynamic Diffraction and Static Structural systems.

Where the attached Geometry includes a large number of parts or bodies (1000 or more), the Mechanical UI may be noticeably unresponsive for some time during Hydrodynamic Pressure Generate and Solve operations. If necessary, you can monitor progress via the Log File, accessible through the Workbench menu bar: **Add-ons** → **View Log File**.

The transferred pressures are displayed in the graphical window when you select a Hydrodynamic Structure in the Outline tree (Figure 8.5: Hydrodynamic Pressures in the Static Structural System (p. 282)), and can be compared to the Structure Panel Pressures (Figure 8.6: Hydrodynamic Pressures in the Hydrodynamic Diffraction System (p. 283)) shown in the **Pressures and Motions** result object in the upstream Hydrodynamic Diffraction system.

Note:

The pressure **Component Selection** in the **Pressures and Motions** object may need to be matched to those included in the **Hydrodynamic Pressure** object for such a comparison to be made.

Figure 8.5: Hydrodynamic Pressures in the Static Structural System

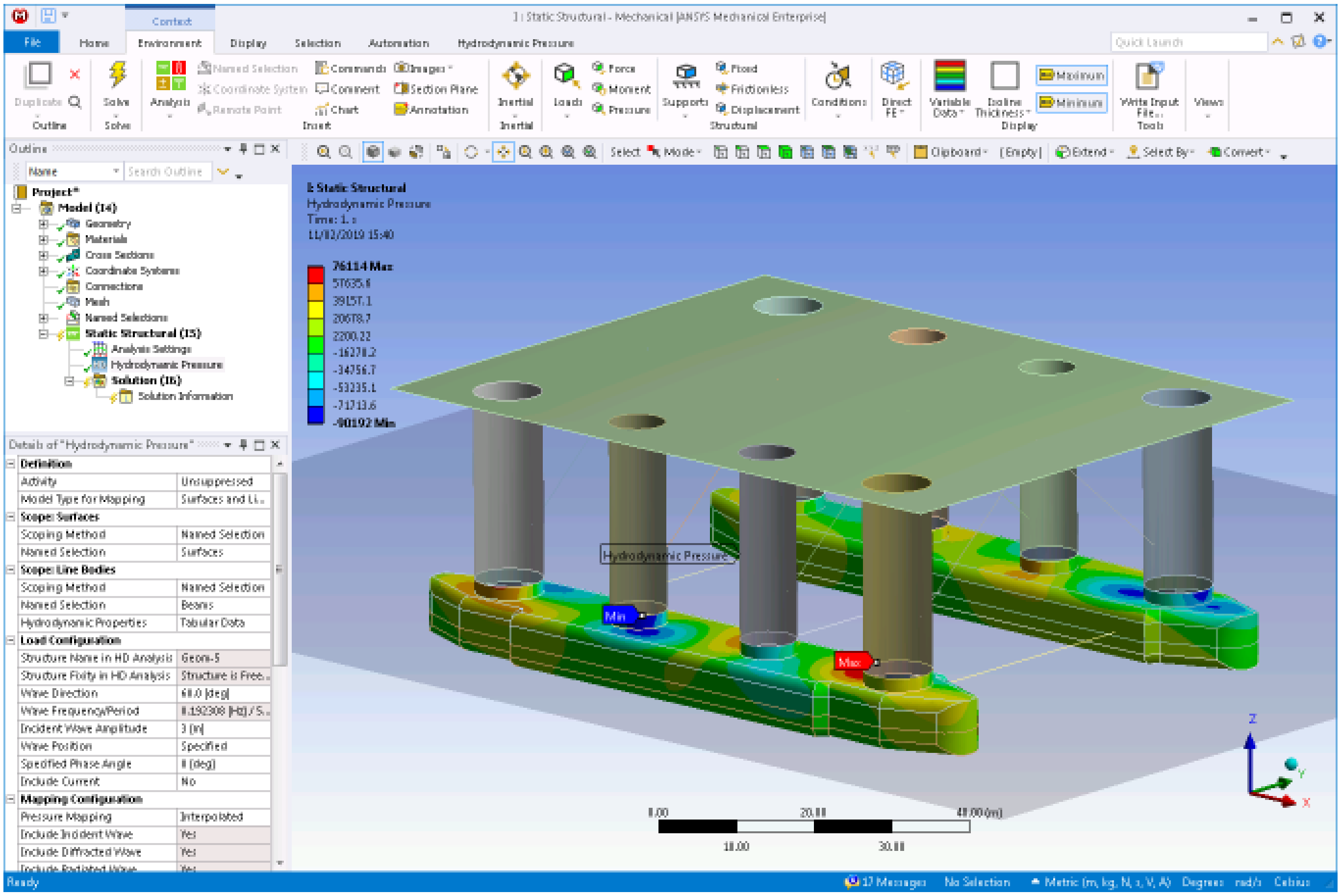
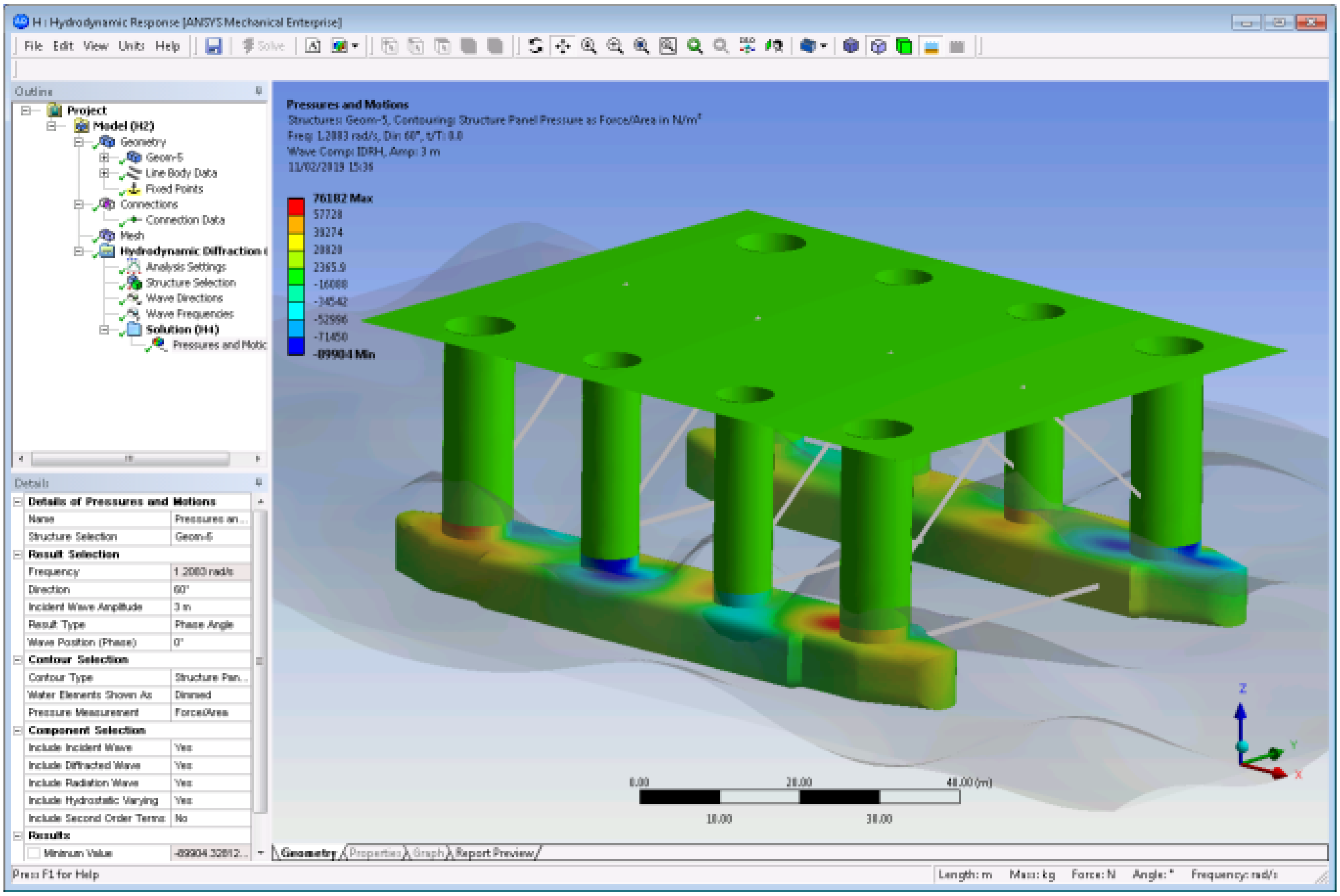


Figure 8.6: Hydrodynamic Pressures in the Hydrodynamic Diffraction System



Where the **Wave Position** option is set to Sequence, the minimum and maximum pressures (on surfaces) and/or beam loads (on line elements) are shown over the wave cycle in graphical and tabular form.

Figure 8.7: Minimum/Maximum Pressures and Beam Loads vs Phase Angle (Graph)

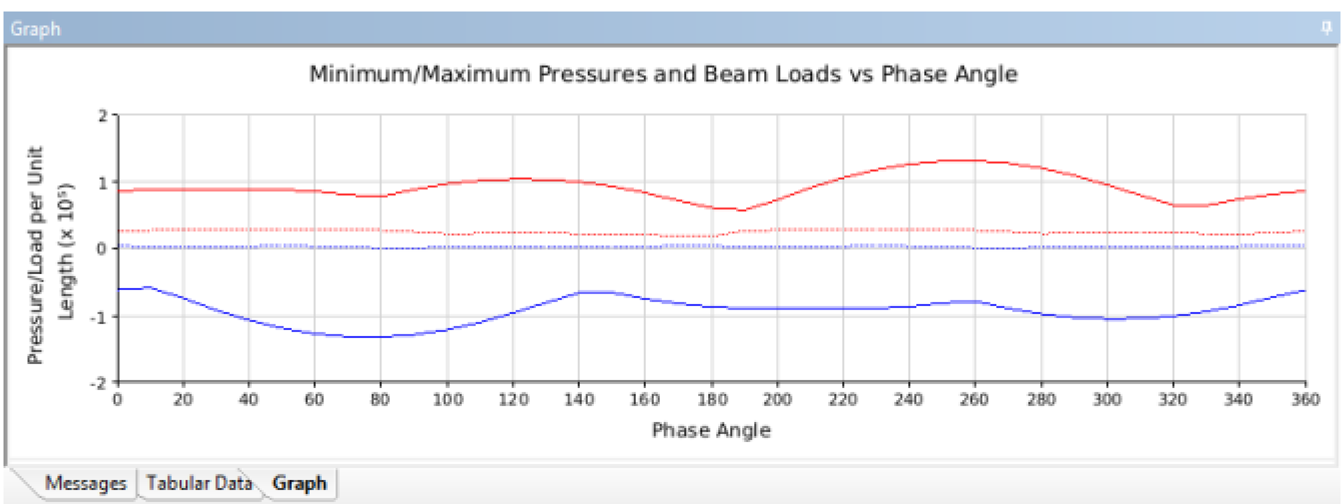


Figure 8.8: Minimum/Maximum Pressures and Beam Loads vs Phase Angle (Table)

Tabular Data

	Phase Angle	Min Pressure [Pa]	Max Pressure [Pa]	Min Beam Load [N m^-1]	Max Beam Load [N m^-1]
1	0.0	-61308.0	86351.0	3961.79355974	26065.2323412
2	10.0	-57820.0	88502.0	3020.79807336	27484.8057115
3	20.0	-73286.0	89082.0	2911.10230840	28364.8179441
4	30.0	-91234.0	88789.0	2051.10018549	29020.7244913
5	40.0	-106410.0	89632.0	3174.02946741	29546.1760639
6	50.0	-118350.0	88405.0	4419.63554267	29845.2292503
7	60.0	-126700.0	85990.0	3332.81131983	29729.9366969
8	70.0	-131200.0	80962.0	2111.85427158	29002.9184911
9	80.0	-131710.0	78014.0	819.891175706	27510.3267463
10	90.0	-128210.0	88878.0	720.611585599	25192.5091924
11	100.0	-120830.0	97041.0	1883.64163354	22429.0161284
12	110.0	-109770.0	102260.0	2216.16212954	22842.9865526

Messages | **Tabular Data** | Graph

8.1.4.3. Structural Analysis Load Steps

When mapping frequency-domain hydrodynamic pressures from Aqwa to Mechanical, you can choose to map for multiple wave frequencies, directions, and phase angles in a single structural analysis. The analysis will include as many load steps as there are wave frequency/direction/phase combinations.

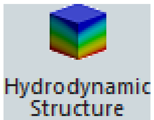
The order of load cases in the structural analysis will be the same for both the Interpolated and Direct pressure mapping methods. The cases are typically organized by wave frequency, direction, then phase angle. For example, in a configuration with 2 frequencies, 3 directions, and 3 phase increments, the load cases will be ordered as follows:

```
Step Number    Frequency No.   Direction No.    Phase Angle (degrees)
     1              1              1               0
     2              1              1             120
     3              1              1             240
     4              1              2               0
     5              1              2             120
     6              1              2             240
     7              1              3               0
     8              1              3             120
     9              1              3             240
    10              2              1               0
    11              2              1             120
    12              2              1             240
    13              2              2               0
    14              2              2             120
    15              2              2             240
    16              2              3               0
    17              2              3             120
    18              2              3             240
```

From this, you can then determine the association between the load step number and the wave case.

8.1.5. Associating a Hydrodynamic Structure from the Hydrodynamic Diffraction Analysis

A Hydrodynamic Structure object is automatically added to the Static Structural analysis when you create a Hydrodynamic Pressure object.

Clicking the 

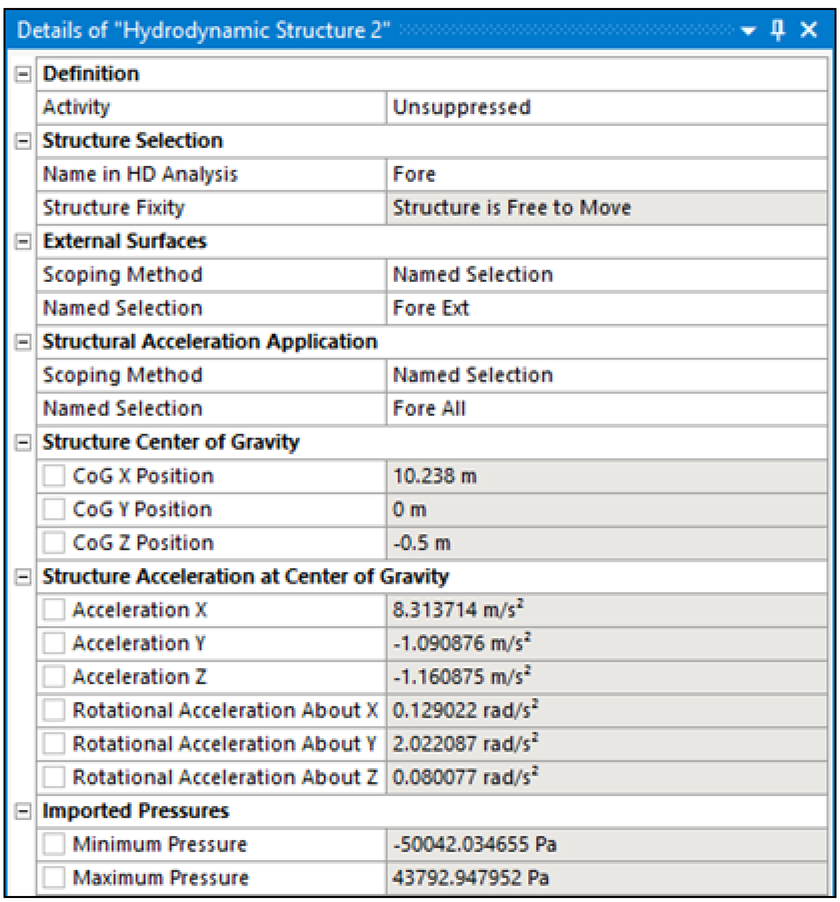
icon creates an additional Hydrodynamic Structure object, and you can repeat this up to the number of structures in the hydrodynamic model.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Hydrodynamic Structure** from the context menu.

8.1.5.1. Configuring the Hydrodynamic Structure Object

The Details of Hydrodynamic Structure panel provides options to configure the Hydrodynamic Structure object. These are described below.

Figure 8.9: Details of Hydrodynamic Structure


Details of "Hydrodynamic Structure 2"

Definition	
Activity	Unsuppressed
Structure Selection	
Name in HD Analysis	Fore
Structure Fixity	Structure is Free to Move
External Surfaces	
Scoping Method	Named Selection
Named Selection	Fore Ext
Structural Acceleration Application	
Scoping Method	Named Selection
Named Selection	Fore All
Structure Center of Gravity	
CoG X Position	10.238 m
CoG Y Position	0 m
CoG Z Position	-0.5 m
Structure Acceleration at Center of Gravity	
Acceleration X	8.313714 m/s²
Acceleration Y	-1.090876 m/s²
Acceleration Z	-1.160875 m/s²
Rotational Acceleration About X	0.129022 rad/s²
Rotational Acceleration About Y	2.022087 rad/s²
Rotational Acceleration About Z	0.080077 rad/s²
Imported Pressures	
Minimum Pressure	-50042.034655 Pa
Maximum Pressure	43792.947952 Pa

The **Activity** field allows you to set the suppression state of the Hydrodynamic Structure object. If you want to exclude the structure from the mapping, set this field to Suppressed.

8.1.5.2. Structure Selection

When the Hydrodynamic Diffraction analysis contains more than one structure, you must select the name of the structure (in the Hydrodynamic Diffraction system) that you intend to map loads from, using the **Name in HD Analysis** option.

The fixity of the selected structure in the Hydrodynamic Diffraction analysis is indicated by **Structure Fixity**. When this field shows Structure is Fixed in Place, the radiated wave pressure component will be zero and no structure accelerations will be calculated. In this case, for consistency, the structure should also be fixed in the Static Structural analysis (for example, by a Fixed Support). If the Structure is Free to Move you are recommended to use the **Weak Springs** option (in the Static Structural Analysis Settings) to mitigate rigid body motions.

If the selected structure has Internal Tanks associated with it in the Hydrodynamic Diffraction analysis, the number of tanks is shown in the read-only **Number of Internal Tanks** field.

If the selected structure has Moonpools associated with it in the Hydrodynamic Diffraction analysis, the number of Moonpools is shown in the read-only **Number of Moonpools** field.

If a forward speed has been defined in the Hydrodynamic Diffraction analysis, this is shown in **Forward Speed in HD Analysis**.

8.1.5.3. External Surfaces

When **Model Type for Mapping** includes External Surfaces, use the External Surfaces options to select the surfaces that you want to map hydrodynamic pressures onto. Surfaces can be selected either by Named Selection, or by Geometry Selection from the graphical window.

8.1.5.4. Line Bodies

When **Model Type for Mapping** includes Line Bodies, use the Line Bodies options to select the beams that you want to map hydrodynamic loads onto. Line Bodies can be selected either by Named Selection, or by Geometry Selection from the graphical window. You must also define viscous drag and inertia coefficients for the different cross sections that are included in your selection. These coefficients are used to calculate distributed loads on beams according to the Morison equation described in Theory. The **Hydrodynamic Properties** table sets coefficients for each cross section type. For non-cylindrical cross sections (solid rectangular and rectangular tube) you must define separate viscous drag and inertia coefficients for each transverse direction.

8.1.5.5. Structural Acceleration Application

When **Perform Mass/Inertia Check** is set to ‘Yes, Selected Bodies Only’ or ‘No, Use Hydrodynamic Model CoG Position’, you must select the bodies in the structural model on to which the hydrodynamic accelerations should be applied. Where relevant, these bodies will also be used in the comparison of mass properties between the hydrodynamic and structural models. Bodies can be selected either by Named Selection, or by Geometry Selection from the graphical window.

8.1.5.6. Output: Structure Acceleration at Center of Gravity, Imported Pressures and Loads

When **Model Type for Mapping** includes External Surfaces, the **Minimum** and **Maximum Pressures** on diffracting panel elements are reported. When Model Type includes Internal Tanks, the **Minimum** and **Maximum Internal Tank Pressures** are provided. When Model Type includes Moonpools, the **Minimum** and **Maximum Moonpool Pressures** are stated. When Model Type includes Line Bodies, the **Minimum** and **Maximum Beam Loads** (as Force per unit Length) are also shown. These quantities are evaluated over this hydrodynamic structure only.

For all cases, the position of the structure's center of gravity (in the Static Structural axis system), and the resultant translational and rotational accelerations about that center of gravity, are displayed. Where applicable, values are displayed for the wave case and phase angle selected in **Display for Wave Case** and **Display Pressures/Accelerations At Phase Angle**, respectively.

8.1.6. Importing Time Domain Hydrodynamic Response Pressures and Loads to Mechanical

Opening the Static Structural system in the Mechanical editor, you will find an additional tab that gives you access to the Hydrodynamic Pressure Add-on.

Clicking the 

icon adds a Hydrodynamic Pressure object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Hydrodynamic Pressure** from the context menu.

Note:

Only one Hydrodynamic Pressure object may be added to a Static Structural analysis.

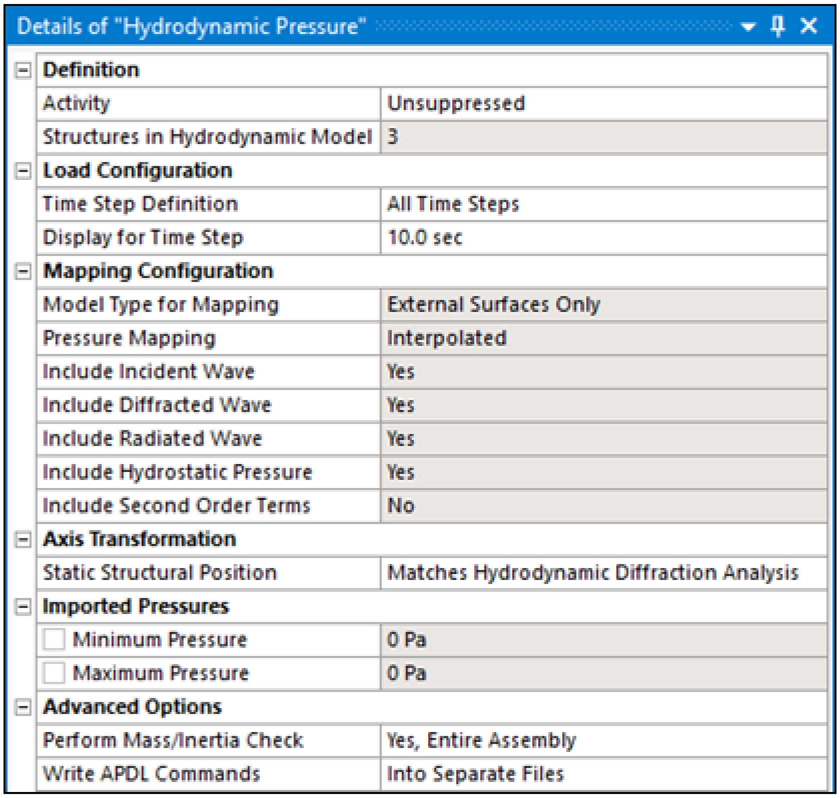
Note:

Moonpool Pressures cannot be included in or mapped from Hydrodynamic Response systems.

8.1.6.1. Configuring the Hydrodynamic Pressure Object

You can configure the Hydrodynamic Pressure object using the options in its Details panel. The fields are described below.

Figure 8.10: Details of Hydrodynamic Pressure



Details of "Hydrodynamic Pressure"

Definition	
Activity	Unsuppressed
Structures in Hydrodynamic Model	3
Load Configuration	
Time Step Definition	All Time Steps
Display for Time Step	10.0 sec
Mapping Configuration	
Model Type for Mapping	External Surfaces Only
Pressure Mapping	Interpolated
Include Incident Wave	Yes
Include Diffracted Wave	Yes
Include Radiated Wave	Yes
Include Hydrostatic Pressure	Yes
Include Second Order Terms	No
Axis Transformation	
Static Structural Position	Matches Hydrodynamic Diffraction Analysis
Imported Pressures	
Minimum Pressure	0 Pa
Maximum Pressure	0 Pa
Advanced Options	
Perform Mass/Inertia Check	Yes, Entire Assembly
Write APDL Commands	Into Separate Files

The **Activity** field allows you to set the suppression state of the Hydrodynamic Pressure object. Set to Suppressed if you want to exclude the object from the analysis.

The **Structures in Hydrodynamic Model** field displays the number of structures that are included in the pressure database from the upstream time domain Hydrodynamic Response system. It is possible to transfer pressures for one, several, or all of these structures. One Hydrodynamic Structure (p. 294) object is automatically added to the analysis when you create the Hydrodynamic Pressure object, but you can create additional Hydrodynamic Structure objects up to the number of structures in the hydrodynamic model.

8.1.6.1.1. Load Configuration

The **Time Step Definition** option allows you to analyze either a single or multiple time steps from the time domain Hydrodynamic Response calculation within a single Static Structural analysis. The time steps available for pressure mapping are defined by the Time Domain Pressure Output (p. 133) options in the time domain Hydrodynamic Response Analysis Settings.

When **Time Step Definition** is set to Single Time Step, use the **Time Step** option to select the time step for pressure mapping.

Setting **Time Step Definition** to Range of Time Steps allows you to define a **First Time Step** and **Last Time Step** for the pressure mapping.

To perform the Static Structural analysis over all available time steps, set **Time Step Definition** to All Time Steps.

Where multiple time steps are to be analyzed, the Static Structural Analysis Setting **Number of Steps** field will be updated automatically.

The additional option **Display for Time Step** can be used to set the time step at which the mapped pressures (in the graphical window) and output values (in the Details panel, described in Output: Imported Pressures, and Loads (p. 290)) are shown.

8.1.6.1.2. Mapping Configuration

Model Type for Mapping indicates the types of loads that will be mapped from the Hydrodynamic Response analysis to the Static Structural analysis. Generally, it is possible to map pressures onto surfaces (Surface Bodies and the outer faces of Solid Bodies) and distributed loads onto beams or pipes (Line Bodies).

The meshes in the Static Structural and Hydrodynamic Response systems are generally not coincident, which necessitates some approach to map the hydrodynamic pressures and Morison loads onto the structural mesh. For pressure mapping from a time domain Hydrodynamic Response system, the **Pressure Mapping** option is automatically set to Interpolated.

Hydrodynamic pressures at each node of the selected surfaces in the structural mesh are interpolated from the element-centered pressures calculated in the time domain Hydrodynamic Response analysis.

The mapped hydrodynamic loads will always include incident, diffracted, radiated and hydrostatic components, as indicated by the **Include Incident**/**Diffracted**/**Radiated Wave** and **Include Hydrostatic Pressure** fields.

For distributed loads on Line Bodies, the forces per unit length along each line element (BEAM188/189 or PIPE288/289) are interpolated from the element-centered Morison forces calculated in the time domain Hydrodynamic Response analysis.

Note:

Any moment acting about the centroid of each hydrodynamic line element is assumed to be negligible. It is therefore recommended to use a relatively small element size for Line Bodies in the hydrodynamic analysis.

Line body loads will only be transferred to line elements in the structural model that are exactly collinear with the Line Bodies of the hydrodynamic model (though the lengths of the line elements can be different between the hydrodynamic and structural meshes).

For Morison discs in the time domain Hydrodynamic Response analysis, forces and moments are directly applied to the nearest (ideally, coincident) mesh node in the structural model. You do not need to scope the Morison discs to the structural geometry. The Hydrodynamic Pressure object will automatically search through all the mesh nodes on Bodies that are included in the load mapping selection.

8.1.6.1.3. Axis Transformation

Although you can share one Geometry source between the Hydrodynamic workflow and the Static Structural system, in the typical use case the Static Structural model will include internal and/or other structural components that are not required for the Hydrodynamic calculations. Depending on the context, the geometries in the Static Structural and Hydrodynamic systems may therefore employ different axis systems. To account for this, it is possible to define an axis transformation from the Static Structural geometry to the Hydrodynamic geometry.

Where the axis systems are consistent between the two analyses, the **Static Structural Position** may be set as Matches Hydrodynamic Diffraction Analysis. Otherwise, this option should be set to Differs from Hydrodynamic Diffraction Analysis, and **Structure Position/Rotation Offsets** should be defined for the translational and rotational freedoms. Offsets are defined relative to the Hydrodynamic Diffraction global axis system (FRA).

The position of the selected structure at the displayed time step is reported in the **Instantaneous Position/Rotation** fields, expressed in the Static Structural axis system (that is, accounting for any defined **Structure Position/Rotation Offsets**). For reference, the mean water surface is drawn relative to the position of the structure.

8.1.6.1.4. Output: Imported Pressures, and Loads

The **Minimum** and **Maximum Pressures** on diffracting panel elements are reported.

When **Model Type for Mapping** includes Internal Tanks, the **Minimum** and **Maximum Internal Tank Pressures** are displayed. When **Model Type** includes Line Bodies, the **Minimum** and **Maximum Beam Loads** (as Force per unit Length) are also displayed. These quantities are evaluated over all of the surfaces and line bodies included for mapping (that is, across all structures); the Hydrodynamic Structure (p. 294) object shows maxima and minima per structure.

Where applicable, values are displayed for the time step selected in **Display for Time Step**.

8.1.6.1.5. Advanced Options

The Hydrodynamic Pressure object accesses the material properties of each Body so that it can estimate the structural mass, inertia, and center of gravity (CoG) position, and compare these to the corresponding properties from the hydrodynamic model. Generally, the force and moment reactions on any defined boundary conditions should be relatively small in the Static Structural system, and this check helps ensure the applied loads and accelerations are well-balanced.

The **Perform Mass/Inertia Check** setting can be used to adjust the behavior of the Hydrodynamic Pressure object in this respect. This setting has 4 options:

- **Yes, Entire Assembly**: The mass/inertia/CoG of the entire assembly, including all Bodies in the Geometry, will be calculated and compared to the corresponding properties of the selected structure in the upstream hydrodynamic analysis. This option is invalid if there is more than one Hydrodynamic Structure in the analysis.

- **Yes, Selected Parts Only**: The mass/inertia/CoG will be calculated and compared only for the Bodies that are included in the **Structural Acceleration Application** scope for each Hydrodynamic Structure.

- **No, Use Hydrodynamic Model CoG Position**: The mass/inertia/CoG will not be calculated. The accelerations included by the Hydrodynamic Pressure object will be applied about the CoG position of the selected structure(s) in the upstream hydrodynamic analysis (accounting for any defined Axis Transformation (p. 279)). These accelerations will be applied to the specific Bodies that are included in the **Structural Acceleration Application** scope for each Hydrodynamic Structure. You must ensure that the mass properties are consistent between the hydrodynamic and structural models.

- **No, Exclude Accelerations**: The mass/inertia/CoG will not be calculated, and the Hydrodynamic Pressure object will not apply any accelerations in the structural analysis. You must ensure that the mass properties are consistent between the hydrodynamic and structural models.

In some cases, a Body's material properties may be inaccessible. For example, where Ansys Composite PrepPost (ACP) has been used to define a composite material, or where the structural analysis employs a material with no density value defined. For these cases, the **Perform Mass/Inertia Check** option must be set to **No, Use Hydrodynamic Model CoG Position** or **No, Exclude Accelerations**. The force/moment reactions on any defined boundary conditions must be checked carefully.

The **Write APDL Commands** option defines where the APDL commands (SFE, SFBEAM, etc.) associated with the mapped hydrodynamic loads are written. By default, these commands will be written **Into Separate Files**, which are referred to within the Mechanical input file using the **/INPUT** command. Alternatively, you can choose to write commands **Direct to Mechanical Input File**, so that no new files are created, though the Mechanical input file may become very large for an analysis with many load cases.

8.1.6.2. Generating Hydrodynamic Pressures and Inertial Loads

Once the Hydrodynamic Pressure object has been configured, and at least one Hydrodynamic Structure (p. 294) has been defined, right-click the **Hydrodynamic Pressure** object and select **Generate** to start the load transfer process. This involves two main steps:

- Using the Ansys solver to determine the structural mass properties, which are compared to the mass properties from the time domain Hydrodynamic Response calculation.
- Reading the output of the time domain Hydrodynamic Response calculation to determine hydrodynamic pressures at the structural mesh nodes and calculate translational and rotational accelerations to balance inertial loads.

Note:

The Hydrodynamic Pressure Add-on will take into account any difference in the unit systems that are employed in the time domain Hydrodynamic Response and Static Structural systems.

Where the attached Geometry includes a large number of parts or bodies (1000 or more), the Mechanical UI may be noticeably unresponsive for some time during Hydrodynamic Pressure Generate and Solve operations. If necessary, you can monitor progress via the Log File, accessible through the Workbench menu bar: **Add-ons** → **View Log File**.

The transferred pressures are displayed in the graphical window when you select a Hydrodynamic Structure in the Outline tree (Figure 8.11: Hydrodynamic Pressures in the Static Structural System (p. 292)), and can be compared to the Structure Panel Pressures (Figure 8.12: Hydrodynamic Pressures in the Time Domain Hydrodynamic Response System (p. 292)) shown in the **Time Domain Pressures** result object in the upstream Hydrodynamic Response system.

Figure 8.11: Hydrodynamic Pressures in the Static Structural System

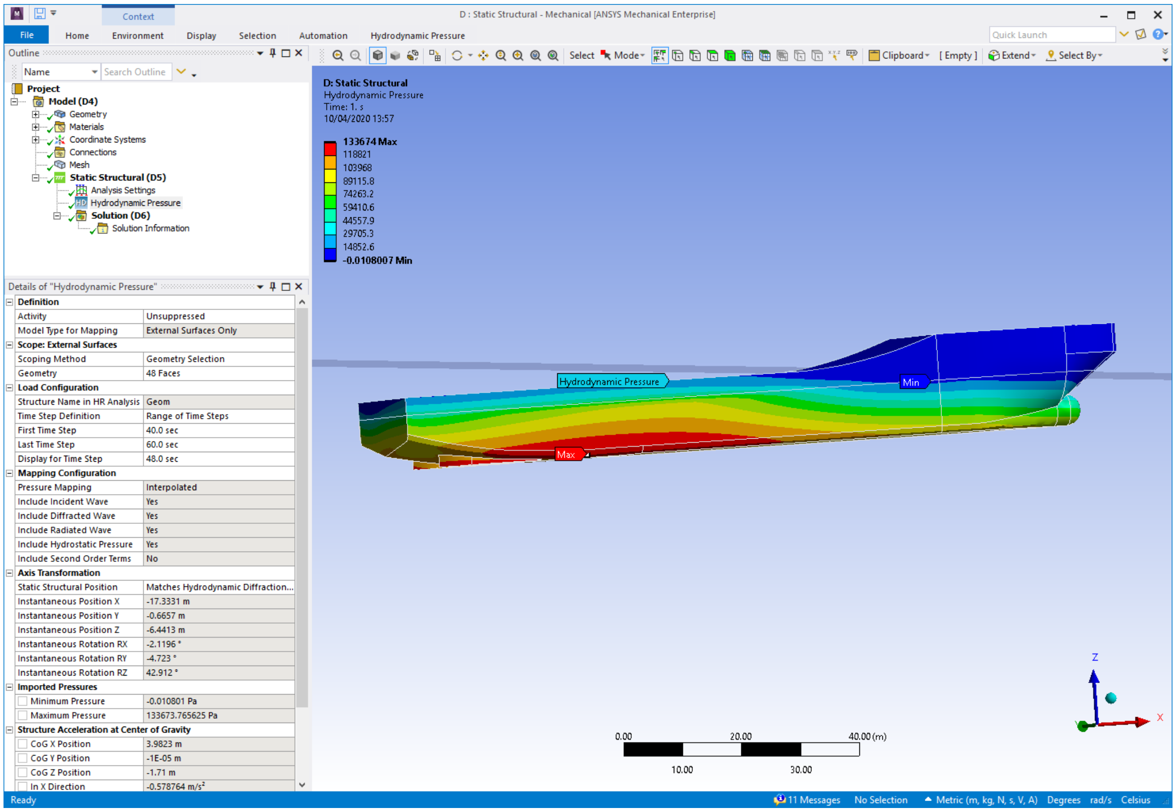
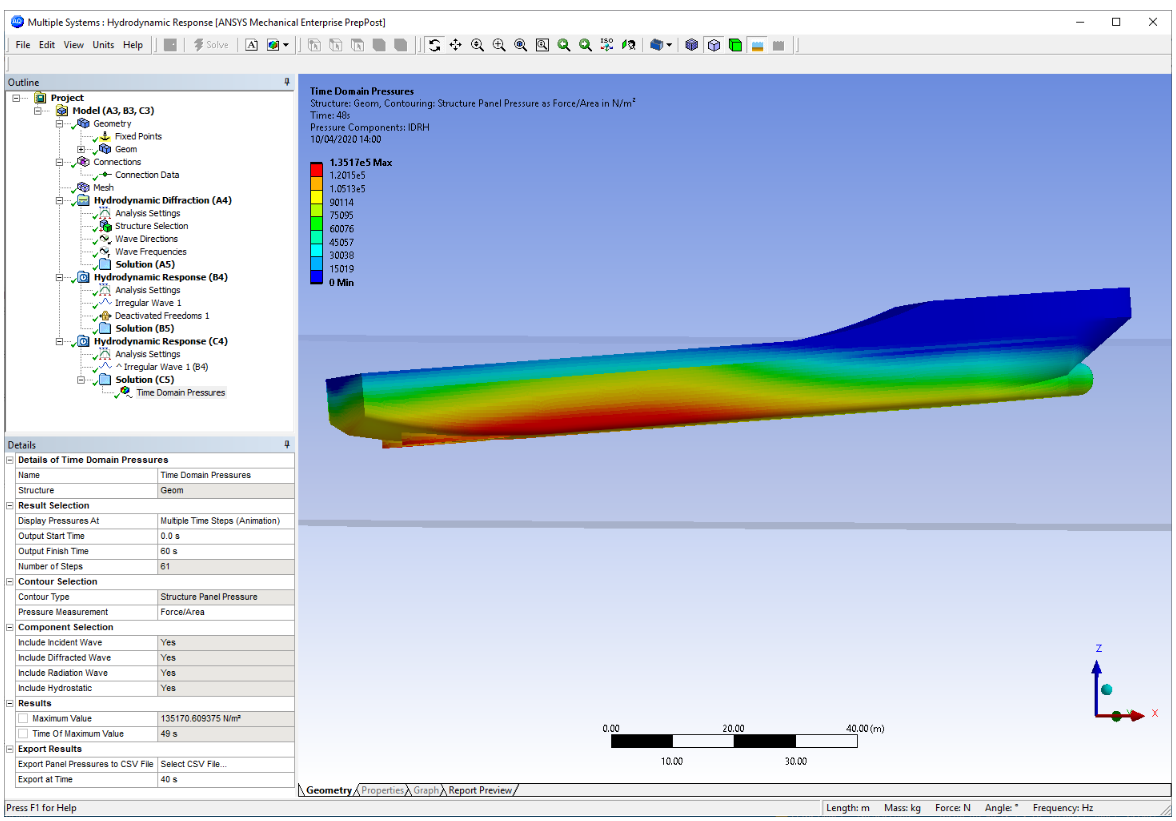


Figure 8.12: Hydrodynamic Pressures in the Time Domain Hydrodynamic Response System



Where the **Time Step Definition** option is set to Range of Time Steps or All Time Steps, the minimum and maximum pressures (on surfaces) are shown over the mapping duration in graphical and tabular form.

Figure 8.13: Minimum/Maximum Pressures vs Time (Graph)

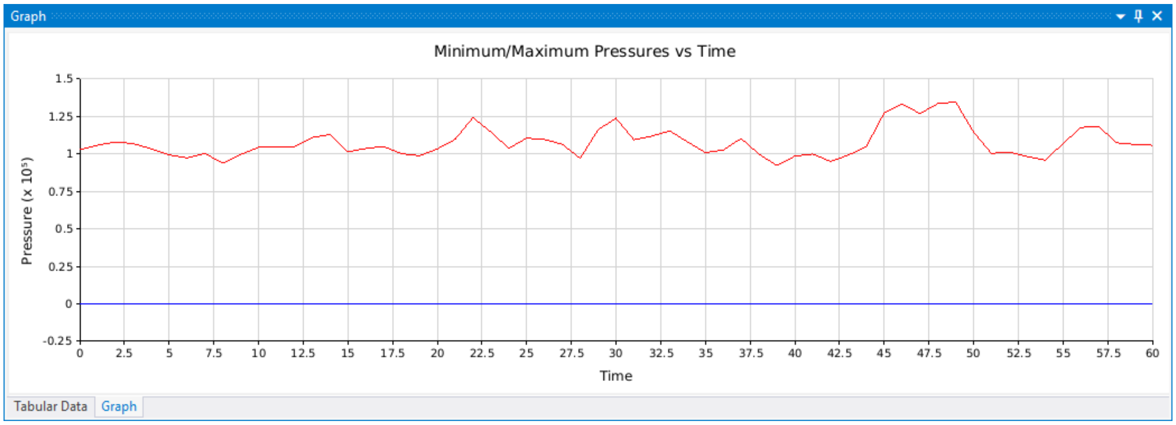
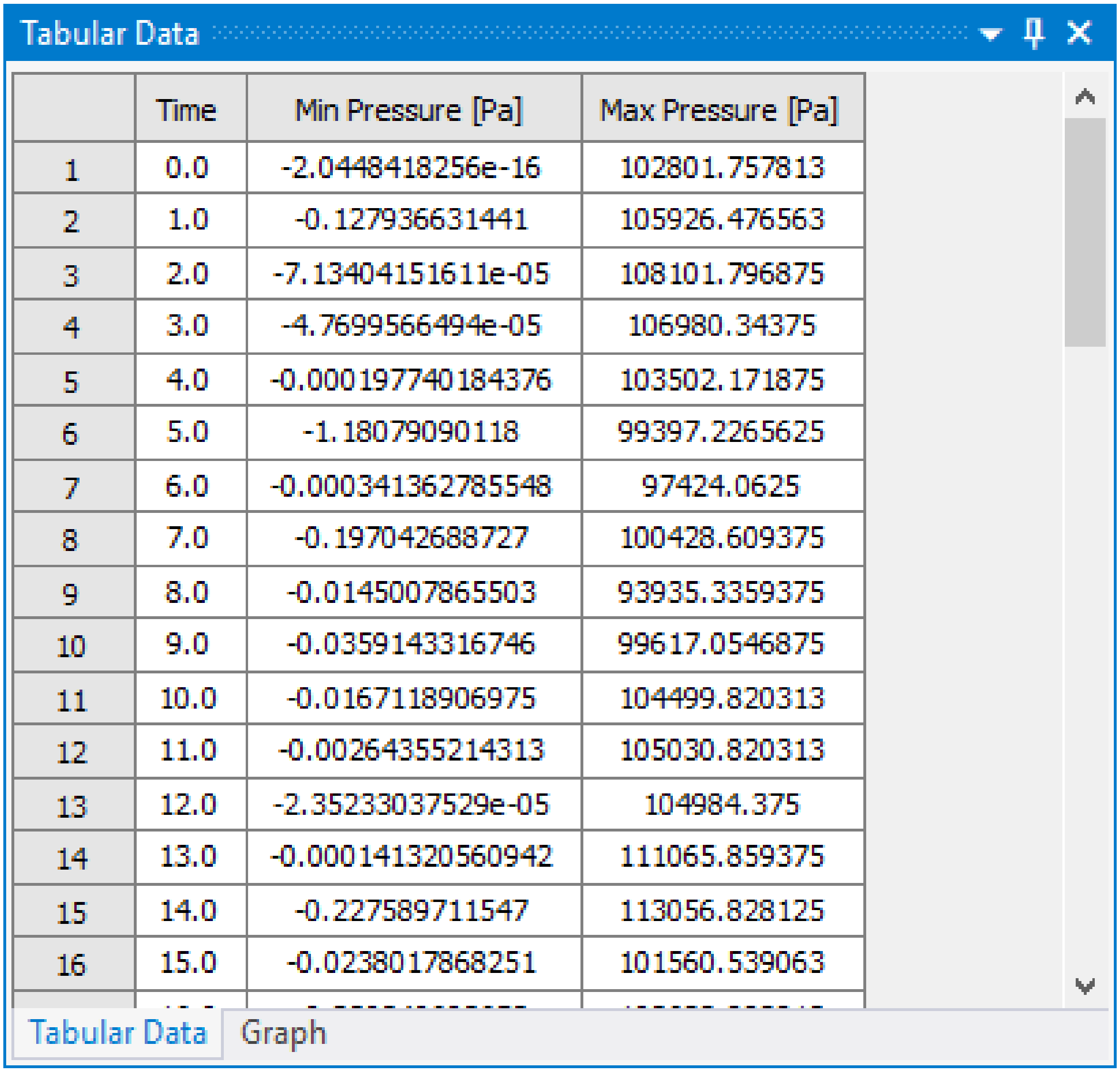


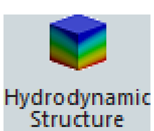
Figure 8.14: Minimum/Maximum Pressures vs Time (Table)



	Time	Min Pressure [Pa]	Max Pressure [Pa]
1	0.0	-2.0448418256e-16	102801.757813
2	1.0	-0.127936631441	105926.476563
3	2.0	-7.13404151611e-05	108101.796875
4	3.0	-4.7699566494e-05	106980.34375
5	4.0	-0.000197740184376	103502.171875
6	5.0	-1.18079090118	99397.2265625
7	6.0	-0.000341362785548	97424.0625
8	7.0	-0.197042688727	100428.609375
9	8.0	-0.0145007865503	93935.3359375
10	9.0	-0.0359143316746	99617.0546875
11	10.0	-0.0167118906975	104499.820313
12	11.0	-0.00264355214313	105030.820313
13	12.0	-2.35233037529e-05	104984.375
14	13.0	-0.000141320560942	111065.859375
15	14.0	-0.227589711547	113056.828125
16	15.0	-0.0238017868251	101560.539063

8.1.7. Associating a Hydrodynamic Structure from the Time Domain Hydrodynamic Response Analysis

A Hydrodynamic Structure object is automatically added to the Static Structural analysis when you create a Hydrodynamic Pressure object.

Clicking the 

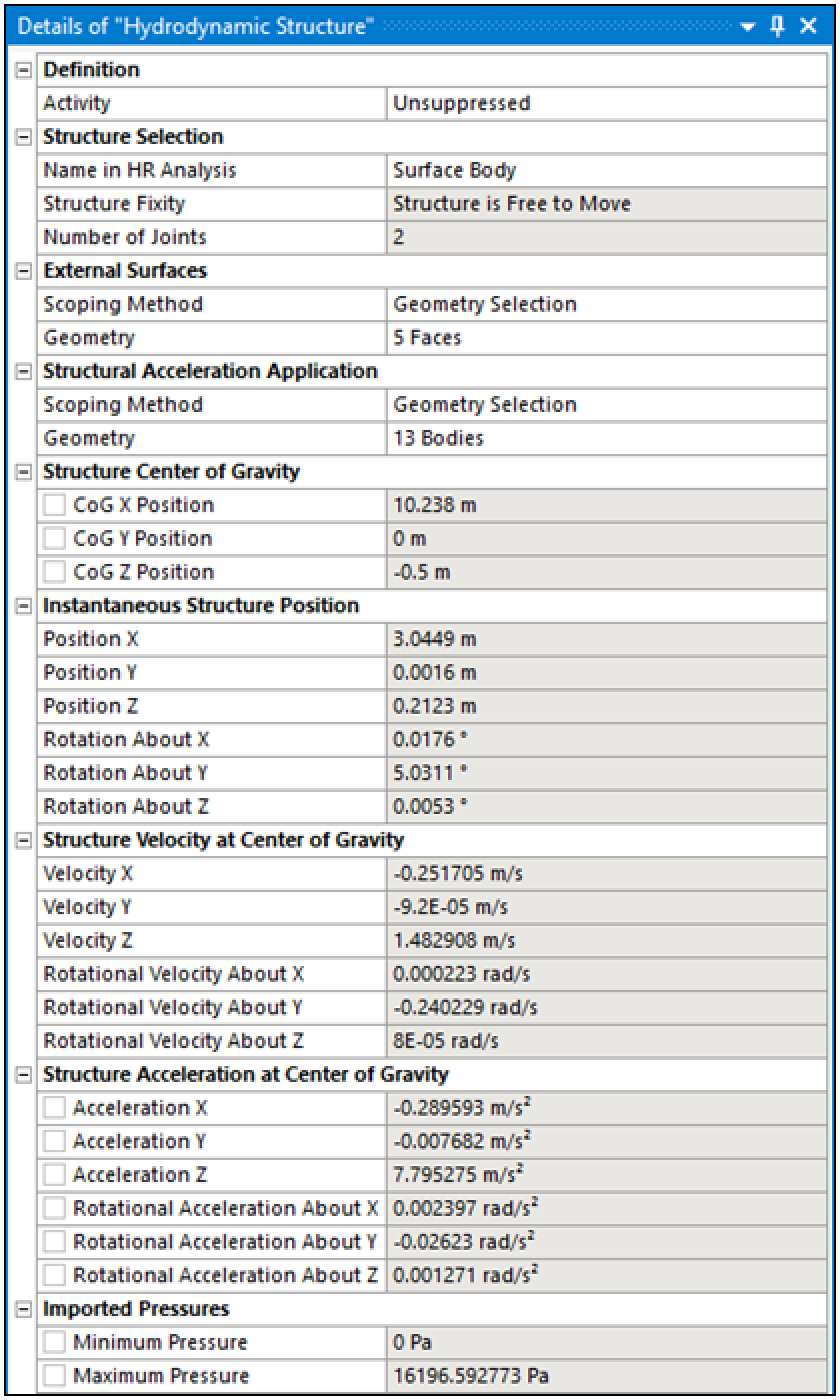
icon creates an additional Hydrodynamic Structure object, and you can repeat this up to the number of structures in the hydrodynamic model.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Hydrodynamic Structure** from the context menu.

8.1.7.1. Configuring the Hydrodynamic Structure Object

The Details of Hydrodynamic Structure panel provides options that allow you to configure the Hydrodynamic Structure object. These are described below.

Figure 8.15: Details of Hydrodynamic Structure



Details of "Hydrodynamic Structure"

Definition	
Activity	Unsuppressed
Structure Selection	
Name in HR Analysis	Surface Body
Structure Fixity	Structure is Free to Move
Number of Joints	2
External Surfaces	
Scoping Method	Geometry Selection
Geometry	5 Faces
Structural Acceleration Application	
Scoping Method	Geometry Selection
Geometry	13 Bodies
Structure Center of Gravity	
CoG X Position	10.238 m
CoG Y Position	0 m
CoG Z Position	-0.5 m
Instantaneous Structure Position	
Position X	3.0449 m
Position Y	0.0016 m
Position Z	0.2123 m
Rotation About X	0.0176 °
Rotation About Y	5.0311 °
Rotation About Z	0.0053 °
Structure Velocity at Center of Gravity	
Velocity X	-0.251705 m/s
Velocity Y	-9.2E-05 m/s
Velocity Z	1.482908 m/s
Rotational Velocity About X	0.000223 rad/s
Rotational Velocity About Y	-0.240229 rad/s
Rotational Velocity About Z	8E-05 rad/s
Structure Acceleration at Center of Gravity	
Acceleration X	-0.289593 m/s²
Acceleration Y	-0.007682 m/s²
Acceleration Z	7.795275 m/s²
Rotational Acceleration About X	0.002397 rad/s²
Rotational Acceleration About Y	-0.02623 rad/s²
Rotational Acceleration About Z	0.001271 rad/s²
Imported Pressures	
Minimum Pressure	0 Pa
Maximum Pressure	16196.592773 Pa

The **Activity** field allows you to set the suppression state of the Hydrodynamic Structure object. Set to Suppressed if you want to exclude the structure from the mapping.

8.1.7.2. Structure Selection

When your time domain Hydrodynamic Response analysis contains more than one structure, you must select the name of the structure (in the Hydrodynamic Response system) that you intend to map loads from using the **Name in HR Analysis** option.

The fixity of the selected structure in the Hydrodynamic Diffraction analysis is indicated by **Structure Fixity**. When this field shows Structure is Fixed in Place, the radiated wave pressure component will be zero and no structure accelerations will be calculated. In this case, for consistency, the structure should also be fixed in the Static Structural analysis (for example, by a Fixed Support). If the Structure is Free to Move you are recommended to use the **Weak Springs** option (in the Static Structural Analysis Settings) to mitigate rigid body motions.

If the selected structure has Internal Tanks associated with it in the Hydrodynamic Response analysis, the number of tanks is shown in the read-only **Number of Internal Tanks** field.

If the selected structure has Cables, Tethers or Joints attached to it in the Hydrodynamic Response analysis, the numbers of these features are shown in the read-only **Number of Cables/Tethers/Joints on Structure** fields.

If the selected structure has Morison Disc elements associated with it in the Hydrodynamic Response analysis, the number of discs is shown in the read-only **Number of Morison Discs on Structure** field.

8.1.7.3. External Surfaces

When **Model Type for Mapping** includes External Surfaces, use the External Surfaces options to select the surfaces that you want to map hydrodynamic pressures onto. Surfaces can be selected either by Named Selection, or by Geometry Selection from the graphical window.

8.1.7.4. Line Bodies

When **Model Type for Mapping** includes Line Bodies, use the Line Bodies options to select the beams that you want to map hydrodynamic loads onto. Line Bodies can be selected either by Named Selection, or by Geometry Selection from the graphical window.

8.1.7.5. Structural Acceleration Application

When the **Perform Mass/Inertia Check** option is set to 'Yes, Selected Bodies Only' or 'No, Use Hydrodynamic Model CoG Position', you must select the bodies in the structural model on to which the hydrodynamic rotational velocities and accelerations should be applied. Where relevant, these bodies will also be used in the comparison of mass properties between the hydrodynamic and structural models. Bodies can be selected either by Named Selection, or by Geometry Selection from the graphical window.

8.1.7.6. Output: Structure Velocity and Acceleration at Center of Gravity, Imported Pressures and Loads

The **Minimum** and **Maximum Pressures** on diffracting panel elements are reported.

When **Model Type for Mapping** includes Internal Tanks, the **Minimum** and **Maximum Internal Tank Pressures** are provided. When Model Type includes Line Bodies, the **Minimum** and **Maximum Beam Loads** (as Force per unit Length) are also shown. These quantities are evaluated over this hydrodynamic structure only. For all cases, the position of the structure's center of gravity (in the Static Structural axis system), the translational and rotational velocities, and the resultant translational and rotational accelerations about that center of gravity, are displayed. Where applicable, values are displayed for the time step selected in **Display for Time Step**.

8.1.8. Internal Tank Pressure from a Hydrodynamic Diffraction Analysis

Clicking the 

icon adds an **Internal Tank Pressure** object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Internal Tank Pressure** from the context menu.

You can only add an **Internal Tank Pressure** object after a Hydrodynamic Pressure object has been added to the Static Structural analysis.

You must ensure that the number of **Internal Tank Pressure** objects in the Static Structural analysis is the same as the number of Internal Tanks defined in the upstream Hydrodynamic Diffraction analysis to account for the weight of the Internal Tank fluid in the force balance in the Static Structural system. Otherwise, there would be an imbalance of vertical forces resulting in an acceleration of the structure out of the water.

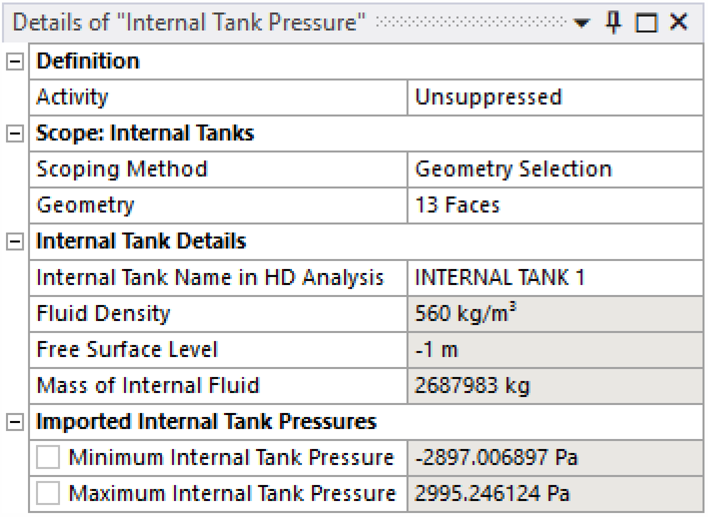
Note:

In the Static Structural system the hydrostatic force on the external surfaces is included via the **Hydrodynamic Pressure** object if the **Include Hydrostatic Pressure** option is set to Yes, while the structure weight is calculated from the material properties and surface body thicknesses defined under Geometry. For the forces to be correctly balanced, the weight of the Internal Tank fluid must also be accounted for. This comes from the Internal Tank hydrostatic force (integral of hydrostatic pressure over the Internal Tank surfaces), which is included via the **Internal Tank Pressure** object when the **Include Hydrostatic Pressure** option is set to Yes in the associated **Hydrodynamic Pressure** object. For this reason, you must ensure that the Hydrodynamic Diffraction model and its linked Static Structural system include the same number of **Internal Tank Pressure** objects.

8.1.8.1. Configuring the Internal Tank Pressure Object

You can configure options for the **Internal Tank Pressure** object in the details panel (see Figure 8.16: Details of Internal Tank Pressure (p. 298)).

Figure 8.16: Details of Internal Tank Pressure



The **Activity** field allows you to set the suppression state of the **Internal Tank Pressure** object. Set it to Suppressed if you want to exclude the object from the analysis.

8.1.8.1.1. Scope: Internal Tanks

Use **Scope: Internal Tanks** to select the surfaces that you want to map internal tank hydrodynamic pressures onto. Surfaces can be selected either by Named Selection, or by Geometry Selection from the graphical window.

8.1.8.1.2. Internal Tank Details

Once you have selected the internal tank surfaces in the Static Structural model, you must associate them with the corresponding Internal Tank in the Hydrodynamic Diffraction analysis. **Internal Tank Name in HD Analysis** lists the names of the Internal Tanks that are defined in the Hydrodynamic Diffraction analysis, and which are associated with the structure selected in the Hydrodynamic Pressure object. Select the appropriate Internal Tank from this list.

The **Fluid Density** and **Free Surface Level** that have been defined in the Hydrodynamic Diffraction analysis for the selected Internal Tank, as well as the calculated **Mass of Internal Fluid**, are displayed for reference.

8.1.8.1.3. Displayed Pressures

In some instances the internal tank pressures shown in the graphical window may be out of date. In this case, the **Refresh Graphical Window** option will appear, in an invalid state, and should be changed from Required to Not Required. The option will then disappear, and the state of the **Internal Tank Pressure** object will change to show that it requires an update. Right-click the **Internal Tank Pressure** object and select Generate to display the correct internal tank pressures.

8.1.8.1.4. Imported Internal Tank Pressures

Once you have Generated the **Internal Tank Pressure** object, the **Minimum** and **Maximum Internal Tank Pressures** on diffracting panel elements are reported. Where applicable, values are displayed for the wave case and phase angle selected in **Display for Wave Case** and **Display**

Pressures/Accelerations At Phase Angle, respectively, as defined in the associated Hydrodynamic Pressure object.

8.1.8.2. Generating Internal Tank Pressures

Once the **Internal Tank Pressure** object has been configured, right-click it and select Generate to start the pressure transfer process. The following parameters for the pressure transfer process are copied from the Hydrodynamic Pressure object:

- Wave Case Definition
- Wave Frequency/Period
- Wave Direction
- Incident Wave Amplitude
- Wave Position
- Specified Phase Angle or Number of Phase Increments
- Include Radiated Wave
- Include Hydrostatic Pressure
- Include Hydrostatic Varying.

The output data files from the Hydrodynamic Diffraction calculation will be read to determine the internal tank hydrodynamic pressures at the structural mesh nodes for the requested wave case(s).

Note:

The Hydrodynamic Pressure Add-on will take into account any difference in the unit systems that are employed in the Hydrodynamic Diffraction and Static Structural systems.

The transferred pressures are displayed in the graphical window (Figure 8.17: Internal Tank Pressures in the Static Structural System (p. 300)), and can be compared to the Internal Tank Panel Pressures (Figure 8.18: Internal Tank Pressures in the Hydrodynamic Diffraction System (p. 300)) shown in the **Internal Tank Pressures** result object in the upstream Hydrodynamic Diffraction system. The pressure Component Selection in the **Internal Tank Pressure** object may need to be matched to those included in the **Hydrodynamic Pressure** object for such a comparison to be made.

Figure 8.17: Internal Tank Pressures in the Static Structural System

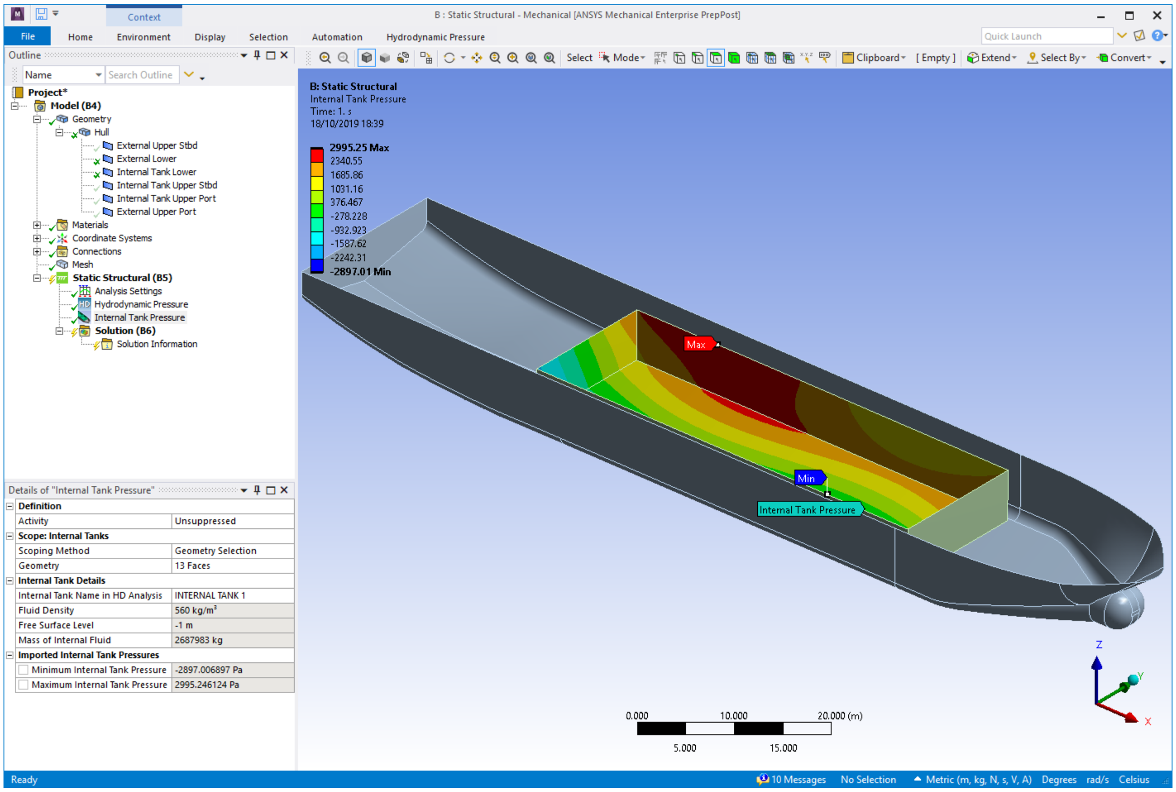
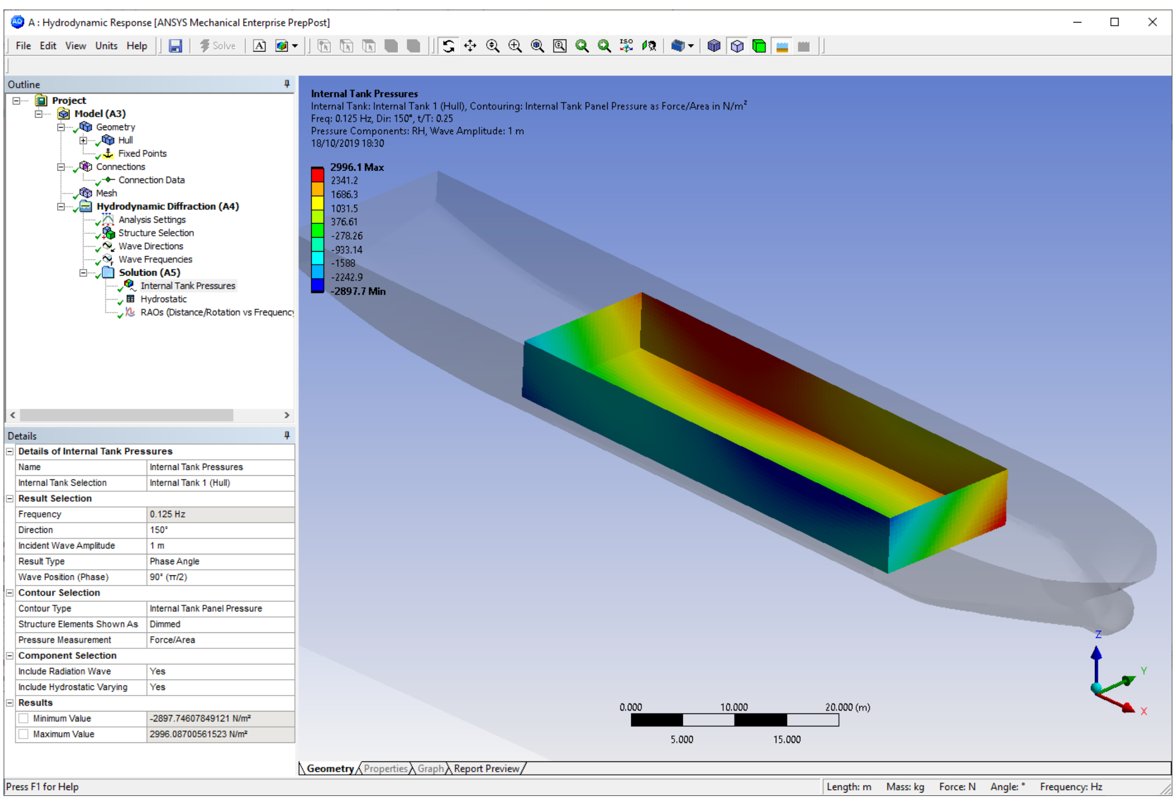


Figure 8.18: Internal Tank Pressures in the Hydrodynamic Diffraction System



8.1.9. Internal Tank Pressure from a Time Domain Hydrodynamic Response Analysis

Clicking the 

icon adds an **Internal Tank Pressure** object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Internal Tank Pressure** from the context menu.

You can only add an **Internal Tank Pressure** object after a Hydrodynamic Pressure object has been added to the Static Structural analysis.

You must ensure that the number of **Internal Tank Pressure** objects in the Static Structural analysis is the same as the number of Internal Tanks defined in the upstream time domain Hydrodynamic Response analysis to account for the weight of the Internal Tank fluid in the force balance in the Static Structural system. Otherwise, there would be an imbalance of vertical forces resulting in an acceleration of the structure out of the water.

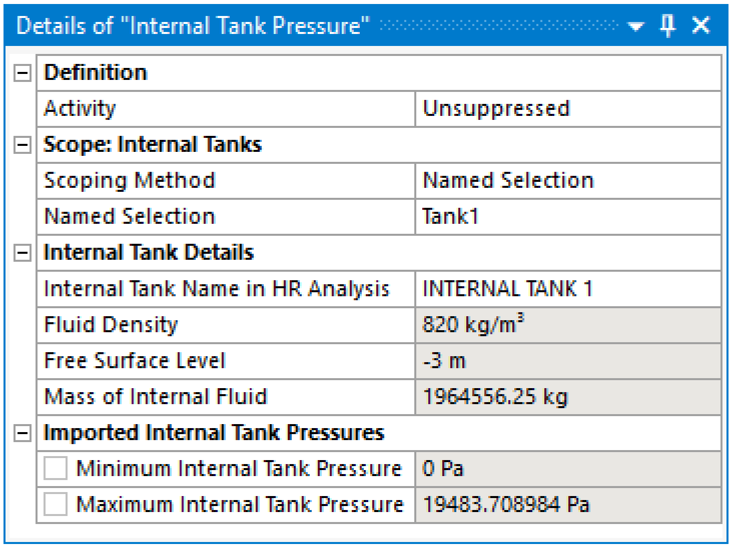
Note:

In the Static Structural system the hydrostatic force on the external surfaces is included via the **Hydrodynamic Pressure** object, while the structure weight is calculated from the material properties and surface body thicknesses defined under Geometry. For the forces to be correctly balanced, the weight of the Internal Tank fluid must also be accounted for. This comes from the Internal Tank hydrostatic force (integral of hydrostatic pressure over the Internal Tank surfaces). For this reason, you must ensure that the timedomain Hydrodynamic Response model and its linked Static Structural system include the same number of **Internal Tank Pressure** objects.

8.1.9.1. Configuring the Internal Tank Pressure Object

You can configure options for the **Internal Tank Pressure** object in the details panel (see Figure 8.19: Details of Internal Tank Pressure (p. 302)).

Figure 8.19: Details of Internal Tank Pressure



Details of "Internal Tank Pressure"

Definition	
Activity	Unsuppressed
Scope: Internal Tanks	
Scoping Method	Named Selection
Named Selection	Tank1
Internal Tank Details	
Internal Tank Name in HR Analysis	INTERNAL TANK 1
Fluid Density	820 kg/m³
Free Surface Level	-3 m
Mass of Internal Fluid	1964556.25 kg
Imported Internal Tank Pressures	
Minimum Internal Tank Pressure	0 Pa
Maximum Internal Tank Pressure	19483.708984 Pa

The **Activity** field allows you to set the suppression state of the **Internal Tank Pressure** object. Set it to Suppressed if you want to exclude the object from the analysis.

8.1.9.1.1. Scope: Internal Tanks

Use **Scope: Internal Tanks** to select the surfaces that you want to map internal tank hydrodynamic pressures onto. Surfaces can be selected either by Named Selection, or by Geometry Selection from the graphical window.

8.1.9.1.2. Internal Tank Details

Once you have selected the internal tank surfaces in the Static Structural model, you must associate them with the corresponding Internal Tank in the time domain Hydrodynamic Response analysis. **Internal Tank Name in HR Analysis** lists the names of the Internal Tanks that are defined in the time domain Hydrodynamic Response analysis, and which are associated with the structure selected in the Hydrodynamic Pressure object. Select the appropriate Internal Tank from this list.

The **Fluid Density** and **Free Surface Level** that have been defined in the time domain Hydrodynamic Response analysis for the selected Internal Tank, as well as the calculated **Mass of Internal Fluid**, are displayed for reference.

8.1.9.1.3. Displayed Pressures

In some instances the internal tank pressures shown in the graphical window may be out of date. In this case, the **Refresh Graphical Window** option will appear, in an invalid state, and should be changed from Required to Not Required. The option will then disappear, and the state of the **Internal Tank Pressure** object will change to show that it requires an update. Right-click the **Internal Tank Pressure** object and select Generate to display the correct internal tank pressures.

8.1.9.1.4. Imported Internal Tank Pressures

Once you have Generated the **Internal Tank Pressure** object, the **Minimum** and **Maximum Internal Tank Pressures** on diffracting panel elements are reported. Where applicable, values are displayed for the time step selected in **Display for Time Step** as defined in the associated Hydrodynamic Pressure object.

8.1.9.2. Generating Internal Tank Pressures

Once the **Internal Tank Pressure** object has been configured, right-click it and select Generate to start the pressure transfer process. The time step(s) for the pressure transfer process are copied from the Hydrodynamic Pressure object:

The output data files from the time domain Hydrodynamic Response calculation will be read to determine the internal tank hydrodynamic pressures at the structural mesh nodes for the requested time step(s).

Note:

The Hydrodynamic Pressure Add-on will take into account any difference in the unit systems that are employed in the time domain Hydrodynamic Diffraction and Static Structural systems.

The transferred pressures are displayed in the graphical window (Figure 8.20: Internal Tank Pressures in the Static Structural System (p. 303)), and can be compared to the Internal Tank Panel Pressures (Figure 8.21: Internal Tank Pressures in the Time Domain Hydrodynamic Response System (p. 304)) shown in the **Time Domain Pressures** result object in the upstream time domain Hydrodynamic Response system.

Figure 8.20: Internal Tank Pressures in the Static Structural System

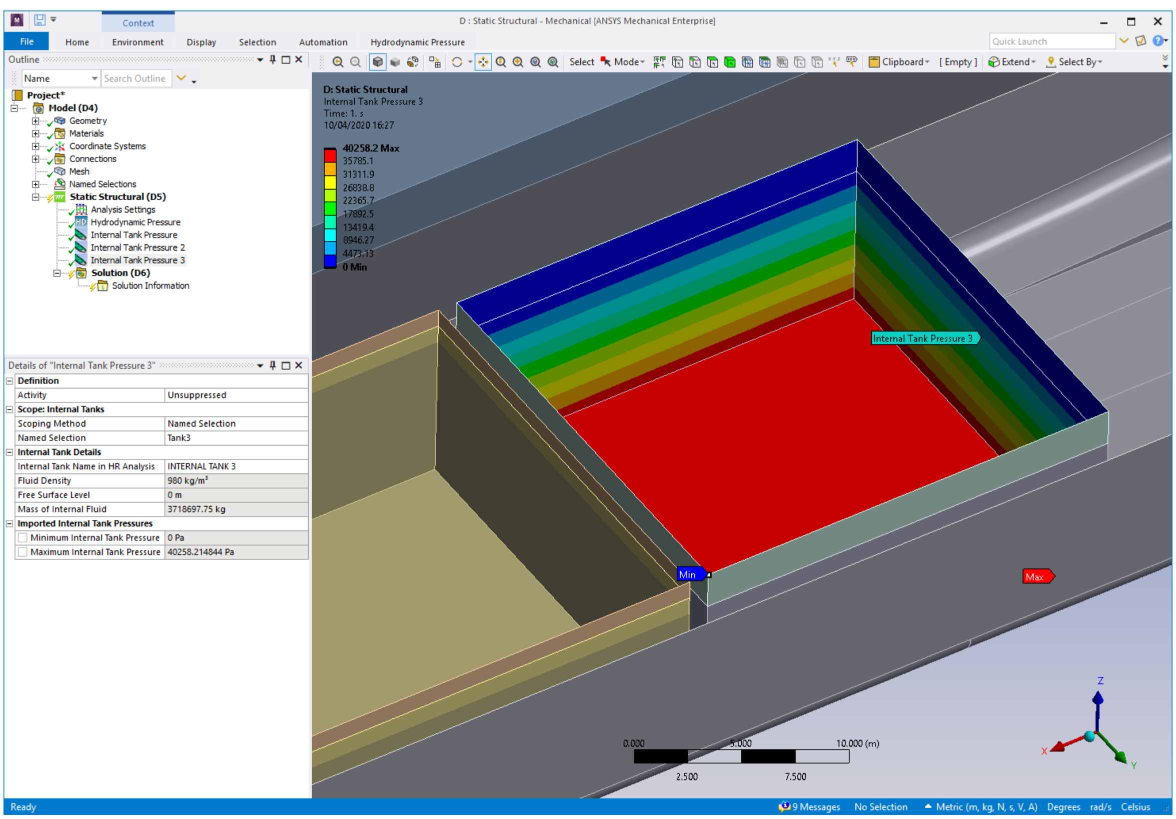
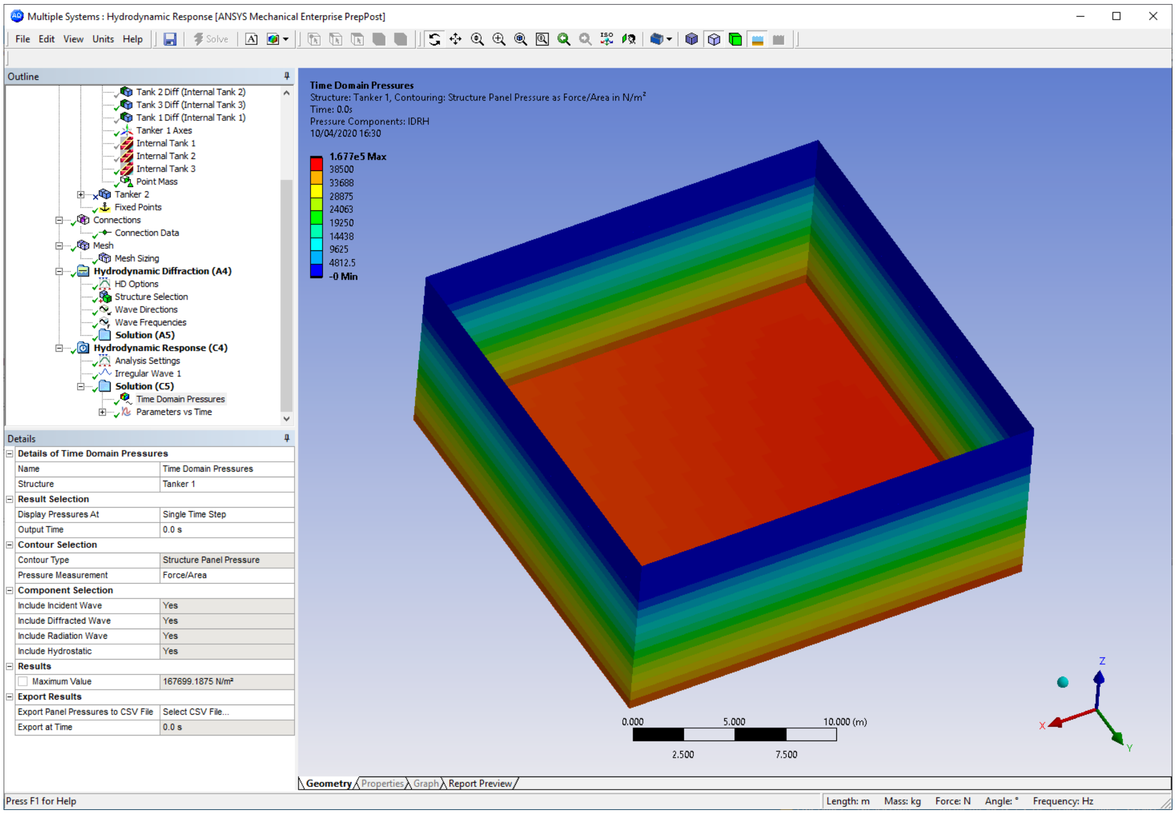


Figure 8.21: Internal Tank Pressures in the Time Domain Hydrodynamic Response System



8.1.10. Moonpool Pressure from a Hydrodynamic Diffraction Analysis

Click the 

icon to add a **Moonpool Pressure** object to the Static Structural analysis.

Alternatively, follow these steps:

1. Right-click the Static Structural analysis.
2. Select **Insert** > **Moonpool Pressure** from the context menu.

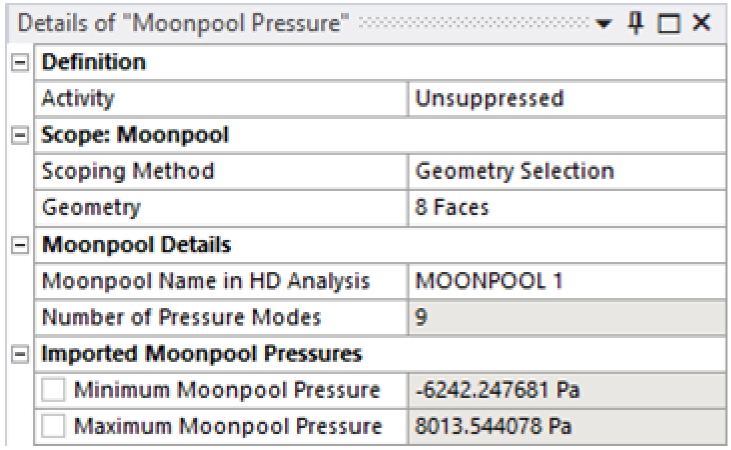
Note:

A **Moonpool Pressure** object may only be added after a **Hydrodynamic Pressure** object has been added to the Static Structural analysis. Ensure that the number of **Moonpool Pressure** objects in the Static Structural analysis is the same as the number of Moonpools defined in the upstream Hydrodynamic Diffraction analysis.

8.1.10.1. Configuring the Moonpool Pressure Object

Configure options for the **Moonpool Pressure** object in the Details panel:

Figure 8.22: Details of Moonpool Pressure



The **Activity** field sets the suppression state of the **Moonpool Pressure** object. Set it to **Suppressed** to exclude the object from the analysis.

8.1.10.1.1. Scope: Moonpool

Use **Scope: Moonpool** to select the surfaces to map moonpool hydrodynamic pressures onto. Surfaces are selected either by Named Selection, or by Geometry Selection from the Graphical window.

8.1.10.1.2. Moonpool Details

After selecting the moonpool surfaces in the Static Structural model, you must associate them with the corresponding Moonpool in the Hydrodynamic Diffraction analysis. **Moonpool Name in HD Analysis** lists the names of the Moonpools that are defined in the Hydrodynamic Diffraction analysis, which are associated with the structure selected in the **Hydrodynamic Pressure** object. Select the appropriate Moonpool from this list.

The **Number of Pressure Modes** defined in the Hydrodynamic Diffraction analysis for the selected Moonpool is displayed for reference.

8.1.10.1.3. Displayed Pressures

In some instances, the Moonpool Pressures shown in the Graphical window may be out of date. In this case, the **Refresh Graphical Window** option will appear, in an invalid state, and should be changed from **Required** to **Not Required**. The option will then disappear, and the state of the **Moonpool Pressure** object will change to show that it requires an update. Right-click the **Moonpool Pressure** object, and select **Generate** to display the correct Moonpool Pressures.

8.1.10.1.4. Imported Moonpool Pressures

After generating the **Moonpool Pressure** object, the **Minimum** and **Maximum Moonpool Pressures** on diffracting panel elements are reported. Where applicable, values are displayed for the wave case and phase angle selected in **Display for Wave Case** and **Display Pressures/Accelerations At Phase Angle**, respectively, as defined in the associated **Hydrodynamic Pressure** object.

8.1.10.2. Generating Moonpool Pressures

Once the **Moonpool Pressure** object is configured, right-click it and select **Generate** to start the pressure transfer process. The following parameters for the pressure transfer process are copied from the **Hydrodynamic Pressure** object:

- Wave Case Definition
- Wave Frequency/Period
- Wave Direction
- Incident Wave Amplitude
- Wave Position
- Specified Phase Angle or Number of Phase Increments
- Include Hydrostatic Pressure

The output data files from the Hydrodynamic Diffraction calculation are read to determine the moonpool hydrodynamic pressures at the structural mesh nodes for the requested wave case(s).

Note:

The Hydrodynamic Pressure Add-on takes into account any difference in the unit systems that are employed in the Hydrodynamic Diffraction and Static Structural systems.

The transferred pressures are displayed in the Graphical window, and can be compared to the Moonpool Pressures shown in the **Pressures and Motions** result object in the upstream Hydrodynamic Diffraction system. The pressure Component Selection may need to be matched to those included in the **Hydrodynamic Pressure** object for such a comparison to be made.

8.1.11. Cable Forces from a Time Domain Hydrodynamic Response Analysis

Clicking the 

icon will add a **Cable Force** object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Cable Force** from the context menu.

You can only add **Cable Force** objects after a Hydrodynamic Pressure object has been added to the Static Structural analysis.

You must ensure that the number of **Cable Force** objects in the Static Structural analysis is the same as the corresponding number of Cables attached to the selected structure in the upstream time domain Hydrodynamic Response analysis. Otherwise, there would be an imbalance of forces resulting in an acceleration of the structure in the Static Structural analysis.

8.1.11.1. Configuring the Cable Force Object

You can configure options for the **Cable Force** object in the details panel (see Figure 8.23: Details of Cable Force (p. 307)).

Figure 8.23: Details of Cable Force

Details of "Cable Force"	
Definition	
Activity	Unsuppressed
Scope: Cable Attachment	
Scoping Method	Geometry Selection
Geometry	1 Face
Applied By	Surface Effect
Selection X (Centroid)	0 m
Selection Y (Centroid)	0 m
Selection Z (Centroid)	-3 m
Cable Details	
Cable Index in HR Analysis	1
Cable Type	Linear
Connected To	Ground
Structure Connection Point X	0 m
Structure Connection Point Y	0 m
Structure Connection Point Z	-3 m
Imported Cable Force	
Force X	-0.010814 N
Force Y	3.151182 N
Force Z	-752247.624994 N

The **Activity** field allows you to set the suppression state of the **Cable Force** object. Set it to Suppressed if you want to exclude the object from the analysis.

8.1.11.1.1. Scope: Cable Attachment

Use **Scope: Cable Attachment** to select the topology that you want to map cable forces onto. Topology can be selected either by Named Selection, or by Geometry Selection from the graphical window. Cable forces can be mapped on to one or more faces, edges, vertices, mesh nodes or mesh element faces.

When the topology selection type is faces or mesh element faces, the **Applied By** option can be used to apply the imported force by Direct or by Surface Effect approach. The Surface Effect option applies force using the surface effect elements created on the top of the scoped geometry. The Direct option applies force directly onto the faces of solid or shell elements in 3D analyses.

Once a topology selection has been made, the **Selection X/Y/Z (Centroid)** is reported and should be checked against the **Structure Connection Point X/Y/Z** position displayed in the Cable Details section.

Note:

A warning will be issued if there is a significant difference between the selection centroid and the connection point position in the hydrodynamic model.

8.1.11.1.2. Cable Details

Once you have selected some topology in the Static Structural model, you must associate this with the corresponding Cable in the time domain Hydrodynamic Response analysis. The **Cable Index in HR Analysis** option lists the indices of the Cables that are defined in the upstream time domain Hydrodynamic Response analysis, and which are attached to the structure selected in the Hydrodynamic Pressure object. Select the appropriate cable index from this list.

Note:

- In the upstream Hydrodynamic system, when you select the Connections object in the Outline tree, a Hydrodynamic Response Connections Summary (p. 100) table will be displayed. This allows you to associate the connections in the hydrodynamic model with their corresponding indices in the analysis.
- For a cable in the upstream Hydrodynamic system that includes at least one pulley attachment, each cable length between the attachments is assigned its own cable index. Where both attachments for a given cable index are located on the structure selected in the Hydrodynamic Pressure object, the **Cable Index in HR Analysis** menu will include an option to select either the First or Second Attachment.

Once a cable index has been selected, the **Structure Connection Point X/Y/Z** definition position will be displayed for reference. The position is transformed into the Static Structural axis system, using the Axis Transformation defined in the Hydrodynamic Pressure object.

Also displayed is the **Cable Type**, which is either 'Linear' or 'Nonlinear', depending on the definition in the hydrodynamic model; and **Connected To**, which will show 'Ground' for a cable between a structure and a Fixed Point, or displays the name of the other structure in the Hydrodynamic Response analysis for a cable between two structures.

8.1.11.1.3. Displayed Forces

In some instances, the imported forces shown in the graphical window may be out of date. In this case, the **Refresh Graphical Window** option will appear, in an invalid state, and should be changed from Required to Not Required. The option will then disappear, and the state of the **Cable Force** object will change to show that it requires an update. Right-click the **Cable Force** object and select Generate to display the correct imported forces.

8.1.11.1.4. Imported Cable Force

Once you have Generated the **Cable Force** object, the imported **Force X/Y/Z** components are reported. Where applicable, values are displayed for the time step selected in **Display for Time Step** as defined in the associated Hydrodynamic Pressure object.

8.1.11.2. Generating Cable Forces

Once the **Cable Force** object has been configured, right-click it and select Generate to start the force import process. The time step(s) for this process are copied from the Hydrodynamic Pressure object, which must be Generated first.

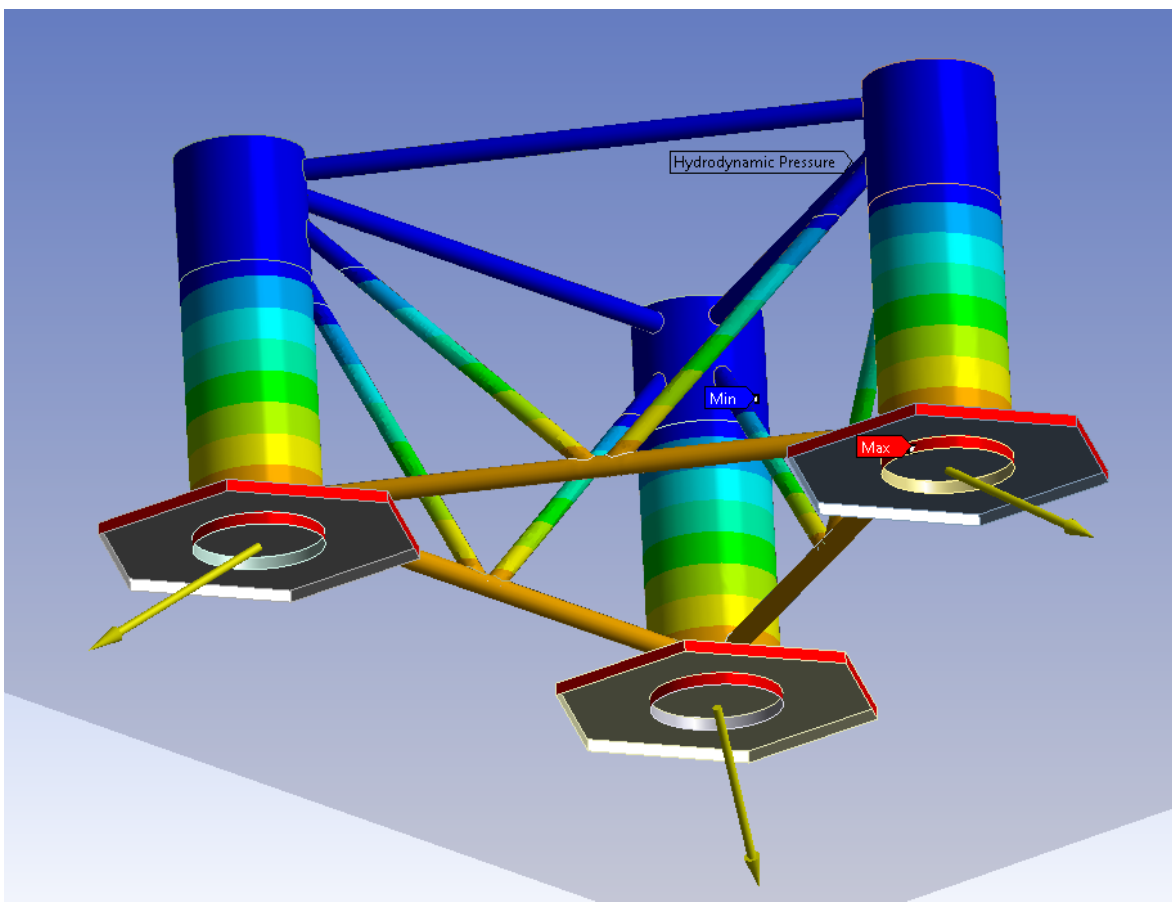
The output data files from the time domain Hydrodynamic Response calculation will be read to determine the cable forces for the requested time step(s).

Note:

The Hydrodynamic Pressure Add-on will take into account any difference in the unit systems that are employed in the time domain Hydrodynamic Diffraction and Static Structural systems.

The transferred cable forces are displayed as yellow arrows in the graphical window (see Figure 8.20: Imported Forces in the Static Structural System).

Figure 8.24: Imported Forces in the Static Structural System



8.1.12. Tether Forces/Moments from a Time Domain Hydrodynamic Response Analysis

Clicking the 

icon will add a **Tether Force** object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Tether Force** from the context menu.

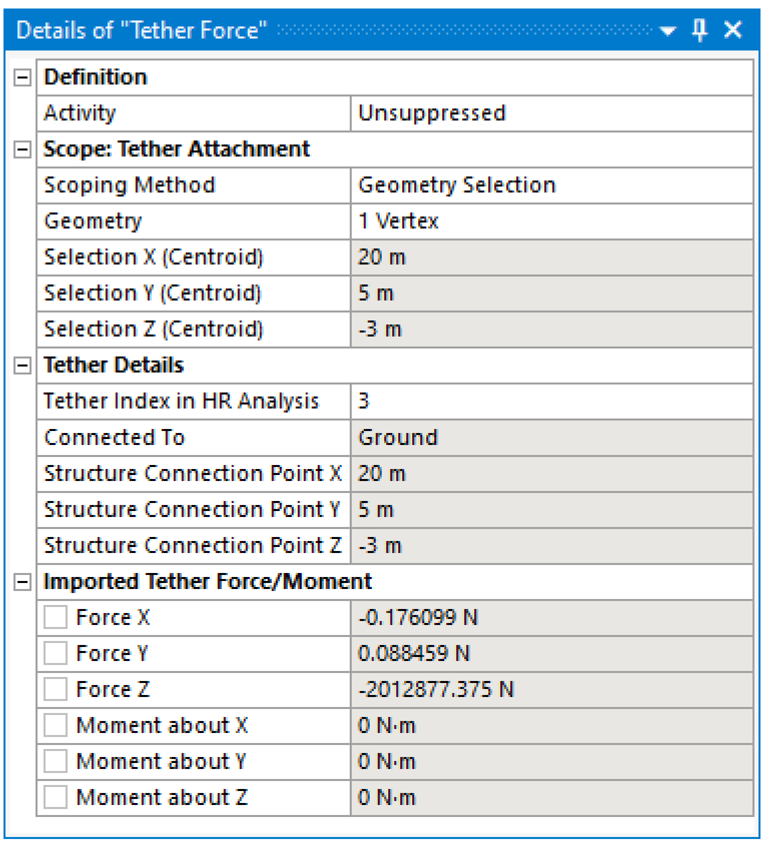
You can only add **Tether Force** objects after a Hydrodynamic Pressure object has been added to the Static Structural analysis.

You must ensure that the number of **Tether Force** objects in the Static Structural analysis is the same as the corresponding number of Tethers attached to the selected structure in the upstream time domain Hydrodynamic Response analysis. Otherwise, there would be an imbalance of forces resulting in an acceleration of the structure in the Static Structural analysis.

8.1.12.1. Configuring the Tether Force Object

You can configure options for the **Tether Force**object in the details panel (see Figure 8.25: Details of Tether Force (p. 310)).

Figure 8.25: Details of Tether Force



Details of "Tether Force"

Definition	
Activity	Unsuppressed
Scope: Tether Attachment	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Selection X (Centroid)	20 m
Selection Y (Centroid)	5 m
Selection Z (Centroid)	-3 m
Tether Details	
Tether Index in HR Analysis	3
Connected To	Ground
Structure Connection Point X	20 m
Structure Connection Point Y	5 m
Structure Connection Point Z	-3 m
Imported Tether Force/Moment	
Force X	-0.176099 N
Force Y	0.088459 N
Force Z	-2012877.375 N
Moment about X	0 N·m
Moment about Y	0 N·m
Moment about Z	0 N·m

The **Activity** field allows you to set the suppression state of the **Tether Force** object. Set it to Suppressed if you want to exclude the object from the analysis.

8.1.12.1.1. Scope: Tether Attachment

Use Scope: Tether Attachment to select the topology that you want to map tether forces/moments on to. Topology can be selected either by Named Selection, or by Geometry Selection from the graphical window. Tether forces/moments can be mapped on to one or more vertices.

Once a topology selection has been made, the **Selection X/Y/Z (Centroid)** is reported and should be checked against the **Structure Connection Point X/Y/Z** position displayed in the Tether Details section.

Note:

A warning will be issued if there is a significant difference between the selection centroid and the connection point position in the hydrodynamic model.

8.1.12.1.2. Tether Details

Once you have selected some topology in the Static Structural model, you must associate this with the corresponding Tether in the time domain Hydrodynamic Response analysis. The **Tether Index in HR Analysis** option lists the indices of the Tethers that are defined in the upstream time domain Hydrodynamic Response analysis, and which are attached to the structure selected in the Hydrodynamic Pressure object. Select the appropriate tether index from this list.

Note:

In the upstream Hydrodynamic system, when you select the Connections object in the Outline tree, a Hydrodynamic Response Connections Summary (p. 100) table will be displayed. This allows you to associate the connections in the hydrodynamic model with their corresponding indices in the analysis.

Once a tether index has been selected, the **Structure Connection Point X/Y/Z** definition position will be displayed for reference. The position is transformed into the Static Structural axis system, using the Axis Transformation defined in the Hydrodynamic Pressure object.

For a **Tether Force** object, the **Connected To** field will always show 'Ground', reflecting the connectivity of tether connections in a hydrodynamic model.

8.1.12.1.3. Displayed Forces

In some instances, the imported forces shown in the graphical window may be out of date. In this case, the **Refresh Graphical Window** option will appear, in an invalid state, and should be changed from Required to Not Required. The option will then disappear, and the state of the **Tether Force** object will change to show that it requires an update. Right-click the **Tether Force** object and select Generate to display the correct imported forces.

8.1.12.1.4. Imported Tether Force/Moment

Once you have Generated the **Tether Force** object, the imported **Force X/Y/Z** and **Moment about X/Y/Z** components are reported. Where applicable, values are displayed for the time step selected in **Display for Time Step** as defined in the associated Hydrodynamic Pressure object.

8.1.12.2. Generating Tether Forces/Moments

Once the **Tether Force** object has been configured, right-click it and select Generate to start the force/moment import process. The time step(s) for this process are copied from the Hydrodynamic Pressure object, which must be Generated first.

The output data files from the time domain Hydrodynamic Response calculation will be read to determine the tether forces/moments for the requested time step(s).

Note:

The Hydrodynamic Pressure Add-on will take into account any difference in the unit systems that are employed in the time domain Hydrodynamic Diffraction and Static Structural systems.

The transferred tether forces are displayed as orange arrows in the graphical window.

8.1.13. Joint Forces/Moments from a Time Domain Hydrodynamic Response Analysis

Clicking the 

icon will add a **Joint Force** object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Joint Force** from the context menu.

You can only add **Joint Force** objects after a Hydrodynamic Pressure object has been added to the Static Structural analysis.

You must ensure that the number of **Joint Force** objects in the Static Structural analysis is the same as the corresponding number of Joints attached to the selected structure in the upstream time domain Hydrodynamic Response analysis. Otherwise, there would be an imbalance of forces resulting in an acceleration of the structure in the Static Structural analysis.

8.1.13.1. Configuring the Joint Force Object

You can configure options for the **Joint Force** object in the details panel (see Figure 8.26: Details of Joint Force (p. 313)).

Figure 8.26: Details of Joint Force

Details of "Joint Force"

Definition	
Activity	Unsuppressed
Scope: Joint Attachment	
Scoping Method	Geometry Selection
Geometry	4 Vertices
Selection X (Centroid)	0 m
Selection Y (Centroid)	0 m
Selection Z (Centroid)	-3 m
Joint Details	
Joint Index in HR Analysis	1
Joint Type	Ball and Socket
Connected To	Ground
Structure Connection Point X	0 m
Structure Connection Point Y	0 m
Structure Connection Point Z	-3 m
Imported Joint Force/Moment	
Force X	-186095.880658 N
Force Y	-273033.10598 N
Force Z	523655.637576 N
Moment about X	-5777.312206 N·m
Moment about Y	-4598.320929 N·m
Moment about Z	1866.961599 N·m

The **Activity** field allows you to set the suppression state of the **Joint Force** object. Set it to Suppressed if you want to exclude the object from the analysis.

8.1.13.1.1. Scope: Joint Attachment

Use **Scope: Joint Attachment** to select the topology that you want to map joint forces/moments on to. Topology can be selected either by Named Selection, or by Geometry Selection from the graphical window. Joint forces/moments can be mapped on to one or more vertices.

Once a topology selection has been made, the **Selection X/Y/Z (Centroid)** is reported and should be checked against the **Structure Connection Point X/Y/Z** position displayed in the Joint Details section.

Note:

A warning will be issued if there is a significant difference between the selection centroid and the connection point position in the hydrodynamic model.

8.1.13.1.2. Joint Details

Once you have selected some topology in the Static Structural model, you must associate this with the corresponding Joint in the time domain Hydrodynamic Response analysis. The **Joint Index in HR Analysis** option lists the indices of the Joints that are defined in the upstream time domain Hydrodynamic Response analysis, and which are attached to the structure selected in the Hydrodynamic Pressure object. Select the appropriate joint index from this list.

Note:

In the upstream Hydrodynamic system, when you select the Connections object in the Outline tree, a Hydrodynamic Response Connections Summary (p. 100) table will be displayed. This allows you to associate the connections in the hydrodynamic model with their corresponding indices in the analysis.

Once a joint index has been selected, the **Structure Connection Point X/Y/Z** definition position will be displayed for reference. The position is transformed into the Static Structural axis system, using the Axis Transformation defined in the Hydrodynamic Pressure object.

Also displayed is the **Joint Type**, which is ‘Rigid’, ‘Hinged’, ‘Universal’ or ‘Ball and Socket’, depending on the definition in the hydrodynamic model; and **Connected To**, which will show ‘Ground’ for a joint between a structure and a Fixed Point, or displays the name of the other structure in the Hydrodynamic Response analysis for a joint between two structures.

8.1.13.1.3. Displayed Forces

In some instances, the imported forces shown in the graphical window may be out of date. In this case, the **Refresh Graphical Window** option will appear, in an invalid state, and should be changed from Required to Not Required. The option will then disappear, and the state of the **Joint Force** object will change to show that it requires an update. Right-click the **Joint Force** object and select Generate to display the correct imported forces.

8.1.13.1.4. Imported Joint Force/Moment

Once you have Generated the **Joint Force** object, the imported **Force X/Y/Z** and **Moment about X/Y/Z** components are reported. Where applicable, values are displayed for the time step selected in **Display for Time Step** as defined in the associated Hydrodynamic Pressure object.

8.1.13.2. Generating Joint Forces/Moments

Once the **Joint Force** object has been configured, right-click it and select Generate to start the force/moment import process. The time step(s) for this process are copied from the Hydrodynamic Pressure object, which must be Generated first.

The output data files from the time domain Hydrodynamic Response calculation will be read to determine the joint forces/moments for the requested time step(s).

Note:

The Hydrodynamic Pressure Add-on will take into account any difference in the unit systems that are employed in the time domain Hydrodynamic Diffraction and Static Structural systems.

The transferred joint forces are displayed as red arrows in the graphical window.

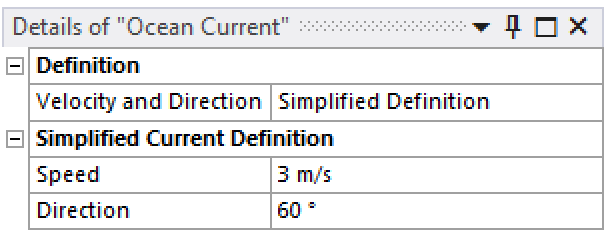
8.1.14. Ocean Current

Clicking the 

icon adds an Ocean Current object to the Static Structural analysis.

Alternatively, you can right-click the Static Structural analysis and select **Insert** → **Ocean Current** from the context menu.

You may define any number of Ocean Current objects in your analysis, provided that the Static Structural system is linked to a Hydrodynamic Diffraction system (not a time domain Hydrodynamic Response system). However, they will only have an effect on the calculation if they are selected in the **Include Current** field of the Hydrodynamic Pressure object.



Use the **Velocity and Direction** property to set the way in which the current is defined. The available options are:

- Simplified Definition: Current is defined by a constant current **Speed** and **Direction**
- Advanced Definition: Current is defined by varying **Speed** and **Direction** values at specified **Depth** increments in the **Advanced Current Definition** tabular data input. Click the  icon to add more rows to the table.

8.1.15. Theory

This section discusses the theory applicable to the Hydrodynamic Pressure Mapping Add-on for the mapping of pressures from a Hydrodynamic Diffraction system.

8.1.15.1. Hydrodynamic and Hydrostatic Pressures on Surfaces

The real and imaginary parts of the incident, diffracted, radiated and hydrostatic-varying pressure components calculated in the Hydrodynamic Diffraction analysis are used to evaluate the total hydrodynamic pressure at each structure node for the specified incident wave amplitude and phase angle:

$$P_\theta = a\left(P_r \cos\theta + P_i \sin\theta\right) \tag{8.1}$$

Where P_θ is the total hydrodynamic pressure at the required phase angle θ, with θ=0° when the incident wave crest passes the center of gravity of the structure; a is the specified incident wave amplitude; and P_r and P_i are the real and imaginary parts, respectively, of the summed hydrodynamic (incident, diffracted, radiated and hydrostatic-varying) pressure components.

Where hydrostatic pressure is also included, the total pressure P is the summation of the hydrodynamic and hydrostatic parts:

$$P = P_\theta + P_s \tag{8.2}$$

Where the hydrostatic pressure P_s is calculated as:

$$P_s = \rho g z \tag{8.3}$$

In which ρ and g are the water density and acceleration due to gravity, respectively, as specified in the Hydrodynamic Diffraction analysis; and z is the depth of the structure node below the waterline in the Static Structural axis system (after any Axis Transformation has been applied).

8.1.15.2. Inertia and Viscous Drag Forces on Line Bodies

Distributed loads on submerged Line Bodies are calculated using the Morison equation:

$$F = \frac{1}{2}\rho D C_d \left(u_f - u_s\right)\left|u_f - u_s\right| + \rho A C_m \dot{u}_f - \rho A C_a \dot{u}_s \tag{8.4}$$

Where:

F is the total Morison force per unit length;

D is the Line Body diameter (which should take into account any marine growth);

C_d is the viscous drag coefficient;

u_f is the flow velocity normal to the Line Body (comprising the diffracted wave particle velocity and any defined current);

u_s is the structure velocity normal to the Line Body;

$\dot{u}_f$ and $\dot{u}_s$ are the flow and structure accelerations, respectively, normal to the Line Body;

A is the cross-sectional area of the Line Body;

C_a is the added mass coefficient, from which:

$$C_m = 1 + C_a \tag{8.5}$$

Where C_m is the inertia coefficient. Setting C_m =0.0 turns off loading due to the diffracted wave and added mass effects (second and third terms of Equation 8.4 (p. 316)); otherwise the valid range of values are $C_m \geq 1.0$. Similarly, viscous drag (first term of Equation 8.4 (p. 316)) can be ignored by setting C_d =0.0. For non-cylindrical cross sections, the Morison equation is applied separately in each direction normal to the Line Body.

The Hydrodynamic Pressure Mapping Add-on determines whether the ends of each beam element are above or below the local water surface and loads the Line Body accordingly. For elements that cut the water surface the loading is applied over the wetted length only.

8.1.16. Limitations

There exist some limitations to the geometry and model that can be used with the Hydrodynamic Pressure Mapping Add-on. These are described below.

In general:

- In the structure mass checks, the only included mass is that which comes from:
  - The structural material itself
  - Point Masses (direct or remote attachment)
  - Distributed Masses
  - Additional Thickness definitions

  Any other mass (for example, due to Layered Sections or Command objects that define additional mass elements directly in APDL) will not be included. This may mean that the center of gravity about which rotational accelerations are applied is incorrect.

Specific to pressure mapping from a Hydrodynamic Diffraction system:

- It is not possible to map loads onto Line Bodies from a Hydrodynamic Diffraction analysis that includes multiple structures.
- When the imported Geometry includes Line Bodies, it is not possible to map hydrodynamic loads onto Cross Sections of Type: ASEC (arbitrary) or MESH (user-defined mesh).
- Line Bodies of Model Type: Link (element type LINK167) are not currently supported.
- Where the Pressure Mapping method is set to Interpolated, you are limited to less than five million mesh nodes.
- When an internal Line Body is used to model the viscous drag acting on part of a structure, and that part of the structure is also modelled by diffracting panels, the wave particle kinematics around the Line Body will include the diffracted wave effects due to that part of the structure.

Specific to pressure mapping from a Hydrodynamic Response system:

- It is not possible to map loads from a frequency domain Hydrodynamic Response system (stability analysis or frequency statistical analysis).
- Apart from cable forces, tether forces/moments and joint forces/moments, which can be mapped using the **Cable Force**, **Tether Force**, and **Joint Force** objects, and Morison loads on Line Bodies or disc elements, the structure selected for time domain Hydrodynamic Response pressure mapping must not include features which will induce additional forces or moments on the structure, such as buoyancy elements, fenders, thrusters, or other viscous drag effects.
- Pressure mapping cannot be performed for hydrodynamic models that include Moonpools.

8.2. The Offshore Add-on

The Offshore Add-on exposes the family of OCEAN commands that are offered in Mechanical APDL. In this way the hydrodynamic loads on submerged beam and pipe elements – resulting from the motion of the structure through the fluid, or from the fluid motion around the structure due to current or waves – can be included in Static Structural, Transient Structural, Modal and Harmonic Response analyses.

8.2.1. Introduction

The purpose of the Offshore Add-on is to expose the OCEAN Mechanical APDL commands in the Mechanical application. The Add-on allows you to define:

- An Ocean Environment (p. 319) object consisting of general ocean properties, global structural coefficients, and incident wave properties (OCTYPE, BASIC)
- Ocean Current (p. 326) objects with either simple or tabulated input of current speed and direction (OCTYPE, CURR)
- User Defined Spectrum (p. 327) objects for non-formulated incident wave spectrum definitions
- Harmonic Wave Load (p. 327) objects for periodic waves in a Harmonic Response analysis (HROCEAN)
- Local adjustments to structural coefficients and member flooding, either by Geometry-Based Variation (p. 328) (OCZONE, ZLOC) or Component-Based Variation (p. 329) (OCZONE, COMP)
- Pipe-in-Pipe (p. 330) objects, where the internal and external pipes may be flooded or filled with an internal fluid (OCZONE, PIP)
- Added Mass (p. 332) objects to define added mass due to hydrodynamic effects on selected surfaces

The Offshore Add-on can be used with the following Mechanical analysis types:

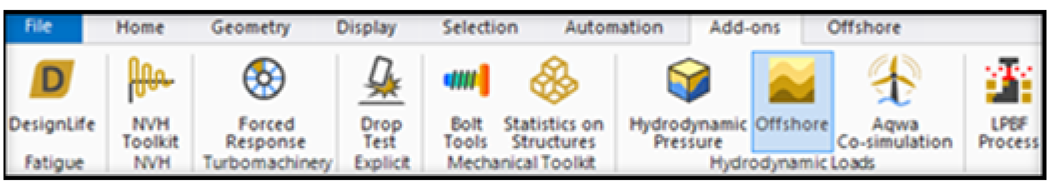
- Static Structural (A in Figure 8.27: Valid Workflows for the Offshore Add-on (p. 319))
- Transient Structural (B in Figure 8.27: Valid Workflows for the Offshore Add-on (p. 319))
- Modal (C in Figure 8.27: Valid Workflows for the Offshore Add-on (p. 319))
- Harmonic Response (using Full Solution Method only, D in Figure 8.27: Valid Workflows for the Offshore Add-on (p. 319))
- Modal with Static Structural pre-stress condition (E-F in Figure 8.27: Valid Workflows for the Offshore Add-on (p. 319))
- Harmonic Response with Static Structural pre-stress condition (G-H in Figure 8.27: Valid Workflows for the Offshore Add-on (p. 319))

Figure 8.27: Valid Workflows for the Offshore Add-on

8.2.2. Loading the Offshore Add-on

To make the capabilities of the Offshore Add-on available, click the **Offshore** icon in the **Add-ons** Ribbon. The icon will be highlighted in blue, indicating that the Add-on is loaded.

Figure 8.28: Offshore Add-on Showing Loaded Status



Note:

Command Snippets may override the OCEAN commands that are written to the Mechanical APDL input file. It is recommended to check the output Solution Information carefully if you run an analysis which contains both Offshore objects and Command Snippets.

8.2.3. Defining an Ocean Environment Object

The first step in running a simulation with Offshore capabilities is to add an Ocean Environment object to the analysis. Once the Ocean Environment is added, you will be able to add other Offshore-specific objects. For a Harmonic Response analysis, you should set the Analysis Settings **Solution Method** option to Full.

You can only have one active Ocean Environment object in an analysis. Additional Ocean Environment objects may only be created once any existing Ocean Environment objects have been deleted or suppressed.

Figure 8.29: Ocean Environment Details

Details of "Ocean Environment"	
Definition	
Activity	Unsuppressed
General	
Ocean Mean Water Level	1 m
Ocean Depth	12 m
Ocean Density	1025 kg/m³
Ocean Viscosity	0.00108
Gravitational Acceleration	9.80665 m/s²
Structural Coefficients	
Default Member Flooding	False
Added Mass Coefficient in Y (CaY)	0.91
Added Mass Coefficient in Z (CaZ)	0.92
Buoyancy Force Ratio	1.01
Tabular Input Options	Z Level
Coefficients by Depth	Tabular Data
Soil-Pile Interaction	
Include Soil-Pile Interaction	No
Wave Options	
Wave Theory	Shell New Wave
Wave Spectrum	Pierson-Moskowitz
Number of Wave Components	50
Wave Case Definition	Tabular Data
Wave Details Output	Yes
Apparent Period Adjustment	No
Stretching Depth Factor	0.5
Delta Stretching Parameter	0.3
Wave Spread Angle	0 °
Time Offset	0 s
Position Offset	0 m
Use Evolving Wave	No

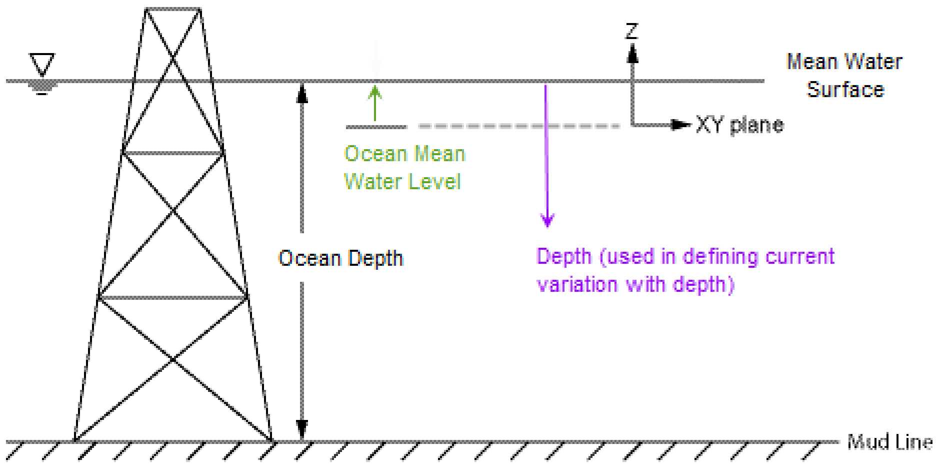
8.2.3.1. Definition Properties

The **Activity** field allows you to set the suppression state of the Ocean Environment object. Set it to Suppressed if you want to exclude the object from the analysis.

8.2.3.2. General Properties

The ocean geometry is defined by the **Ocean Mean Water Level** (mean water surface Z position with respect to the global origin) and the **Ocean Depth** (vertical distance from the Ocean Mean Water Level to the seabed).

Figure 8.30: Ocean Geometry



The material properties of the seawater are defined by the **Ocean Density** and **Ocean Viscosity**. The latter is only used where the global structural coefficients are set to vary by Reynolds number (see Structural Coefficients (p. 321)). The default values of Ocean Density and Ocean Viscosity are set to 1025 kg/m³ and 0.00108 Pa s, respectively, which are applicable for seawater at a temperature of 20°C and a salinity of 0.035 kg/kg.

The vertical acceleration due to gravity is defined by the **Gravitational Acceleration** property, which defaults to the standard value of 9.80665 m/s².

8.2.3.3. Structural Coefficients

These coefficients apply to all submerged line bodies in the Geometry.

Default Member Flooding controls whether tubular line bodies are considered to be flooded by the surrounding seawater or not. (An internal fluid for tubular line bodies can also be defined with Component-Based Variation (p. 329) or Pipe-in-Pipe (p. 330) objects.)

Added Mass Coefficient in Y (CaY) and **Added Mass Coefficient in Z (CaZ)** control the ratio of added mass to displaced mass, where the added mass is the mass of seawater that moves with the line body in a dynamic analysis, in the two directions perpendicular to the line body axis.

Buoyancy Force Ratio can be used to adjust the buoyancy of line bodies relative to the buoyancy force based on the line body displacement and the defined Ocean Density. This may be useful when accounting for external pipe features that are not explicitly modelled.

Drag and inertia coefficients are also defined for all submerged line bodies. These coefficients can be set to vary by vertical Z position, or by the Reynolds number of the relative fluid flow around the line body; set **Tabular Input Options** to Z Level for the former, or Reynolds Number for the latter.

Depending on your selection, you can then set **Coefficients by Depth** or **Coefficients by Reynolds Number** using the tabular data input. The independent variable is entered in the first column, followed by the **Drag Coefficient in Y (CdY)**, **Drag Coefficient in Z (CdZ)**, **Drag Coefficient in X (CtX)**, **Inertia Coefficient in Y (CmY)** and **Inertia Coefficient in Z (CmZ)**. All coefficients are defined in the local line body axes. Click the  icon to add more rows to the table. Duplicate entries of **Z Level** or **Reynolds Number** are ignored.

Note:

At least one row of tabular data must be defined.

The values for CaY/CaZ will be used as defined; there is no assumption that $C_a = C_m - 1.0$.

8.2.3.4. Soil-Pile Interaction

You can choose to **Include Soil-Pile Interaction**. Setting this option to Yes allows you to specify a **Soil-Pile Data File**, which may be a Mechanical APDL macro (`.mac`) file or some other APDL code snippet. The specified file is included in the solver input file via the /INPUT command.

Examples of soil-pile interaction macros may be obtained through a Service Request on the Ansys customer site.

8.2.3.5. Wave Options

The following options are used to define wave and current conditions for the simulation.

Wave Theory

In Static Structural and Transient Structural analyses, you can define a **Wave Theory** to be used in the calculation. The available options are:

- No Water Motion: no fluid motion will be accounted for in the calculation
- Current Forces Only: no wave motion will be accounted for, but the effect of an Ocean Current (p. 326) may be included
- Airy: small amplitude first-order periodic wave
- Wheeler: Airy wave with Wheeler stretching to account for the variation of water depth with wave surface
- Stokes: fifth-order wave theory for periodic waves in intermediate or deep water
- Stream Function: stream function wave theory suited to periodic waves in shallower water
- Random: a random, but repeatable, non-periodic combination of Airy waves fitted to a selected wave spectrum
- Shell New Wave: similar to a random wave, but with a modified surface coefficient to better represent the most probable maximum condition of a real sea
- Constrained: embeds a Shell new wave into a random wave
- Diffracted: allows pressures and motions to be imported from a Hydrodynamic Diffraction (p. 33) analysis

- User Defined Wave: calls the Mechanical APDL subroutine userPartVelAcc for the input of user-defined wave and current information

A more complete description of these wave theories is provided in Hydrodynamic Loads in the *Mechanical APDL Theory Reference* .

In a Modal analysis, or where the imported Geometry contains only solid bodies, water motion is not permitted. In a Harmonic Response analysis, the wave information is entered via the Harmonic Wave Load (p. 327) object. Harmonic Response analyses are restricted to periodic waves (Airy, Wheeler, Stokes, Stream Function or User Defined Wave).

For Static Structural and Transient Structural analyses, if the imported Geometry does not contain any line bodies but does contain surface bodies, you can only choose between No Water Motion or Diffracted. If the imported Geometry does contain line bodies, you are free to select from any of the wave theories listed above.

Wave Spectrum

Where the Wave Theory is selected as Random, Shell New Wave or Constrained, an additional **Wave Spectrum** option is shown. This is used to select a wave spectrum to which the combination of linear waves is fitted. The available options are:

- Pierson-Moskowitz: a formulated spectrum defined by peak wave period and significant wave height
- JONSWAP (Joint North Sea Wave Project): a formulated spectrum defined by peak period, significant wave height and a peak enhancement factor
- User Defined: an unformulated spectrum defined by the frequency/spectral energy density data in a User Defined Spectrum (p. 327) object

Wave Case Definition

For any Wave Theory other than No Water Motion, the **Wave Case Definition** tabular data input is used to define the current and/or wave properties for the calculation. The number of rows in the Wave Case Definition table will match the **Number of Steps** defined in the Static Structural or Transient Structural Analysis Settings; a separate load case may be defined for each step of the analysis.

Figure 8.31: Wave Case Definition Tabular Data Input

Wave Case Definition

Load Step	Time Duration	Wave Direction	Current Definition	Wave Period	Wave Height	Phase Shift	Stream Function Order
1	1	30 [deg]	Ocean Current	8.4 [sec]	3.5 [m]	0 [deg]	3
2	2	60 [deg]	Ocean Current 2	9.8 [sec]	4.7 [m]	0 [deg]	10

All wave theories require a **Current Definition** at every step. Use the drop-down menu to select from any of the Ocean Current (p. 326) objects that you have defined in your analysis, or choose Disable Current to exclude current effects for that step.

For all wave theories that include wave motion, a **Wave Direction** may be defined at each step. This sets the direction that the incident wave travels towards, relative to the direction of the global X axis.

For wave theories of Airy, Wheeler, Stokes, Stream Function, Diffracted and User Defined Wave, the wave properties are specified by the **Wave Period**, **Wave Height** and **Phase Shift**. Setting the Wave Period or Wave Height to zero will disable the wave motion for a given step. For a Stream Function wave, the **Stream Function Order** may also be defined; for a User Defined Wave, the **Wave Length** may be defined.

For Random, Shell New Wave and Constrained waves, the wave property definitions depend on the selected **Wave Spectrum**. These are summarized in Table 1.

Table 8.1: Wave Properties for Non-Periodic Wave Theories

		Wave Spectrum		
		Pierson-Moskowitz	**JONSWAP**	**User Defined**
Wave Theory	**Random**	Peak Period Significant Wave Height	Peak Period Significant Wave Height Peak Enhancement Factor	User Defined Spectrum
	Shell New Wave	Peak Period Max Wave Crest Amplitude	Peak Period Max Wave Crest Amplitude Peak Enhancement Factor	User Defined Spectrum
	Constrained	Peak Period Significant Wave Height	Peak Period Significant Wave Height Peak Enhancement Factor	User Defined Spectrum

Where applicable, in the **User Defined Spectrum** column of the **Wave Case Definition** table, you should use the drop-down menu to select from any of the User Defined Spectrum (p. 327) objects that you have defined in your analysis.

Additional Options

For periodic waves (Airy, Wheeler, Stokes, Stream Function or User Defined Wave) the **Force Application** option allows you to set the position/orientation of elements relative to the wave surface for the force calculation. The available options are:

- Act on Elements at their Actual Location
- Use Wave Peak: elements are assumed to be positioned at the wave peak

- Use Wave Trough: elements are assumed to be positioned at the wave trough
- Vertical Upward Force Only
- Vertical Downward Force Only

The **Wave Details Output** option indicates that a description of the defined wave should be written to the Solution Information. For non-periodic waves, this includes details of the wave components.

For Airy, Wheeler, Stokes and Stream Function waves, the **MacCamy-Fuchs Adjustment** option allows you to include or exclude a correction to the structural inertia coefficients to account for diffraction effects. This is relevant for larger diameter line bodies in shorter wavelength waves.

For Random, Shell New Wave and Constrained waves:

- The **Number of Wave Components** allows you to set the number of constituent waves for the wave combination
- The **Apparent Period Adjustment** option allows you to include or exclude a calculation of the apparent period where a wave is superimposed over a current
- The **Stretching Depth Factor** is used to provide the wave kinematics under the wave crest, in a form of Wheeler stretching known as delta stretching
- The **Delta Stretching Parameter** sets the shape of the delta stretching function

For Random and Constrained waves:

- The **Wave Kinematics Factor** is used to account for wave spreading by modifying the horizontal wave kinematics, where a value of 1.0 corresponds to no spreading
- The **Initial Seed** sets the seed for random phase angle generation

For Shell New Wave and Constrained waves, the **Time Offset** and **Position Offset** set the time and position offsets at which the maximum wave crest will occur.

For Shell New Waves the **Wave Spread Angle** is used to compute a wave spreading factor to modify the horizontal wave kinematics for near-unidirectional seas, while the **Use Evolving Wave** option may be used to modify the component wave phase angles over the simulation.

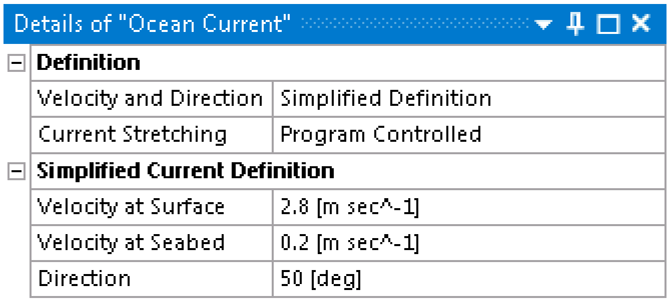
8.2.3.6. Pressure Mapping

Where the Wave Theory is set to Diffracted, additional pressure mapping options are displayed. Use the **Hydrodynamic Data File** selection dialog box to choose an Aqwa-generated hydrodynamic database (.ahd) file, which will be used to determine hydrodynamic loads on line body elements. Where the imported Geometry includes surface bodies, use the **Scoping Method** and **Geometry/Named Selection** options to define the surfaces onto which the hydrodynamic pressures will be transferred.

8.2.4. Ocean Current

You may define any number of Ocean Current objects in your analysis, but they will only have an effect on the calculation if they are selected in the **Current Definition** column of the Ocean Environment Wave Case Definition (p. 323) table.

Figure 8.32: Ocean Current Details


Details of "Ocean Current"

Definition	
Velocity and Direction	Simplified Definition
Current Stretching	Program Controlled
Simplified Current Definition	
Velocity at Surface	2.8 [m sec^-1]
Velocity at Seabed	0.2 [m sec^-1]
Direction	50 [deg]

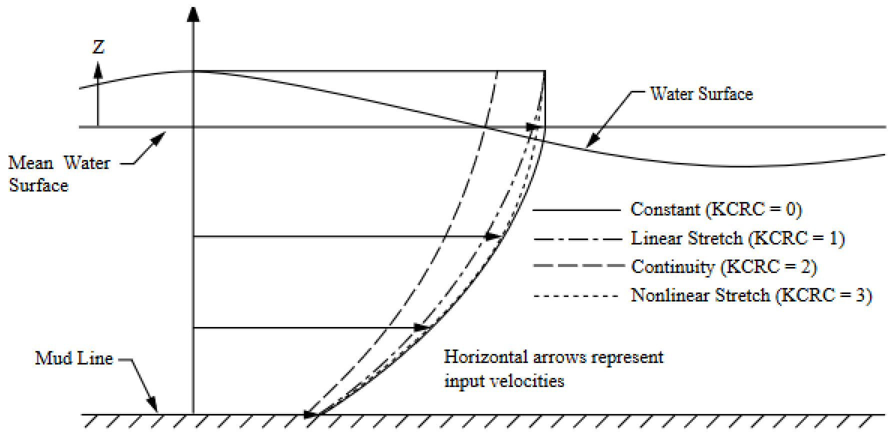
Use the **Velocity and Direction** property to set the way in which the current is defined. The available options are:

- Simplified Definition: Current is defined by a **Velocity at Surface**, a **Velocity at Seabed** and a constant current **Direction**
- Advanced Definition: Current is defined by varying **Velocity** and **Direction** values at specified **Depth** increments in the **Advanced Current Definition** tabular data input. Click the icon to add more rows to the table.

For both types of current definition the **Current Stretching** option allows adjustments to the current profile to account for cases where the wave amplitude is large relative to the **Ocean Depth**, such that significant wave-current interactions exist. The available options are:

- Off: no profile adjustment, with constant current velocity above the **Ocean Mean Water Level**.
- Linear: linearly stretch or compress the profile from the seabed to the wave surface.
- Linear and Flow Continuity: as for Linear, but with a further adjustment to maintain flow continuity. This option requires the current direction to be constant with depth.
- Nonlinear: where **Wave Theory** is set to Random, Shell New Wave or Constrained, perform a nonlinear stretching or compression of the current profile from the seabed to the wave surface.
- Program Controlled: sets **Current Stretching** to:
  - Nonlinear when the **Wave Theory** is Random, Shell New Wave or Constrained
  - Linear and Flow Continuity if **Velocity and Direction** is set to Simplified Definition, or if current direction is constant with depth in the **Advanced Current Definition** table
  - Linear otherwise

Figure 8.33: Current Stretching Formulations



8.2.5. User Defined Spectrum

You may define any number of User Defined Spectrum objects in your analysis, but they will only be used in the calculation if they are selected in the **User Defined Spectrum** column of the Ocean Environment **Wave Case Definition** (p. 323) table.

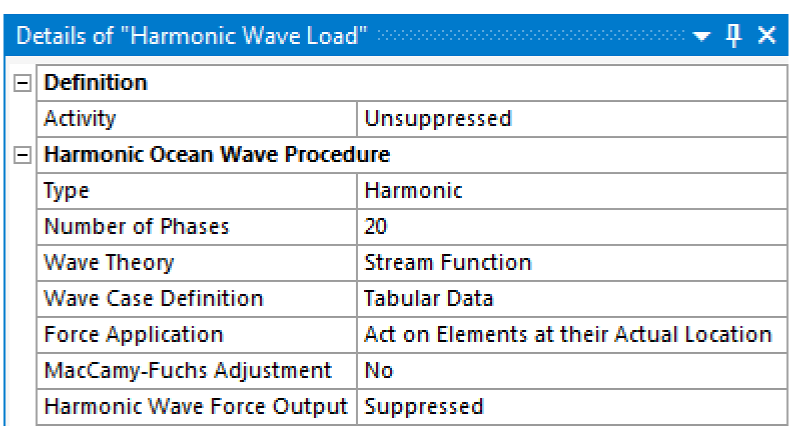
Use the **Spectrum Data** tabular data input to define **Angular Frequency** ordinates and their corresponding **Spectral Energy Density** values. Click the  icon to add more rows to the table. The Spectrum Data table must have at least two pairs of frequency/energy density values.

8.2.6. Harmonic Wave Load

In a Harmonic Response analysis, you should use a **Harmonic Wave Load** object to set up the periodic incident waves to which the structure will be subjected.

You can only have one active Harmonic Wave Load object in an analysis. Additional Harmonic Wave Load objects may only be created once any existing Harmonic Wave Load objects have been deleted or suppressed.

Figure 8.34: Harmonic Wave Load Details


Details of "Harmonic Wave Load"

Definition	
Activity	Unsuppressed
Harmonic Ocean Wave Procedure	
Type	Harmonic
Number of Phases	20
Wave Theory	Stream Function
Wave Case Definition	Tabular Data
Force Application	Act on Elements at their Actual Location
MacCamy-Fuchs Adjustment	No
Harmonic Wave Force Output	Suppressed

The **Activity** field allows you to set the suppression state of the Harmonic Wave Load object. Set it to Suppressed if you want to exclude the object from the analysis.

The Harmonic Wave Load **Type** specifies how the ocean wave information should be included in the Harmonic Response analysis. The available options are:

- Harmonic: calculates real and imaginary loads by the Harmonic Ocean Wave Procedure. The Harmonic Response analysis is performed at a frequency determined from the **Wave Period** specified in the **Wave Case Definition** table (see below).
- Static: equivalent to a Harmonic analysis, but at a frequency of zero.
- Off: deactivates the Harmonic Wave Load.

When the **Type** option is set to Harmonic or Static, use the **Number of Phases** option to specify the number of phases over which forces will be calculated. Select a **Wave Theory** from the periodic Airy, Wheeler, Stokes, Stream Function or User Defined Wave options, as described in Wave Options (p. 322).

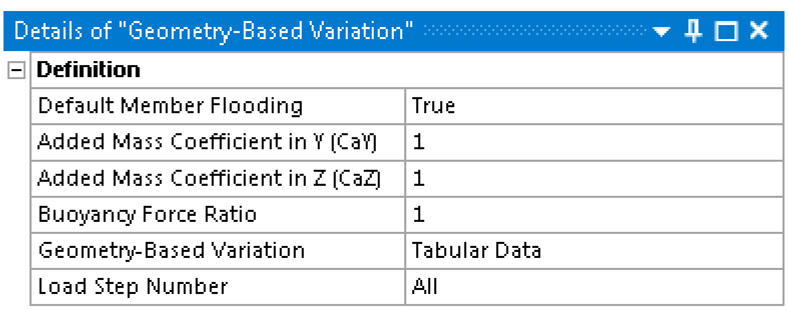
The **Wave Case Definition** tabular data input, **Force Application** option and **MacCamy-Fuchs Adjustment** provide the same functionality as described in Wave Options (p. 322).

Use the **Harmonic Wave Force Output** option to include or exclude additional Harmonic Ocean Wave Procedure output in the Solution Information.

8.2.7. Geometry-Based Variation

You may choose to override the structural coefficients (p. 321) defined in the Ocean Environment object, for a specified range of water depths, using the Geometry-Based Variation object. You may add as many of these to the analysis as required.

Figure 8.35: Geometry-Based Variation Details



Set the **Default Member Flooding** option to True, False or Program Controlled, where the latter inherits the Default Member Flooding defined in the Ocean Environment Structural Coefficients (p. 321) details. The **Added Mass Coefficient in Y (CaY)**, **Added Mass Coefficient in Z (CaZ)** and **Buoyancy Force Ratio** options described in Structural Coefficients apply only over the range of depths defined in the **Geometry-Based Variation** table.

The **Geometry-Based Variation** tabular data input allows you to set the drag and inertia properties at different vertical Z positions. The **Z Level** is entered in the first column, followed by the **Drag Coefficient in Y (CdY)**, **Drag Coefficient in Z (CdZ)**, **Drag Coefficient in X (CtX)**, **Inertia Coefficient in Y (CmY)** and **Inertia Coefficient in Z (CmZ)**. All coefficients are defined in the local line body axes.

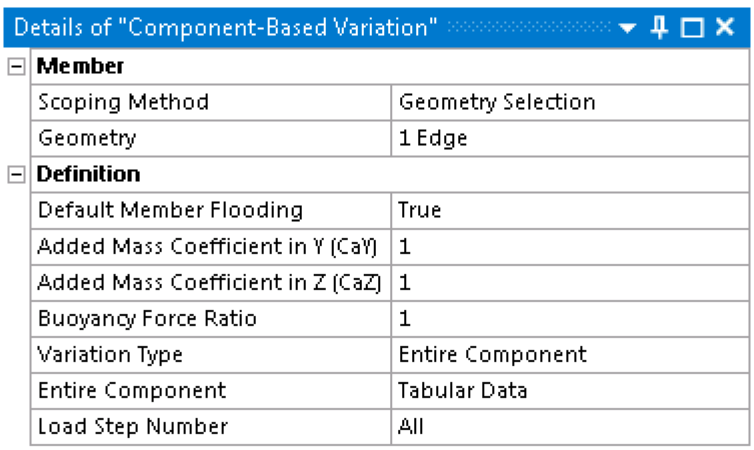
Click the  icon to add more rows to the table. You may add duplicate Z Level values to create a discontinuous variation in structural coefficients with depth, but you should not have more than two rows with the same Z Level. You may also define **Bio-Fouling Added Mass** and **Bio-Fouling Added Thickness** for each water depth to account for the presence of marine growth.

In Static Structural and Transient Structural analyses, you may use the **Load Step Number** option to set the steps at which the Geometry-Based Variation object is applied. By default the object is applied for All steps; if required, change this to Range and specify the **Starting Load Step** and **Ending Load Step** to define a range of steps.

8.2.8. Component-Based Variation

You may choose to override the structural coefficients (p. 321) defined in the Ocean Environment object, for a selection of line bodies, using the Component-Based Variation object. You may add as many of these to the analysis as required.

Figure 8.36: Component-Based Variation Details


Details of "Component-Based Variation"

Member	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Default Member Flooding	True
Added Mass Coefficient in Y (CaY)	1
Added Mass Coefficient in Z (CaZ)	1
Buoyancy Force Ratio	1
Variation Type	Entire Component
Entire Component	Tabular Data
Load Step Number	All

Use the **Scoping Method** and **Geometry**/**Named Selection** options to select the component (line body) to which the structural coefficient adjustments will be applied.

For the Component-Based Variation object, **Default Member Flooding** has four available options:

- True: the selected component is flooded with seawater.
- False: the selected component is not flooded.
- Program Controlled: the option inherits the Default Member Flooding defined in the Ocean Environment Structural Coefficients (p. 321) details.
- Internal Fluid: the selected component is flooded with a fluid, of a density defined by the **Fluid Density**, up to the specified **Free Surface Level**. No other modifications to the structural coefficients of the component may be made with this object.

The **Added Mass Coefficient in Y (CaY)**, **Added Mass Coefficient in Z (CaZ)** and **Buoyancy Force Ratio** options are described in Structural Coefficients (p. 321), but apply only to the selected component.

By default the variation is applied to the whole of the selected component, and the structural coefficients (**Drag Coefficient in Y (CdY)**, **Drag Coefficient in Z (CdZ)**, **Drag Coefficient in X (CtX)**, **Inertia**

Coefficient in Y (CmY), **Inertia Coefficient in Z (CmZ)**, **Bio-Fouling Added Mass** and **Bio-Fouling Added Thickness**) are defined in the **Entire Component tabular data** input.

However, the **Variation Type** can be changed from Entire Component to Piecewise Variation Over Component. This allows you to define drag and inertia properties at different vertical Z positions along the length of the component, in the **Piecewise Variation Over Component** tabular data input. The **Z Level** is entered in the first column, followed by the drag and inertia coefficients, and the bio-fouling properties. All coefficients are defined in the local line body axes. Click the icon to add more rows to the table.

The **Variation Type** can also be set to Program Controlled, which causes the drag and inertia properties to be inherited from those defined in the Ocean Environment Structural Coefficients (p. 321) or any applicable Geometry-Based Variation (p. 328) object. This also applies to regions of the component which lie outside of the range of vertical Z positions defined in the **Piecewise Variation Over Component** table when the **Variation Type** is set to Piecewise Variation Over Component.

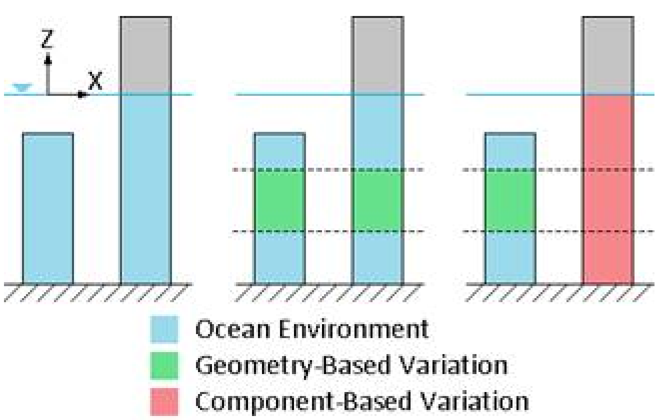
In Static Structural and Transient Structural analyses, you may use the **Load Step Number option** to set the steps at which the Component-Based Variation object is applied. By default the object is applied for All steps; if required, change this to Range and specify the **Starting Load Step** and **Ending Load Step** to define a range of steps.

Precedence of Local Adjustments to Structural Coefficients

Where there are multiple definitions of structural coefficients applied to a range of water depths, the order of precedence is:

- Component-Based Variation (most significant)
- Geometry-Based Variation
- Ocean Environment (least significant)

Figure 8.37: Precedence of Local Adjustments to Structural Coefficients



8.2.9. Pipe-in-Pipe

For the special case of a pipe with an internal pipe running through it, the flooding options defined in the Ocean Environment (p. 321) object may be overridden by a Pipe-in-Pipe object. You may add as many of these to the analysis as required.

Figure 8.38: Pipe-in-Pipe Details

Details of "Pipe-in-Pipe Configuration"	
Internal Member	
Scoping Method	Geometry Selection
Geometry	1 Edge
External Member	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Internal Member Flooding	True
External Member Flooding	Internal Fluid
External Pipe Fluid Density	820 [kg m^-1 m^-1 m^-1]
External Pipe Free Surface Level	-5 [m]
Load Step Number	All

Under the heading of Internal Member, use the **Scoping Method** and **Geometry**/**Named Selection** options to select the line body which represents the internal pipe.

Under the heading of External Member, use the **Scoping Method** and **Geometry**/**Named Selection** options to select the line body which represents the external pipe.

Note:

The line bodies selected to represent the internal and external pipes must have their **Model Type** property set to Pipe.

The **Internal Member Flooding** option sets the flooding state of the internal pipe. The available options are:

- True: the internal pipe is flooded with a fluid, of a density defined by the **External Pipe Fluid Density**
- False: the internal pipe is not flooded
- Internal Fluid: the internal pipe is flooded with a fluid, of a density defined by the **Internal Pipe Fluid Density**, up to the specified **Internal Pipe Free Surface Level**

The **External Member Flooding** option sets the flooding state of the external pipe. Where the **Default Member Flooding** option in the Ocean Environment object Structural Coefficients (p. 321) is set to True, this option must also be True. Otherwise, the available options are:

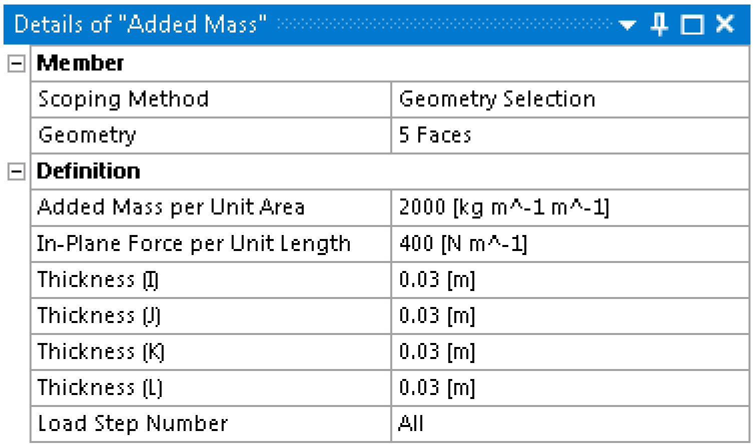
- False: the external pipe is not flooded
- Internal Fluid: the external pipe is flooded with a fluid, of a density defined by the **External Pipe Fluid Density**, up to the specified **External Pipe Free Surface Level**

In Static Structural and Transient Structural analyses, you may use the **Load Step Number** option to set the steps at which the Pipe-in-Pipe object is applied. By default the object is applied for All steps; if required, change this to Range and specify the **Starting Load Step** and **Ending Load Step** to define a range of steps.

8.2.10. Added Mass

Added Mass objects are used to define the added mass of surfaces due to hydrodynamic effects. You may add as many of these to the analysis as required.

Figure 8.39: Added Mass Details



Use the **Scoping Method** and **Geometry**/**Named Selection** options to select the surfaces bodies to which the added mass will be applied.

The **Added Mass per Unit Area** defines the amount of added mass that will be included over the surface, while the **In-Plane Force per Unit Length** accounts for surface tension acting on the surface. The four **Thickness** values define the thicknesses at the I, J, K and L nodes of each surface element, as described in SURF154 in the *Mechanical APDL Element Reference*.

In Static Structural and Transient Structural analyses, you may use the **Load Step Number** option to set the steps at which the Added Mass object is applied. By default the object is applied for All steps; if required, change this to Range and specify the **Starting Load Step** and **Ending Load Step** to define a range of steps.

8.3. The Aqwa Co-simulation Add-on

A co-simulation system with Ansys Rigid Dynamics is available for the Aqwa solver. This combination of the Rigid Dynamics and Aqwa solvers allows you to create a more powerful simulation. For example, this co-simulation system can solve marine structures which have complex contact, shaft, and gear components, such as a point absorber. This co-simulation system can also conduct Hydro-Aero-Multi-body coupling analyses for floating offshore wind turbines by including AeroDyn, a third-party aerodynamic module from OpenFAST v3.0.0, that provides the ability to compute wind turbine (horizontal axis) aerodynamic responses.

Note:

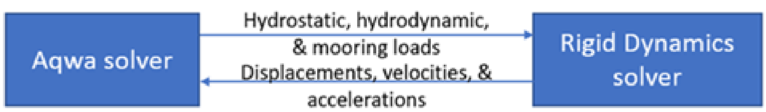
To access the OpenFAST User Documentation for the version of AeroDyn included in the Ansys installation, see Developing Documentation for more information on how to build the documentation. If you encounter any errors, contact OpenFAST GitHub support for assistance.

As the co-simulation system involves several products communicating simultaneously, some of the setup procedures can be complex. Therefore, the Aqwa Co-simulation Add-on has been developed to simplify the setup process for the co-simulation system.

8.3.1. Introduction

In the co-simulation system of Aqwa and Rigid Dynamics, the Rigid Dynamics solver is responsible for solving the equations of motion and passes the motion responses (displacements, velocities, and accelerations) to Aqwa (Aqwa-Naut, Hydrodynamic Response > Time Response Analysis > Regular/Irregular Wave Response) to calculate the hydrostatic, hydrodynamic, and mooring loads based on the updating motion responses. Aqwa will pass the total force and moment of hydrostatic, hydrodynamic and mooring loads to Rigid Dynamics to calculate the motion responses. The data transfer between the Rigid Dynamics and Aqwa solvers happens at each co-simulation time step, as presented in Figure 8.40: Aqwa-Rigid Dynamics Solver Relationship (p. 333). The transferred motion and load data are with respect to each structure's center of gravity (COG) in the Aqwa model.

Figure 8.40: Aqwa-Rigid Dynamics Solver Relationship



In the current version, the data transfer between Aqwa and Rigid Dynamics is conducted via the Functional Mock-up Interface (FMI). Before running a co-simulation between Aqwa and Rigid Dynamics, you must generate two Functional Mock-up Unit (FMU) packages, which contain the Aqwa and Rigid Dynamics project data for the co-simulation analysis, respectively. These FMU packages can be created via the Rigid Dynamics and Aqwa Workbench projects. The generation of these FMU packages is a process of packaging and saving the current model data and setups in the Workbench project into a compressed file.

For the co-simulation analysis, some special setups must be included in the Aqwa and Rigid Dynamics projects. In the Aqwa Workbench project, the Output Aqwa FMU Package (p. 140) option must be switched on while the unit and coordinate system information of the Rigid Dynamics model need to be defined if they are different from the Aqwa model. In the Rigid Dynamics Workbench project, there are extra setups which need to be included based on the Aqwa model data:

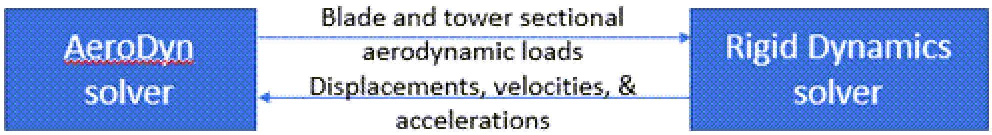
1. The initial positions and rotations of the Rigid Dynamics structures must be set the same as the Aqwa structures.

2. The project should have co-simulation pins that are based on the COG positions of the Aqwa structures.

3. The fluid added mass matrix of the Aqwa structures at the infinite frequency must be added to the Rigid Dynamics model.

If AeroDyn is selected to be included in the co-simulation workflow, the connection with AeroDyn also must be set up and activated in the Rigid Dynamics project. Between the AeroDyn program and Rigid Dynamics, a wrapper program enables them to communicate directly, like wrapping AeroDyn as a sub-module of Rigid Dynamics. Once the AeroDyn inputs are properly set in the Rigid Dynamics model, AeroDyn will provide Rigid Dynamics with the aerodynamic forces and moments on the sectional nodes of the blades and tower during each time integration step of the Rigid Dynamics solver. These forces and moments are calculated based on the motion responses obtained from the Rigid Dynamics solver at each time integration step. The data exchange between the two solvers is presented in Figure 8.41: Aerodyn-Rigid Dynamics Solver Relationship (p. 334).

Figure 8.41: Aerodyn-Rigid Dynamics Solver Relationship



When the Aqwa and Rigid Dynamics FMU packages are ready, Ansys Twin Builder or other platforms which support the FMI (v1.0) system can be employed to load, connect, and run both FMU packages. The Aqwa and Rigid Dynamics models packed in the FMU packages can have different numbers of structures, but the Aqwa and Rigid Dynamics FMU packages should have matching input and output co-simulation pins in order to connect properly. The output pins of the Rigid Dynamics FMU package should transfer the motion responses (position, velocity, and acceleration on six degrees of freedom) at each Aqwa structure's COG while its input pins are for the forces and moments on six degrees of freedom with respect to each Aqwa structure's COG. Equally, the Aqwa FMU package's output pins are for the total forces and moments of hydrostatic, hydrodynamic and mooring loads at each Aqwa structure's COG while its input pins are for the motion responses at the Aqwa structure's COG.

In general, the steps to run the co-simulation analysis are as follows:

1. Set up an Aqwa project, solve the Hydrodynamic Diffraction analysis and generate the Output Aqwa FMU Package (p. 140) by solving the Hydrodynamic Response analysis

2. Set up a Rigid Dynamics project:

   a. Set up structures, connections, and other Rigid Dynamics basic settings

   b. Set up the connection with the AeroDyn inputs (optional)

   c. Map the Aqwa model data into the Rigid Dynamics project

   d. Generate the Rigid Dynamics FMU package

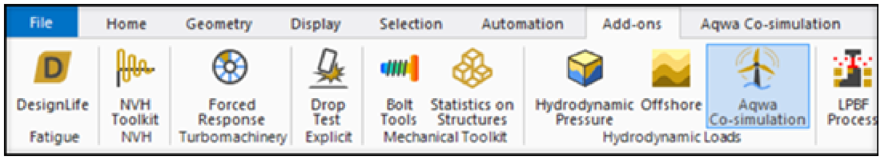
3. Set up and run a Twin Builder project for the co-simulation analysis

Among these steps, the Aqwa Co-simulation Add-on is provided to automatically conduct the exclusive steps to the co-simulation analysis, which are steps 2.b, 2.c and 3.

8.3.2. Loading the Aqwa Co-simulation Add-on

To make the capabilities of this Add-on available, click the **Aqwa Co-simulation** icon in the **Add-ons** Ribbon. The icon will be highlighted in blue, indicating that the Add-on is loaded.

Figure 8.42: Aqwa Co-simulation Add-on Showing Loaded Status



8.3.3. Aqwa Co-simulation Toolbar

The **Aqwa Co-simulation** toolbar includes four options:

1. Update/Import Aqwa File (p. 336)
2. Mapping Aqwa Data (p. 336)
3. Clear (p. 336)
4. Co-simulation Wizard (p. 337)

Figure 8.43: Aqwa Co-simulation Toolbar

Update/Import Aqwa Files and **Mapping Aqwa Data** can be used to write Aqwa model data into the Rigid Dynamics project if the user chooses not to do so via the **Co-simulation Wizard**.

There are two steps to write the Aqwa model data into the Rigid Dynamics project:

1. Load the Aqwa input `*.DAT` file and `*.ADM` file by clicking **Update/Import Aqwa Files**. Once the files have been successfully loaded, there is no need to further import them unless the input files have been changed.

2. Click **Mapping Aqwa Data** for the Add-on to write the Aqwa data into the Rigid Dynamics project.

Clear is used to remove any objects created by the Aqwa Co-simulation Add-on.

Co-simulation Wizard provides a three-step wizard, which guides you through the process of establishing the connection with the AeroDyn inputs and importing the Aqwa data into the Rigid Dynamics project, setting up a co-simulation Twin Builder project and running the co-simulation analysis.

Note:

The **Aqwa Co-simulation** toolbar options are only active in the Rigid Dynamics project when the project tree object (except the Solution branch and its subbranches) is selected.

8.3.3.1. Update/Import Aqwa File

The Aqwa data needed by the Add-on are in the Aqwa *.ADM file and *.DAT file. The Add-on will parse these two imported files. If any problem occurs, warnings or errors will be shown in both the message panel and the Extensions Log File.

8.3.3.2. Mapping Aqwa Data

Mapping Aqwa Data can write Aqwa model data (including the COG positions and initial positions of Aqwa structures, the fluid added mass matrix at the infinite frequency, etc.) into the Rigid Dynamics project and generate the co-simulation pins which match the Aqwa FMU package's co-simulation pins. This requires that the data in the Aqwa input `*.DAT` file and `*.ADM` file have already been imported into the Add-on (see Update/Import Aqwa File (p. 336)). If the Add-on does not find the required Aqwa data files or sufficient Aqwa data, the Add-on will request a re-selection of the Aqwa files. The Aqwa data will be mapped into the Rigid Dynamics project in the forms of:

- An added mass script and an Aqwa Co-simulation option script
- Pin joints for the co-simulation pins
- Co-simulation pins

The added mass and Aqwa Co-simulation option script will be written to the first Rigid Dynamic system in the project tree.

The pin joints for the co-simulation pins are general joints, which have been set to **Free** in all degrees of freedom. They are located at the Aqwa structures' COGs. During the co-simulation analysis, the hydrostatic, hydrodynamic and mooring loads on the COG of each Aqwa structure will be transferred onto the pin joint at the corresponding Aqwa structure's COG in the Rigid Dynamics model. The initial displacements and rotations of each Aqwa structure are also applied through those pin joints, and this Add-on will automatically set the scope of the pin joint on the surface of its corresponding Aqwa structure. However, in some cases when the mesh data in the Aqwa input `*.DAT` file do not match the geometry in the Rigid Dynamics project, you need to manually select the scope for the pin joints. Apart from this case, it is strongly recommended not to change anything the Add-on creates unless you are fully aware of the resulting effects.

Note:

If you plan to use the **Co-simulation Wizard**, the sequence of the co-simulation pins created by this Add-on should not be changed, and any additional co-simulation pins should be added at the end of the existing co-simulation pins created by this Add-on.

8.3.3.3. Clear

The **Clear** button can help you to remove unwanted objects created by this Add-on quickly. There are three options:

1. **Clear Aqwa Data**: Removes the tree objects (added mass script, pin joints, and the co-simulation pins) created by the **Mapping Aqwa Data** option or the **Aqwa Modeling Data** step of the Aqwa Co-simulation Setup Wizard.

2. **Clear AeroDyn Marker**: Removes all the coordinate systems created for the AeroDyn marker in the **Aerodyn Connection** step of the Aqwa Co-simulation Setup Wizard.

3. **Clear AeroDyn Scripts**: Removes all the scripts created in the **AeroDyn Connection** step of the Aqwa Co-simulation Setup Wizard.

> **Note:**
>
> The **Clear** button cannot remove the objects created by the Aqwa Co-simulation Add-on during previous use (that is, before the Add-on has been previously unloaded).

8.3.3.4. Co-simulation Wizard

The Co-simulation Wizard opens the **Aqwa Co-simulation Setup Wizard** interface, when selected. You can also continue with previous wizard progress, if it exists. The Aqwa Co-simulation Setup Wizard is designed to automate some of the co-simulation steps.

A prerequisite for running a co-simulation via the Aqwa Co-simulation Setup Wizard is locally installing the Ansys Twin Builder.

8.3.4. Aqwa Co-simulation Setup Wizard

The Aqwa Co-simulation Setup Wizard comprises three steps:

8.3.4.1. AeroDyn Connection
8.3.4.2. Aqwa Modeling Data
8.3.4.3. Co-simulation Settings
8.3.4.4. Run Co-simulation

The wizard will go through the process of setting up the connection between the AeroDyn inputs and the Rigid Dynamics project, creating Aqwa data objects in the Rigid Dynamics project, defining the remaining information for the co-simulation analysis, establishing the co-simulation Twin Builder project and finally, running the co-simulation.

8.3.4.1. AeroDyn Connection

This step aims to generate scripts in the Rigid Dynamics model to set up and activate the connection between the Rigid Dynamics solver and the AeroDyn program. Aerodyn from OpenFAST version 3.0.0 is used for this feature. The Aerodyn input files must be prepared for this specific version, including the AeroDyn primary input file, InflowWind input files, Blade Data input file, and Airfoil Data input files. If you want to skip this process, select **No** for the **Include AeroDyn Connection** option and proceed to the next step. If **Yes** is selected (see Figure 8.44: AeroDyn Connection Setup (p. 340)), additional AeroDyn and turbine information must be provided. The required information includes the following:

- **AeroDyn Primary Input File**: The path of the AeroDyn primary input file.
- **InflowWind Input File**: The path of the InflowWind input file. For more information, refer to the InflowWind Users Guide.

- **Write AeroDyn Result Files**: Choose whether to generate AeroDyn Result files.
- **Turbine Data**: The following parameters follow the same definition described in the Driver Inputs and Options section of the AeroDyn User Guide:
  - **Number of Blades**: The amount of blades which the turbine model has, with a selection of 1 to 3.
  - **Hub Radius**: The radius of the hub, measured from the center-of-rotation to the blade root along the (possibly preconed) blade-pitch axis.
  - **Blade Length**: The length of the blade, which is measured from the blade root to the blade tip.
  - **Hub Height**: The height of the hub center measured from the origin of the Global coordinate system of the Rigid Dynamics project, which is the global Z-coordinate value of the hub center.
  - **Overhang**: The global X-coordinate value of the hub center. Overhang is positive downwind, so use a negative number for upwind rotors.
  - **Shaft Tilt**: The angle between the rotor shaft and the horizontal plane. Positive shaft tilt means the downwind end of the shaft is the highest.
  - **Precone**: The angle between a flat rotor disk and the cone swept by the blades, positive downwind (upwind turbines have negative Precone for improved tower clearance).

Note:

To ensure the consistency of the AeroDyn connection scripts and the AeroDyn input files, the path of the AeroDyn primary input file must be re-selected if any settings in the AeroDyn input files have been modified.

The data transfer between AeroDyn and Rigid Dynamics is conducted through the nodes, which represent the turbine geometry parts in a condensed form. For each node, we use a marker to indicate it. The marker consists of a position and an orientation. Namely, it is equivalent to the local coordinate system of the geometry part represented. For example, the hub marker is located at the hub center, and the shaft tilt will influence its orientation. The hub marker, blade marker and blade node maker are determined using the work described by Jonkman & Jonkman [1].

When expressed in the coordinate system terms, the **hub marker** orientation is:

- X-axis: Pointing along the hub centerline in the nominally downwind direction
- Y-axis: Orthogonal with the x- and z-axes such that they form a right-handed coordinate system
- Z-axis: Perpendicular to the hub centerline with the same azimuth as Blade 1

Similarly, the **blade marker** is located at the intersection of the pitch axis and the blade root of each blade. Its orientation is as follows:

- X-axis: Orthogonal with y- and z-axes such that they form a right-handed coordinate system

- Y-axis: Pointing towards the trailing edge of the blade and parallel to the chord line at the zero-twist blade station
- Z-axis: Pointing along the pitch axis towards the tip of the blade

The **blade node marker** is located at the aerodynamic analysis node listed in the Blade Data Input file. Its orientation is as follows:

- X-axis: Orthogonal with y- and z-axes such that they form a right-handed coordinate system
- Y-axis: Aligned with the local chord line pointing towards the trailing edge
- Z-axis: Directed along the blade towards the tip

Note:

The turbine, which can be simulated by this Add-on, is limited to the single basic horizontal axis wind turbine. The initial orientation of Blade 1 should have the same azimuth of the global z-axis and should not have any rotation about the hub rotation axis. The following blades (Blade 2 and Blade 3) spread equally in the rotation plane in an anti-clockwise order, viewing towards the downwind direction. The nacelle and hub should not have any yaw angle.

Three buttons are available to help the inspection of the correctness of the AeroDyn input being set up. They are **Show Hub Marker**, **Show Blade Markers**, and **Show Blade Node Markers**. By clicking on them, the coordinate systems of the corresponding markers will be generated in the outline tree of the Rigid Dynamics project.

The setup scripts for AeroDyn will be written into the Rigid Dynamics project when the **Next** button is clicked. The scripts will activate the connection between the AeroDyn program and the Rigid Dynamics solver while the Rigid Dynamics project is being solved.

Note:

The AeroDyn primary input file specifies the path of the Blade Data Input file and the Airfoil Data Input file while the InflowWind Input file specifies the path of the wind file if necessary. All the AeroDyn and InflowWind files will not be embedded into the Rigid Dynamics FMU package. However, they should be available and accessible in their claimed directories while the Rigid Dynamics project or the co-simulation project is being solved.

Note:

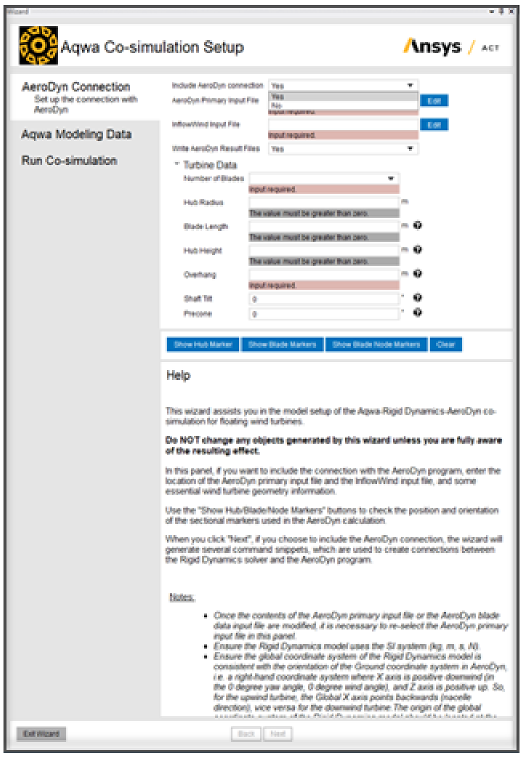
The wizard assumes the blade and tower sectional nodes will follow the requirements in the AeroDyn User Guide. The local elevation of the tower nodes must increase from the lowest (tower-base) to the highest (tower-top) elevation. The first and last blade nodes must be located at the most inboard (blade root) and most outboard (blade tip) points, respectively. The tower nodes should not be located at the multiple structures that have relative motions, and neither should the blade nodes of each blade. Otherwise, the AeroDyn connection scripts could be inaccurate.

Note:

As AeroDyn uses the SI system (kg, m, s, N), the Rigid Dynamics project must also use the same unit system if the AeroDyn connection is included.

More notes about this step can be found in the Help section of the wizard.

Figure 8.44: AeroDyn Connection Setup

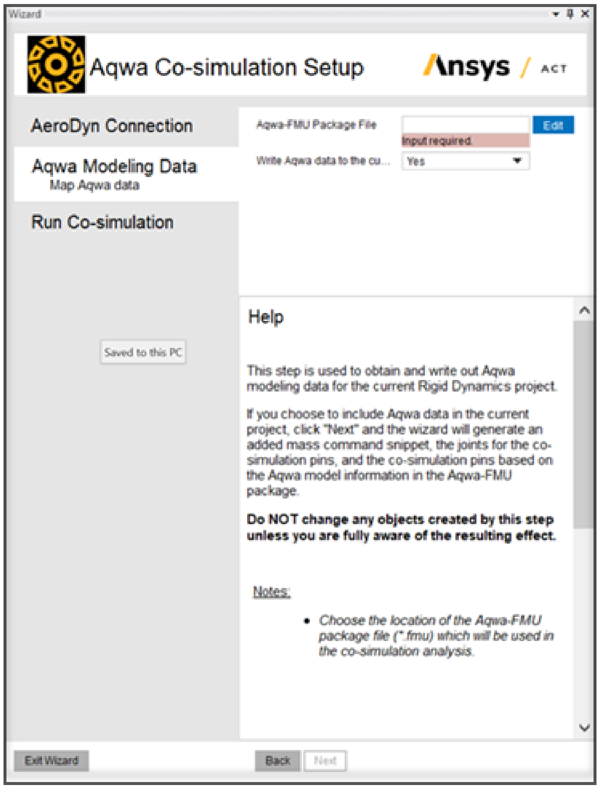


8.3.4.2. Aqwa Modeling Data

The Aqwa Modeling Data step is used to record the file location of the Aqwa FMU package file, which will be used in the co-simulation analysis.

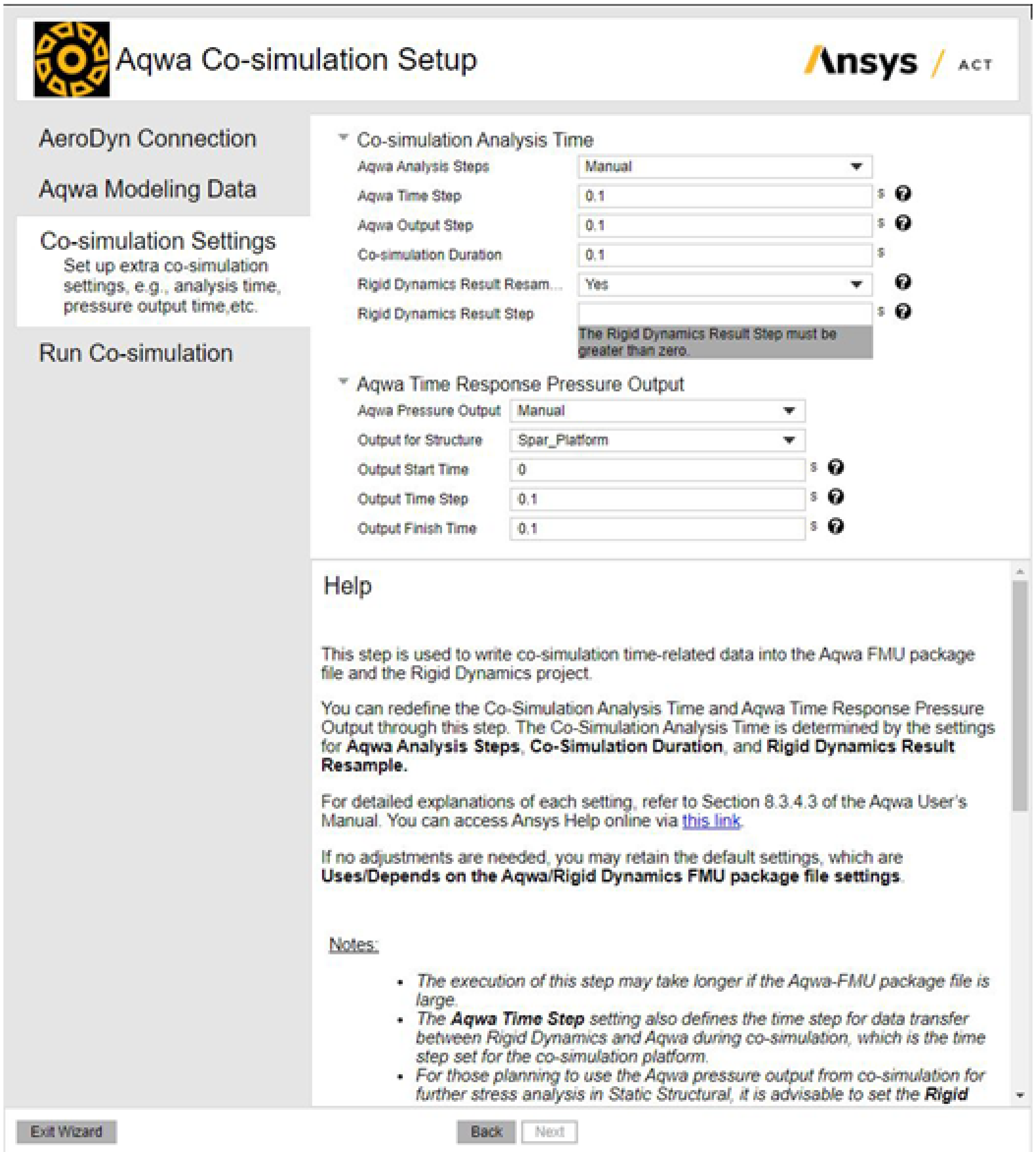
Based on the Aqwa input files in the Aqwa FMU package, if **Yes** is selected for **Write Aqwa data to the current project?**, this step can map Aqwa data into the current Rigid Dynamics project when **Next** is clicked, similar to the Mapping Aqwa Data (p. 336) option in the **Aqwa Co-simulation** toolbar. If **No** is selected, there will be no changes to the current Rigid Dynamics project when proceeding to the next step. **No** can be selected when a Rigid Dynamics FMU package file that matches the selected Aqwa FMU Package File in this wizard step is already available or can be obtained without writing the Aqwa data into the current project.

Figure 8.45: Aqwa Modeling Data Setup



8.3.4.3. Co-simulation Settings

Use the Co-simulation Settings step to configure the Co-simulation Analysis Time (p. 342) and the Aqwa Time Response Pressure Output (p. 343) for the co-simulation analysis. The settings defined in this step will be written into the Aqwa FMU package file selected during the Aqwa Modelling Data (p. 340) step or applied to the **Rigid Dynamics** project when you select the **Next** button of the wizard.



Important:

If the **Aqwa Analysis Steps** or the **Aqwa Time Response Pressure Output** options are manually redefined during this step, the original values in the selected Aqwa FMU package file will be overwritten with the new inputs when this step is executed. Ansys recommends that you back up the original Aqwa-FMU package file if necessary.

8.3.4.3.1. Co-simulation Analysis Time

The **Aqwa Analysis Steps** option refers to both the Aqwa analysis time step and the Aqwa output time step. The Aqwa analysis time step determines the analysis time interval for Aqwa as well as the data transfer interval between the Rigid Dynamics analysis and Aqwa during the co-simulation process. This time step sets the analysis time step for the co-simulation platform. Options include:

- **Manual**: When selected, you can redefine the **Aqwa Time Step** and **Aqwa Output Step** for the co-simulation properties. When this step is executed, these new settings will replace the original values in the selected Aqwa-FMU package file.
- **Use Aqwa-FMU package file settings**: Use this option to retain the settings defined originally in the Aqwa-FMU package file.

The **Co-simulation Duration** specified in the wizard determines the overall duration of the co-simulation analysis. This setting automatically overrides any co-simulation duration previously defined in the Aqwa FMU and Rigid Dynamics FMU package files. During co-simulation, the Rigid Dynamics analysis will adhere to the analysis settings defined in the Rigid Dynamics project, which generates the Rigid Dynamics FMU package file.

However, using the **Rigid Dynamics Result Resample** option, you can resample Rigid Dynamics result points through a resample command. Settings for the **Rigid Dynamics Result Resample** option include:

- **Yes**: This setting automatically inserts a Commands Snippet with a result resample command into the Rigid Dynamics project, that enables the application to select the closest suitable result points among all of the initially computed result points based on the time interval set by **Rigid Dynamics Result Step** option of the wizard. This occurs during post-processing and does not involve recomputing or interpolating the results.
- **No**: This setting omits the result resample command written by the wizard from the Rigid Dynamics project.

8.3.4.3.2. Aqwa Time Response Pressure Output

Specifying the **Aqwa Time Response Pressure Output** is equivalent to the Time Response Pressure Output (p. 136) settings in Aqwa Workbench. The Aqwa-FMU package file initially contains the pressure output settings from the Aqwa Workbench project that generated it. If you set the **Aqwa Pressure Output** option to **Depends on Aqwa-FMU package file settings**, the original settings will be retained, except the Output Finish Time (p. 136) that will be adjusted to match the **Co-simulation Duration**, provided a structure was originally selected for pressure output.

If you use the **Manual** setting, you can redefine the **Output for Structure**, **Output Start Time**, **Output Time Step**, and **Output Finish Time** options for the co-simulation analysis. When this step is executed, these new inputs will replace the original values in the selected Aqwa-FMU package file.

8.3.4.4. Run Co-simulation

The **Run Co-simulation** step sets up the remaining information required to run the co-simulation and the result saving directories for co-simulation results. Once all the requested information is provided, selecting the **Run Co-simulation** button will trigger the establishment of the co-simulation project and the running of the co-simulation analysis. Upon completion, click **Finish** to close the wizard, which will clear the values recorded or provided in the wizard.

8.3.4.4.1. Information Setup

The remaining information required to run the co-simulation includes:

- **Rigid Dynamics FMU Package File Location**: For the co-simulation with Aqwa and AeroDyn, the Rigid Dynamics project should have the Aqwa data mapped and the connection with the AeroDyn inputs established before generating the FMU package. The Rigid Dynamics FMU package can then be generated using the Write Input File option.

- **Co-simulation Platform**: Select between Twin Builder and System Coupling as the co-simulation running platform. If Twin Builder is selected, further Twin Builder project-related options will appear.

- **Twin Builder Project**: The version of Twin Builder used to run a co-simulation analysis is the same as Ansys Workbench. Selecting either **Save the Twin Builder project file** or **Keep the Twin Builder project open after the co-simulation** can decide whether the Twin Builder co-simulation project will be saved or kept open after the co-simulation ends, respectively. If **Yes** is selected for the **Keep the Twin Builder project open after the co-simulation** option, the Twin Builder project must be closed manually before continuing to work in the wizard. You should keep the Twin Builder project open the first time you run the co-simulation to ensure that no error messages show in the Twin Builder message panel.

> **Note:**
>
> System Coupling requires FMU version 2.0 for both Aqwa and Rigid Dynamics FMU package files. Twin Builder supports both FMU version 1.0 and 2.0.

8.3.4.4.2. Result Saving Directory Setup

The co-simulation results files generated by the Aqwa and Rigid Dynamics solvers will be saved respectively in the selected **Aqwa Co-simulation Result Directory** and **Rigid Dynamics Co-simulation Result Directory**. These are the available directory options:

- **Project Solving File Directory**: Specifically for the Aqwa results, it refers to the solver files directory of the Aqwa project where the Aqwa FMU package was created. For the Rigid Dynamics results, it refers to the solver files directory of the first Rigid Dynamics system in the project where the wizard is open. The full path of the directory will be shown underneath the option once it is selected.

- **FMU Package File Directory**: It refers to the directory that contains the corresponding (Aqwa/Rigid Dynamics) FMU package file. The full path of the directory will be shown underneath the option once it is selected.

- **Manual**: It enables the manual selection for the result file saving directory.

> **Note:**
>
> Irrespective of the selected result saving option, the result files are saved in a folder named **ARACosimulation.Aqwa** for Aqwa or **ARACosimulation.RBD** for Rigid Dynamics inside the selected result directory.

8.3.4.4.3. Running the Co-simulation Analysis

The **Run Co-simulation** button can work on the condition that the first two wizard steps passed without any error and all the requested information is provided in the current wizard step. In the

Twin Builder or System Coupling project created by the wizard, the Aqwa FMU package and the Rigid Dynamics FMU package selected in the wizard will be linked and run as a co-simulation system. The System Coupling project will run via Command Prompt without a User Interface. However, the **Open Log File** button can work after the co-simulation finishes. It shows the running log file of the selected Co-simulation Platform (p. 344).

After the co-simulation analysis finishes successfully, the wizard will try to copy back the co-simulation results created by the Aqwa and Rigid Dynamics solvers from the Twin Builder or System Coupling project temporary directory to the Aqwa and Rigid Dynamics project directories. When Twin Builder is used, the detailed copying paths for the different solver result files will be shown in the Twin Builder message panel and the Twin Builder running log file if the file copying process succeeds.

The Rigid Dynamics results can be manually imported back into the Rigid Dynamics project via Read Result Files while the Aqwa results can be accessed via **AqwaGS**, **AqwaReader** and **Aqwa Workbench** where the Import Co-simulation Results (p. 191) option can be selected to import the co-simulation results to the Hydrodynamic Response Solution. Once loaded, the Twin Builder or System Coupling project can also show the co-simulation pin values along with the co-simulation duration.

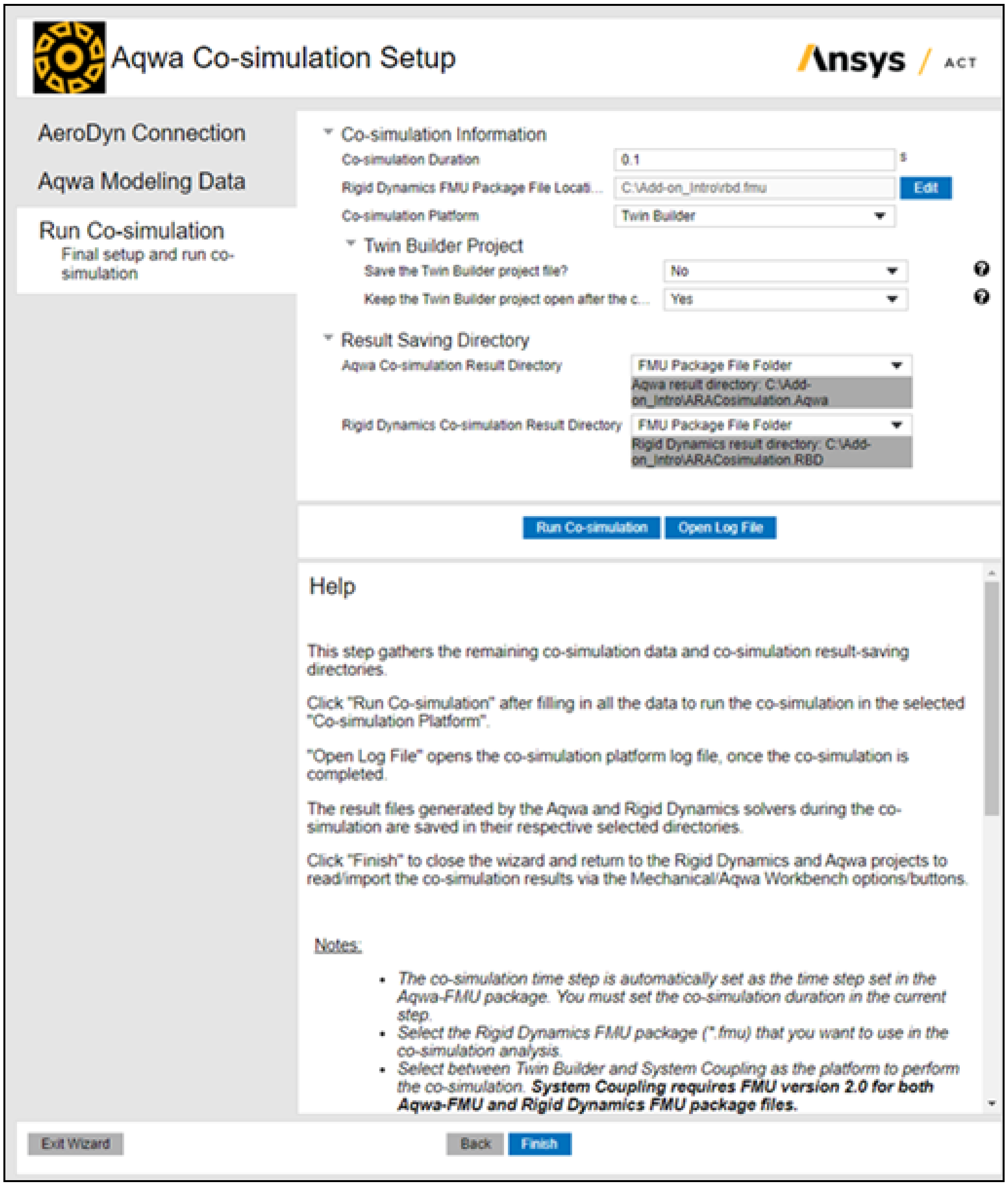
Note:

If you encounter any error messages or a stop in the progress bar while working in the Twin Builder project, you should navigate to the Aqwa `INPUTFILE.MES` file and Rigid Dynamics `solve.out` file under the subfolders of the **temp** folder in the Twin Builder project directory and check the messages in them.

Note:

On the Windows system, the 'Opens with' application must be set with a text editor for the `.out` and `.log` files first to enable the Open Log File button to function properly. To set the 'Opens with' application on a Windows system, right-click an `*.out` file, select Properties in the list, and in the General tab of the Properties window, click the 'Change...' button in the 'Opens with' property, then select Notepad or any other text editor application.

Figure 8.46: Run Co-simulation Setup



8.3.5. Additional Notes

Messages

The message from the Add-on is issued in both the message window of the Rigid Dynamics project and the Extensions Log File. If errors were seen, the Extensions Log File could include detailed information about the problem. To open the Extensions Log File, go to the Workbench toolbar and select **Extensions > View Log File**.

Co-simulation Time Integration Method

The time integration method for the co-simulation analysis can be chosen in the Rigid Dynamics project before generating the Rigid Dynamics FMU package.

Model Differences/Requirements

The Add-on and co-simulation program do not request the full consistency between the Aqwa and Rigid Dynamics models except for the model-defined position and orientation. The Add-on can only define the scope which represents the Aqwa structure correctly in the Rigid Dynamics project when the Rigid Dynamics model contains the geometry surface of the Aqwa model. Otherwise, you must manually define the scope of the pin joints, which represents the Aqwa structure.

Co-simulation Time

The time interval of the co-simulation analysis set in the Twin Builder project should be equivalent to the time interval set for the Aqwa FMU package. The start time for the co-simulation analysis will always be zero. The co-simulation time duration will be decided by the Twin Builder project. For more information, see **Co-simulation Duration** in the Information Setup (p. 343) section.

Co-simulation Platform

Instead of Ansys Twin Builder, other platforms that support FMI 1.0 standards could be attempted to run this co-simulation system while Aqwa and Rigid Dynamics FMU packages are already available.

Manual Definition of the Scope of Pin Joints

When the geometry shape of the model is irregular or you open a project of an older version in the recent version of Workbench, the Aqwa Co-simulation Add-on may be unable to find the joint scope for the pin joints created for the co-simulation. A warning message will pop up instructing you to manually define the corresponding scope for the joint. For the pin joints that have automatically defined joint scopes, it is also advisable to double-check if the scopes of those pin joints have been defined correctly.

8.3.6. Bibliography

1. Jonkman, B.J. & Jonkman, J.M. *Documentation of Updates to FAST, A2AD, and AeroDyn*. (Released March 31, 2010, Including the Revised AeroDyn Interface). Unpublished NREL Technical Report: 2010.

Chapter 9: Aqwa Appendix

9.1. Information for Existing Aqwa Users

If you are familiar with the Aqwa data files and help system, then you can use this section to find where an existing command is located in the Aqwa Editor tree objects.

Table 9.1: Cross reference of tree objects with Aqwa commands

Aqwa Manual Reference	*Aqwa Editor Tree Object*
4.0 - Deck 0 – Preliminary Deck	
4.0.0 General Description	Project (p. 52)
4.0.1 JOB Card	Analysis (p. 143)
4.0.2 TITLE Card	Project (p. 52)
4.0.3 OPTIONS Card	Analysis Settings (p. 129)
4.0.4 RESTART Card	Analysis (p. 143)
4.1 - Deck 1 – (COOR) – Coordinate Positions	
4.1.0 General Description	
4.1.1 Deck Header	
4.1.2 COORDINATE Card	Mesh (p. 46), Mesh Sizing (p. 52)
4.1.3 COORDINATE Card with Rotation Node Generation	Not Supported
4.1.4 COORDINATE STRUCTURE Card	Parts (Structures) (p. 54)
4.1.5 COORDINATE OFFSET card	Not Supported
4.1.6 COORDINATE card with TRANSLATION	Not Supported
4.1.7 COORDINATE Card with Mirror Node Gen	Not Supported
4.1.8 NOD5 card - 5-digit node numbers	Always enabled
4.2 Deck 2 (ELM*) – Element Topology	
4.2.0 General Description	
4.2.1 Deck Header	
4.2.2 ELEMENT TOPOLOGY Card	Mesh (p. 46), Mesh Sizing (p. 52), Line Bodies (p. 58), Point Mass (p. 60), Point Buoyancy (p. 63), Disc (p. 63)
4.2.3 SYMX and SYMY - X and Y Symmetry Cards	Not Supported
4.2.4 HYDI card - Hydrodynamic Interaction	Structure Selection (p. 143)
4.2.5 RMXS/RMYS cards - Remove Symmetry	Not Supported
4.2.6 MSTR Card - Move Structure	Not Supported
4.2.7 FIXD Card - Fix Structure	Parts (Structures) (p. 54)

Aqwa Manual Reference	*Aqwa Editor Tree Object*
4.2.9 VLID Card - Suppression of Standing Waves	Parts (Structures) (p. 54), Surface Bodies (p. 58)
4.2.10 ASYM Axi-Symmetric Structure Generation	Not Supported
4.2.11 ILID Card - Suppression of Irregular Frequencies	Surface Bodies (p. 58)
4.2.12 ZLWL Card - Waterline Height	Geometry (p. 42)
4.3 Deck 3 (MATE) – Material Properties	
4.3.0 General Description	
4.3.1 Deck Header	
4.3.2 MATERIAL PROPERTY Card	Line Bodies (p. 58), Point Mass (p. 60), Point Buoyancy (p. 63), Disc (p. 63)
4.4 Deck 4 (GEOM) – Geometric Properties	
4.4.0 General Description	
4.4.1 Deck Header	
4.4.2 GEOMETRIC Property and CONTINUATION Card	Line Bodies (p. 58), Point Masses (p. 60), Point Buoyancy (p. 63), Disc (p. 63)
4.5 Deck 5 (GLOB) – Global Parameters	
4.5.0 General Description	
4.5.1 Deck Header	
4.5.2 DPTH Card (Optional) - Water Depth	Geometry (p. 42)
4.5.3 DENS Card (Optional) - Water Density	Geometry (p. 42)
4.5.4 ACCG Card (Optional) – Gravitational Acceleration	Geometry (p. 42)
4.6 Deck 6 (FDR*) – Frequencies and Directions Table	
4.6.0 General Description	
4.6.1 Deck Header	
4.6.2 FREQ/PERD/HRTZ Card - Frequencies/Periods at which the Parameters are Defined	Wave Frequencies (p. 145)
4.6.3 DIRN Card - Directions at which the Parameters are Defined	Wave Directions (p. 144)
4.6.4 MVEF Card - Move Existing Freq' Parameters	Not Supported
4.6.5 DELF Card - Delete Frequency Parameters	Not Supported
4.6.6 CSTR Card - Copy from Structure Number	Not Supported
4.6.7 FILE Card - Copy from File Unit	Not Supported
4.6.8 CPYF/CPYP/CPYH Cards - Copy Freq' Param's	Not Supported
4.6.9 CPYS Card - Copy Stiffness Matrix	Not Supported
4.6.10 CPDB Card - Copy Data Base	Not Supported
4.6.11 FWDS Card - Define Forward Speed	Wave Directions (p. 144)
4.7 DECK 7 (WFS*) – Wave Frequency Dependent Parameters and Stiffness Matrix	Parts (Structures) (p. 54), Additional Stiffness (p. 72), Additional Damping (p. 73), Additional Added Mass (p. 73), Connection Stiffness (p. 112)

Aqwa Manual Reference	***Aqwa Editor Tree Object***
4.8 DECK 8 (DRC*) – Drift Force Coefficients	Not Supported
4.9 DECK 9 (DRM*) – Drift Motion Parameters	Not Supported
4.10 DECK 10 (HLD*) – Hull Drag Coefficients and Thruster Forces	Current Force Coefficients (p. 77), Wind Force Coefficients (p. 80)
4.11 DECK 11 (ENVR) – Environmental Parameters	Wind (non spectral definition) (p. 178), Current (p. 172)
4.12 DECK 12 (CONS) – Constraints	Not Supported
4.13 Deck 13 (SPEC) – Spectral Parameters	Irregular Wave (p. 156), Wind (spectral definition) (p. 178)
4.13N Deck 13 (WAVE) – Regular Wave Parameters	Regular Wave (p. 155)
4.14 Deck 14 (MOOR) – Mooring Lines Description	Cables (p. 101), Cable Winch (p. 183), Cable Failure (p. 187), Analysis Settings (p. 129)
4.15 Deck 15 (STRT) – Starting Conditions	Not Supported
4.15N Deck 15 (STRT) – Starting Conditions (NAUT)	Not Supported
4.15D Deck 15 (STRT) – Starting Conditions (DRIFT)	Not Supported
4.16 Deck 16 (STRT) – Starting Conditions	Not Supported
4.16L Deck 16 (STRT) – Starting Conditions (LINE)	Not Supported
4.16B Deck 16 (STRT) – Starting Conditions (LIBRIUM)	Analysis Settings (p. 129)
4.17 Deck 17 (HYDC) – Hydrodynamic parameters for non diffracting elements	Analysis Settings (p. 129), Parts (Structures) (p. 54)
4.18 Deck 18 (PROP) – Printing Options	Not Supported
4.21 Deck 21 (ENLD) – Element and Nodal Loads	Not Supported
All AqwaWave commands	Not Supported

Index

A

Ansys Product Improvement Program, 17
APIP (Ansys Product Improvement Program see)
aqwa
    attach geometry, 40
    create analysis system, 39
attaching geometry, 40

G

geometry
    attach - aqwa, 40