

## **Sample of a user's guide for software**

The RAPPID™ User's Guide was written to accompany the RAPPID™ software when distributed to customers. It was a dynamic help system, constantly being updated as the software changed. This document was created to reorganize the content into a more task-oriented format from an AuthorIT project. This user's guide was to be imported into HelpServer, a new tool to which we were switching for our help development and intranet site maintenance; however, the documentation was put on hold and then the project was completely halted before I could finish the document.

User's Guide

# RAPPID

Multi-disciplinary, modeling and simulation tool

Version 3

**Rotordynamics-Seal Research**

Copyright © 1988-2009 Rotordynamics-Seal Research.

All rights reserved

Rotordynamics-Seal Research makes no warranty or representation, either expressed or implied, with respect to this program or documentation, their quality, performance, merchantability, or fitness for a particular purpose. As a result, this program and documentation are licensed "as is" and you, the licensee, are assuming the entire risk as to their quality and performance. In no event shall Rotordynamics-Seal Research be liable for incidental or consequential damages, even if Rotordynamics-Seal Research has been advised of the possibility of such damages.

**Rotordynamics-Seal Research**

3302 Swetzer Rd.

Loomis, CA 95650

Phone: (916) 660-0444

Fax: (916) 660-0222

E-Mail: [rsr@rsr.com](mailto:rsr@rsr.com)

Website: <http://www.rsr.com>

## Table of Contents

<b>INTRODUCTION.....</b>	<b>1</b>
ABOUT RAPPID™.....	1
ABOUT THIS MANUAL .....	3
<i>Typographical conventions.....</i>	3
<i>Computer terminology.....</i>	3
User interface conventions.....	4
Keyboard terms.....	4
General terms.....	5
<b>PART I: THE RAPPID™ SYSTEM.....</b>	<b>6</b>
CHAPTER 1: GETTING STARTED .....	6
<i>Starting RAPPID™.....</i>	6
<i>Introducing the main RAPPID™ screen .....</i>	7
<i>Specifying the location of projects and archives.....</i>	8
<i>Working with unit systems in RAPPID™.....</i>	9
Changing the default system of units.....	10
<i>Getting help in RAPPID™.....</i>	12
Specifying the location of the web browser used for displaying online help .....	12
CHAPTER 2: WORKING WITH PROJECTS.....	14
<i>Introducing the Project Manager.....</i>	14
Accessing active and archived projects.....	14
Interpreting the dates on projects.....	14
<i>Creating a new project.....</i>	15
<i>Copying a project.....</i>	15
<i>Renaming a project.....</i>	16
<i>Moving an active project.....</i>	16
<i>Deleting a project .....</i>	17
<i>Locking and unlocking projects .....</i>	17
Locking an entire project .....	18
Locking existing analysis cases in a project .....	18
Using passwords.....	18
Unlocking a project .....	19
CHAPTER 3: WORKING WITH ANALYSIS CASES .....	20
<i>Introducing the Analysis Manager.....</i>	20
<i>Creating a new analysis case.....</i>	22
<i>Renaming an analysis case.....</i>	22
<i>Copying an analysis case .....</i>	23
<i>Moving an analysis case.....</i>	23
<i>Transferring case inputs.....</i>	24
<i>Exporting analysis cases .....</i>	26
<i>Importing an analysis case .....</i>	28
<i>Deleting analysis cases .....</i>	30
<i>Locking and unlocking analysis cases.....</i>	33
Locking the selected analysis case.....	34
Locking all existing analysis cases in a project .....	34
Using a password .....	35
Unlocking an analysis case .....	36
<i>Introducing a sample case manager.....</i>	36
Specifying common options.....	37
Adding comments to analysis cases.....	37
Specifying the system of units for an analysis case .....	38
Defining the Dynamic Solution .....	39
Adding materials and fluids .....	39
Entering tabular input .....	40
<i>Accessing pages in an analysis case .....</i>	40
<i>Using a grid to enter data.....</i>	40

# RAPPID User's Guide

---

Inserting rows into a grid .....	41
Copying a row in a grid .....	42
Moving a row in a grid .....	43
Deleting a row in a grid .....	43
Importing data into an analysis case.....	44
Entering variable data .....	47
Entering a constant value .....	48
Entering variable values.....	48
Running analysis cases.....	49
Validating analysis case input .....	49
Monitoring analysis case runs .....	49
Starting runs for more than one analysis case at a time.....	51
Creating a batch run.....	53
Renaming a batch run .....	54
Copying a batch run.....	54
Moving a batch run.....	54
Deleting a batch run.....	55
Defining and launching a batch run .....	55
Specifying the order of analysis cases in a batch run .....	57
Adding multiple analysis cases to a batch run .....	57
Locking and unlocking batch runs.....	58
Viewing and printing results.....	61
Comparing analysis case results.....	61
Changing the default comparison tool .....	61
Opening results from another analysis case.....	62
Comparing text from two results files.....	63
Deleting analysis case results.....	63
CHAPTER 4: ADDING NOTES AND ATTACHMENTS .....	65
Working with notes.....	65
Adding or editing notes associated with a project.....	65
Adding or editing notes associated with an analysis case.....	66
Setting the appearance of the text in notes .....	67
Finding text and phrases in notes.....	68
Printing notes .....	68
Opening an output file in the Notes dialog box .....	69
Copying text from notes .....	70
Comparing text from a note with text from an output file .....	73
Working with attachments .....	74
Adding attachments to a project or an analysis case .....	74
Viewing the contents of an attached file .....	75
Renaming an attached file.....	76
Copying an attached file .....	76
Exporting an attached file .....	76
Deleting an attached file .....	78
CHAPTER 5: WORKING WITH MATERIALS.....	79
Using the Materials Library .....	80
Accessing the Materials Library.....	80
Choosing the unit system to apply to all material properties .....	81
Adding a new material to the Materials Library .....	81
Moving a material in the Materials Library.....	82
Deleting a material from the Materials Library .....	82
Modifying a material in the Materials Library.....	84
Using the Materials page .....	85
Adding a material directly into an analysis case.....	85
Linking to a material from the Materials Library .....	86
Linking to a different material in the Materials Library .....	87
Breaking a link to a material in the Materials Library.....	88
Modifying a material on the Materials page .....	88
CHAPTER 6: WORKING WITH FLUIDS .....	89
Using the Fluids Library .....	90
Accessing the Fluids Library.....	90
Choosing the unit system to apply to all fluid properties .....	91

# RAPPID User's Guide

---

Adding a new fluid to the Fluids Library .....	91
Defining fluids in the Fluids Library .....	92
Viewing the basis of a generated Real Properties fluid .....	93
Reassigning a Real Properties fluid .....	93
Moving a fluid in the Fluids Library .....	94
Deleting a fluid in the Fluids Library .....	94
Archiving a fluid in the Fluids Library .....	95
Unarchiving a fluid in the Fluids Library .....	96
<i>Using the Fluids page for adding a single fluid</i> .....	97
Specifying the type of fluid and the fluid name .....	97
Defining a fluid on the Fluids page for a single fluid .....	98
Linking a fluid on the Fluids page for single fluids .....	99
<i>Using the Fluids page for adding multiple fluids</i> .....	99
Specifying the type of fluid and the fluid name .....	100
Defining a fluid on the Fluids page for multiple fluids .....	100
Linking a fluid on the Fluids page for multiple fluids .....	101
<i>Defining fluid properties</i> .....	102
Defining properties for an Incompressible fluid .....	102
Defining properties for a Compressible fluid .....	104
Defining a Real Properties fluid .....	107
Pre-defined fluid table files .....	107
User-Defined fluid table files .....	110
<i>Linking to fluids in the Fluids Library</i> .....	116
CHAPTER 7: PLOTTING RESULTS .....	119
<i>Accessing plots</i> .....	119
<i>Displaying plots</i> .....	120
<i>Customizing plots</i> .....	121
<i>Printing and saving plots</i> .....	123
CHAPTER 8: BACKING UP .....	124
<i>Archiving a project</i> .....	124
<i>Unarchiving a project</i> .....	125
<i>Refreshing the list of archived projects</i> .....	126
<b>PART II – ANALYSIS CASE TYPES</b> .....	<b>127</b>
CHAPTER 9: MODEL/EIGENVALUE COMPONENTS .....	127
<i>Overview</i> .....	127
<i>Options page</i> .....	130
Active Degrees of Freedom box .....	130
Load Path box .....	131
Other box .....	131
<i>Materials page</i> .....	133
<i>Nodes page</i> .....	133
Initialize Using Shaft Sections .....	134
Node # .....	134
Interconnection .....	134
Stiffness to Ground .....	134
Connectivity .....	135
XYZ Coordinates .....	135
Added Mass .....	135
Added Ip .....	136
Added It .....	136
Active Load Paths .....	136
<i>Beams page</i> .....	136
Beam # .....	136
Left Node .....	136
Right Node .....	137
Material .....	137
Beam Type .....	137
Left O.D. .....	137
Left I.D. .....	137
Right O.D. .....	137

# RAPPID User's Guide

---

Right ID.....	137
ID Text .....	137
CHAPTER 10: 0-D FLOW ELEMENTS – FITTING.....	138
<i>Overview</i> .....	138
<i>Options page</i> .....	138
Analysis Parameter box .....	138
Other box .....	138
CHAPTER 11: 0-D FLOW ELEMENTS – MISC.....	140
<i>Overview</i> .....	140
<i>Options page</i> .....	140
Analysis Parameter box .....	140
Other box .....	140
CHAPTER 12: 0-D FLOW ELEMENTS – PIPE .....	142
<i>Overview</i> .....	142
<i>Options page</i> .....	142
Analysis Parameter box .....	142
Type .....	142
Geometry box .....	142
Other box .....	143
CHAPTER 13: 0-D FLOW ELEMENTS – CENTRIFUGAL PUMP AND TURBINE .....	144
<i>Overview</i> .....	144
<i>Options page</i> .....	144
Analysis Parameter box .....	144
Other box .....	144
CHAPTER 14: JOURNAL BEARING – HYDROSTATIC.....	146
<i>Overview</i> .....	146
<i>Options page</i> .....	147
Analysis Parameter box .....	147
Output Format .....	147
Grid .....	147
Dynamic Solution box .....	147
<i>Materials page</i> .....	148
<i>Geometry page</i> .....	148
Bearing Geometry box.....	148
Supply Ports Definition dialog box.....	151
<i>Fluids page</i> .....	154
<i>Operating Conditions page</i> .....	154
Rotor Speed .....	155
Pressure/Temperature box.....	155
Exit Inflow Conditions box.....	156
Loading box .....	157
Misalignment box.....	159
CHAPTER 15: JOURNAL BEARING – 2-D FIXED GEOMETRY.....	161
<i>Overview</i> .....	161
Bearing Geometry .....	161
Thermal model .....	162
Thermal and centrifugal growth .....	162
Groove model .....	162
Applied Load/Prescribed Eccentricity .....	163
Fluid Film gridding .....	163
Coordinate system.....	164
Conventions for defining sectors.....	164
Sector Definition - Single Arc Sectors.....	165
Sector Definition - Recessed Sectors .....	167
Conventional Bearing Configurations (Axial Grooves only, Single Arc Bearing Surface) .....	168
Bearing Configurations with Axial and Circumferential Grooves .....	173
Bearing Configurations with Recessed Sectors (Pressure Dam and Tapered Land).....	176
<i>Options page</i> .....	178
Analysis Parameter box .....	178
Output Format .....	179
Grid .....	179

# RAPPID User's Guide

---

Groove Temp .....	179
Clearance Input .....	179
<i>Materials page</i> .....	180
<i>Geometry page</i> .....	180
Bearing Geometry box .....	180
Sector Definition dialog box .....	181
<i>Fluids page</i> .....	187
<i>Operating Conditions page</i> .....	187
Rotor Speed .....	187
Pressure/Temperature box .....	187
Loading box .....	189
<i>Sample analysis cases</i> .....	192
Manual Example 1: 360° bearing at three speed points .....	192
Manual Example 2a/2b: 2- lobe bearing with circumferential groove in upper half .....	193
Manual Example 3: 3-lobe bearing with gear loading .....	194
Manual Example 4a/4b: 4 lobe bearing with circumferential groove .....	194
Manual Example 5: Pressure Dam Bearing (recessed sector) .....	195
Manual Example 6: Tapered Land Bearing (recessed sectors) .....	195
CHAPTER 16: JOURNAL BEARING – 2-D TILTING PAD .....	197
<i>Overview</i> .....	197
Bearing Geometry .....	197
Thermal Model .....	197
Thermal and centrifugal growth .....	198
Groove Model .....	198
Pivot Model .....	199
Applied Load/Prescribed Eccentricity .....	202
Fluid Film gridding .....	202
Coordinate system .....	203
Rotordynamic coefficients .....	203
Implications of Non-Linear Transfer Function on Rotordynamic Analysis .....	206
Conventions for defining pads .....	207
<i>Options page</i> .....	208
Analysis Parameter box .....	208
Output Format .....	208
Grid .....	208
Groove Temp .....	209
Clearance Input .....	209
<i>Materials page</i> .....	209
<i>Geometry page</i> .....	210
Bearing Geometry box .....	210
Pad Definition dialog box .....	211
<i>Fluids page</i> .....	217
<i>Operating Conditions page</i> .....	217
Rotor Speed .....	217
Pressure/Temperature box .....	217
Loading box .....	219
<i>Sample analysis cases</i> .....	222
Manual Example #1: 5 pad bearing linearized rotordynamic coefficients .....	222
Manual Example #2: 4 pad bearing with non-linear transfer function .....	223
CHAPTER 17: THRUST BEARING – HYDROSTATIC .....	225
<i>Overview</i> .....	225
<i>Options page</i> .....	225
Analysis Parameter box .....	225
Output Format .....	226
Grid .....	226
<i>Materials page</i> .....	226
<i>Geometry page</i> .....	226
Bearing Geometry box .....	226
Supply Ports Definition dialog box .....	228
<i>Fluids page</i> .....	231
<i>Operating Conditions page</i> .....	231

# RAPPID User's Guide

---

Rotor Speed .....	231
Pressure/Temperature box.....	232
Exit Inflow Conditions box.....	233
Loading box.....	234
Misalignment box.....	235
CHAPTER 18: THRUST BEARING – 2-D TILTING PAD .....	237
<i>Overview</i> .....	237
Bearing Geometry .....	237
Thermal model.....	237
Thermal and centrifugal growth .....	238
Groove model .....	238
Pivot model.....	238
Applied Load/Prescribed Eccentricity .....	243
Fluid Film gridding .....	243
Coordinate system.....	244
Conventions for defining pads.....	244
<i>Options page</i> .....	245
Analysis Parameter box .....	245
Output Format.....	245
Grid .....	245
Groove Temp .....	245
<i>Materials page</i> .....	246
<i>Geometry page</i> .....	246
Bearing Geometry box.....	246
Pad Definition dialog box .....	247
<i>Fluids page</i> .....	253
<i>Operating Conditions page</i> .....	254
Rotor Speed .....	254
Pressure/Temperature box.....	254
Loading box.....	256
CHAPTER 19: FLOW ELEMENTS COMPONENTS.....	258
<i>Overview</i> .....	258
<i>Options page</i> .....	258
Analysis Parameter box .....	258
Output Format.....	258
Wall Thermal Model .....	259
Stability Criteria.....	259
Grid .....	259
Solution Type.....	259
<i>Materials page</i> .....	259
<i>Geometry page</i> .....	260
Bearing Geometry box.....	260
<i>Fluids page</i> .....	260
<i>Operating Conditions page</i> .....	261
Boundary Conditions box .....	261
Loading box.....	262
<i>Output Options page</i> .....	263
CHAPTER 20: MISCELLANEOUS – TABULAR INPUT .....	264
<i>Overview</i> .....	264
<i>Options page</i> .....	264
Analysis Parameter box .....	264
Other box .....	264
<i>Dynamic Forces page</i> .....	264
CHAPTER 21: ANNULAR SEALS COMPONENTS .....	268
<i>Overview</i> .....	268
Labyrinth Configurations.....	268
<i>Options page</i> .....	271
Analysis Parameter box .....	271
Output Format.....	271
Grid .....	271
<i>Geometry page</i> .....	271



# RAPPID User's Guide

---

Settings box – Damping.....	271
Settings box – Labyrinth.....	272
Seal Geometry box – Damping and Smooth .....	273
Seal Geometry box – Labyrinth.....	276
<i>Fluids page</i> .....	277
<i>Operating Conditions page</i> .....	278
CHAPTER 22: DYNAMICS NETWORKS .....	280
<i>Overview</i> .....	280
Eigenvalue .....	280
Steady State Forced Response.....	280
Transient Forced Response .....	281
<i>Options page – Eigenvalue</i> .....	281
Analysis Parameter box .....	281
Other box .....	281
<i>Options page – Steady State Forced Response</i> .....	281
Analysis Parameter box .....	281
Other box .....	282
<i>Options page – Transient Forced Response</i> .....	282
Analysis Parameter box .....	282
<i>Rotors and Housings pages</i> .....	282
Description.....	283
Run Mode .....	283
Speed Ratio/Rotor Speed .....	285
Modal Damping .....	285
<i>Interconnection page</i> .....	285
Component Category.....	286
Component Type.....	286
Case Name.....	286
Define button .....	286
Interconnection Definition dialog box .....	286
<i>Forcing Function page</i> .....	289
Type .....	289
DOF (Degrees-of-Freedom).....	289
Forcing Function Name .....	290
Define button .....	290
Forcing Function Definition dialog box .....	290
<i>Output Options page – Eigenvalue</i> .....	293
Other box .....	293
Frequency Range for Reported Results box .....	294
Output Locations box .....	295
Rotor/Housing/Interconnection Output Locations dialog box .....	295
Forcing Function Output dialog box.....	296
<i>Output Options page – Steady State Forced Response</i> .....	296
Other box .....	296
Output Locations box .....	298
Rotor Output Locations dialog box.....	298
Housing Output Locations dialog box .....	298
Interconnection Output Locations dialog box.....	299
<i>Output Options page – Transient Forced Response</i> .....	300
Other box .....	300
Output Locations box .....	301
Rotor Output Locations dialog box.....	301
Housing Output Locations dialog box .....	302
Interconnection Output Locations dialog box.....	302
Forcing Function Output dialog box .....	303
<i>Plot Options – Eigenvalue</i> .....	304
Select Stability Parameter .....	304
Limit Eigenvalue Data to Display (Root Pre-sort).....	304
Sort Roots on Natural Frequency and Stability Plots .....	304
Eliminate Mode Shape Plots With Deflections < xx.xx% of Maximum Deflection .....	306
Add Harmonic Lines to Natural Frequency Plot.....	306
<i>Plot Options – Steady State Forced Response</i> .....	306

# RAPPID User's Guide

---

Select coordinate systems for all nodal location plots .....	306
Display forcing function location and magnitude on plots .....	306
Set displacement plots maximum ordinate values to .....	307
Set housing acceleration plots maximum ordinate values to .....	307
Set dynamic load plots maximum ordinate values to .....	307
Set relative displacement plots maximum ordinate values to .....	307
Eliminate response shapes with maximum < xx.xx% of Maximum Response .....	307
Calculate separation margins on X'-Y' Response plots .....	307
Show amplification factors > 'value' on X'-Y' Response plots .....	308
Remove grid lines from X'-Y' Response plots .....	308
Plot Multiple curves on the Response plots .....	308
Define Plot Curves dialog box .....	309
<i>Plot Options – Transient Forced Response</i> .....	<i>311</i>
FRF Plots .....	314
Waterfall plots .....	314
CHAPTER 23: TRANSIENT FLOW NETWORK .....	315
<i>Overview</i> .....	<i>315</i>
<i>Options page</i> .....	<i>315</i>
Analysis Parameter box .....	315
Loops .....	315
<i>Fluids page</i> .....	<i>315</i>
<i>Nodes page</i> .....	<i>316</i>
ID Text .....	316
Type .....	316
Define button .....	317
Define Internal/Boundary Node dialog box .....	317
View button .....	317
Active Loops .....	318
<i>Component Connectivity page</i> .....	<i>318</i>
Inlet Node .....	318
Exit Node 1 .....	318
Exit Node 2 .....	318
Component Category .....	318
Component Type .....	318
Case Name .....	319
Run Mode .....	319
<i>Output Options page</i> .....	<i>320</i>
Sample Rate .....	320
CHAPTER 24: INTEGRATED NETWORK .....	321
<i>Overview</i> .....	<i>321</i>
<i>Options page</i> .....	<i>321</i>
System Composition box .....	321
Simulation Control box .....	321
Analysis Parameter box .....	322
<i>Interactions page</i> .....	<i>322</i>
Network Category .....	322
Network Case .....	323
Interaction Type .....	323
Interactive column .....	323
Define button .....	324
Interactions dialog box .....	324
Flow Network and Component columns .....	326
<i>Components Settings page</i> .....	<i>326</i>
Network Category .....	326
Network Case .....	326
Update Profiles Library button .....	326
Update Profiles Library dialog box .....	327
Define button .....	327
Component Settings dialog box .....	327
<b>PART III – APPENDICES .....</b>	<b>330</b>

# RAPPID User's Guide

---

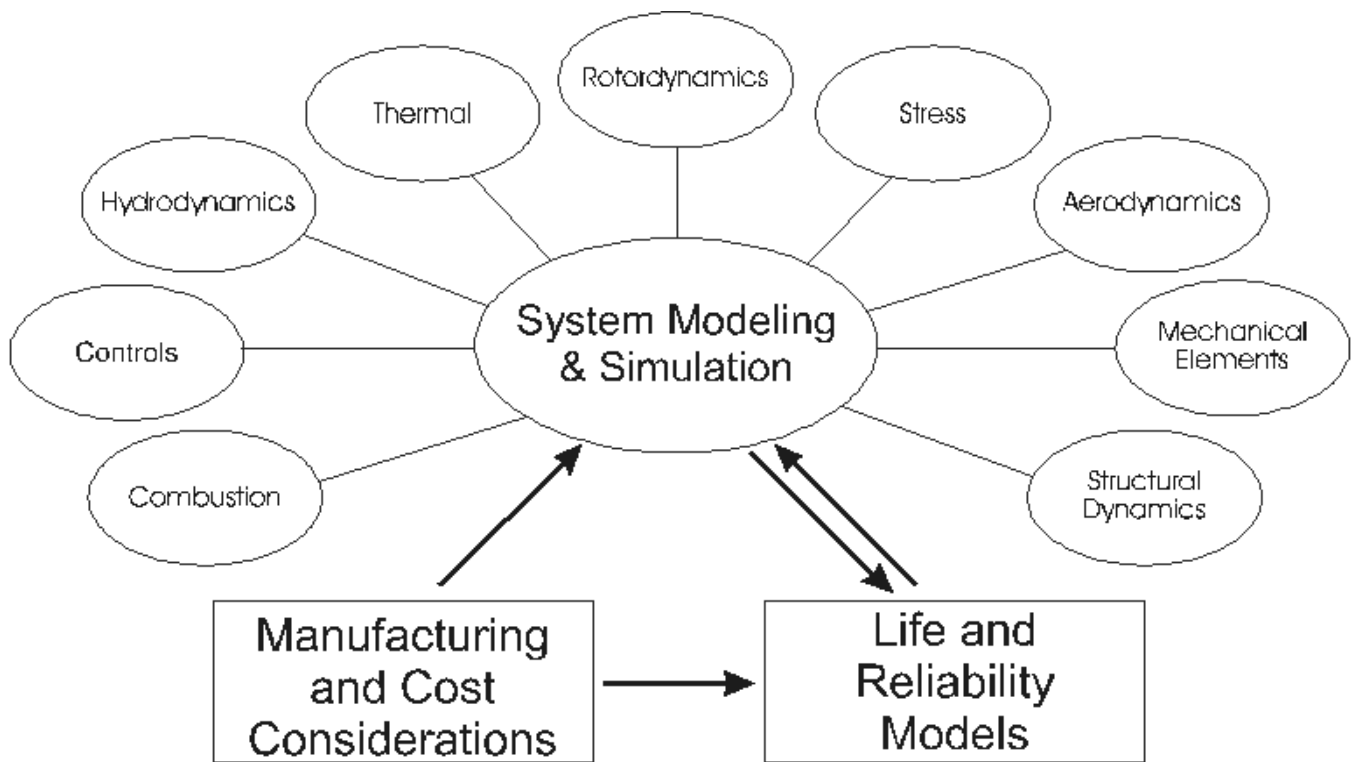
APPENDIX A: SCREEN ELEMENTS FOR ENTERING DATA .....	330
<i>Introduction</i> .....	330
<i>Edit boxes</i> .....	330
<i>Spin edit boxes</i> .....	330
<i>Radio buttons</i> .....	330
<i>List boxes</i> .....	331
<i>Drop-down menus</i> .....	331
<i>Data grids</i> .....	331
<i>Button-like cells in data grids</i> .....	332
<i>Cancel and OK/Done buttons</i> .....	332
<i>Other buttons</i> .....	333
APPENDIX B: TROUBLESHOOTING .....	333
APPENDIX C: SUPPORT INFORMATION .....	333
<b>GLOSSARY</b> .....	<b>335</b>
<b>INDEX</b> .....	<b>336</b>

## Introduction

This section is intended to provide an overview of the RAPPID™ software and to introduce you to the conventions used throughout the manual. In addition, this section includes directions for contacting RSR for technical assistance.

### About RAPPID™

RSR's RAPPID™ software, illustrated in the figure below, is an integrated, multi-disciplinary (FEA, CFD), multi-fidelity (0-D to 3-D), rotating machinery and propulsion modeling and simulation tool capable of accurate prediction of system vibration and performance characteristics and all related fluid flow conditions.



The focus of the software is to enable the creation of an integrated network analysis capability to fully couple fluid and structural interactions for proper phasing of forcing functions and accurate boundary conditions for transfer functions. The RAPPID™ package consists of an integrated library of computational fluid dynamic (CFD) analysis and finite element analysis (FEA) code modules. The levels of analysis for each module varies from rudimentary 0-D approximations all the way up to complete 3D analysis. The integration of the physical disciplines and distribution of fidelity enables a “virtual test” capability.

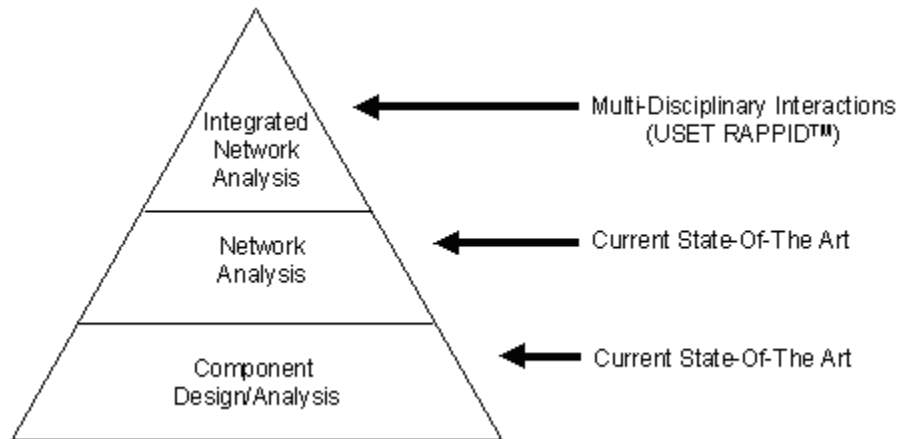
The Program's integrated architecture classifies solvers into one of the following three categories (see the **Analysis Pyramid** figure):

- Component Level Solvers

# RAPPID User's Guide

---

- Network Level Solvers
- Integrated Network Solver



Component level solvers are utilized to perform independent analysis of a specific system component by treating the component's interfaces with the system as user input boundary conditions. Specific component level solvers to be included in this version of RAPPID™ are:

- Hydrostatic Journal and Thrust Bearing Solvers
- Annular, Damper, and Labyrinth Seal Solvers
- Generalized Flow Passage Transfer Function Solver (CFD)
- Pipes
- Generalized Resistances
- Pumps
- Turbines

Network level solvers are utilized to perform analysis within a specific discipline (flow or structural) by treating the discipline's interfaces with interfaces from the other disciplines as user input boundary conditions. Network models are generated by directly defining the relationships between the various component level models and user-defined inputs used to describe the elements of the network. Specific network level solvers included in the integrated version of RAPPID™ are:

- Flow Network Solver
- Rotordynamics Network Solver (rotating & stationary structures)

The integrated network solver is utilized to perform multi-disciplinary analysis on the system of interest by treating the system's interfaces with the operating environment as user input boundary conditions. Integrated network models are generated by directly defining the relationships between the flow and/or rotordynamic networks used to define the system.

# RAPPID User's Guide

---

## About this manual

**Part I** of this manual explains how to use the RAPPID™ system to create, maintain, and run analysis cases.

**Part II**, **Part III**, and **Part IV** of this manual provide information about the modules you have bought with RAPPID™.

While going through the chapters in the first four parts of the manual, you may also want to refer to **Appendix A** in **Part V** for explanations about the various screen elements used to enter data into RAPPID™.

**Part V** of this manual contains appendices explaining the screen elements used to enter data, information about troubleshooting, and support information.

## Typographical conventions

Before you start using this manual, it is important to understand the terms and typographical conventions used in the documentation.

For more information on specialized terms used in the documentation, see the Glossary at the end of this manual.

The following kinds of formatting in the text identify special information.

Formatting convention	Type of Information
<b>Bold</b>	Items you must select, such as menu options, command buttons, or items in a list.
<i>Emphasis</i>	Names of modules within RAPPID™
Monospace	Text to type literally
CAPITALS	Names of keys on the keyboard. for example, SHIFT, CTRL, or ALT.
KEY+KEY	Key combinations for which the user must press and hold down one key and then press another, for example, CTRL+P, or ALT+F4.

**NOTE:** Throughout the documentation, "RAPPID™" and "the Program" are used interchangeably, both referring to the software application.

## Computer terminology

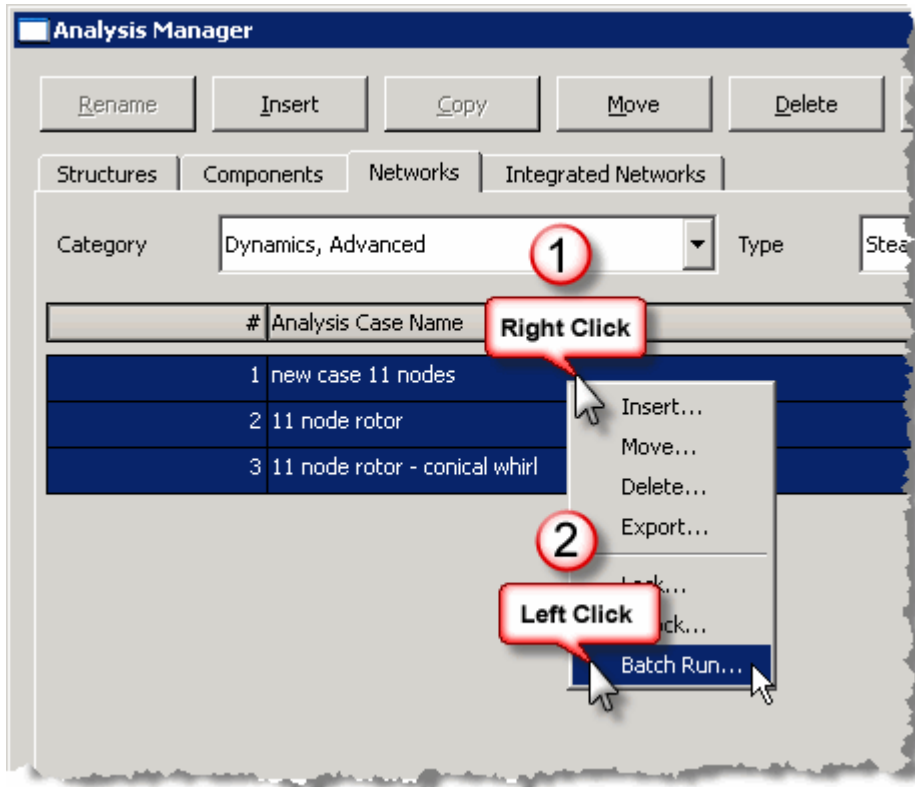
There are several conventions and terms used throughout this manual, which must be understood. For any Fedora-specific information, please refer to the Fedora documentation.

# RAPPID User's Guide

---

## User interface conventions

- A mouse *click* means a single press of the left (or primary) button.
- A *double click* means two clicks of the left (or primary) mouse button in rapid succession.
- *Right-clicking* in some areas of RAPPID™ accesses a popup menu of options applicable to the current window and selection. The following picture displays an example of a popup menu.



- *Dragging* the mouse means clicking and holding the left (or primary) button while moving the mouse in the desired direction.
- *Point* means to move the pointer to a specific object.
- The *active window* is the window in which you are currently working - it always has a highlighted border and title bar.

## Keyboard terms

- *Press* means to press a specific key on your keyboard. For example “press ENTER” means “press the ENTER key.”
- *Type* tells you specific information to type from the keyboard. For example, “type C:\RAPPID\PROJECT” means to type those characters on the keyboard, exactly as given (without the quotes).

- *Enter* is a general term referring to information that cannot be expressed with explicit instructions. For example, “Enter the current date” means if today is February 9, 2009, type 2/9/09.

### General terms

- *Select* means to pick a specific menu command, drop-down list item, or object. For example, to “Select **Manager** from the **Project** menu” use the mouse to click the **Project** menu on the menu bar and then click the **Manager** command.
- *Click* means to pick an object. For example, to “click the **Options** tab” means to click the tab with the mouse.



## Part I: The RAPPID™ system

This part of the user's guide illustrates how to use the RAPPID™ system to creating and running projects and analysis cases. Reference chapters about specific analysis case types are available in **Part II – Analysis case types** on page 127.

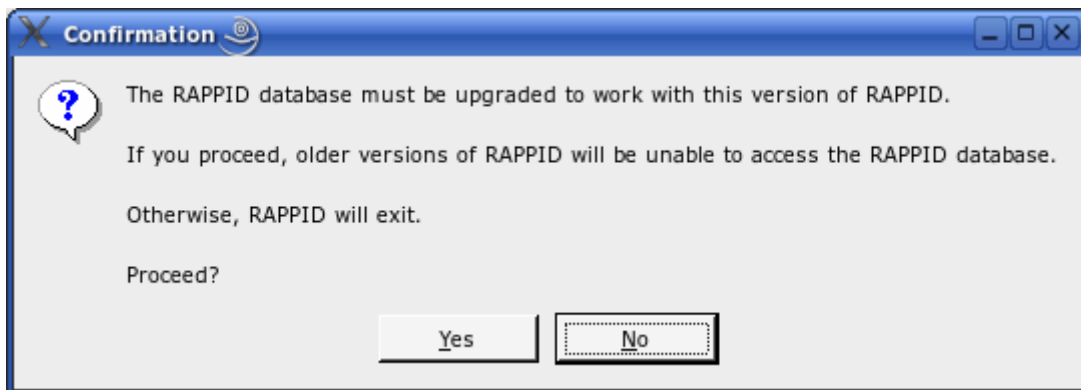
### ***Chapter 1: Getting Started***

This chapter discusses how to start RAPPID™, what is available on the main RAPPID™ screen, unit systems used within RAPPID™, and getting help within RAPPID™.

#### **Starting RAPPID™**

Double clicking the RAPPID™ icon on the Windows desktop brings up the **Welcome to RAPPID™!** splash screen, containing license and contact information. Click **Ok** to continue.

The data you enter in the Program is stored in a database. The database structure may be updated from time to time within the Program. If you are starting the Program for the first time or if you obtained an updated version of the Program, you may see the dialog box shown in the following figure.

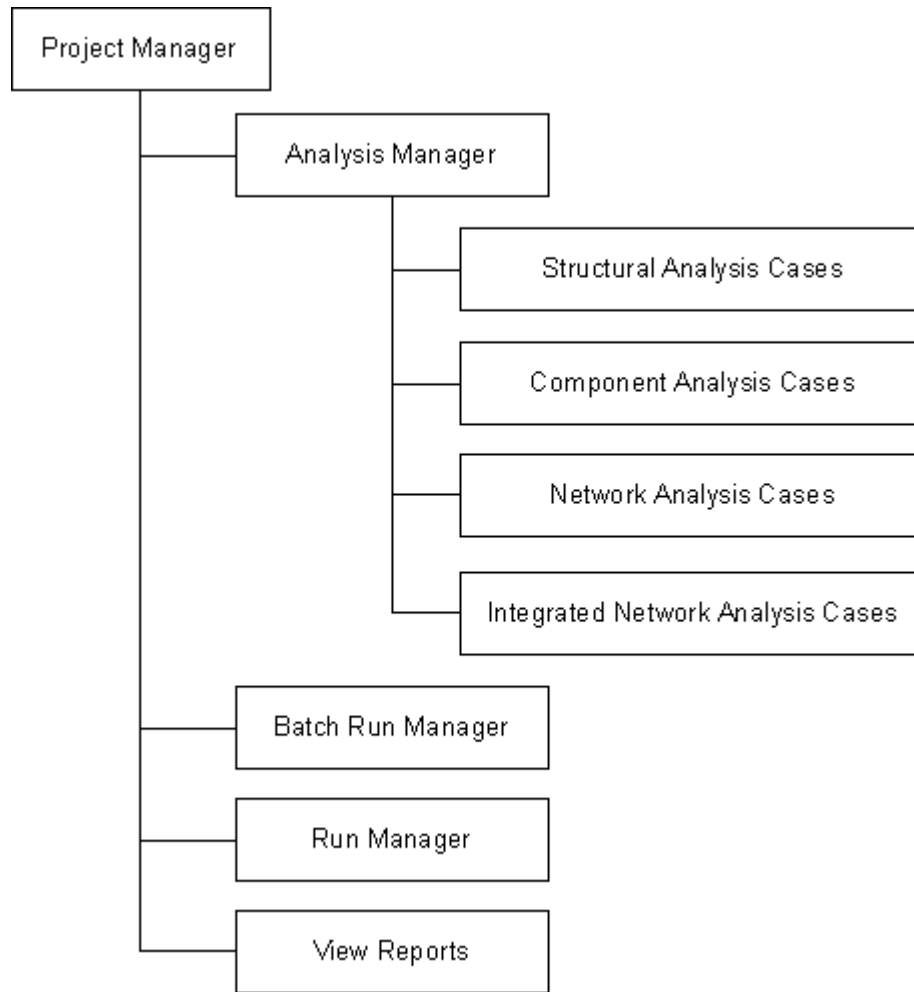


If you click **No**, RAPPID™ will not open. In order to continue to use the Program, you must click **Yes**. When you click **Yes**, the Program opens with the **Project Manager** open. See

**Chapter 2: Working with** projects on page 14 for more information about the **Project Manager**.

## Introducing the main RAPPID™ screen

RAPPID™ is separated into managers used to create and manage analysis cases. The following figure maps out the basic structure of the Program.



This structure is also pictured on the main RAPPID™ screen and each box is a button that is available at different times to access different parts of the Program.

The **Project Manager** (see page 14) allows you to create (insert) new projects, rename, copy, move, delete, and open existing projects, and archive and unarchive projects.

The **Analysis Manager** allows you to create (insert) new analysis cases of the following types:

- Structural
- Component
- Network

---

# RAPPID User's Guide

---

- Integrated network

You can also rename, copy, move, delete, and open existing analysis cases.

The **Batch Run Manager** allows you to run multiple cases, either sequentially or in parallel, without having to open each separate case.

The **Run Manager** allows you to monitor currently running analysis cases.

## Specifying the location of projects and archives

As mentioned in **Starting RAPPID™** on page 6, RAPPID™ utilizes a database to store the data you enter. The files for this database need to be stored on your hard drive. When you archive a project that file is also stored on your hard drive.

When RAPPID™ is installed and run for the first time, the project files are stored in the same directory as the Program. A directory for storing archived project files is also created in the Program directory at the same level as the projects directory.

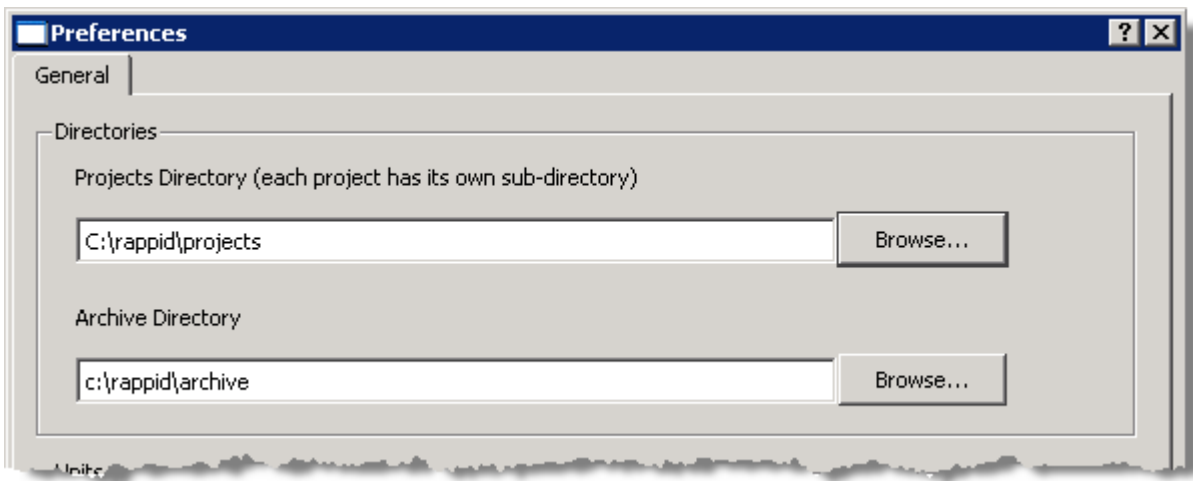
**NOTE:** Do not run the Program on read-only media, such as a CD or DVD. The Program needs to be able to initially write to the program directory.

To change the location for storing the project files and any archive files you create, follow these steps:

**NOTE:** The directories for the project files and archive files do not have to be located within the same directory. The archive directory will be the only directory you will be accessing outside of the Program.

1. Select **Preferences** from the **Project** menu.

The **Preferences** dialog box displays.

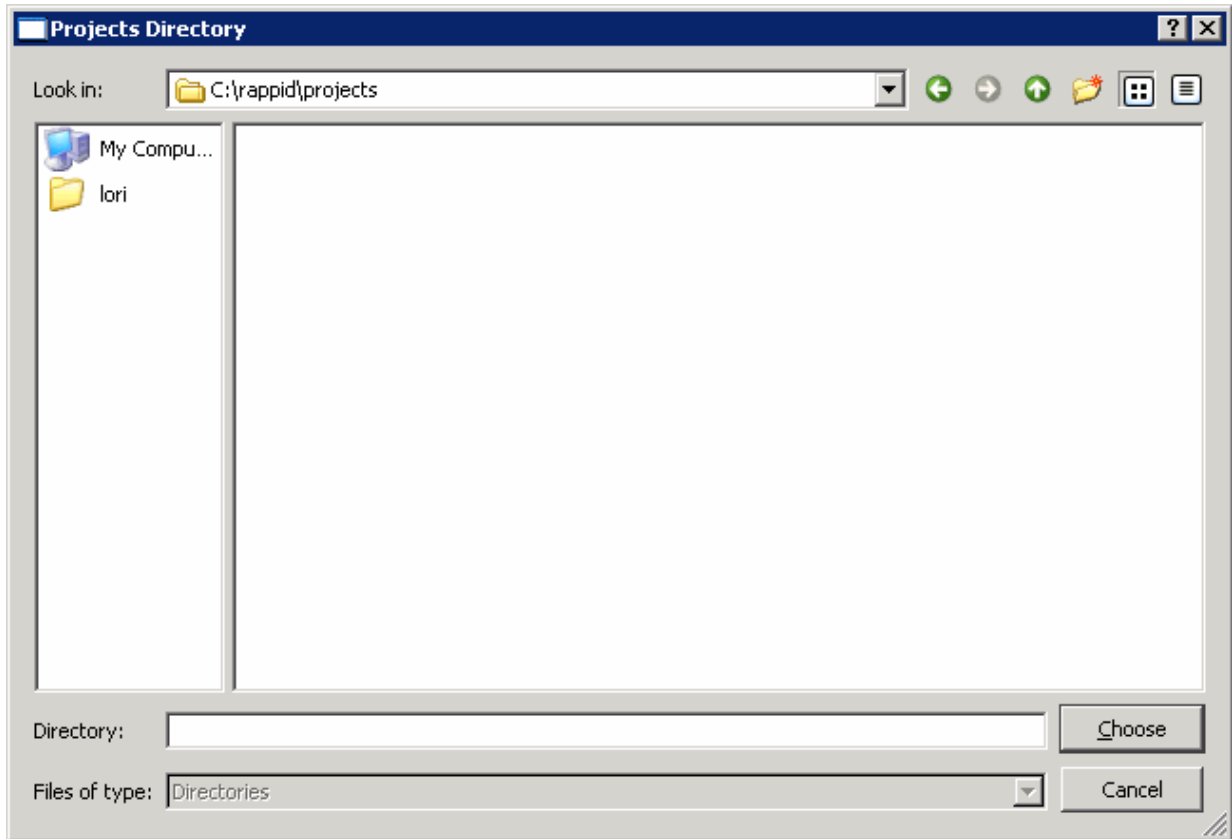


2. In the **Directories** box, click the **Browse** button next to the **Projects Directory** edit box to select a directory in which to store all of your active projects.

# RAPPID User's Guide

**NOTE:** If you know the full path of the directory you want to use, you can type it directly into the **Projects Directory** edit box.

The **Projects Directory** dialog box displays.



3. Navigate to the directory you want to use or create a new directory.
4. Click **Done** to accept the selected directory and return to the **Preferences** dialog box.

**NOTE:** When you change the **Directories**, the Program exits and you must restart the Program for the changes to the directories to take effect.

5. Follow steps 1-4 to select the **Archive Directory**.

## Working with unit systems in RAPPID™

Unit selections in the Program include **English**, **SI-m**, and **SI-mm**. The units for many parameters are given in the following table.

Parameter	English	SI-m	SI-mm
Length	inch	meter	millimeter
Force	pound	Newton	Newton

# RAPPID User's Guide

Parameter	English	SI-m	SI-mm
Head	feet	meter	meter
Pressure	psi	Bar	Bar
Temperature	°F	°C	°C
Angle	degree	degree	degree
Speed	rpm	cpm	cpm
Velocity	feet/s	meter/s	meter/s
Flow Rate	gallon/minute	liter/second	liter/second
Density	lbm/ft <sup>3</sup>	kg/m <sup>3</sup>	kg/m <sup>3</sup>
Viscosity	lbm/ft-s	N-s/m <sup>2</sup>	centiPoise
Specific Heat	BTU/lbm-°R	kJ/kg-K	kJ/kg-K
Power	horsepower	kiloWatt	kiloWatt
Specific Speed	rpm, gpm, ft	cpm, l/s, m	cpm, l/s, m

**NOTE:** Inputs that do not have units associated with them are defined as **Dimensionless**, and a dash ‘(-)’ displays for those fields.

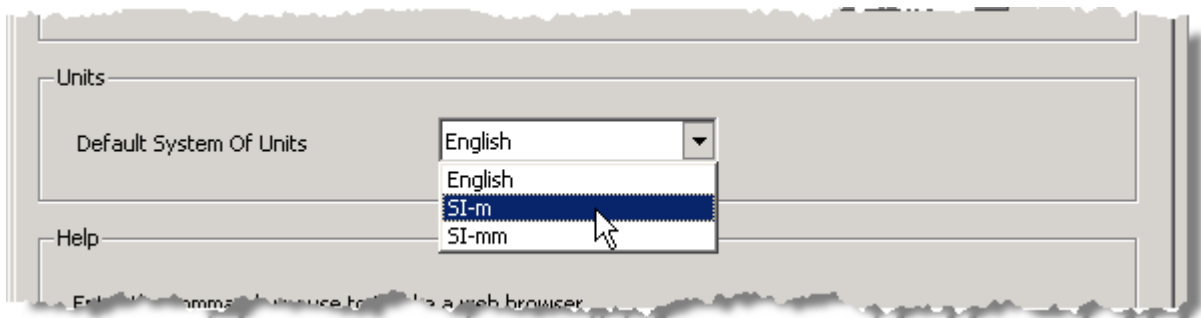
## Changing the default system of units

You can choose the unit system used by default when a new analysis case is created. To do this, follow these steps:

- Select **Preferences** from the **Project** menu.

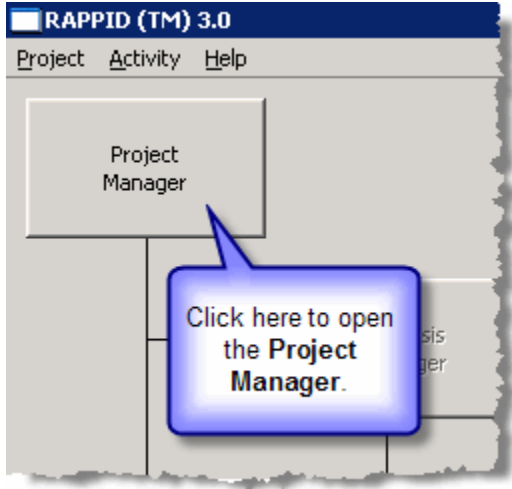
The **Preferences** dialog box displays.

- In the **Units** box, select an option from the **Default System of Units** drop-down menu to specify the unit system you want selected by default in new analysis cases.



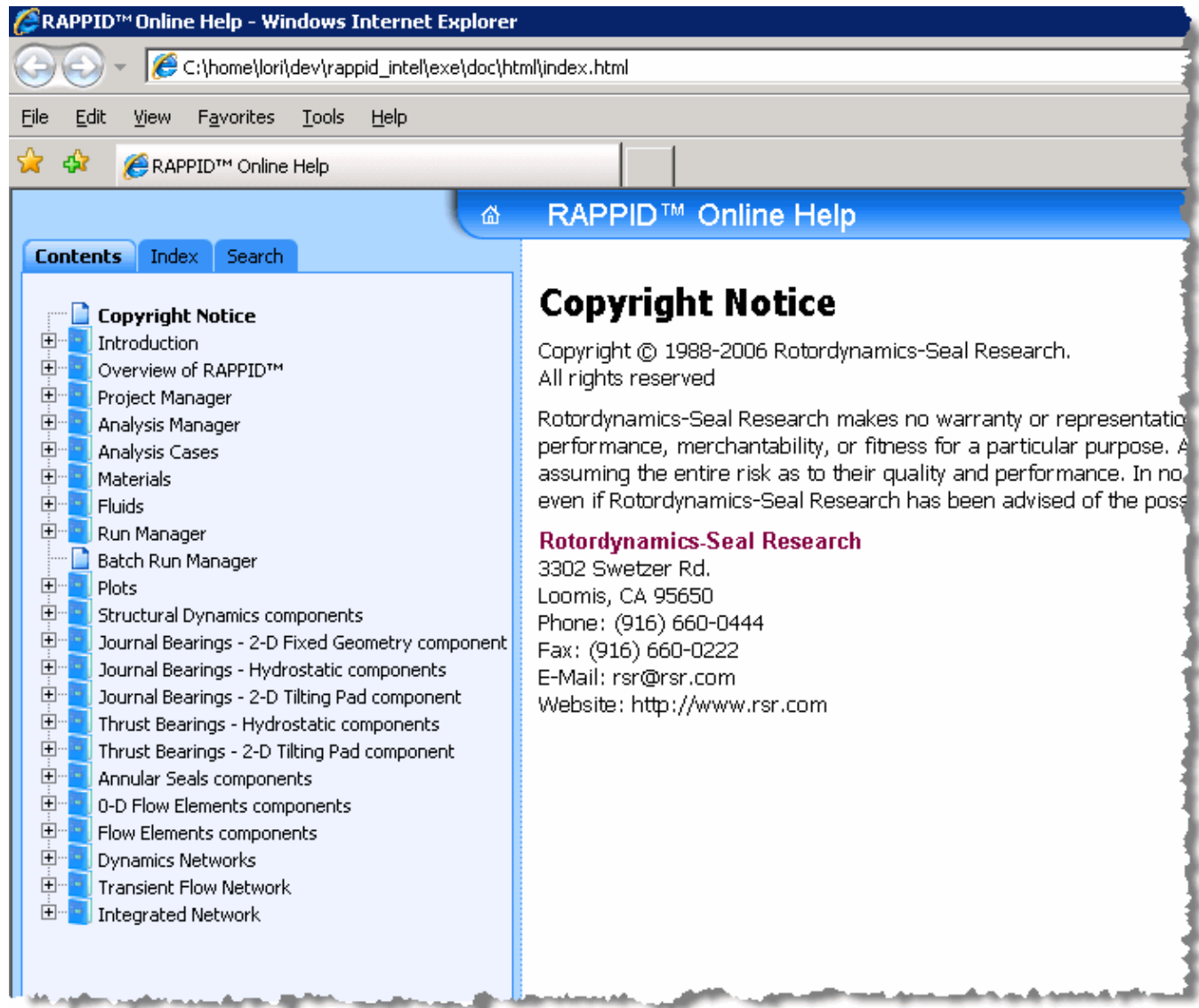
- Click **Done**.

**NOTE:** If you accessed the **Preferences** dialog box while a project was open, the project is closed before the dialog box is opened. When you are done setting preferences, you are returned to the main Program window. Click the **Project Manager** button on the main Program window or select **Manager** from the **Project** menu to open the **Project Manager** again to open a project and access analysis cases.



## Getting help in RAPPID™

Most screens and dialog boxes within RAPPID™ have a **Help** button that accesses a help topic in the main help system containing information about the current screen or dialog box. You can also access any other help topic using the **Contents** and **Index** tabs in the left pane.



## Specifying the location of the web browser used for displaying online help

You can choose which web browser RAPPID™ uses to display the online help. To do this, follow these steps:

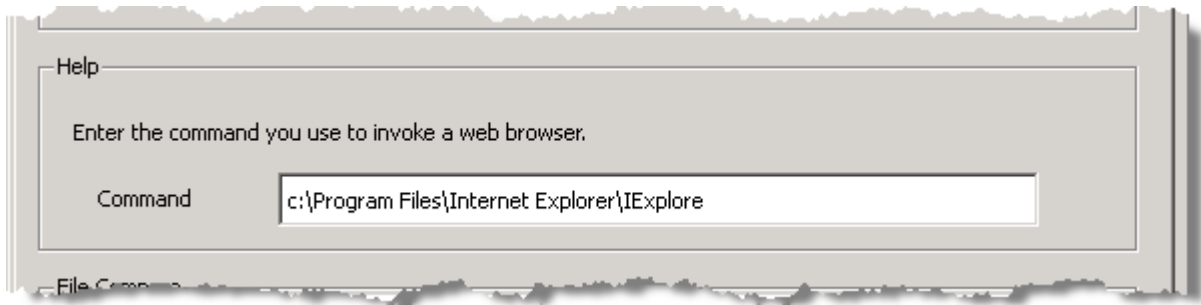
1. Select **Preferences** from the **Project** menu.

The **Preferences** dialog box displays.

## RAPPID User's Guide

---

2. In the **Help** box, enter the command in the **Command** edit box used to access the web browser you want to use to view the RAPPID™ online help.



**NOTE:** When entering the command to start a web browser, enter the command you would use on the command line. See your operating system documentation for details on running programs from the command line.



## Chapter 2: Working with projects

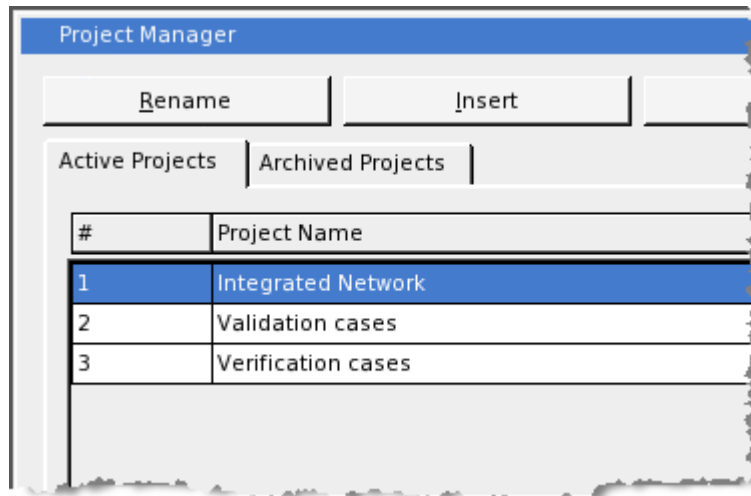
This chapter introduces you to the **Project Manager** and discusses the various actions available to perform on projects.

### Introducing the Project Manager

Use the **Project Manager** to create and manage projects that contain groups of analysis cases. You can have multiple projects allowing you to group your analysis cases in a manner that suits your needs.

### Accessing active and archived projects

Two tabs are available on the **Project Manager** window: **Active Projects** and **Archived Projects**, as shown in the figure below.



The **Active Projects** tab contains a list of projects you can use to manage analysis cases.

**NOTE:** Until a project is created or retrieved from an archive, the **Active Projects** tab does not display any projects.

The **Archived Projects** tab contains a list of projects that have been archived in RAPPID™'s archive format for use if you want to backup or move one or more projects.

See **Chapter 8: Backing up** on page 124 for information on archiving and unarchiving projects.

### Interpreting the dates on projects

Two dates are listed for each project on the **Active Projects** tab:

- **Date Last Entered**

The **Date Last Entered** indicates the last date on which the project was opened.

# RAPPID User's Guide

---

**NOTE:** The **Date Last Entered** date changes even if you do not add or delete any cases within the project. The only requirement for the **Date Last Entered** date to change is for the project to be opened.

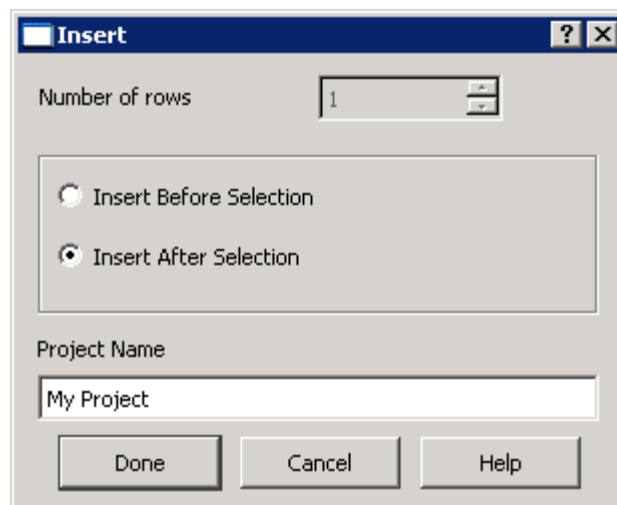
- **Date Created**

The **Date Created** lists the date on which the project was initially created.

On the **Archived Projects** tab, the **Date Created** is the only date that displays for archives.

## Creating a new project

Use the **Insert** button to create a new project on the **Active Projects** tab. You can only add one new project at a time.

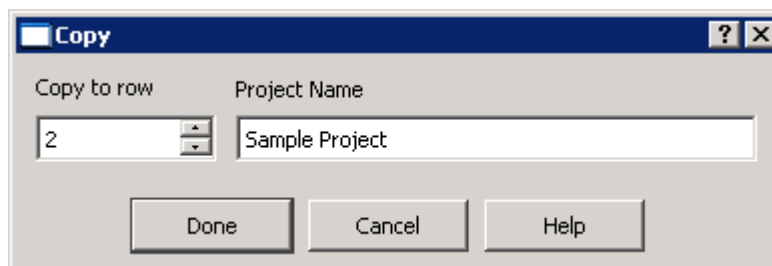


The screenshot shows a dialog box titled "Insert". It has a "Number of rows" field with the value "1". Below this are two radio buttons: "Insert Before Selection" (unselected) and "Insert After Selection" (selected). At the bottom is a "Project Name" text field containing "My Project". There are three buttons at the bottom: "Done", "Cancel", and "Help".

**NOTE:** The **Insert** button is the only button available until a project has been created or un-archived.

## Copying a project

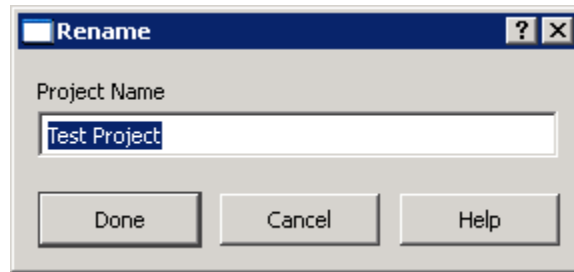
If you want to create a new project containing all the same cases as another project, use the **Copy** button to copy the desired project into a new project.



The screenshot shows a dialog box titled "Copy". It has two fields: "Copy to row" with the value "2" and "Project Name" with the value "Sample Project". There are three buttons at the bottom: "Done", "Cancel", and "Help".

## Renaming a project

Use the **Rename** button to rename an active project or an archived project.

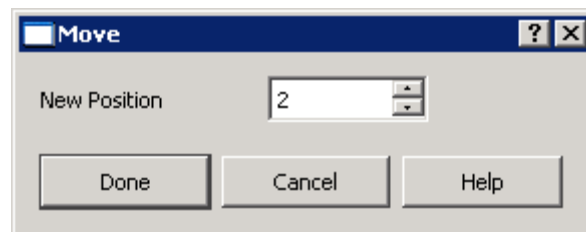


**NOTE:** When you rename a project on the **Archived Projects** tab, the archive file in the archive directory is also renamed.

## Moving an active project

When you add a new project, it is inserted at the end of the list of projects on the **Active Projects** tab. However, you can rearrange the order of the active projects. Use the **Move** button to move a selected project to any other location in the list of projects.

On the **Move** dialog box, specify the **New Position** for the selected project by typing in a new number or using the up and down arrows to select a value.

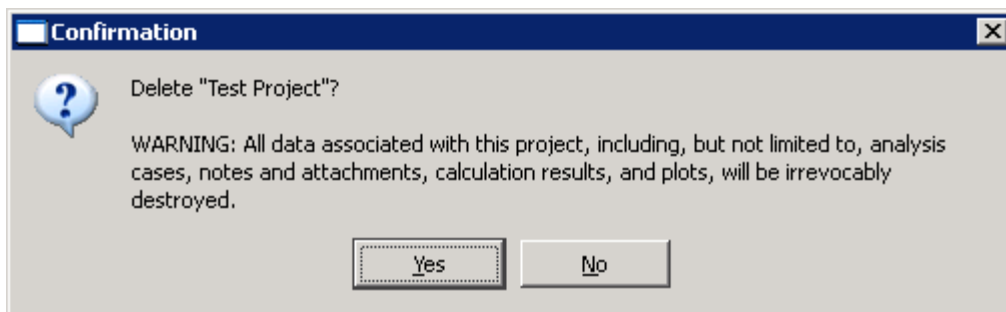


**NOTE:** You cannot move projects listed on the **Archived Projects** tab, therefore, the **Move** button is not available. Archived projects are listed in the same alphabetical order as they are listed in the archive directory on your hard disk.

## Deleting a project

Use the **Delete** button to delete an active project or an archived project.

A confirmation dialog box containing a warning displays stating that all data within this project, including analysis cases, calculation results, and plots will be permanently deleted. Be sure this is what you want to do.

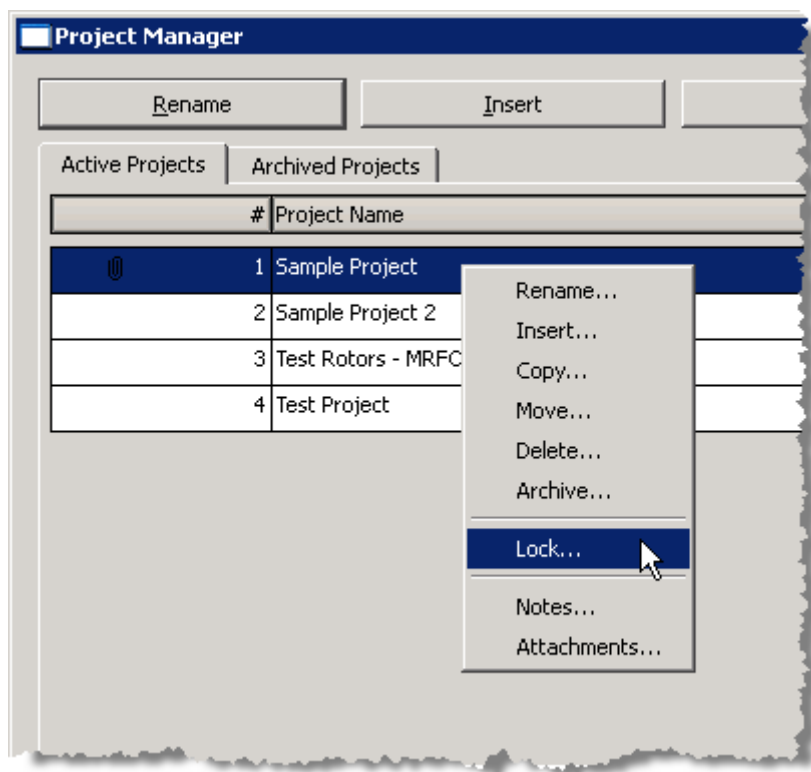


**NOTE:** When you delete an archived project, the archive file is deleted from your hard drive.

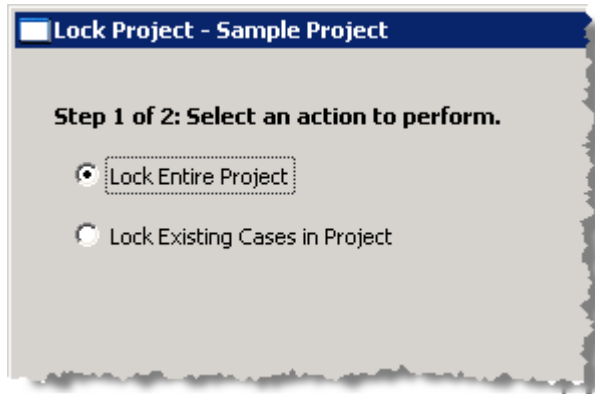
## Locking and unlocking projects

Locking a project prevents inadvertent changes to a project from being saved to the project and to all the cases in the project. Use this feature if you want to preserve the values and settings within all of the analysis cases within a project.

To lock a project, right-click on the project you want to lock and select **Lock** from the popup menu.



The **Lock Project** dialog box displays.



## Locking an entire project

Use the **Lock Entire Project** option in **Step 1** on the **Lock Project** dialog box to lock the project itself in addition to all the analysis cases in the project. You will not be able to add any new analysis cases to the project if you select this option.

## Locking existing analysis cases in a project

Use the **Lock Existing Cases in Project** option in **Step 1** on the **Lock Project** dialog box to lock all the analysis cases in the project but NOT the project itself. This can be used to prevent changes to existing analysis cases in the project, but it will allow you to add new cases to the project.

When locking a project, you can also choose to lock all of the analysis cases in that project. If you do lock all of the analysis cases when locking the project, and you assign a password to the locked project, the project-locking password applies to all the locked analysis cases, as well. The two locking passwords in the project would be the same.

## Using passwords

There are two passwords that can be associated with each project:

- one for locking the project
- one for locking the analysis cases in the project (see x-ref) for more information)

Both of these passwords are optional. You can lock a project and/or the analysis cases within the project using no password.

**NOTE:** The password(s) you enter when locking a project and all the cases in a project is intended to provide protection against inadvertent changes. Do not expect high security!

## RAPPID User's Guide

---

If you selected the **Lock Entire Project** option in **Step 1**, both sets of password edit boxes are available as displayed in the following picture:

The screenshot shows a Windows-style dialog box titled "Lock Project - Sample Project". Inside, it says "Step 2 of 2: Enter optional password(s)." Below this is a note: "NOTE: The passwords you enter below are intended to provide protection against inadvertent changes. Do not expect high security! The unencrypted passwords can easily be recovered from the 'password.txt' file in the project directory." There are two sections for password entry. The first section is "Password to unlock individual cases in project" with "Enter once:" and "Enter again:" labels followed by text input boxes. The second section is "Password to unlock project" with "Enter once:" and "Enter again:" labels followed by text input boxes. At the bottom are four buttons: "Cancel", "< Back", "Finished" (which is highlighted with a dashed border), and "Help".

The **Password to unlock project** edit boxes are not available if the **Lock Existing Cases in Project** option in **Step 1** was selected.

**NOTE:** Once you enter a password to lock a project, this password exists whether or not you have any projects locked. If you have no projects currently locked, and you want to lock a project, RAPPID™ will ask you to enter the old password in addition to entering the new password twice or applying no password (leave the **Enter new password** edit boxes blank). If you apply no password after entering the old password, the project-locking password is cleared.

### Unlocking a project

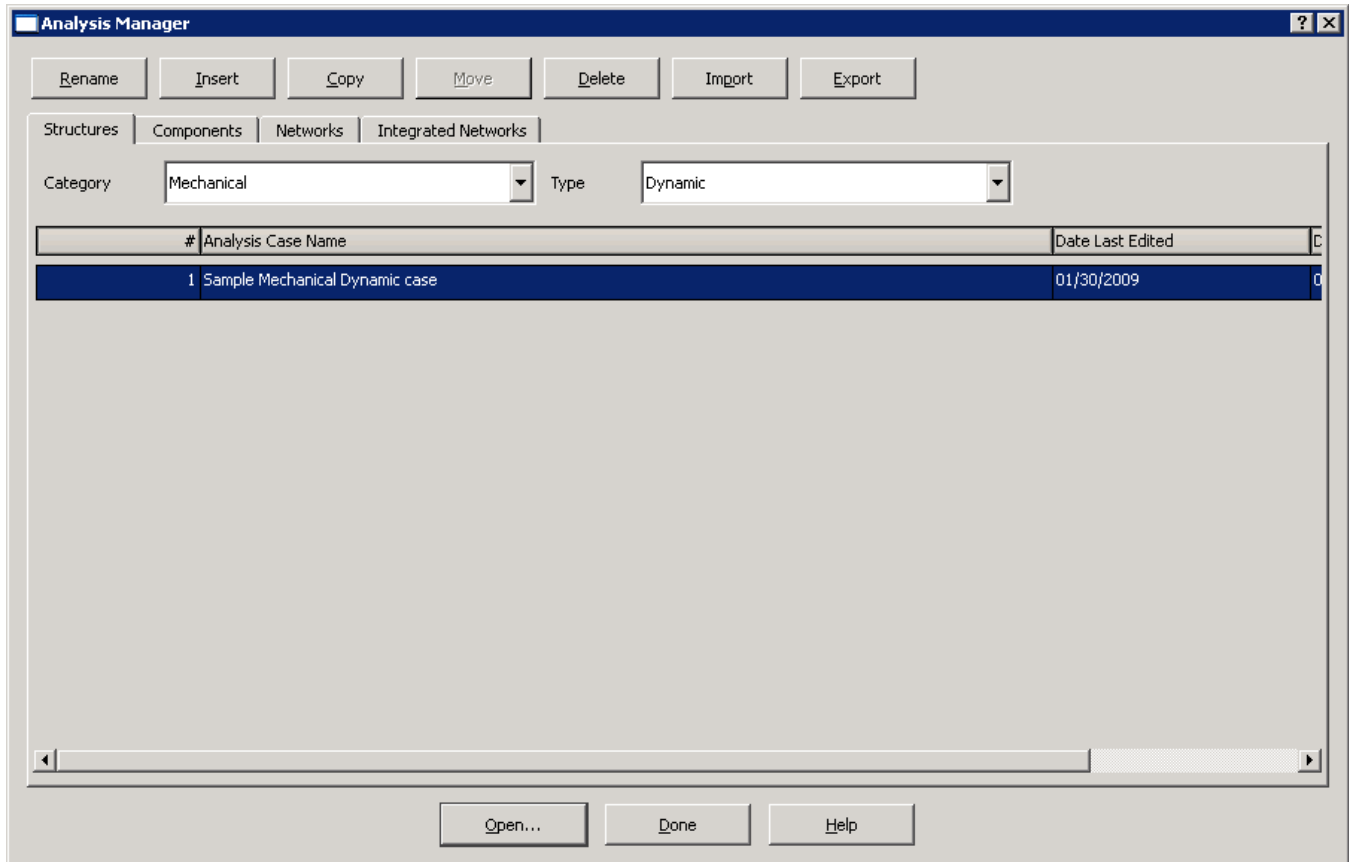
When you unlock a project, the individual analysis cases are not automatically unlocked. You must navigate to one of the cases in the project to unlock that case or all of the cases in the project.

## Chapter 3: Working with analysis cases

This chapter introduces you to the **Analysis Manager** and discusses the various actions available to perform on analysis cases and within the case manager.

### Introducing the Analysis Manager

When you open a project in the **Project Manager**, the **Analysis Manager** displays. Use the **Analysis Manager** to create and manage analysis cases in the currently open project.



There are four tabs in the **Analysis Manager**, specifying the four levels of analysis cases RAPPID™ supports:

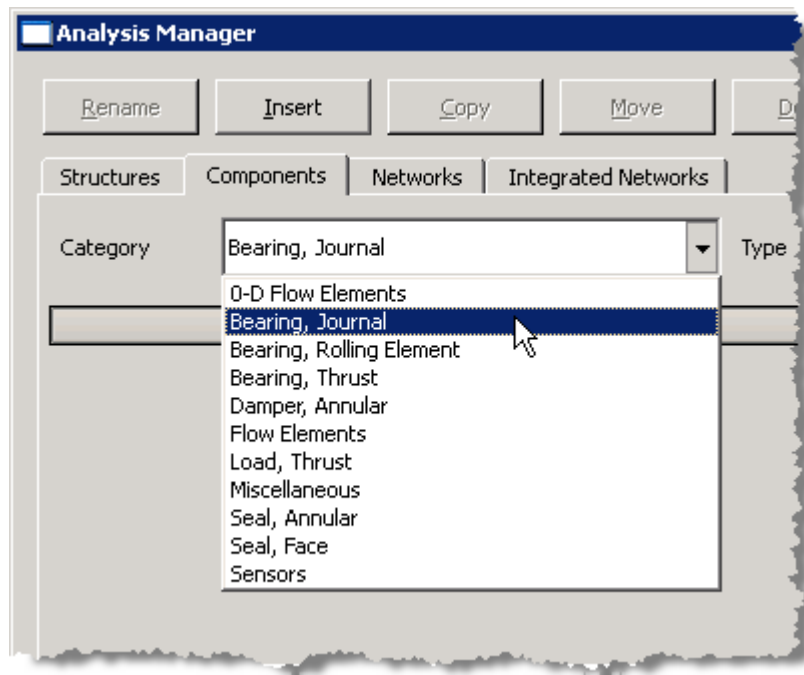
- **Structures**
- **Components**
- **Networks**
- **Integrated Networks**

There are three drop-down menus at the top of each tab, allowing you to choose the specific type of analysis case under each general type listed above:

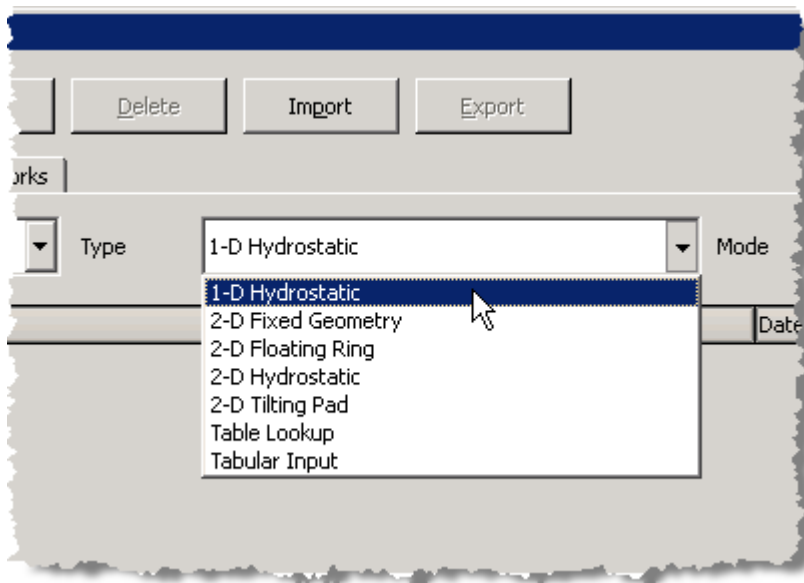
# RAPPID User's Guide

- **Category**
- **Type**
- **Mode**

For each **Category** there may be multiple **Types** available. For example, when you select the **Bearing, Journal** category on the **Components** tab,



the following types are available:



The **Category** and **Type** drop-down menus contain different options for each of the four tabs.



---

## RAPPID User's Guide

---

The **Mode** drop-down menu allows you to specify whether the case being created is a **Deterministic** case or a **Probabilistic** case.

Two dates are listed for each analysis case:

- **Date Last Edited**

The **Date Last Edited** indicates the last date on which the analysis case was saved, or when the notes or attachments were last modified.

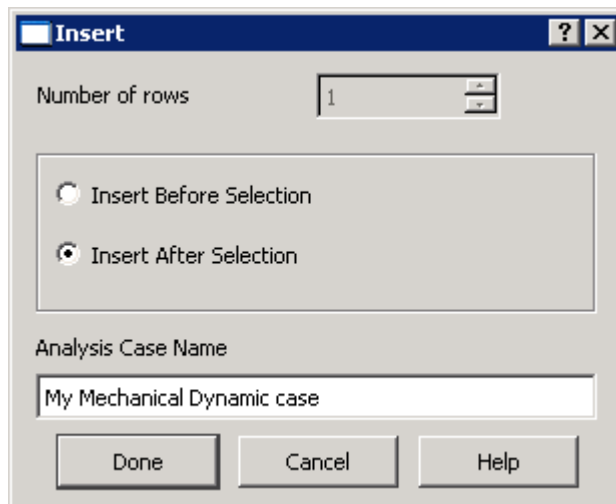
- **Date Created**

The **Date Created** lists the date on which the analysis case was initially created.

### Creating a new analysis case

Before adding a new analysis case to your project, you must select a **Category** and **Type** for the desired analysis case type and specify the **Mode**. The default **Mode** is **Deterministic**, which is the most commonly used mode.

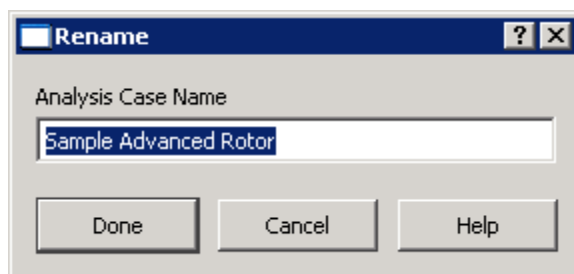
Once you have selected the desired **Category** and **Type** for the analysis case, use the **Insert** button to add a new analysis case to the project.



Specify whether you want to add the new case before or after the currently selected analysis case and enter a unique name for the case. No two analysis cases can have the same name.

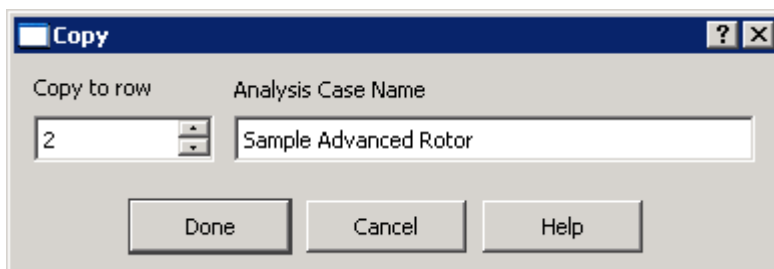
### Renaming an analysis case

Use the **Rename** button to rename the currently selected analysis case.



### Copying an analysis case

Use the **Copy** button to copy the currently selected analysis case as a new analysis case.



Enter or select the row number to which you want to copy the new analysis case in the **Copy to row** spin edit box and specify a new name for the new analysis case.

**NOTE:** You cannot copy an analysis case to a new analysis case using the same name as the original analysis case.

In the **Copy to row** spin edit box, you can specify any row number from 1 through one past the current last row number. As an example, there are three existing analysis cases. You want to select the second analysis case to copy and put it at the end of the list of analysis cases. Enter **4** in the **Copy to row** spin edit box and enter a new name for the new analysis case.

**NOTE:** If you specify a row number in the **Copy to row** spin edit box that corresponds with an analysis case that already exists, the existing analysis case is NOT replaced. The new analysis case is placed in that position and the existing analysis cases are shifted to accommodate the new analysis case.

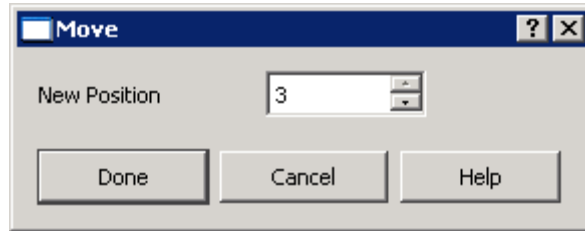
### Moving an analysis case

Use the **Move** button to move a selected analysis case to any other location in the list of analysis cases of the same type.

## RAPPID User's Guide

---

On the **Move** dialog box, specify the **New Position** for the selected analysis case by typing in a new number or using the up and down arrows to select a value.



**NOTE:** The **Move** button is only available when there are multiple analysis cases of the currently selected analysis case type in the **Analysis Manager**.

### Transferring case inputs

The **Transfer Case Inputs** button in the upper-right corner of the **Analysis Manager** window only displays when the **Networks** tab is active and is only available for *Dynamics* network cases. This feature allows you to import data from one *Dynamics* network case to a different *Dynamics* network case, e.g., from a *Steady State Forced Response* case (source case) to a *Transient Forced Response* case (target case).

To transfer inputs from one *Dynamics Network* analysis case into another, follow these steps:

1. On the **Networks** tab in the **Analysis Manager**, select **Dynamics, Advanced** in the **Category** drop-down menu and select one of the three types of **Dynamics, Advanced** analysis case types in the **Type** drop-down menu.
2. Select an analysis case INTO which you want to transfer inputs.

3. Click the **Transfer Case Inputs** button.

The **Transfer Case Inputs** dialog box displays.

The screenshot shows the 'Transfer Case Inputs' dialog box with a blue title bar and standard window controls. It is divided into three steps. Step 1: 'Select the type of the case that contains the inputs you want to transfer.' It features a 'Type' label and a drop-down menu currently showing 'Eigenvalue'. Step 2: 'Select the name of the case that contains the inputs you want to transfer.' It includes a label 'Cases Containing Input Data' above a list box containing 'Sample Dynamics Eigenvalue case'. Below the list box is a note: 'NOTE: If you want to transfer inputs from a case in another project, you must first import the case into this project.' Step 3: 'Confirm your intent to REPLACE THE CONTENTS of the case listed below.' It has a label 'Steady State Forced Response Case for transferred inputs' and a checked checkbox with the text 'Transfer case inputs from the case I selected in Step 2 into the case listed above.' At the bottom are three buttons: 'Done', 'Cancel', and 'Help'.

4. In **Step 1**, use the **Type** drop-down menu to select the type of case FROM which you want to transfer inputs.

**NOTE:** The types available in the **Type** drop-down list are all types of *Dynamics*, *Advanced* cases not including the type selected on the **Networks** tab in the **Analysis Manager**.

5. In **Step 2**, use the **Cases Containing Input Data** list box to select the specific case FROM which you want to transfer inputs.
6. In **Step 3**, make sure the name of the case corresponds to the case INTO which you want to transfer the inputs.

## RAPPID User's Guide

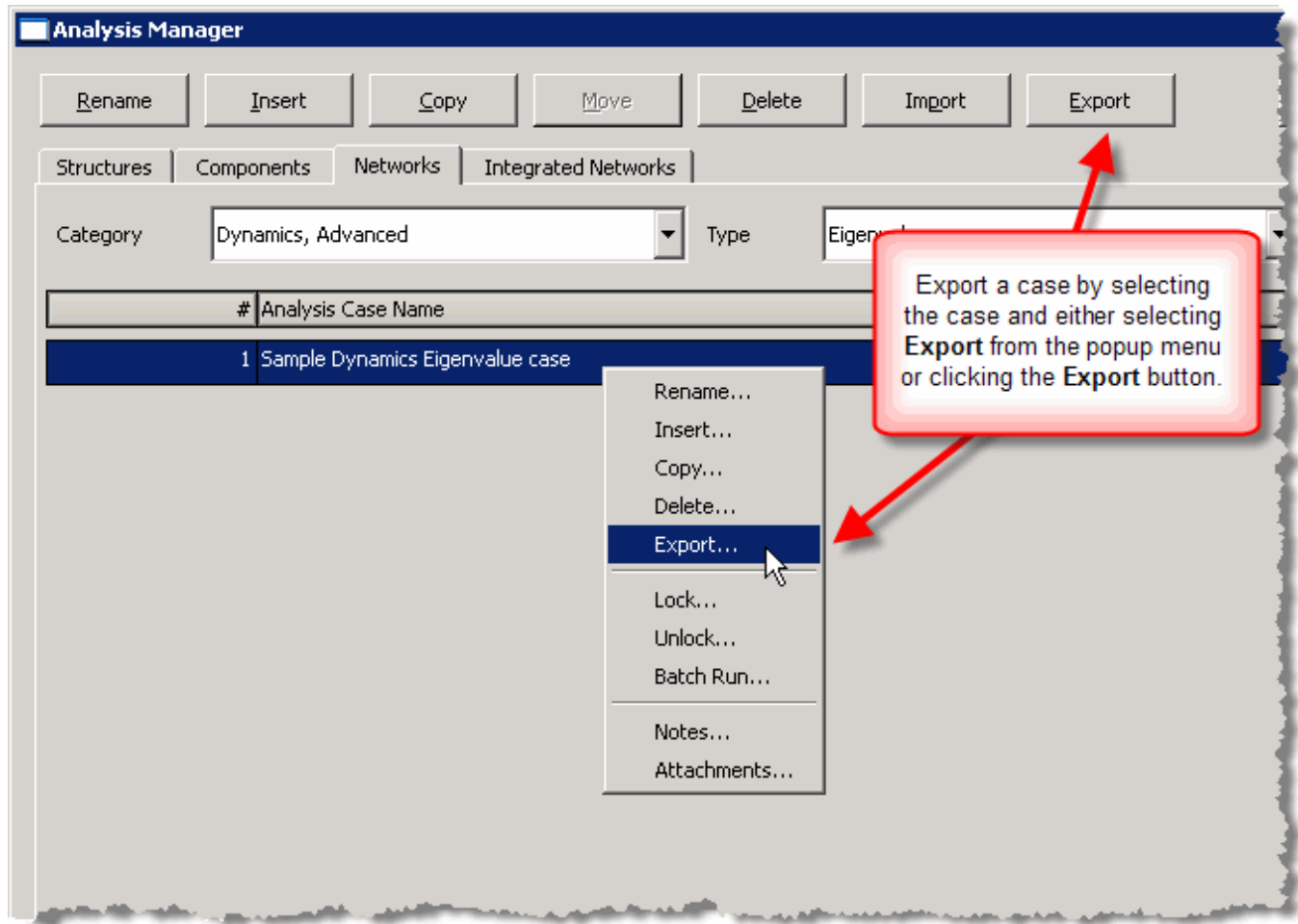
- Click the **Transfer case inputs from the case I selected in Step 2 into the case listed above** check box to confirm your selections.

**NOTE:** Clicking the check box does not transfer the inputs yet.

- Click the **Done** button to transfer the inputs.

### Exporting analysis cases

You can export one or more analysis cases into a new project archive, a new project or an existing project. When you export a network or an integrated network analysis case, all the referenced cases are also exported.



You can choose whether you want to include fluids with the exported cases. If the cases being exported use fluids, it would be a good idea to export the fluid(s) with the case(s) so the case(s) will still run when accessed from another project or archive.

**NOTE:** If your case(s) use **Real Properties** fluids, the files for these fluids are large (on the order of hundreds of megabytes).

# RAPPID User's Guide

Exporting analysis cases into a new project archive is similar to creating a project archive. The difference is that archiving selected analysis cases using the **Export** feature allows you to archive only certain analysis cases and not all of the analysis cases in a project.

To export an analysis case, follow these steps:

1. In the **Analysis Manager**, click one of the tabs for the type of analysis case you want to export.
2. Select the **Category** and **Type** of the analysis case you want to export.
3. Select one or more analysis cases from the resulting list.
4. Click the **Export** button.

The **Export Analysis Case(s)** dialog box displays.

**Export Analysis Case(s)**

Export Source

Project Name: Sample Project

Analysis Case Name: **Multiple Analysis Cases Selected**

Date Last Edited: 00/00/0000

Date Created: 00/00/0000

Export Destination

☒ Export analysis case(s) to a new project archive

Archive Name:

☒ Exclude fluid tables from archive.

☐ Export analysis case(s) to a new project

Project Name:

☐ Export analysis case(s) to an existing project

Select Project... No Project Selected

Done Cancel Help

5. There are three ways you can export analysis cases:
  - a. Use the **Export analysis case(s) to a new project archive** radio button if you want to export the analysis case(s) to a new project archive that will be created during this process.
    - i. Use the **Archive Name** edit box to specify the name of the archive file.
    - ii. Use the **Exclude fluid tables from archive** check box if you do not want to include fluid tables in the archive.

- b. Use the **Export analysis case(s) to a new project** radio button if you want to export the analysis case(s) to a new project that will be created during this process.
    - i. Use the **Project Name** edit box to specify a name for the new project.
  - c. Use the **Export analysis case(s) to an existing project** radio button if you want to export the analysis case(s) to a project that already exists.
    - i. Use the **Select Project** button to select the project to which the analysis case(s) will be exported.
6. Click **Done** to export the analysis case(s).

### Importing an analysis case

Use the **Import** button to import one or more analysis cases of the current **Category** and **Type** from another project into the current project.

**NOTE:** The case(s) you are importing must be of the same **Category** and **Type** you currently have selected.

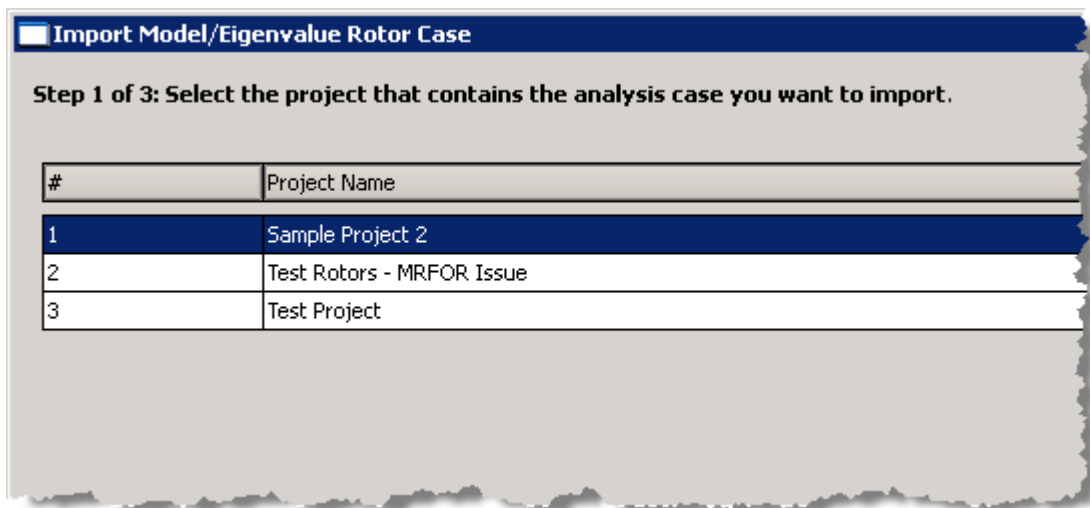
To import an analysis case from another project, follow these steps:

1. Select the **Category** and **Type** corresponding to the category and type of analysis case you want to import.

**NOTE:** It does not matter which analysis case is selected when you import an analysis case.

2. Click the **Import** button.

The **Step 1 of 3** screen on the **Import** dialog box displays. The title of the dialog box corresponds to the type of analysis case you are importing. The following figure displays part of the **Import** dialog box for importing a *Structural Dynamics Rotor* case.



## RAPPID User's Guide

---

3. Select the project that contains the analysis case you want to import from the list.
4. Click the **Next** button.

The **Step 2 of 3** screen displays.

The screenshot shows a software window titled "Import Model/Eigenvalue Rotor Case". Below the title bar, it says "Step 2 of 3: Select the analysis case you want to import." There is a table with two columns: "#" and "Analysis Case Name". The first row has the value "1" in the first column and "Another Sample Advanced Rotor case" in the second column. The row is highlighted with a blue background.

#	Analysis Case Name
1	Another Sample Advanced Rotor case

5. Select the analysis case you want to import from the list.
6. Click the **Next** button.

The **Step 3 of 3** screen displays.

The screenshot shows a software window titled "Import Model/Eigenvalue Rotor Case". Below the title bar, it says "Step 3 of 3: Specify the import destination." There are two main sections: "Import Source" and "Import Destination".

**Import Source**

Project Name:	Sample Project 2
Analysis Case Name:	Another Sample Advanced Rotor case
Date Last Edited:	02/10/2009
Date Created:	02/10/2009

**Import Destination**

Project Name:	Sample Project
Analysis Case Name:	Another Sample Advanced Rotor case

☐ Keep dates from original analysis case.

Information from the source analysis case (the case being imported) displays in the **Import Source** box.



# RAPPID User's Guide

---

The **Import Destination** box displays the current project name. The current name of the analysis case displays in the **Analysis Case Name** edit box.

7. If you want to use a different name for the case once it is imported into the current project, enter a new name in the **Analysis Case Name** edit box.
8. The **Date Last Edited** and **Date Created** are set to the current date. To use the dates from the original analysis case, select the **Keep dates from original analysis case** check box.
9. If you need to change your selections on the previous screens, click the **Back** button. To advance to the next screen again, click the **Next** button.
10. Click the **Finished** button to import the case.

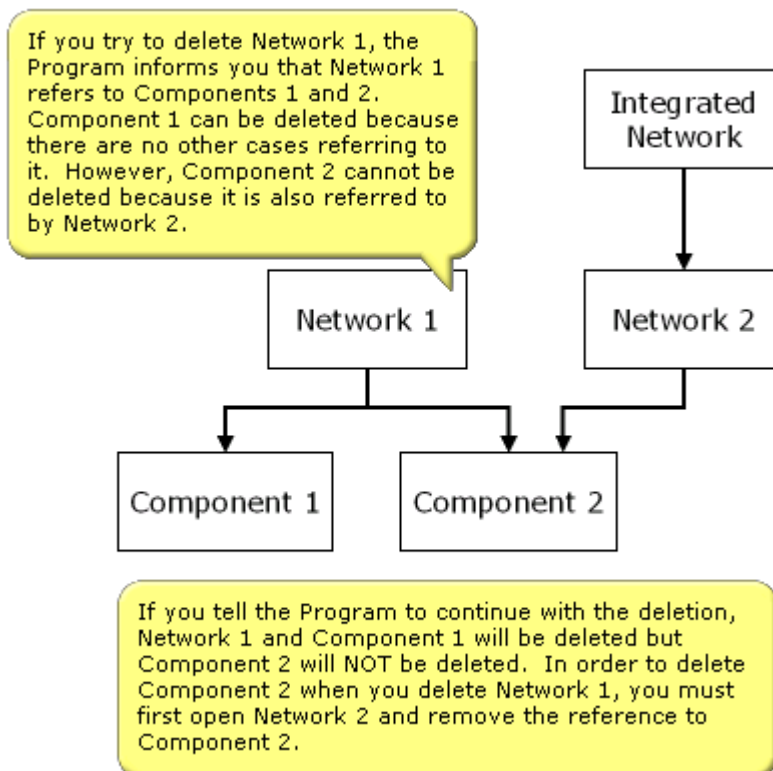
The imported case displays in the list of cases. If there were already existing cases for the current category and type, the newly imported case displays at the end of the list.

## Deleting analysis cases

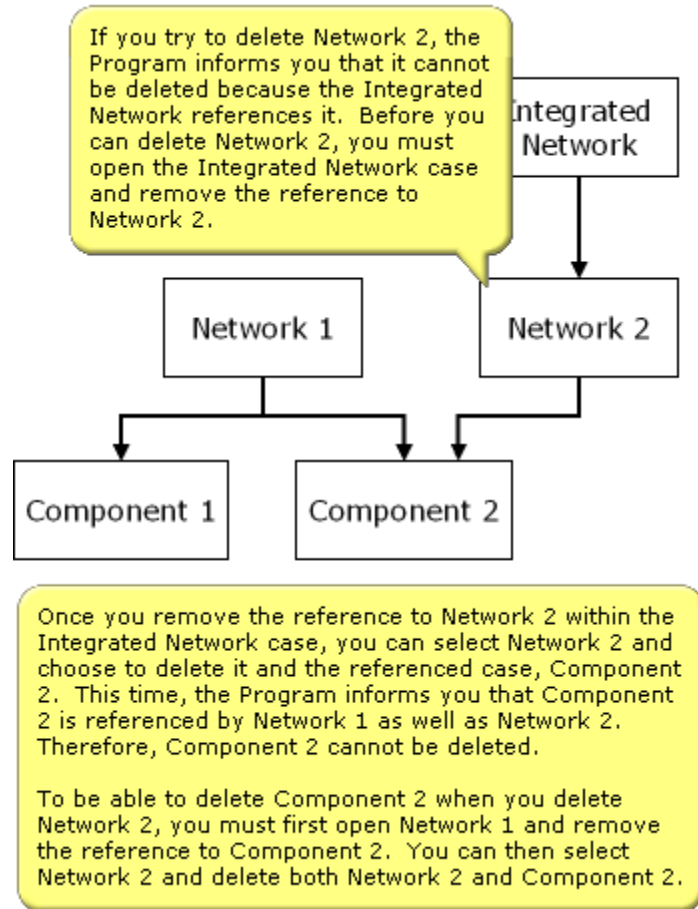
When select one or more analysis cases for deletion, the Program looks for cases that reference any of the selected cases and that are referenced by any of the selected cases. The following figures explain the behavior when you delete the different cases in the sample case structure displayed.

**WARNING:** When you delete one or more analysis cases, be sure that is what you want to do. This action is not reversible.

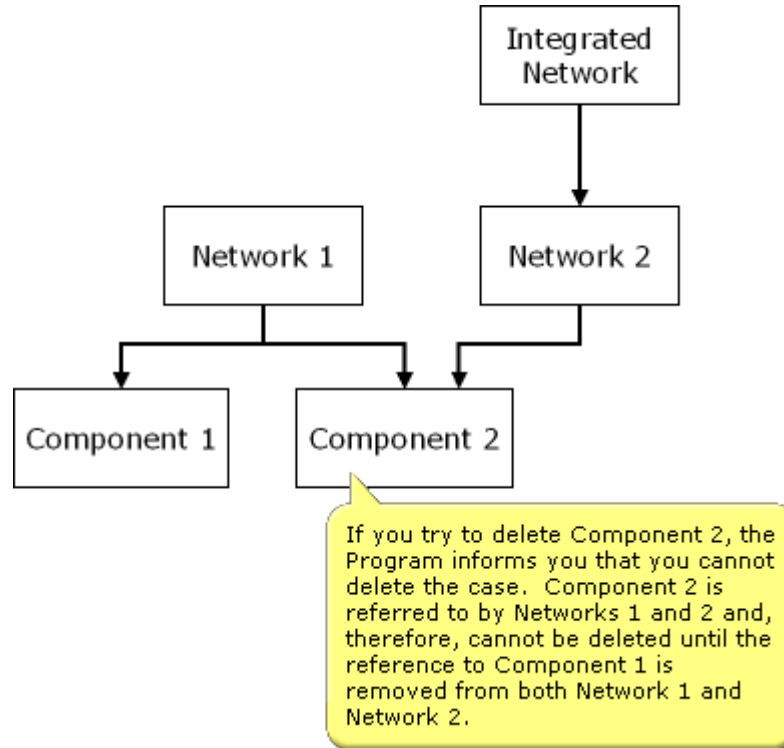
### Deleting Network 1:



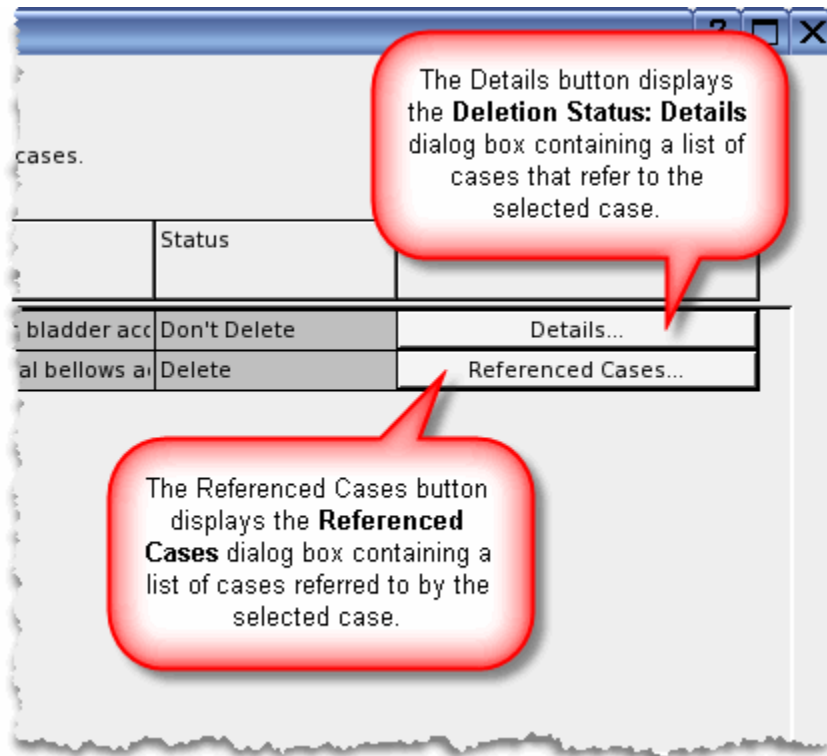
### Deleting Network 2:



## Deleting **Component 2**:



To delete one or more analysis cases, select the analysis case(s) that you want to delete on the **Analysis Manager** window and click the **Delete** button. See the figure below for information about the buttons available on the **Delete Analysis Case(s)** dialog box that displays.



# RAPPID User's Guide

---

**NOTE:** The analysis cases listed on the **Referenced Cases** dialog box might be referred to by other analysis cases or refer to other analysis cases themselves, in which case, **Details** buttons and/or **Referenced Cases** buttons might be available for the referenced cases.

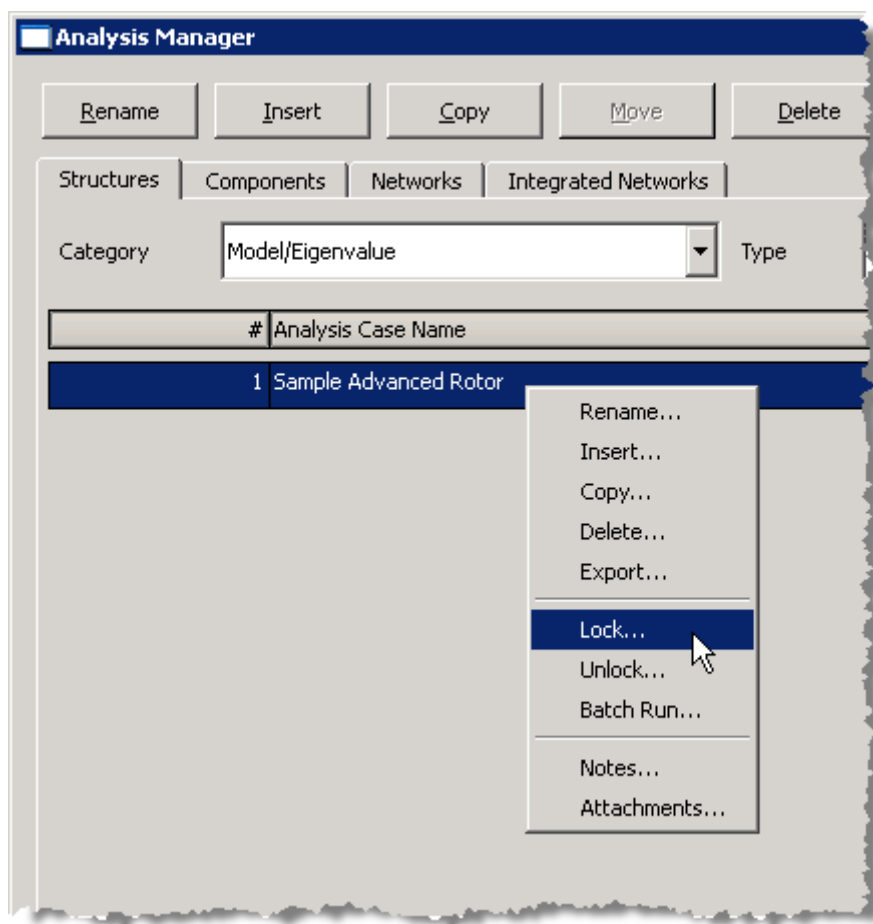
Use the **Deletion Status: Details** and **Referenced Cases** buttons on any applicable dialog boxes to determine which analysis cases you must change to be able to delete the analysis cases you want to delete.

To change analysis cases, you must click **Done** on the **Delete Analysis Case(s)** dialog boxes and return to the **Analysis Manager** to access the analysis cases to change.

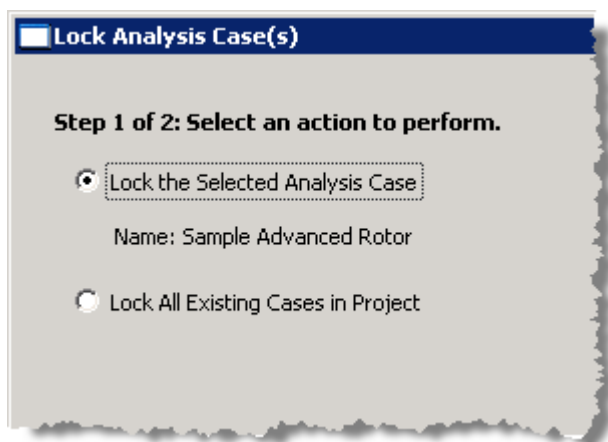
## Locking and unlocking analysis cases

Locking an analysis case prevents inadvertent changes to the analysis case from being saved. Use this feature if you want to preserve the values and settings in your analysis cases.

To lock one or more analysis cases, right-click on the analysis case(s) you want to lock and select **Lock** from the popup menu.



The **Lock Analysis Case(s)** dialog box displays.



### Locking the selected analysis case

Use the **Lock the Selected Analysis Case** option in **Step 1** on the **Lock Analysis Case(s)** dialog box to only lock the currently selected analysis case.

### Locking all existing analysis cases in a project

Use the **Lock All Existing Cases in Project** option in **Step 1** on the **Lock Analysis Case(s)** dialog box to lock all the analysis cases in the project. This can be used to prevent changes to all existing analysis cases in the currently open project, but it will allow you to add new cases to the project.

## Using a password

Use **Step 2** of the **Lock Analysis Case(s)** dialog box to assign a password to the locked case(s).

The screenshot shows a Windows-style dialog box titled "Lock Analysis Case(s)". It has a blue title bar with a question mark icon and a close button. The main area is light gray. At the top, it says "Step 2 of 2: Enter optional password(s)." followed by a note: "NOTE: The passwords you enter below are intended to provide protection against inadvertent changes. Do not expect high security!". Below this, it says "Password to unlock individual cases in project". There are two text input fields. The first is labeled "Enter new password once:" and the second is labeled "Enter new password again:". At the bottom, there are four buttons: "Cancel", "< Back", "Finished" (which is highlighted with a dashed border), and "Help".

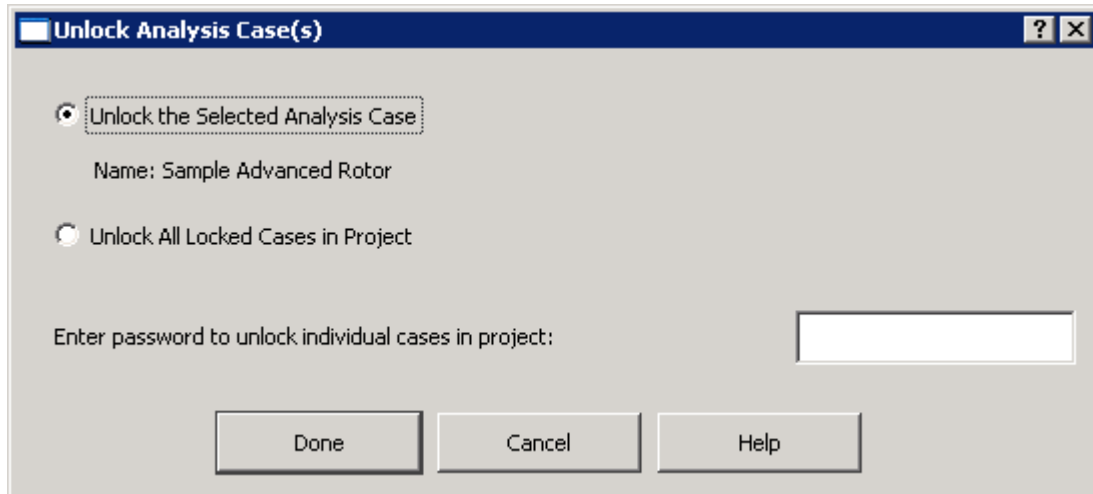
**NOTE:** Once you enter a password to lock an analysis case, this password exists whether or not you have any analysis cases locked. If you have no analysis cases currently locked, and you want to lock an analysis case, RAPPID™ will ask you to enter the old password in addition to entering the new password twice or applying no password (leave the **Enter new password** edit boxes blank). If you apply no password after entering the old password, the analysis case-locking password is cleared.

This password is optional. You can lock one or more analysis cases using no password.

**NOTE:** The password(s) you enter when locking a project and all the cases in a project is intended to provide protection against inadvertent changes. Do not expect high security!

## Unlocking an analysis case

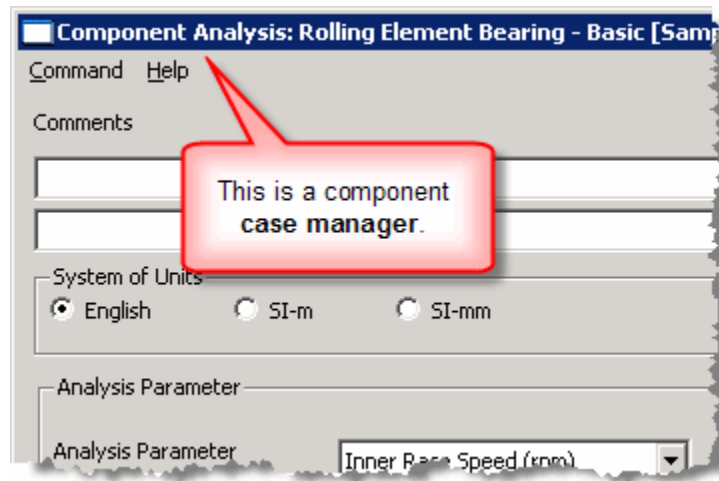
When you unlock an analysis case, similar choices are available as when you locked the analysis case, as shown on the following picture.



If you assigned a password to the locked cases, enter the password in the **Enter password to unlock individual cases in project** edit box to unlock the case(s).

## Introducing a sample case manager

When you open an analysis case from one of the four tabs on the **Analysis Manager**, a case manager displays. The title bar at the top of the screen indicates the type of analysis case currently open.



Use **case managers** to specify input values for an analysis case, run the case to obtain results, and to plot the results.

Although the specific features and functions of the **case manager** vary between the CFD and FEA codes, there are common fields and pages.

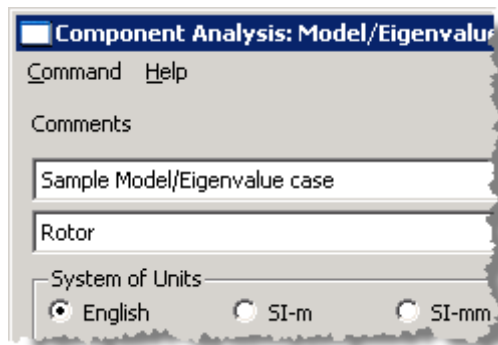
## Specifying common options

The first page of all analysis cases is always the **Options** page. For some cases, such as *0-D Flow Elements* cases, this is the only page available for the case.

All cases have two **Comments** boxes, a **Notes** button, an **Attachments** button, and a **System of Units** box on the **Options** page. The **Dynamic Solution** box discussed in this section is only available for certain cases.

## *Adding comments to analysis cases*

Use the **Comments** edit boxes to enter text to be used as labels for viewed output and displayed plots.



The length of the **Comments** text for each entry is limited to 72 characters. This limit was imposed mainly for the purposes of producing clean plots. For a more extensive text description of the case, you are encouraged to make use of the **Notes** button (see



**Chapter 4: Adding notes and** attachments on page 65). However, the **Notes** are not printed in the viewed output or on the displayed plots.

Use the **Attachments** button to attach external files to the current analysis case.

### *Specifying the system of units for an analysis case*

Use the **System of Units** box to specify the system of units to be used when the results of the case are calculated. The chosen **System of Units** displays in the output file.

Three systems of units are available:

- **English**
- **SI-m**
- **SI-mm**

For every new case, the default system of units is set by the selection entered in the **Preferences** dialog box (accessed from the **Project** menu). To change the system of units for the selected case, click one of the radio buttons in the **System of Units** box.



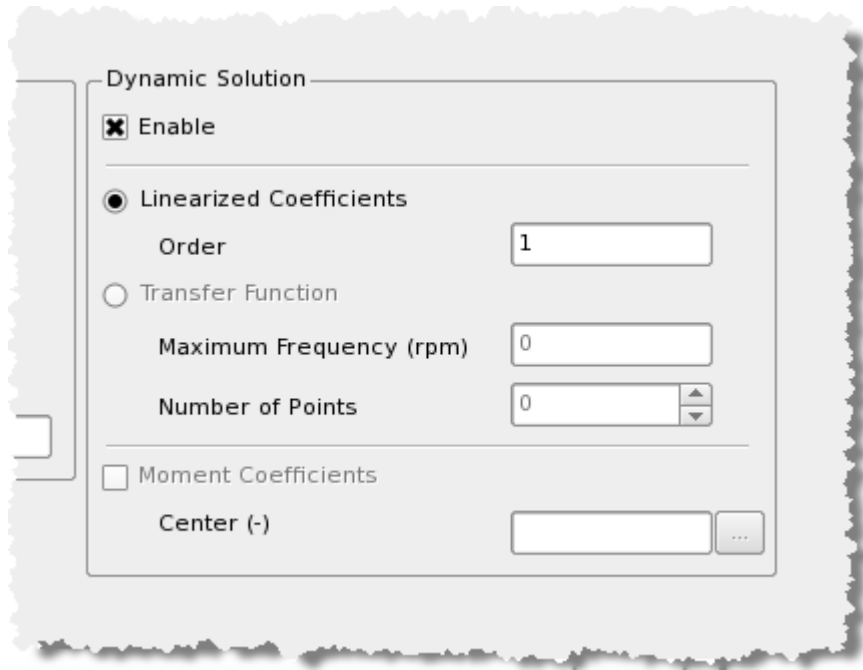
The radio buttons in the **System of Units** box are used for selecting the system of units to be used when the results of the case are calculated. The chosen **System of Units** displays in the output file.

See **Working with unit systems** in RAPPID™ on page 9 for more information about units.

## *Defining the Dynamic Solution*

Use the input fields in the **Dynamic Solution** box to obtain rotordynamic coefficients at different frequency points and for translation.

Calculating the **Dynamic Solution** in analysis cases, causes longer running times. Use the **Enable** check box to specify whether or not to include the rotordynamic coefficients in the results of your analysis case. If you want to shorten the running time and don't need the rotordynamic coefficients in the results, click the **Enable** check box such that there is no **X** in the box.



The screenshot shows a dialog box titled "Dynamic Solution". It contains the following controls:

- An "Enable" checkbox with an "X" inside it, indicating it is checked.
- Two radio buttons: "Linearized Coefficients" (selected) and "Transfer Function".
- Under "Linearized Coefficients":
  - An "Order" text box with the value "1".
  - A "Maximum Frequency (rpm)" text box with the value "0".
  - A "Number of Points" spinner box with the value "0".
- A "Moment Coefficients" checkbox, which is currently unchecked.
- Below "Moment Coefficients":
  - A "Center (-)" text box.
  - A small button with three dots "..." to the right of the text box.

Currently, only the **Linearized Coefficient** radio button is active. The **Transfer Function** radio button is not available. Use the **Order** edit box to obtain linearized coefficients at frequencies other than rotor speed. The value entered can be any positive floating point value and is multiplied by the rotor speed to obtain the frequency at which the rotordynamic coefficients are calculated.

The **Moment Coefficients** check box and the **Center** edit box are only available for certain cases. **Moment Coefficients** are the rotordynamic characteristics that relate forces to angular displacements. The **Center** is the center point of the perturbation for the moment coefficients.

## **Adding materials and fluids**

**Materials** and **Fluids** pages are common among some analysis case types that use materials and fluids. See

**Chapter 5: Working with materials** on page 79 and **Chapter 6: Working with fluids** on page 89.

## Entering tabular input

**Tabular Input** is available for several different analysis case types to allow you to enter coefficients manually. See **Chapter 20: Miscellaneous – Tabular Input** on page 264 for more information.

## Accessing pages in an analysis case

Each **analysis case manager** offers two methods for user-controlled paging, as shown in the following figure.



- Use the **Previous** and **Next** buttons to “turn” the pages of the case.

This accesses the pages of the case like you would turn the pages of a book.

- Use the **Page** drop-down menu to jump to a specific page in the case.

This is useful if you want to go from the **Options** page directly to the last page of the case.

## Using a grid to enter data

Many cases in the Program contain data grids for entering rows of data. These are tables containing cells in rows and columns. All grids generally contain the same capabilities. You can:

- **Insert, Copy, Move, and Delete** rows.
- **Import** data into grids.

See **Importing data into** on page 44 for more information.

# RAPPID User's Guide

The following figure displays an example of a data grid on the **Nodes** page of a *Model/Eigenvalue – Rotor* case.

Rotor

Insert Copy Move Delete Import

Node #	ID Text	Interconnection	Connectivity	X (in)	Y (in)
1		<input type="checkbox"/>	View...	0	0
2		<input type="checkbox"/>	View...	0	0
3	Bearing 1	<input checked="" type="checkbox"/>	View...	0	0
4		<input type="checkbox"/>	View...	0	0
5		<input type="checkbox"/>	View...	0	0
6		<input type="checkbox"/>	View...	0	0
7		<input type="checkbox"/>	View...	0	0
8		<input type="checkbox"/>	View...	0	0
9	Bearing 2	<input checked="" type="checkbox"/>	View...	0	0
10		<input type="checkbox"/>	View...	0	0
11		<input type="checkbox"/>	View...	0	0

## Inserting rows into a grid

Use the **Insert** button to add rows to the grid. If the data grid already contains a row or rows, select the row in the data grid before or after which you want to insert the row(s).

**NOTE:** If the data grid already contains a row or rows, you must select a row before you insert more rows. The **Insert** button will not work if you don't select a row. Selecting a row is how you tell the Program where to insert the new row(s).

**Insert** ? X

Number of rows

☐ Insert Before Selection

☒ Insert After Selection

Node ID Text

Done Cancel Help

Use the **Number of rows** spin edit box to specify the number of rows you want to insert into the grid.

---

## RAPPID User's Guide

---

Use the **Insert Before Selection** and **Insert after Selection** radio buttons to specify whether to insert the new row(s) before or after the currently selected row.

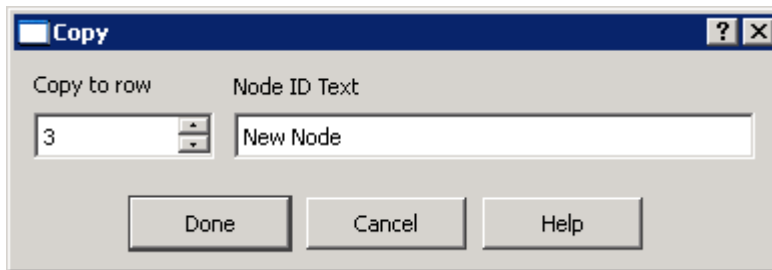
**NOTE:** When inserting a row or set of rows into a data grid that does not contain any rows, the **Insert Before Selection** and **Insert After Selection** radio buttons are not available.

If there is a column in the grid in which you can enter identifying text, enter it in the edit box right above the buttons at the bottom of the dialog box. The title above this edit box changes depending on the data grid into which you are inserting rows.

**NOTE:** The edit box at the bottom of the Insert dialog box is not available if you are entering more than one row.

### Copying a row in a grid

Select the row you want to copy and use the **Copy** button above a grid to duplicate the row of data into another row in the same grid.



**WARNING:** When copying a row, you are copying the data in the row to another *existing* row. Therefore, if you don't want to replace an existing row that currently has data, insert a new row to contain the copied row of data.

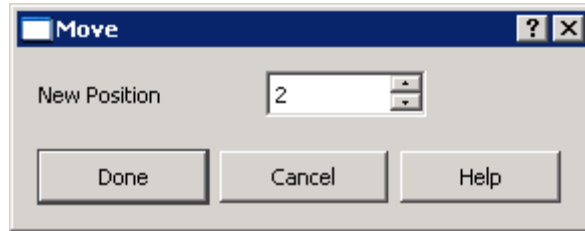
Use the **Copy to row** spin edit box to specify the row number to which you want to copy the selected row.

If there is a column in the grid in which you can enter identifying text, enter it in the edit box next to the **Copy to row** edit box. The title above this edit box changes depending on the data grid in which you are copying rows.

**NOTE:** Using the edit box, you can change the identifying text in the row into which you are copying the selected row.

## Moving a row in a grid

Select the row you want to move and use the **Move** button to move the row to a different location in the grid.

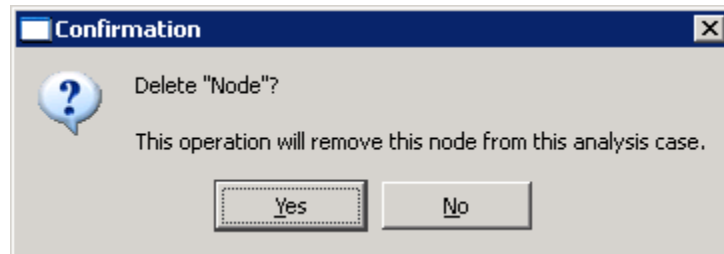


Enter a number in the **New Position** edit box for the row number to which you want to move the selected row.

**NOTE:** The row being moved does NOT replace the row specified in the **New Position** edit box. The rows in the data grid are shifted as necessary to accommodate the move.

## Deleting a row in a grid

Select the row you want to delete and use the **Delete** button to remove it from the grid.



Clicking the Delete button displays a **Confirmation** dialog box, asking if you want to delete the row and warning you that the row will be deleted from the case.

**WARNING:** Once a row is deleted, the action cannot be reversed. If you need to get the data back, you can close the analysis case without saving it by clicking the **Cancel** button at the bottom of the **case manager**. However, if you made any other changes when you click **Cancel** to close the case, these changes will be lost.

Click **Yes** to delete the selected row. The remaining rows shift to fill the gap where needed.

### Importing data into an analysis case

Use the **Import** button above a data grid to import data contained in an ASCII-formatted file into a grid in an analysis case. This can save you time manually entering values such as coefficients in a **Tabular Input** module.

The screenshot shows a Windows-style dialog box titled "Import ASCII File". It contains three main sections. The first section, "What kind of file do you want to import?", has three radio buttons: "Space-delimited" (selected), "Comma-delimited", and "Tab-delimited". Below these is a note: "NOTE: Individual data elements within your file cannot contain the delimiter character you specified above." The second section, "What is the import filename?", features a text field labeled "Filename" with a value of "1" and a browse button "...". The third section, "Which lines from your file do you want to import?", has two radio buttons: "All lines" (selected) and "Just these lines (inclusive)". Below the second radio button are "From" and "To" spinners, both set to "1". At the bottom of this section is a label "Skip lines starting with this character" followed by an empty text field. At the very bottom of the dialog are three buttons: "Import", "Cancel", and "Help".

**Import ASCII File**

What kind of file do you want to import?

☒ Space-delimited

☐ Comma-delimited

☐ Tab-delimited

NOTE: Individual data elements within your file cannot contain the delimiter character you specified above.

What is the import filename?

Filename: 1 ...

Which lines from your file do you want to import?

☒ All lines

☐ Just these lines (inclusive)

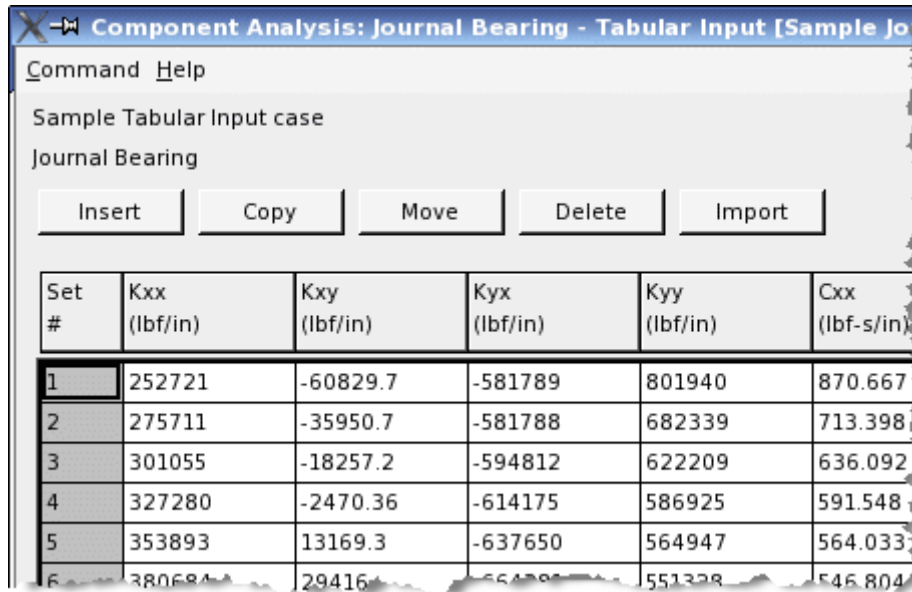
From: 1 To: 1

Skip lines starting with this character:

Import Cancel Help

# RAPPID User's Guide

The following picture is an example of a **Tabular Input** analysis case into which values were imported from a text file.



Set #	Kxx (lbf/in)	Kxy (lbf/in)	Kyx (lbf/in)	Kyy (lbf/in)	Cxx (lbf-s/in)
1	252721	-60829.7	-581789	801940	870.667
2	275711	-35950.7	-581788	682339	713.398
3	301055	-18257.2	-594812	622209	636.092
4	327280	-2470.36	-614175	586925	591.548
5	353893	13169.3	-637650	564947	564.033
6	380684	29416	-654288	551338	546.804

The ASCII-formatted file must be **Space-delimited**, **Comma-delimited**, or **Tab-delimited**. These choices indicate what character is used between the values to be imported (a space, a comma, or a tab).

**NOTE:** As mentioned on the **Import ASCII file** dialog box, make sure the values in your file do not contain the delimiter character you select.

To import data from an ASCII-formatted file into a data grid in on a page in an analysis case or on the **Variable Analysis Input** dialog box, follow these steps:

**NOTES:** Make sure your file is formatted correctly, using a space, a comma, or a tab between the values on each line in the file.

1. Insert the number of rows in the data grid corresponding to the number of rows of data you are importing from the text file.

**NOTE:** If you do not insert enough rows in the data grid, not all your data will be imported.

2. Select the rows in the grid into which the data will be imported in one of two ways:
  - a. Highlight all the rows into which the data will be imported.
  - b. Select the first row in the group of rows into which the data will be imported.

**NOTE:** If you are selecting only the first row, make sure there are enough rows after the selected row to contain the data when it is imported. If there are not enough rows, not all your data will be imported.

3. Click the **Import** button.



---

## RAPPID User's Guide

---

4. Choose the type of delimiter you used between the data values in the text file you are importing from the **What kind of file do you want to import?** box.
5. Specify the name of the text file to import in the **Filename** edit box in one of two ways:
  - a. Manually enter the full path of to the file in the edit box.
  - b. Browse your hard drive or network for the file. Follow these steps:
    - i. Click the ... button to the right of the **Filename** edit box.  
  
The **Locate File to Import** dialog box displays.
    - ii. Navigate to the directory in which your data file is located.
    - iii. Select the file in the list.
    - iv. Click the **Open** button.

The full path of the file is entered in the **Filename** edit box for you.

6. Select from the following two options in the **Which lines from your file do you want to import?** box:
  - a. All lines
  - b. Just these lines (inclusive)
    - i. Enter the number of the first line from the group of lines you want to import in the **From** edit box. You can also use the up and down arrow buttons to select a number.
    - ii. Enter the number of the last line from the group of lines you want to import in the **To** edit box. You can also use the up and down arrow buttons to select a number.

**NOTE:** Remember that the line numbers you enter are inclusive. If you enter “5” in the **From** edit box and “10” in the **To** edit box, lines 5 and 10 will be imported along with the lines in between. Line numbers in your file include lines that you skip when importing. For example, a file containing the following lines would have four lines. If you only want to import the first line of data (9000 and 4), enter “2” in both the **From** and **To** edit boxes.

\* Variable input for Left Diameter

9000 4  
13000 5  
20000 6

7. In addition to specifying lines to import, a simple filter can be used to skip lines in your data file. This can be useful if you want to put comments in your data file to describe the data.

# RAPPID User's Guide

---

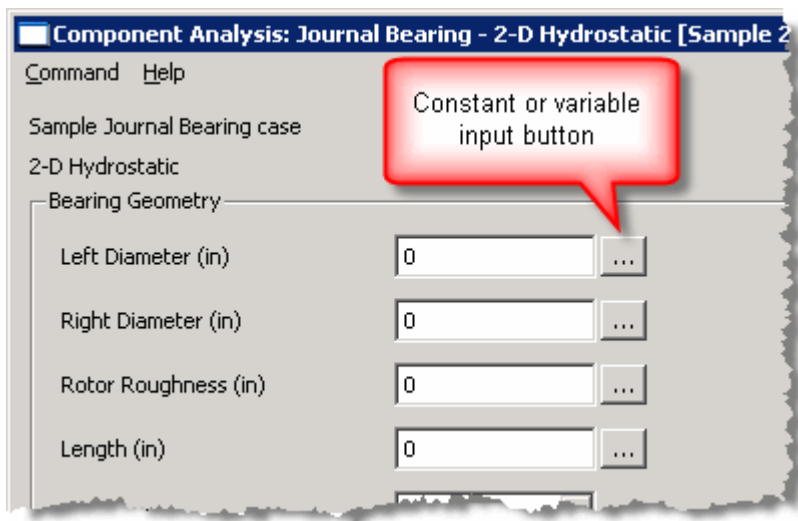
For example, consider the lines listed in step 4 as being the contents of your input file. The first line is meant to be a comment reminding you for which field this data is used. Therefore, you do not want to import this line. To avoid importing the first line in the example in the note under step 6, enter an asterisk (“\*”) in the **Skip lines starting with this character** edit box.

8. Click the **Import** button.

The data from the text file is inserted into the selected rows in the data grid.

## Entering variable data

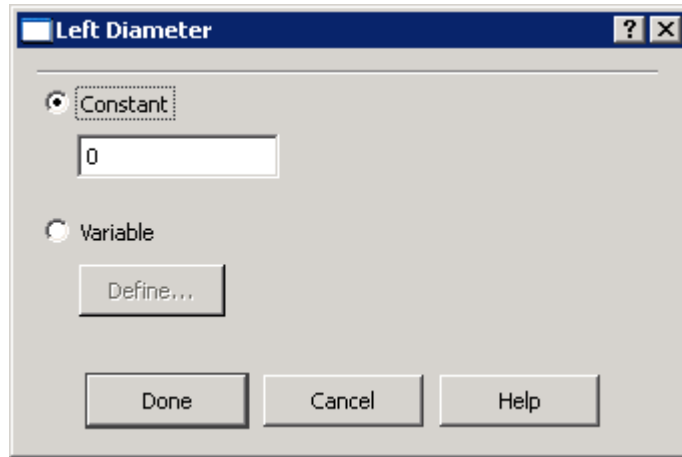
In some analysis cases, such as *Bearings*, some fields have a ... button next to the edit box for an input field, as shown in the following figure:



**NOTE:** In *Flow Elements – Circular Branch* analysis cases, there may be multiple constant values or multiple sets of variable values corresponding to multiple radii entered. See

**Chapter 19: Flow Elements** components on page 258 for more information.

Use the ... buttons to enter either a constant value or values that vary as a function of the analysis parameter. A dialog box displays titled with the name of the field for which you are entering data.



## Entering a constant value

You have two choices for entering a constant value for a field with a ... button available.

- Enter the constant value directly into the field page in the case manager, or
- Click the ... button, select the **Constant** radio button on the dialog box that displays, and enter the constant value in the edit box below the **Constant** radio button.

## Entering variable values

If you want to enter values that vary as a function of the analysis parameter, e.g., rotor speed, you must use the ... button.

Click the ... button and select the **Variable** radio button on the resulting dialog box. Click the **Define** button that becomes available.

On the **Variable Analysis Input** dialog box, select the method of interpolation from the **Relationship Definition** drop-down menu. Enter the values for the analysis parameter in using one of the following two ways:

- Enter the values manually in the data grid.  
See **Using a grid to enter data** on page 40 for more information.

- Import the variable values from a text file into the data grid.

See **Importing data into an analysis case** on page 44 for more information.

## Running analysis cases

Use the **Run** button to calculate the results of the data your entered into the analysis case.

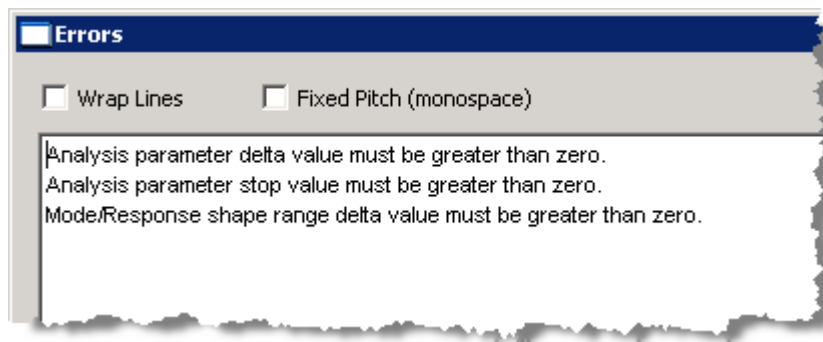
**NOTES:** You can also select **Run** from the **Command** menu.

While the case is running, the **Run**, **View**, and **Plot** buttons (and the **Run**, **View**, and **Plot** options on the **Command** menu) are not available.

Clicking the **Run** button saves the analysis case, just as the **Done** button or the **Save and Exit** option on the **Command** menu does. Therefore, if you do not want to save your analysis case, do not run the case.

## Validating analysis case input

When you run an analysis case, the **Analysis Case Errors** dialog box may display informing you that errors were found in your analysis case. This occurs when you have entered data that does not comply with the limits of the various input fields. The dialog box informs you of the limits so you can correct your errors.



Use the **Print** button on the **Errors** dialog box to print the list of errors.

Click the **Done** button to close the window and return to the **analysis case manager** to correct your errors.

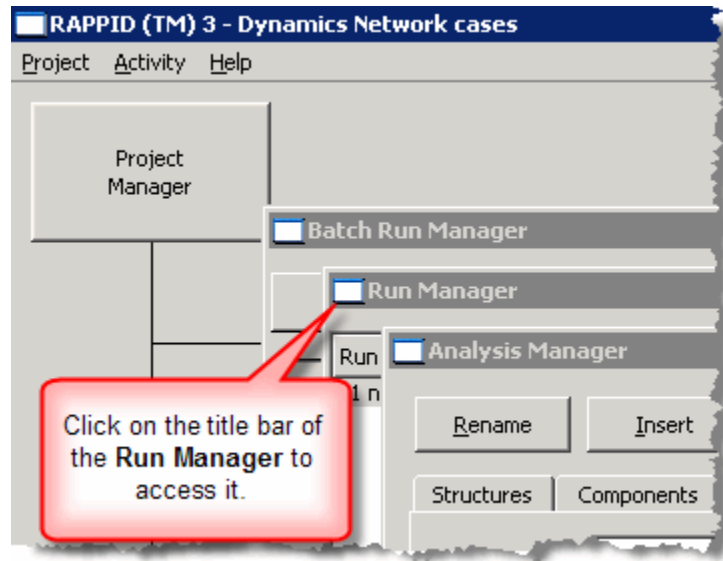
## Monitoring analysis case runs

Since RAPPID™ has the capability to execute multiple cases simultaneously, a **Run Manager** has been implemented to view the status of cases being run. The **Run Manager** also displays the status of any fluids being generated.

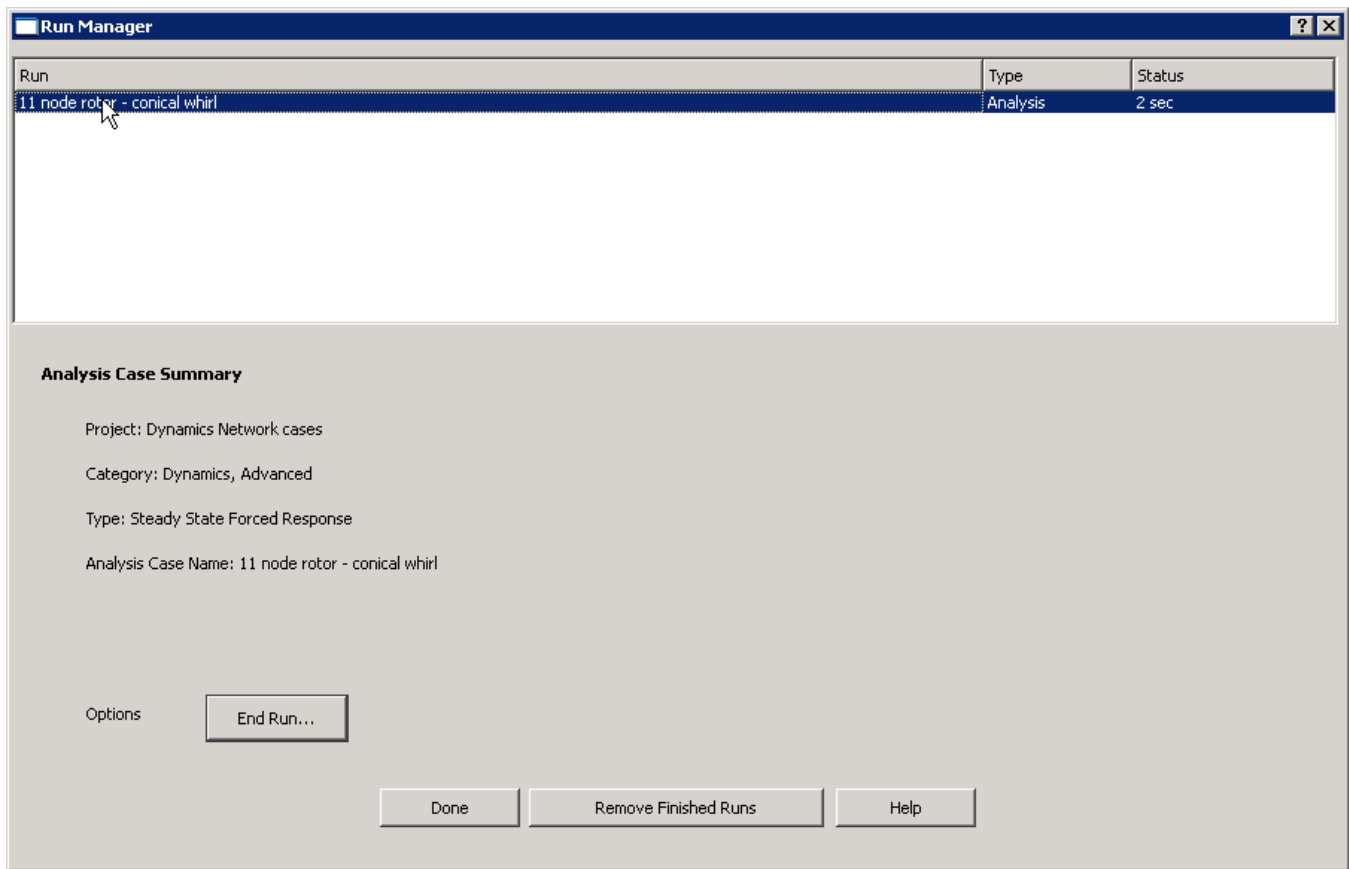
The **Run Manager** is one of three managers that are automatically opened when a project is opened. Clicking on any visible portion of a manager window activates that manager. If you close the **Run Manager** by clicking **Done**, you may re-open the **Run Manager** by selecting **Open Run Manager** from the **Activity** menu.

# RAPPID User's Guide

**NOTE:** Sometimes you may have to move other managers out of the way to access the desired manager. Use the title bar of a window to move it.



**NOTE:** Once you exit RAPPID™, the cases listed in the **Run Manager** are automatically removed from the list.

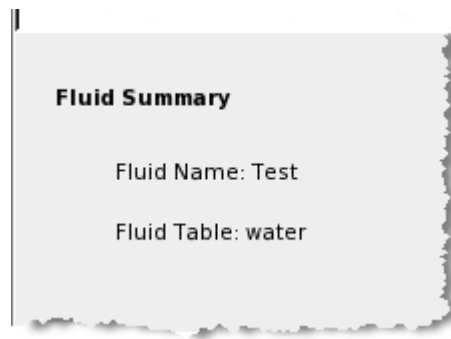


# RAPPID User's Guide

The **Run Manager** indicates each run that has been attempted during the current session of RAPPID™. For each run, the **Run**, **Type**, and **Status** are listed.

- The **Run** column displays the name of the analysis case or fluid.
- The **Type** column displays **Analysis** for analysis cases being run or **Fluid** for fluids being generated.
- The **Status** lists either the elapsed clock time for the run or **Finished**.

Select a run in the list box to view an **Analysis Case Summary** below the list box of that analysis case. If you select a fluid in the list box, a **Fluid Summary** displays.



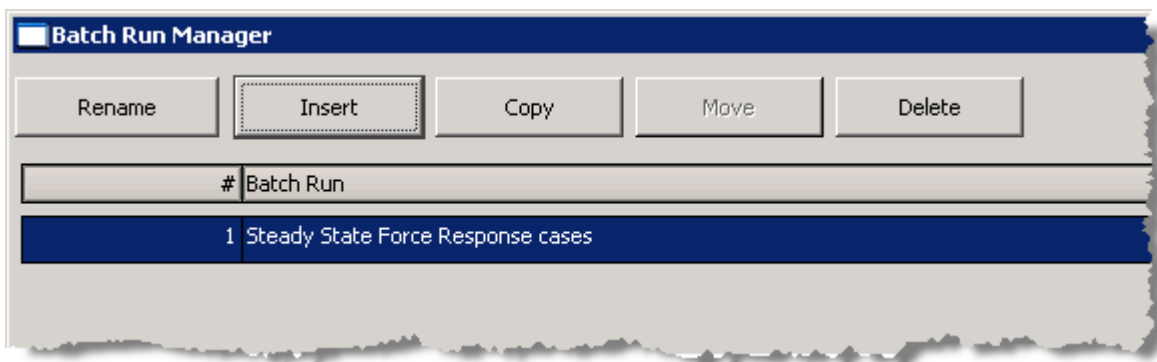
Use the **End Run** button to abort the run.

**NOTE:** The **End Run** button is only available for analysis cases being run. You cannot stop a fluid during generation.

Use the **Remove Finished Runs** to remove any analysis cases that have finished running from the list box. This is useful if you have many analysis cases running and you want to only see the ones that are still running.

## Starting runs for more than one analysis case at a time

Use the **Batch Run Manager** to run multiple analysis cases within one project without having to open each analysis case separately. You can have multiple batch runs that are set up separately.

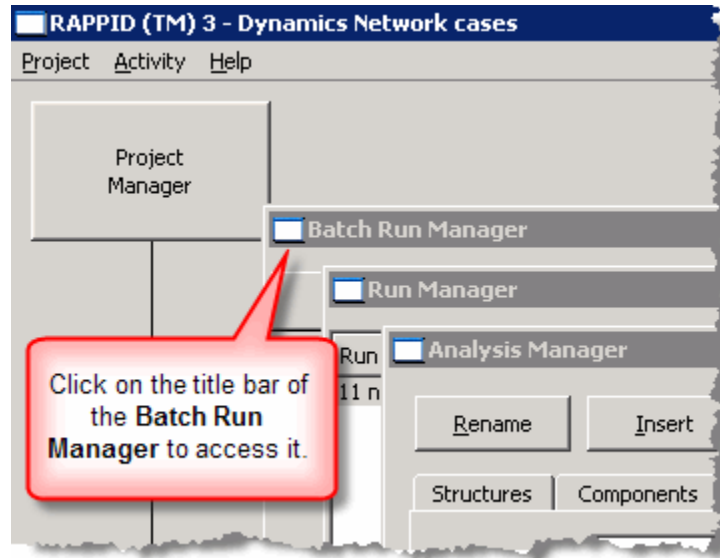


## RAPPID User's Guide

---

The **Batch Run Manager** is one of three managers that are automatically opened when a project is opened. Clicking on any visible portion of a manager window activates that manager. If you close the **Batch Manager** by clicking **Done**, you may re-open the **Batch Run Manager** by selecting **Open Batch Run Manager** from the **Activity** menu.

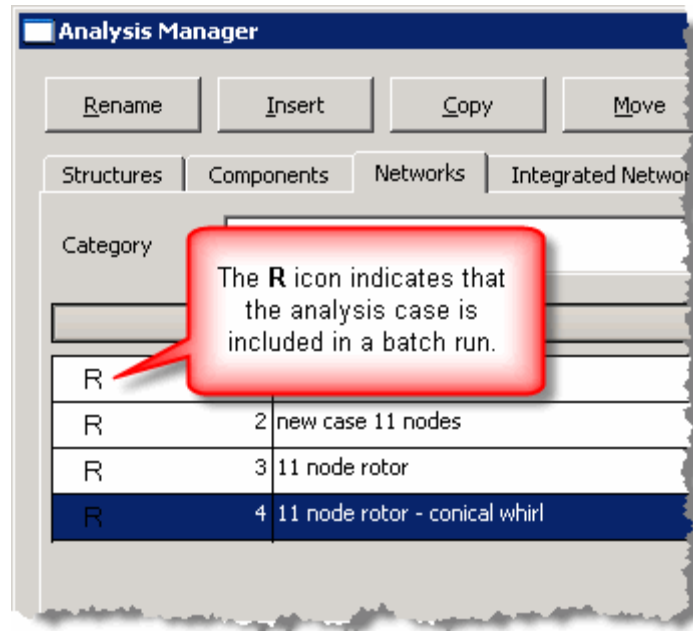
**NOTE:** Sometimes you may have to move other managers out of the way to access the desired manager. Use the title bar of a window to move it.



Once you have created, added cases to, and launched a batch run, the cases in the batch that are currently running are listed in the **Run Manager** (see **Monitoring analysis case runs** on page 49). If you have chosen to run the cases in parallel, one group of cases is added to the **Run Manager** and when these cases are finished running, the next group of cases is added.

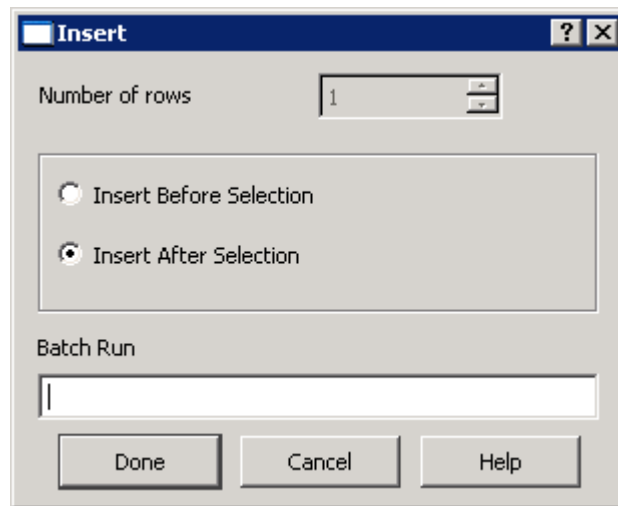
# RAPPID User's Guide

If an analysis case is part of a batch run, that fact is indicated with an 'R' icon next to the case name in the **Analysis Manager**.



## Creating a batch run

Use the **Insert** button on the **Batch Run Manager** to add a new batch run.



The **Number of rows** spin edit box is not available because you can only add one batch run at a time.

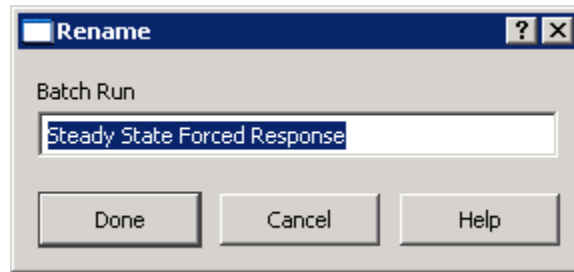
If you have any existing batch runs when you open the **Batch Run Manager**, the first case is automatically selected. Select the batch run before or after which you want to add the new batch run.

Use the **Insert Before Selection** and **Insert After Selection** radio buttons to specify where to add the new batch run. These radio buttons are not available if you have no existing batch runs before adding the new one.



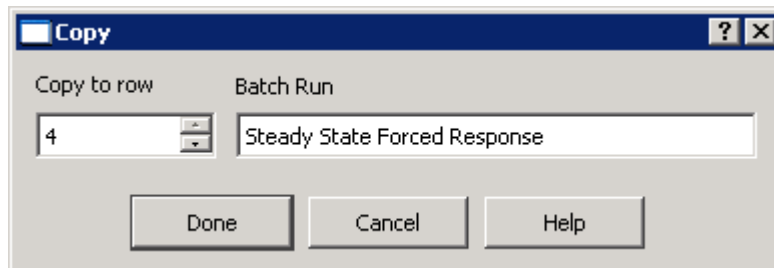
## ***Renaming a batch run***

Use the **Rename** button to rename the currently selected batch run.



## ***Copying a batch run***

Use the **Copy** button to copy a batch run as a new batch run.



Enter or select the row number to which you want to copy the new batch run in the **Copy to row** spin edit box and specify a new name for the new batch run.

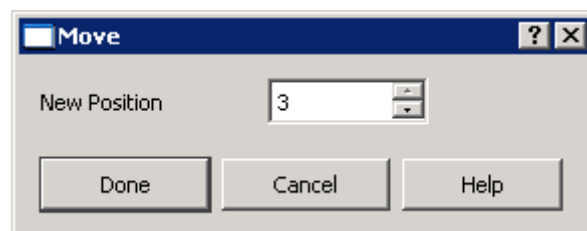
**NOTE:** You cannot copy a batch run to a new batch run using the same name as the original batch run.

In the **Copy to row** spin edit box, you can specify any row number from 1 through one past the current last row number. As an example, there are three batch runs currently defined. You want to select the second batch run to copy and put it at the end of the list of batch runs. Enter **4** in the **Copy to row** spin edit box and enter a new name for the new batch run.

**NOTE:** If you specify a row number in the **Copy to row** spin edit box that corresponds with a batch run that already exists, the existing batch run is NOT replaced. The new batch run is placed in that position and the existing batch runs are shifted to accommodate the new batch run.

## ***Moving a batch run***

Use the **Move** button to move the currently selected batch run to a different location in the list.

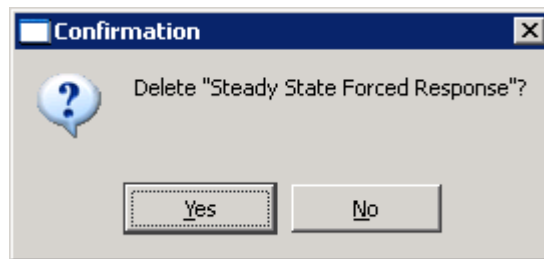


# RAPPID User's Guide

**NOTE:** The **Move** button is only available when there are multiple batch runs in the **Batch Run Manager**.

## Deleting a batch run

Use the **Delete** button to delete the currently selected batch run. A **Confirmation** dialog box displays asking you if you want to delete the batch run named on the dialog box. Click **Yes** if you are sure you want to delete that batch run.



**NOTE:** Be sure you want to delete the batch run specified on the **Confirmation** dialog box. This action is irreversible.

## Defining and launching a batch run

Once you have created a batch run (see **Creating a batch run** on page 53), you need to define it by adding analysis cases to it.

**Define Batch Run**

Options

Batch Run Name:

Comments

Steady State Forced Response cases **1**

From the Dynamics Network cases project

Notes... **2**

Batch Run Mode: Run Cases in Parallel **3**

Maximum number of cases to run in parallel (Enter zero to allow unlimited parallelism) **5** 0

Run Locked Analysis Cases: Yes **5**

**6** Insert **8** Move **9** Delete

Case #	Locked?	Component Category	Component Type	Name
1	No	Dynamics, Advanced	Steady State Forced Response	new case 11 nodes
2	No	Dynamics, Advanced	Steady State Forced Response	11 node rotor - conical whirl
3	No	Dynamics, Advanced	Steady State Forced Response	11 node rotor

Follow these steps:

1. Enter comments and notes about the batch run in the two **Comments** edit boxes, if desired.

## RAPPID User's Guide

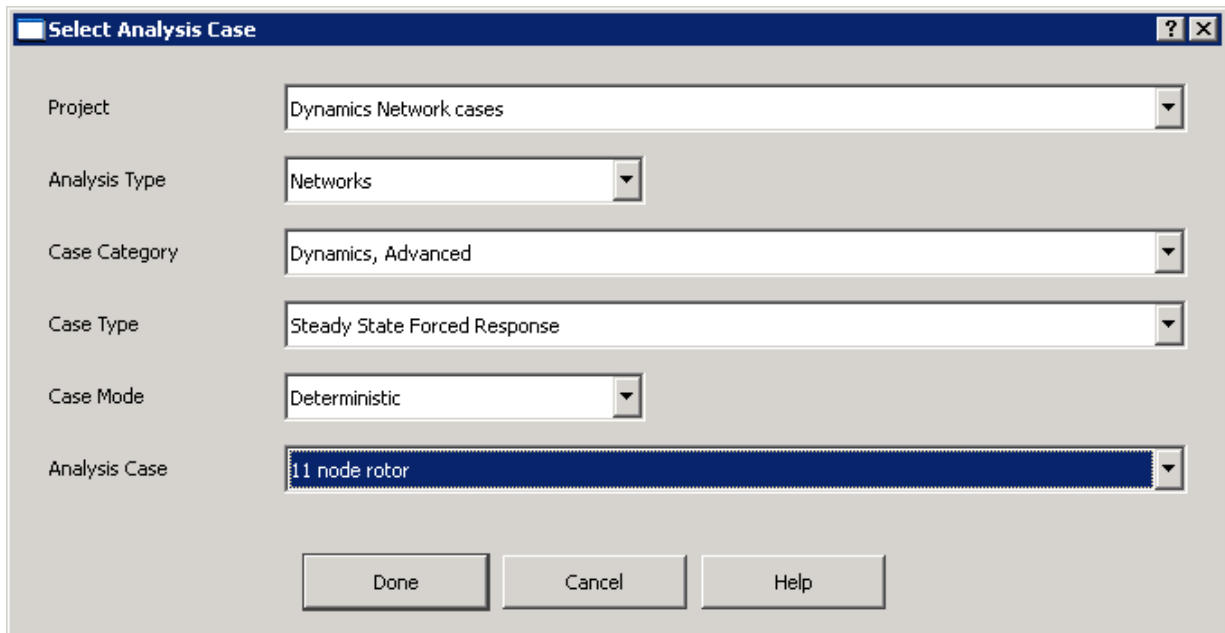
2. Use the **Notes** button to add notes about the currently open batch run, if desired.
3. Use the **Batch Run Mode** drop-down menu to specify whether to run the analysis cases in the currently open batch run one after the other (**Run Cases Sequentially**) or multiple cases at the same time (**Run Cases in Parallel**).
4. If you choose **Run Cases in Parallel**, use the **Maximum number of cases to run in parallel** spin edit box to specify how many cases to run at a time. To let RAPPID™ decide for you, enter or select zero.

The Program runs the cases in the batch in the order listed in the grid. When one group runs, the next group of cases from the list start running.

**NOTE:** For more efficient use of run time, when adding your cases to the list, order them in the list in order of expected run time. For example, if you add 10 cases to the list in the batch, and 5 cases run very quickly and 5 cases run very slowly, put the 5 faster cases first before the 5 slower cases.

5. Use the **Run Locked Analysis Cases** drop-down menu to specify whether to run locked analysis cases within the project. Normally, locked analysis cases cannot be run, however, they can be run when part of a batch run. See **Locking and unlocking analysis cases** on page 33 for more information.
6. Click the **Insert** button to add an analysis case to the batch run.

The **Select Analysis Case** screen displays.



The screenshot shows a dialog box titled "Select Analysis Case". It contains several dropdown menus for selecting analysis parameters. The "Project" dropdown is set to "Dynamics Network cases". The "Analysis Type" dropdown is set to "Networks". The "Case Category" dropdown is set to "Dynamics, Advanced". The "Case Type" dropdown is set to "Steady State Forced Response". The "Case Mode" dropdown is set to "Deterministic". The "Analysis Case" dropdown is set to "11 node rotor". At the bottom of the dialog box, there are three buttons: "Done", "Cancel", and "Help".

7. Use the drop-down menus on the **Select Analysis Case** screen to select the analysis case you want to add from the currently open project:

**NOTE:** Select these options in order because the values in each drop-down menu change depending on the option selected in the drop-down menus above.

8. Use the **Move** button to rearrange the order of the analysis cases in the list.

See **Specifying the order of analysis cases in a batch run** on page 57 for more information.

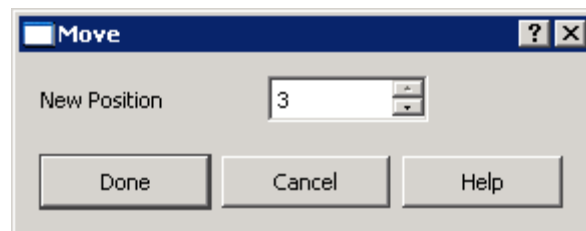
9. Click **Done** to add the selected analysis case to the batch run.

10. Click the **Launch** button on the **Batch Run Manager** to start the batch run.

### *Specifying the order of analysis cases in a batch run*

Analysis cases in a batch run are run in the order they are listed on the **Define Batch Run** dialog box when you define the batch run. If you select to run your analysis cases in parallel, the order of the cases still determines the order in which they are run. For example, you have 10 analysis cases in your batch run, you select **Run Cases in Parallel** from the **Batch Run Mode** drop-down menu, and specify **2** in the **Maximum number of cases to run in parallel** spin edit box, the first two cases in the list will be run at the same time. When both of those are finished, the next two cases in the list are run, etc.

To rearrange the order of analysis cases in a batch run, access the **Batch Run Manager**, select the batch run you want to change, and click **Open**. Select an analysis case on the **Define Batch Run** dialog box and click the **Move** button.



Enter or select the new position for the selected analysis case in the **New Position** spin edit box. The other analysis cases in the list are shifted to accommodate the move.

Select and move analysis cases as often as it takes to get the order in which you want to run your analysis cases.

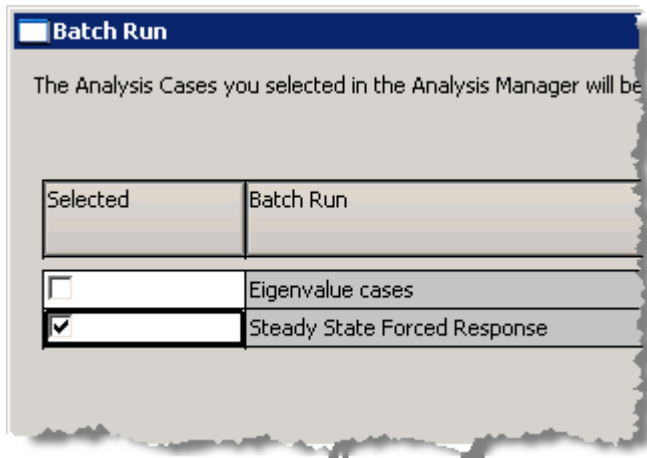
### *Adding multiple analysis cases to a batch run*

You can add one or more analysis cases to an existing batch run directly from the **Analysis Manager**.

On the **Analysis Manager**, follow these steps:

1. Select the cases you want to add.
2. Right click on the selected cases and select **Batch Run** from the popup menu that displays.

3. On the **Batch Run** dialog box, use the check boxes in the **Selected** column to select any batch runs to which you want to add the selected analysis case(s).



4. Click the **Done** button to add the selected cases to the batch run and return to the **Analysis Manager**.

**NOTE:** Analysis cases added to a batch run using this method are appended to the end of the list of cases in the batch run. To rearrange the order of analysis cases

5. To launch the batch run, access the **Batch Run Manager** and click the **Launch** button.

See **Defining and launching a batch run** on page 55 for more information.

### ***Locking and unlocking batch runs***

Locking a batch run prevents cases from inadvertently being added or removed.

You have two choices when you lock a batch run. You may lock

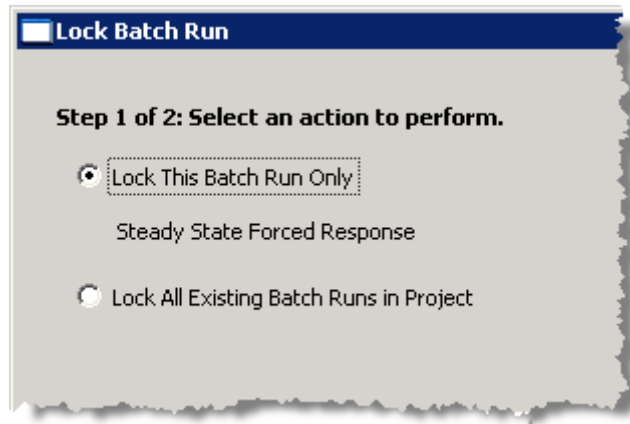
- the selected batch run only, or
- all batch runs in the current project.

To lock a batch, access the **Batch Run Manager**, right-click on the batch run you want to lock and select **Lock** from the popup menu.

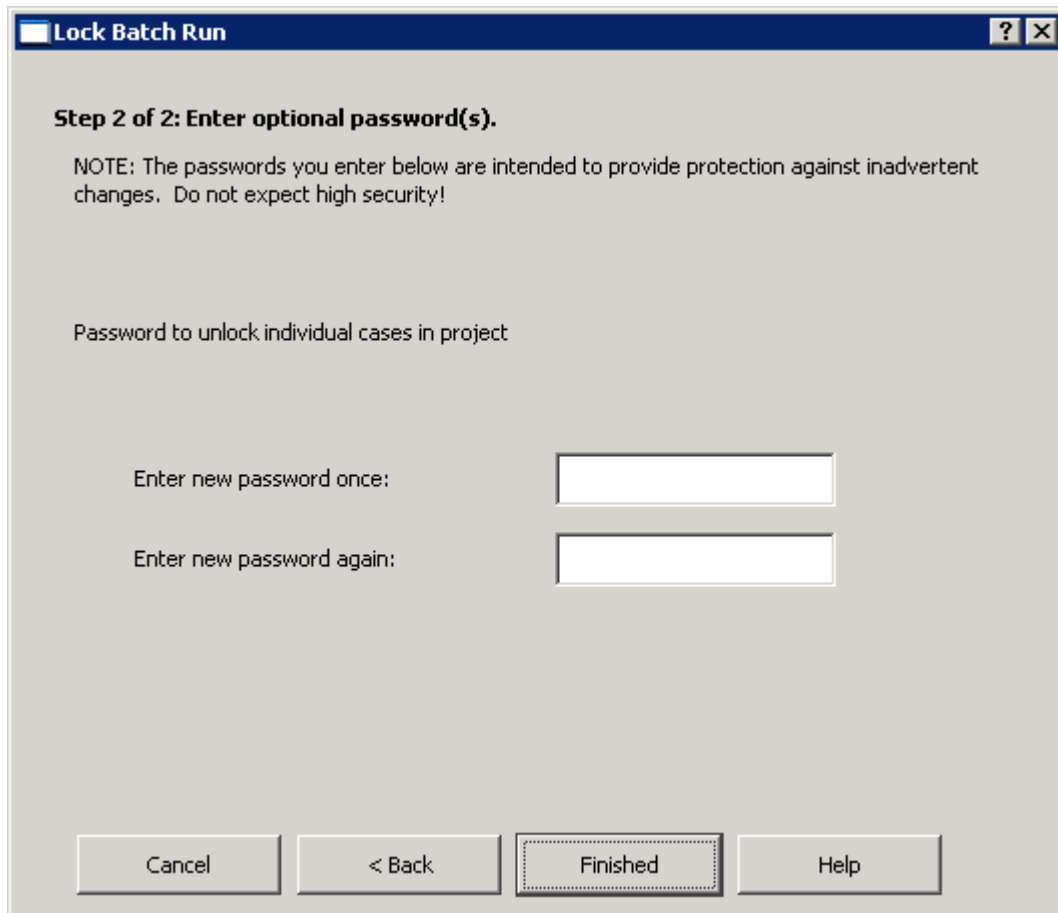
## RAPPID User's Guide

---

Under **Step 1 of 2** on the **Lock Batch Run** dialog box, select whether you want to only lock the currently selected batch run or all batch runs in the currently open project.



Use **Step 2 of 2** of the **Lock Batch Run** dialog box to assign a password to the locked batch run(s).



**NOTE:** Once you enter a password to lock a batch run, this password exists whether or not you have any batch runs locked. If you have no batch runs currently locked, and you want to lock a batch run, RAPPID™ will ask you to enter the old password in addition to entering the new password twice or applying no password (leave the **Enter new password** edit boxes blank). If you apply no password after entering the old password, the batch run-locking password is cleared.

## RAPPID User's Guide

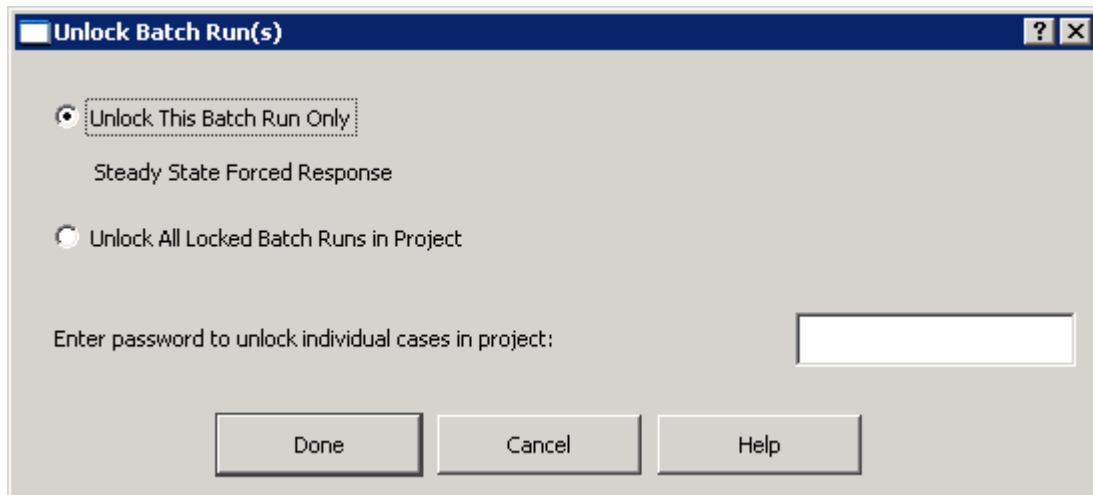
---

This password is optional. You can lock one or more batch runs using no password.

**NOTE:** The password(s) you enter when locking a project and all the cases in a project is intended to provide protection against inadvertent changes. Do not expect high security!

When you unlock a batch run, similar choices are available as when you locked the batch run. You may either unlock:

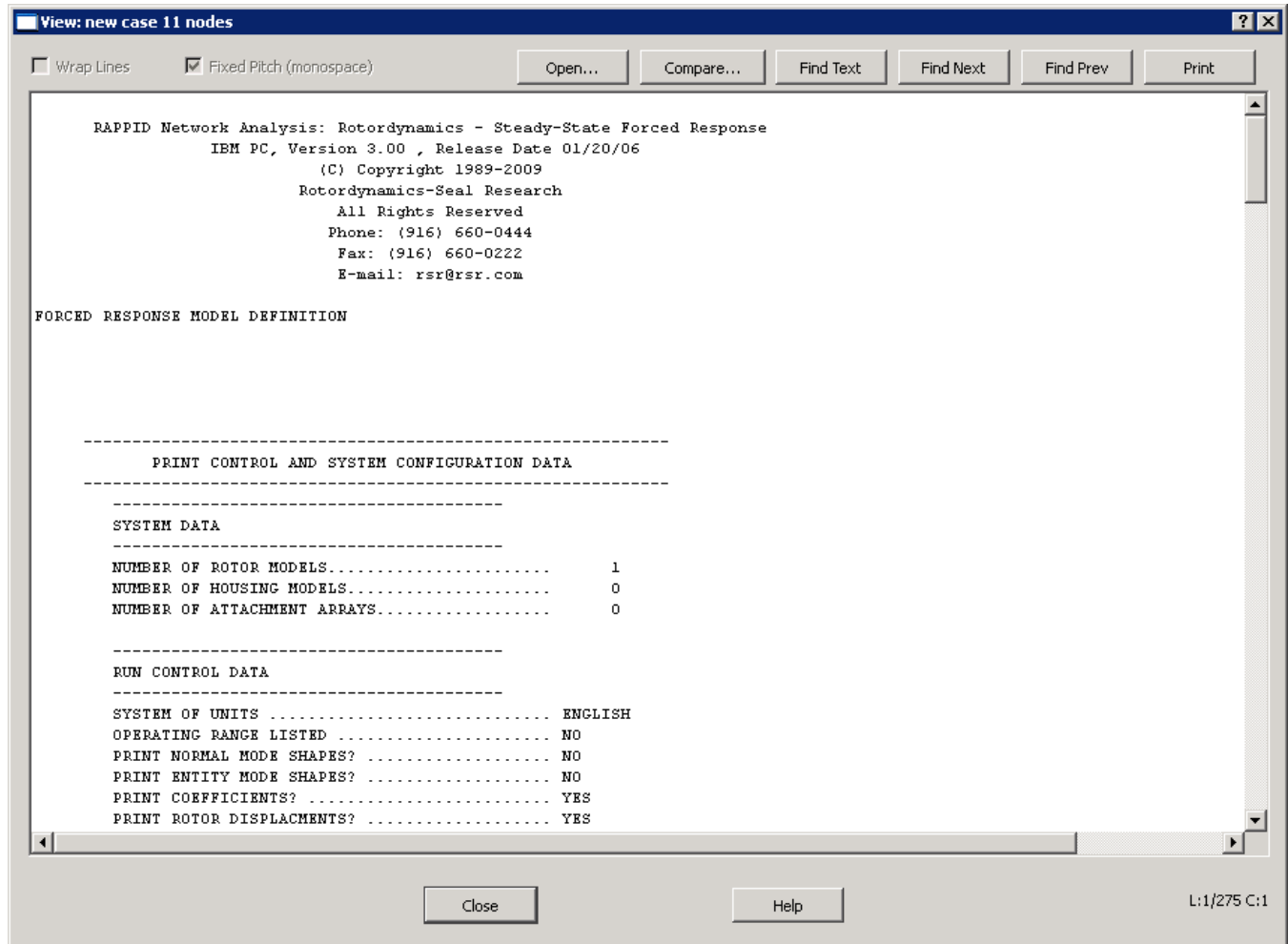
- the selected batch run, or
- all locked batch runs in the project



If you assigned a password to the locked batch runs, enter the password in the **Enter password to unlock individual cases in project** edit box to unlock the batch run(s).

## Viewing and printing results

Use the **View** button on the case manager or select **View** from the **Command** menu to view the results of the analysis case run.



**NOTE:** The text displayed on the **View** dialog box cannot be edited; therefore, the **Wrap Lines** and **Fixed Pitch (monospace)** check boxes at the top of the dialog box are not available.

Use the **Print** button to print the results displayed in the dialog box.

## Comparing analysis case results

Once you have run an analysis case, you can compare all or part of the results of that analysis case with all or part of output from another analysis case.

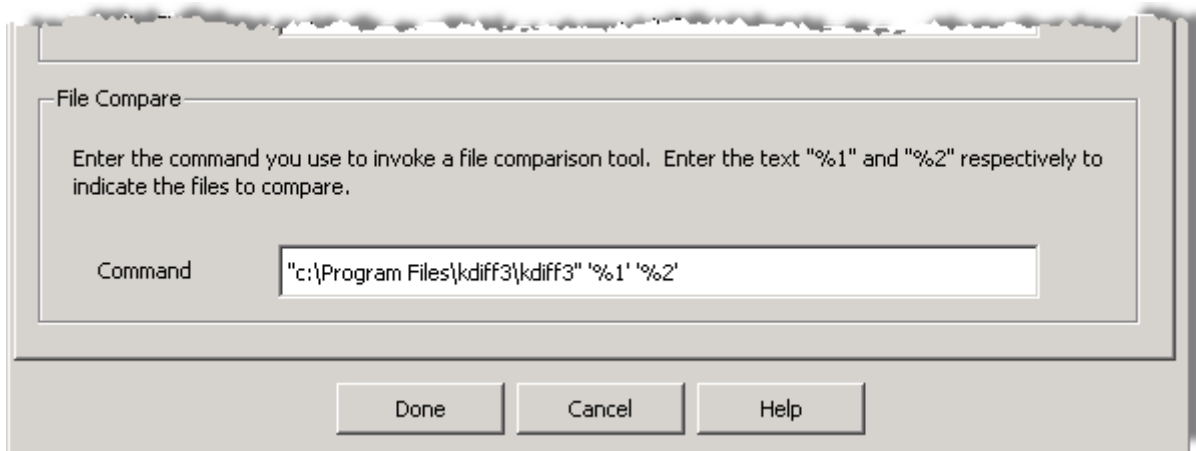
## Changing the default comparison tool

RAPPID™ provides an option to change the tool used when comparing output files.



## RAPPID User's Guide

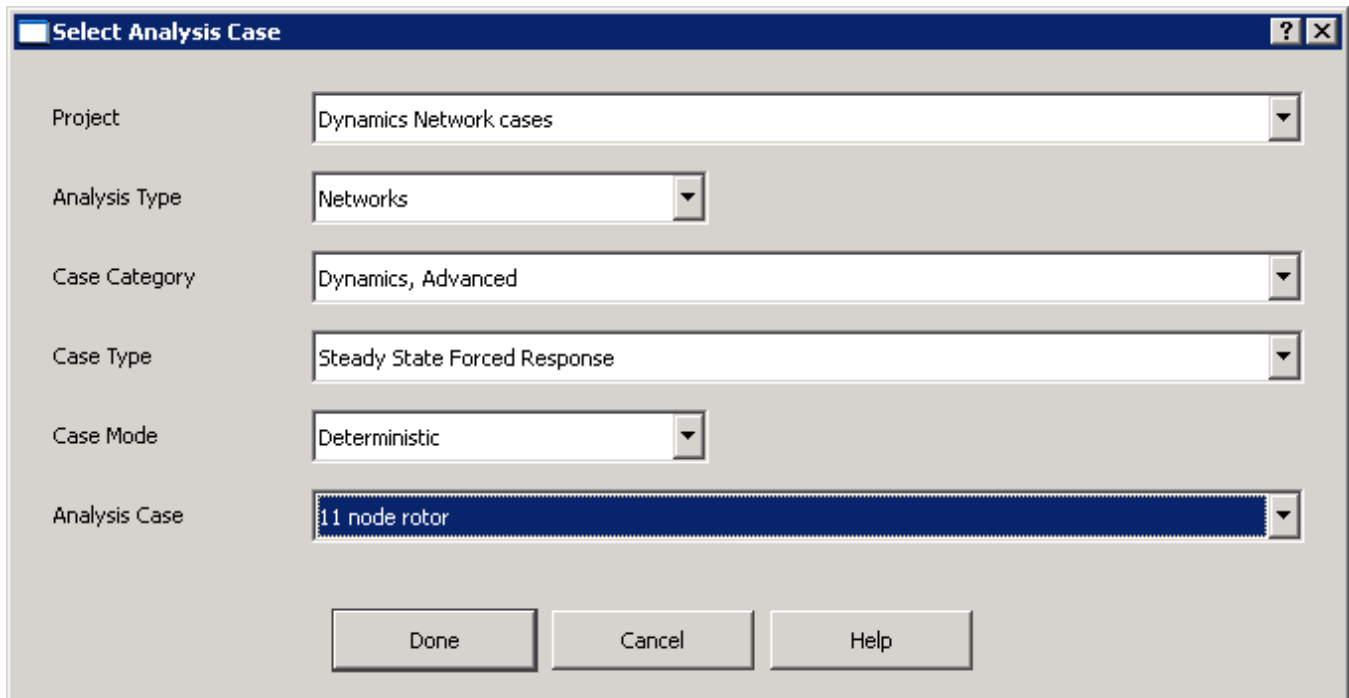
Select **Preferences** from the **Project** menu. In the **Command** edit box in the **File Compare** box on the **General** tab, specify the command to start the differencing tool you want to use when comparing output in the **View** dialog box.



**NOTE:** After the command used to start the file comparison program, be sure to put "%1" and "%2" for the two files to be used in the comparison. Be sure to specify the full path to the location of the file comparison program or make sure the path is defined in the PATH environment variable in the current operating system.

### Opening results from another analysis case

Use the **Open** button on the **View** dialog box to open the case results for any analysis case in any project in a second text box.



---

## RAPPID User's Guide

---

Use the drop-down menus to select the **Project**, **Analysis Type**, **Case Category**, **Case Type**, **Case Mode**, and **Analysis Case** for the second analysis case.

Click **Done** to display the second results file in the text box of the **View** dialog box.

Press **F8** to switch between the two results files.

### Comparing text from two results files

To compare all or part of the results from the currently open analysis case with all or part of results from another analysis case, follow these steps:

**NOTE:** The two sets of text being compared are compared as files (temporary files are created) and are referred to as such in this procedure. When the **View** dialog box is closed, these two files are deleted.

1. In the currently open analysis case, click the **View** button on the **case manager** and open the results from another analysis case.

See **Opening results from another analysis case** on page 62 for more information.

2. Press **F8** if necessary to go back to the results file for the currently open analysis case.
3. If you want to compare only parts of each results file, press **F8** to switch between the results files, and highlight the text in each that you want to compare.

**NOTE:** If no text is selected, all of the text in that file is used in the comparison. This is different from highlighting text to copy.

4. Click **Compare** to start the comparison tool specified in the **Preferences** (see **Changing the default comparison tool** on page 61) to compare the two files.

### Deleting analysis case results

RAPPID™ provides the ability to delete the results from running an analysis case. This feature is useful when you are working with **Network** analysis cases and you want the **Network** case to re-run the **Component** analysis case and use the new results. See **Run Mode** on page 283 in

**Chapter 22: Dynamics** Networks for more information.

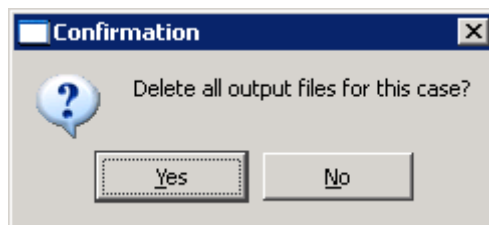
Output files for analysis cases include the following:

- A file containing the calculation results viewed using the **View** button on the **case manager**.
- A file containing the data used to generate the plots using the **Plot** button on the **case manager**.

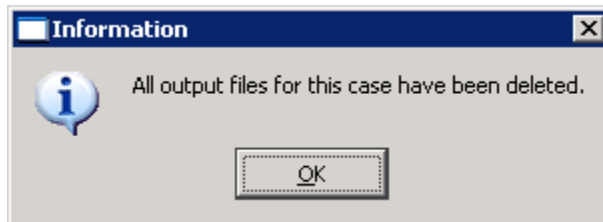
To delete output files for an analysis case, you must first open the analysis case. Use the **Delete Output Files** option on the **Command** menu to delete output files for currently open analysis case.

**NOTE:** It does not matter which page is active in the analysis case when you select this option.

A **Confirmation** dialog box displays asking if you want to delete all the output files for the case.



Click **Yes** to delete the output files. An **Information** dialog box displays informing you that all the output files have been deleted.



Click **OK** to return to the page that was active when you selected the **Delete Output Files** option.

## Chapter 4: Adding notes and attachments

Projects and analysis cases can hold extra information in addition to comments and data values.

1. You can store textual information internally in projects and analysis cases in the form of notes.

See **Working with notes** on page 65 for more information.

2. You can store external information in the form of files attached to projects and analysis cases.

See **Working with attachments** on page 74 for more information.

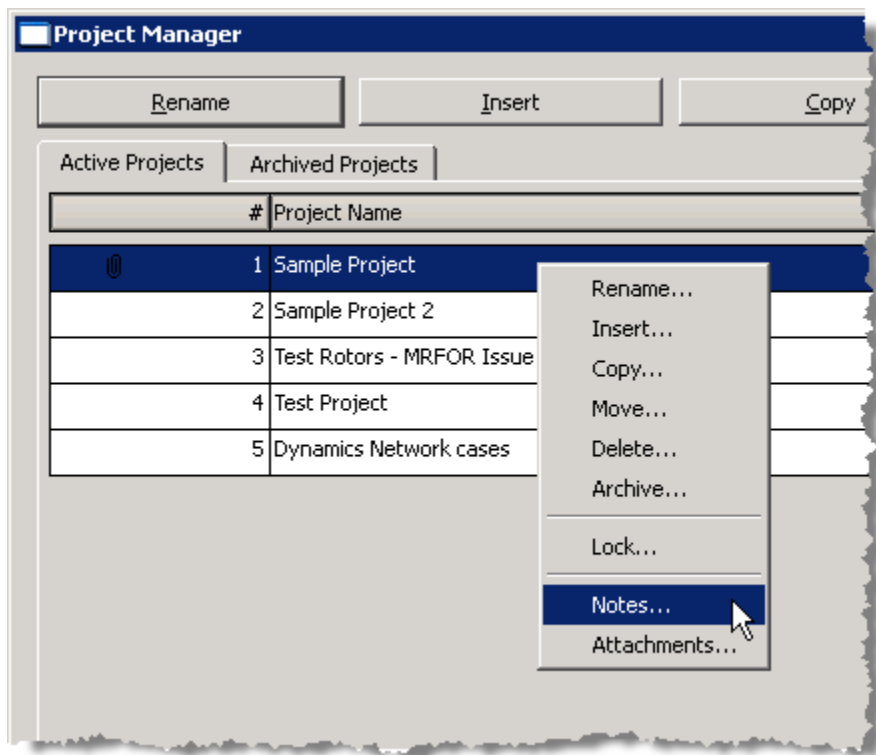
### Working with notes

Additional information may be stored within projects and analysis cases in the form of notes. Text entered as notes in a project or analysis case are stored directly in the project or analysis case. The same dialog box is used for entering notes in both projects and analysis cases, but it is accessed differently for each.

**NOTE:** The text entered in the **Notes** dialog box is for your reference only and is not used by the Program.

### Adding or editing notes associated with a project

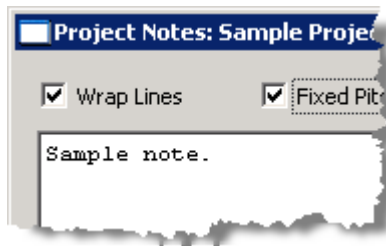
To add notes to a project, select a project in the **Project Manager**, right-click on it, and select **Notes** from the popup menu.



# RAPPID User's Guide

---

Enter any notes you want to store with the project in the text box and click **Done** to save your text.

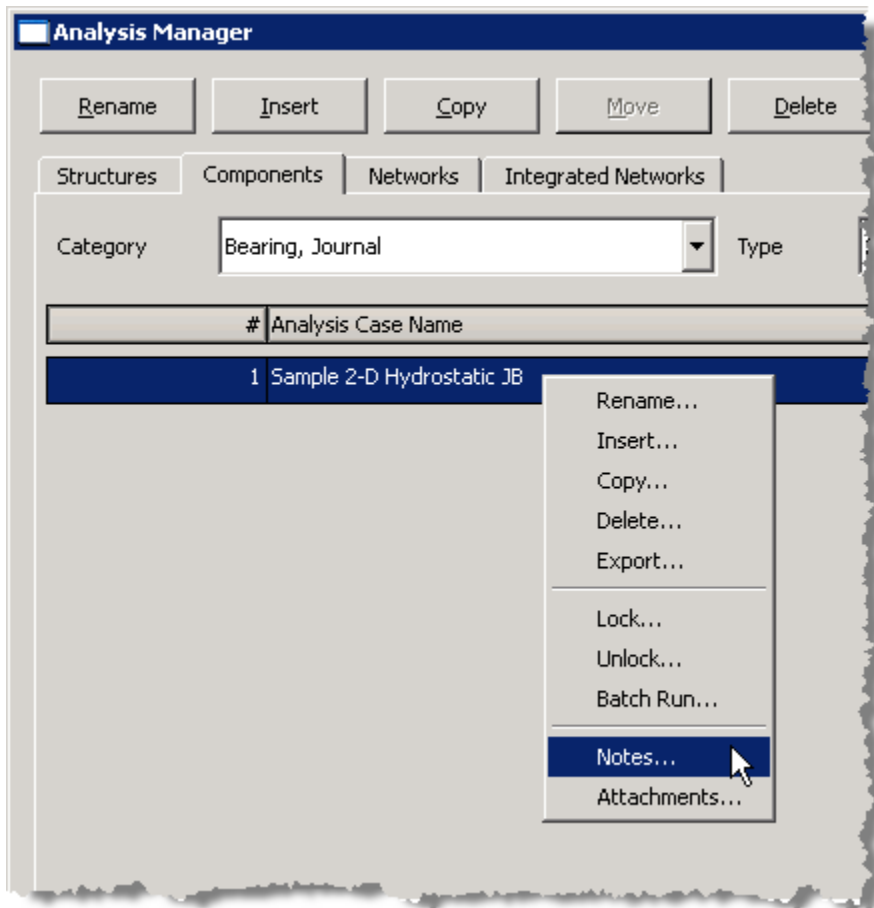


**NOTE:** You can enter two sets of notes using **F8** to switch between them. The notes text currently displaying when you click **Done** is what RAPPID™ saves with the project. You will lose the other set of notes.

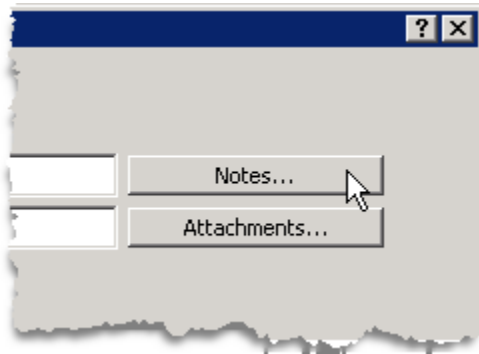
## Adding or editing notes associated with an analysis case

To add notes to an analysis case, use one of the following methods in the **Analysis Manager**:

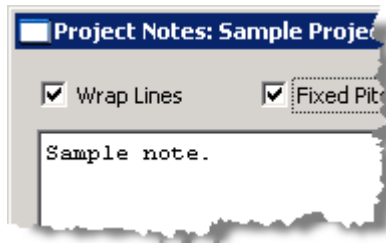
- Select an analysis case, right-click on it, and select **Notes** from the popup menu.



- Open the analysis case into which you want to add notes and click the **Notes** button on the **Options** page.



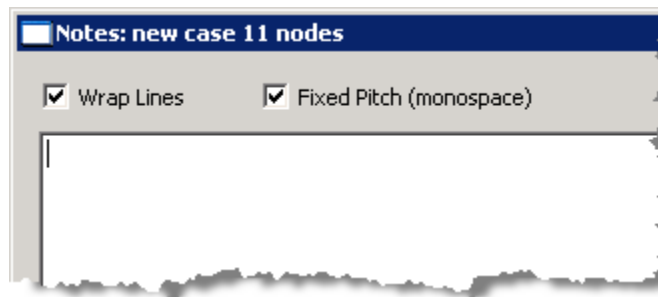
Enter any notes you want to store with the analysis case in the text box and click **Done** to save your text.



**NOTE:** You can enter two sets of notes using **F8** to switch between them. The notes text currently displaying when you click **Done** is what RAPPID™ saves with the project. You will lose the other set of notes.

### Setting the appearance of the text in notes

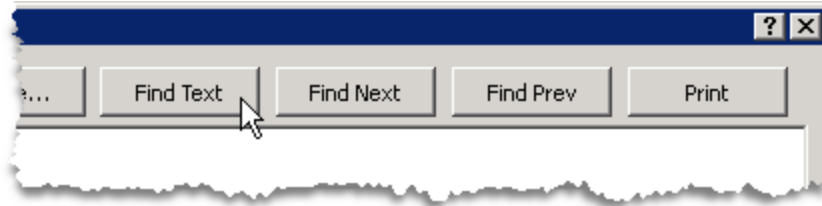
Use the **Wrap Lines** option to automatically wrap lines of text that exceed the width of the text box on the **Notes** dialog box.



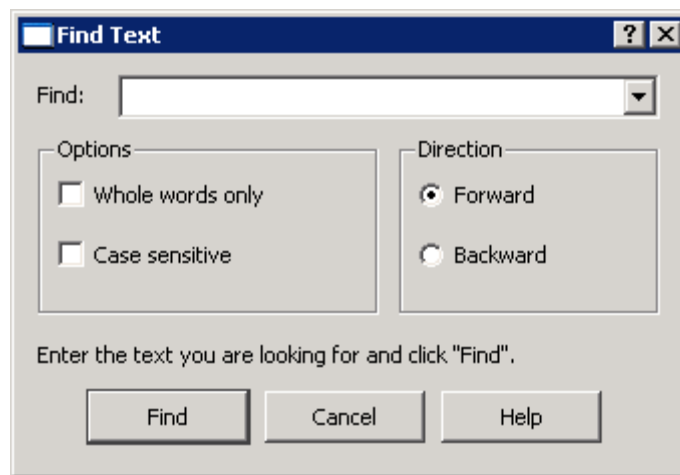
Use the **Fixed Pitch (monospace)** option changes the font of the text to a monospace font.

## Finding text and phrases in notes

To find text or a phrase in notes within a project or an analysis case, click the **Find Text** button on the **Notes** dialog box.



Enter the word or phrase you want to find, specify any other options you want to apply to the search, and click **Find**.



To find the next occurrence of the word or phrase, click **Find Next**.

To find the previous occurrence of the word or phrase, click **Find Prev**.

Click **Done** to close the **Notes** dialog box.

## Printing notes

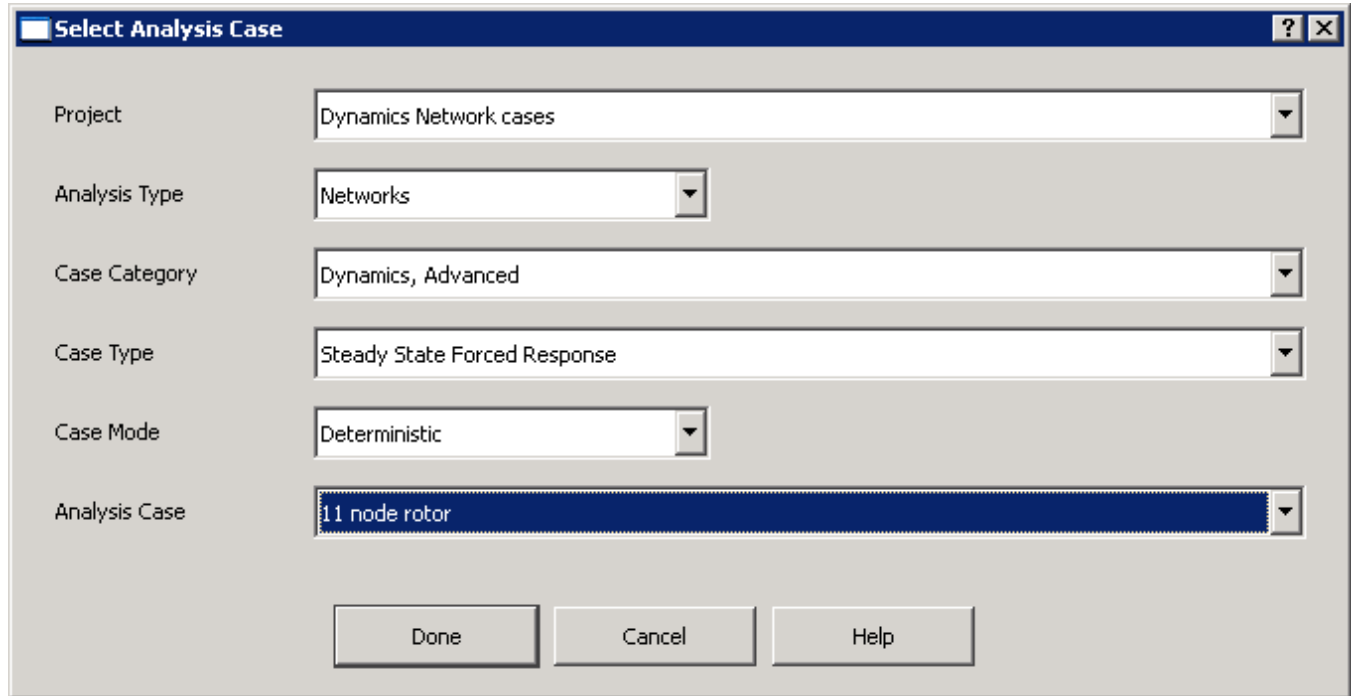
Use the **Print** button on the **Notes** dialog box to print the notes associated with a project or an analysis case.

The standard printing dialog box displays for your operating system, such as Windows or Linux. Refer to the documentation for the operating system you are using and for the printer you are using if you need assistance with the printing options available.

## Opening an output file in the Notes dialog box

Use the **Open** button to open the case output, or results, for any analysis case in any project in a second text box on the **Notes** dialog box.

**NOTE:** The **Open** button is only available on the **Notes** dialog box for analysis case notes, not for project notes.



The screenshot shows a dialog box titled "Select Analysis Case". It features six drop-down menus for selection: "Project" (Dynamics Network cases), "Analysis Type" (Networks), "Case Category" (Dynamics, Advanced), "Case Type" (Steady State Forced Response), "Case Mode" (Deterministic), and "Analysis Case" (11 node rotor). The "Analysis Case" menu is currently expanded, showing "11 node rotor" as the selected item. At the bottom of the dialog are three buttons: "Done", "Cancel", and "Help".

Use the drop-down menus to select the **Project**, **Analysis Type**, **Case Category**, **Case Type**, **Case Mode**, and **Analysis Case** for the analysis case containing the output you wish to view.

Click **Done** to display the output file in the text box of the **Notes** dialog box.

Press **F8** to switch between the notes and the output file.

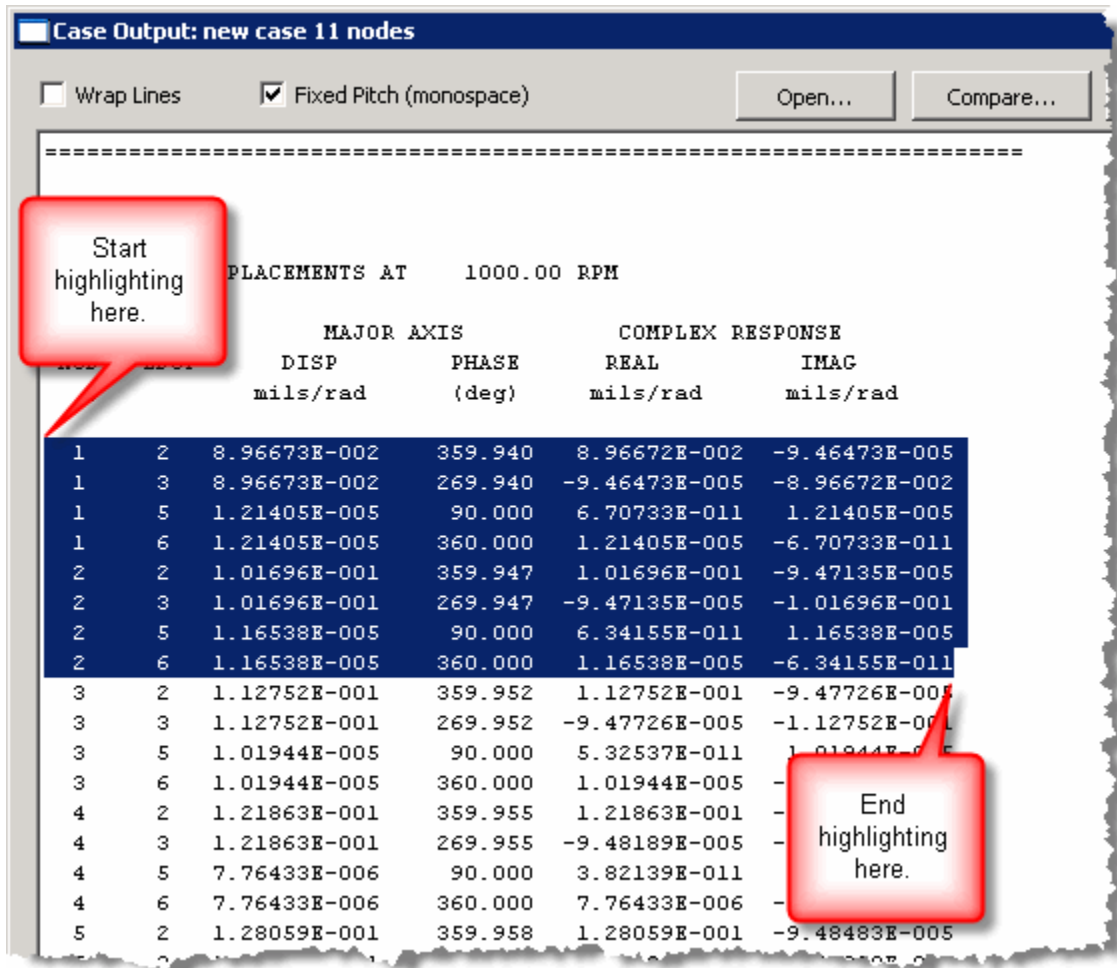
**WARNING:** When you click **Done** on the **Notes** dialog box, the text currently visible is saved as your notes in the current analysis case. If you want to preserve notes you have entered, be sure the text box containing your notes is visible, pressing **F8** if they are not. If the output file is visible, that is saved as your notes for that analysis case and the notes you entered are lost.



## Copying text from notes

To copy all or part of the text in the **Notes** dialog box, follow these steps:

1. On the **Notes** dialog box, highlight the text you want to copy.

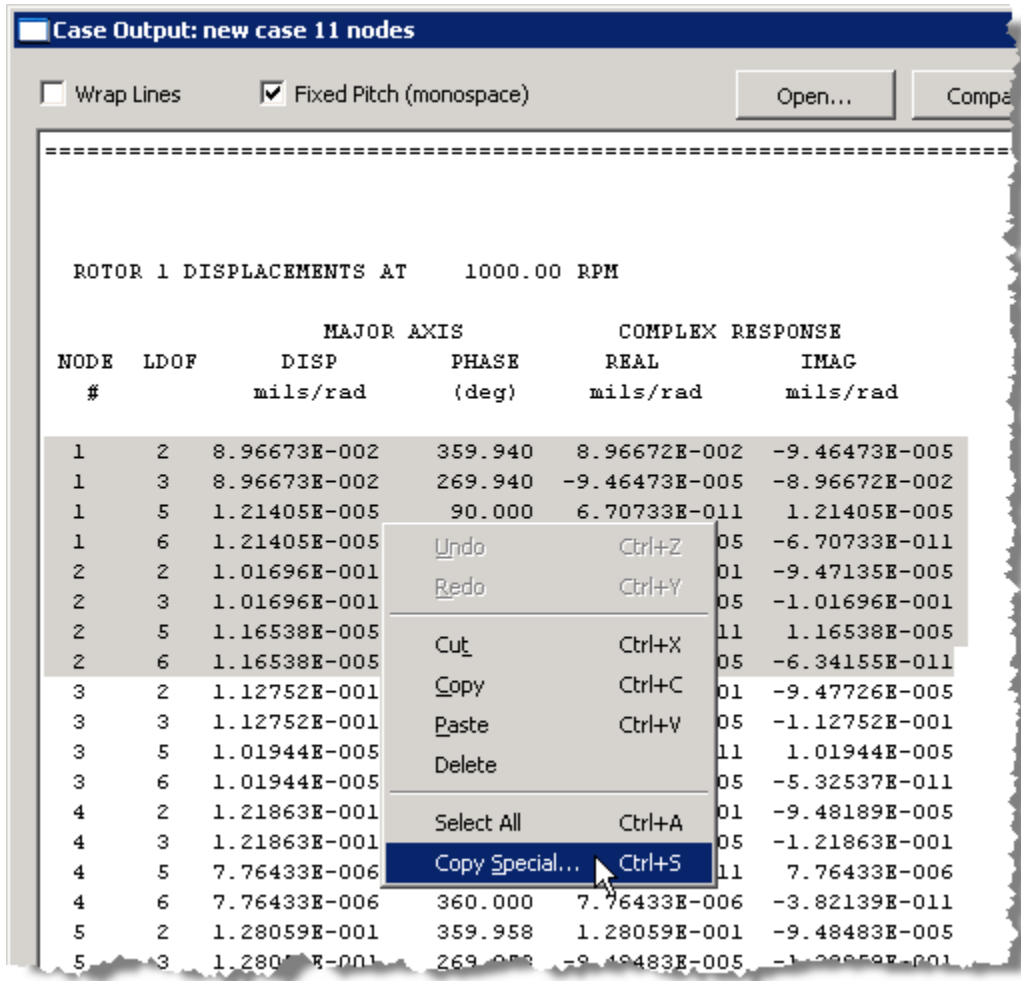


In the example in the picture above, the highlighting starts before the first column in the first row to be copied and ends after the last column in the last row to be copied. This will copy the eight rows and six columns into column format.

2. The following are guidelines to consider when copying text in the **Notes** dialog:
  - a. If you do not select any text, the **Copy Special** command will copy text from the cursor location to the next blank line. Therefore, if the cursor is on a blank line, nothing will be copied.
  - b. If you select text for which the starting column number is the same as the ending column number, nothing will be copied. If you want to copy several complete lines of text, whether they be columns of text or paragraphs, highlight from the first character to right after the last character, similar to the above example.

# RAPPID User's Guide

- Right-click on the selected text and select **Copy Special** from the popup menu.



You may also press **Ctrl-S** to access the **Copy Special** dialog box.

The **Copy Special** dialog box displays.

**Copy Special** ? X

Specify a block of text to copy.

**Start Location**

Line Number 175

Column Number 1

**End Location**

Line Number 182

Column Number 66

**Options**

Delimiter Tab

Done Cancel Help

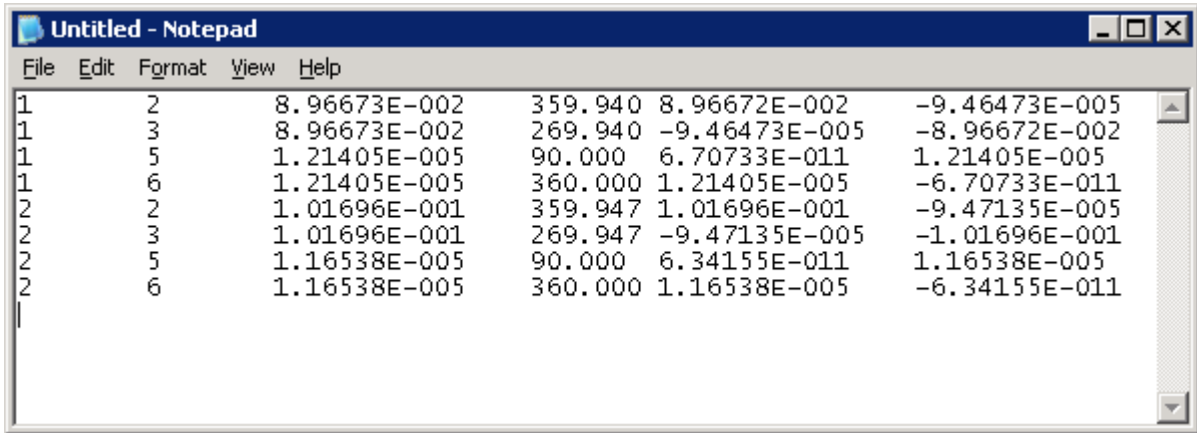
The line numbers and column numbers for the start and end locations are initially automatically entered from the selection but they can be changed manually, if desired.

4. In the **Options** box, use the **Delimiter** drop-down list to specify whether you want columnar text to separated by tabs, spaces, or commas when you paste it into a text editor, spreadsheet, or other software program.
5. Click **Done** to finish copying the text.
6. Paste the text into a document in your destination application.

You can also paste the text into certain places within RAPPID™. For example, you may want to copy a table of values from an output file into a grid in the analysis case or into the notes for the analysis case.

# RAPPID User's Guide

The following example shows the text highlighted in step 1 pasted into **Notepad** in Windows using tabs as delimiters.



## Comparing text from a note with text from an output file

To compare all or part of one set of notes text with all or part of output from an analysis case (in analysis case notes) or another set of notes text (in project notes), follow these steps:

**NOTES:** The two sets of text being compared are compared as files (temporary files are created) and are referred to as such in this procedure. When the **Notes** dialog box is closed, these two files are deleted.

See **Changing the default comparison tool** on page 61 for information on specifying which differencing tool you want RAPPID™ to use when comparing files.

1. On the **Notes** dialog box, highlight the text in one file you want to compare.

**NOTE:** If no text is selected, all of the text in that file is used in the comparison. This is different from highlighting text to copy. See **Copying text from notes** on page 70 for more information.

2. Press **F8**.

A blank text box displays on the **Notes** dialog box. The text from the original text box is preserved and accessible by pressing **F8** again.

- If you are entering notes for an analysis case, you can open an output file from another analysis case to use for comparison.

See **Opening an output file in the Notes dialog box** on page 69.

- If you are entering notes for a project, you can enter a second set of notes in the blank screen or paste text from a different source.
3. In both files, highlight the text you want to use in the comparison, or, to compare both whole files, do not highlight any text in either file.

4. Click the **Compare** button to open the differencing tool displaying the two files.

**NOTE:** The current file displaying in the **Notes** dialog box is the file that will be considered the first file in the comparison and is typically placed on the left side (for side-by-side display) or the top (for top-bottom) of the differencing tool.

When you are done with the comparison and you close the differencing tool, the **Notes** dialog is still available.

5. To close the **Notes** dialog box, click **Done**.

### Working with attachments

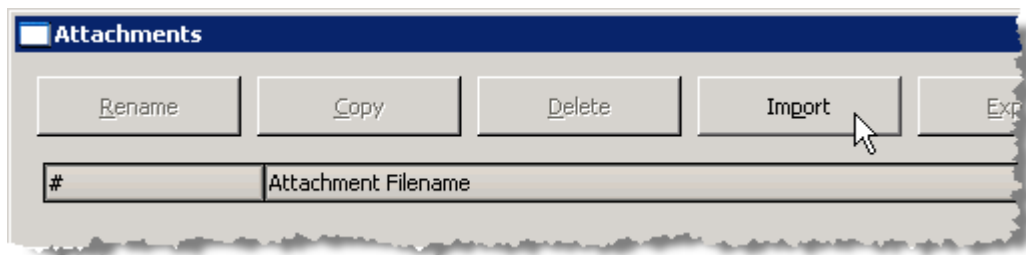
Use **Attachments** to store information with a project or an analysis case that resides in one or more external files.

When you add files as attachments in projects or analysis cases, copies of the attached files are stored with the case, separately from the original location.

### Adding attachments to a project or an analysis case

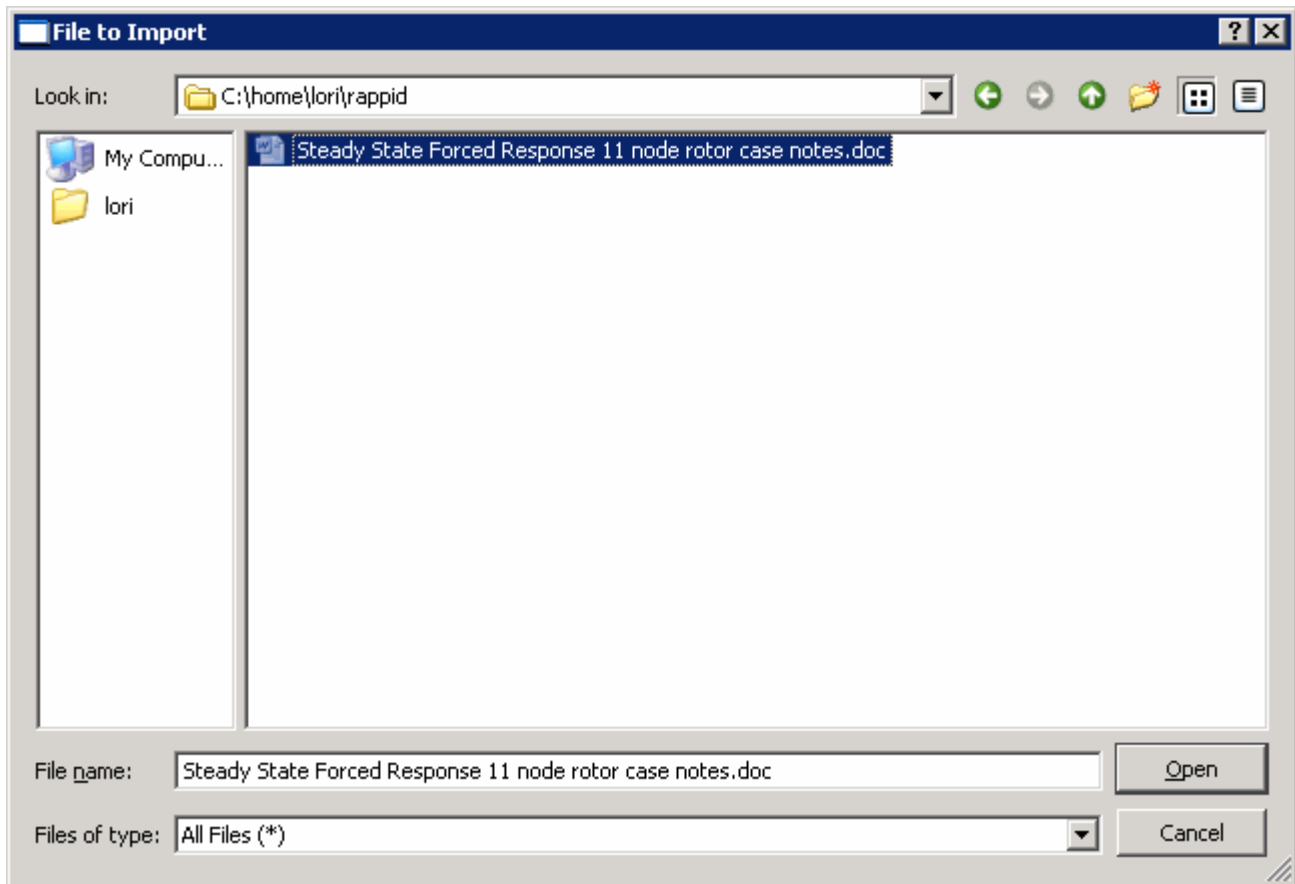
When adding attachments to projects, access the **Attachments** dialog box by selecting the **Attachments** option on the pop-up menu that displays when you right-click on the name of a project on the **Active Projects** tab in the **Project Manager**. The same procedure can be applied when adding attachments to analysis cases. However, for analysis cases, you may also click the **Attachments** button next to the second **Comments** edit box once you open the analysis case.

Use the **Import** button to select a file to attach to the project or analysis case.

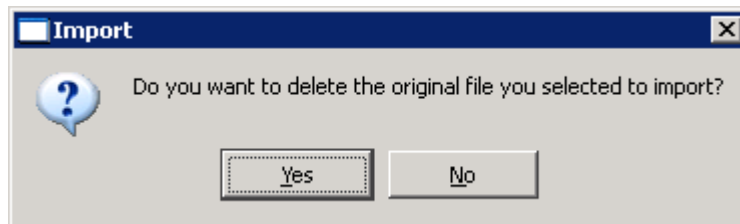


# RAPPID User's Guide

On the **File to Import** dialog box, select the file you want to import and click **Open**.



When you import a file as an attachment in a project or an analysis case, you have the option to delete the original file you selected to import. To preserve the original file, click **No**.



**NOTE:** A copy of the file has been included with the project or analysis case at this point. However, the file may only be accessible within RAPPID™ if you delete the original file and you do not have another copy of the file.

## Viewing the contents of an attached file

Files attached to projects and analysis cases can be opened from within RAPPID™.

On the **Attachments** dialog box, select the attached file you want to open and click the **Open** button. The file is opened in the program in which it was created.

### Renaming an attached file

Use the **Rename** button to rename a file attached to a project or analysis case within RAPPID™. Since a copy of the file is included with the project or analysis case, when you rename an attached file, you are renaming the copy of the file not the original file. If you want to rename the original file, use the **Export** button. See **Exporting an attached file** on page 76 for more information.

### Copying an attached file

Use the **Copy** button to make a copy of a selected attached file. The copy is also stored with the project or analysis case in the same location as the file being copied. Therefore, you need to specify a different filename for the copied file. If you want to copy an attached file to a different location, use the **Export** button. See **Exporting an attached file** on page 76 for more information.

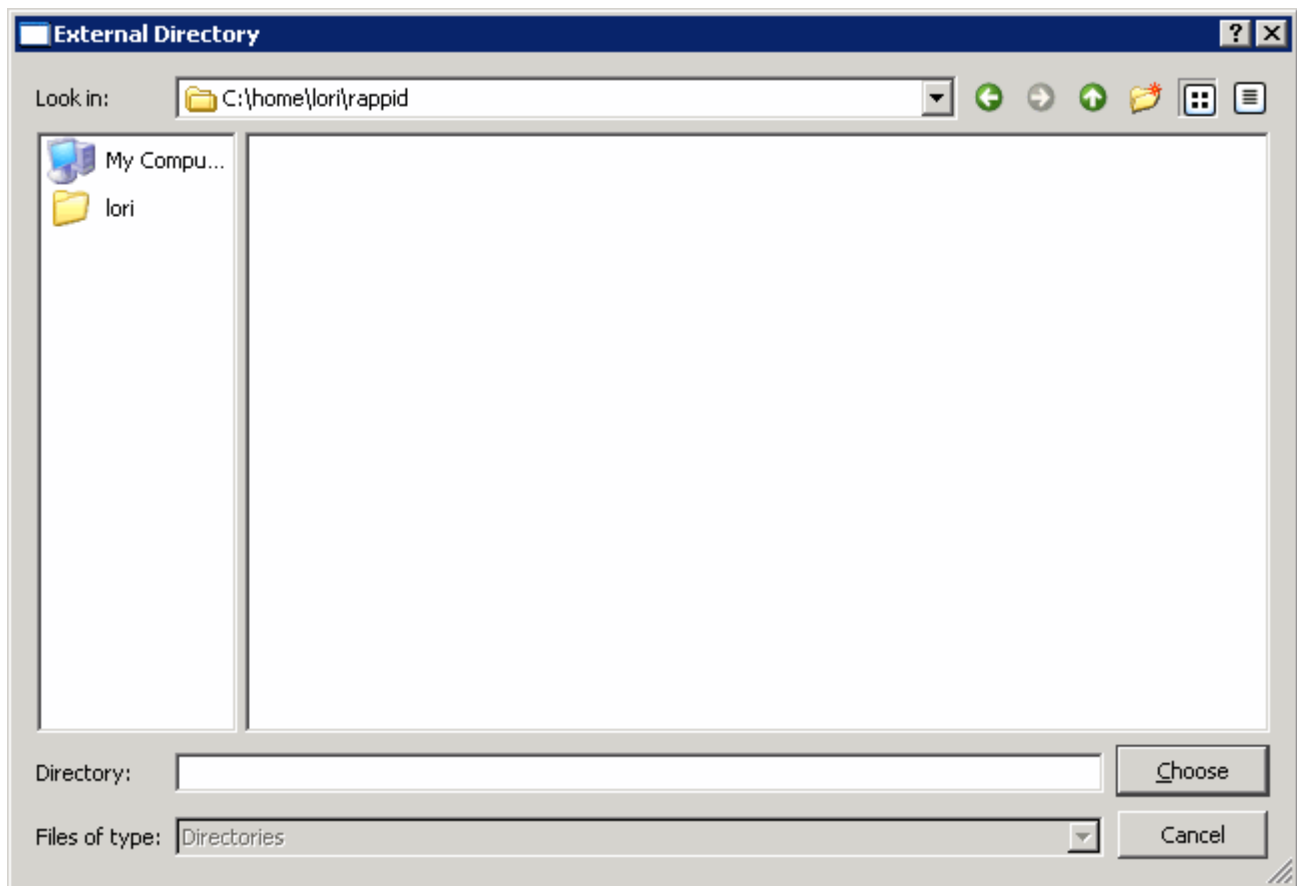
### Exporting an attached file

Use the **Export** button to copy an attached file to a different location outside of the project or analysis case.

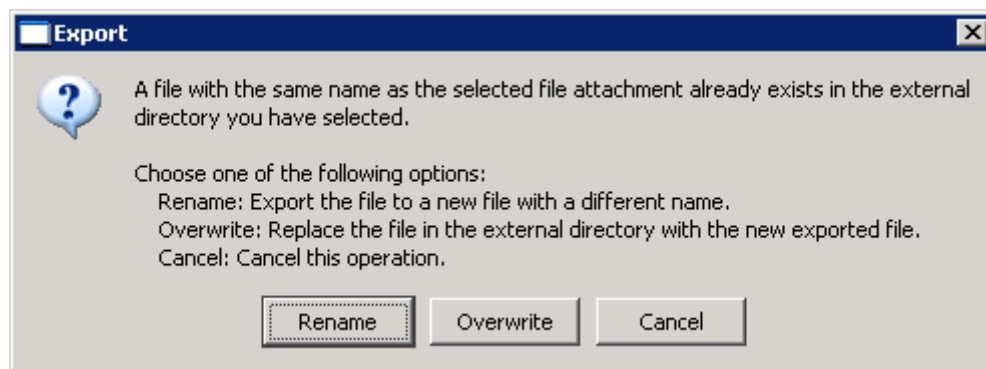
This feature is useful if you have inadvertently deleted the original file, or you have changed the copy of the file RAPPID™ stores with the project or analysis case and you want to save it externally under a new name.

## RAPPID User's Guide

On the **External Directory** dialog box, select the directory in which you want to save the new file.



If you select the same directory as the original file, if it still exists, RAPPID™ displays the **Export** dialog box.

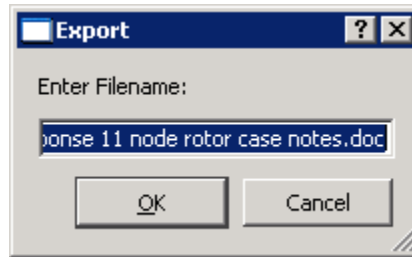




## RAPPID User's Guide

---

To preserve the original file and save the new file with a different name, click the **Rename** button. A dialog box displays prompting you to specify a new filename.



To replace the original file, click the **Overwrite** button.

### Deleting an attached file

Use the **Delete** button to delete the selected attached file.

**NOTE:** This action deletes only the copy of the file stored in RAPPID™, not the original file.

## ***Chapter 5: Working with materials***

Use the **Materials Library** and the **Materials** page to define the material properties list that will be available for developing a finite element model for the analysis case. Use one of the following methods to add one or more materials to an analysis case.

- Use the **Materials** page in an analysis case to add new materials and store the material properties within the analysis case. The properties available on the **Materials** page depends on the type of analysis case currently open.

See **Using the Materials page** on page 85.

- Use the **Materials Library** to add and store materials for use in multiple analysis cases in any active project. On the **Materials** page in an analysis case, link to a material from the **Materials Library**.

See **Using the Materials Library** on page 80 and **Linking to a material from the Materials Library** on page 86.

- Embed an existing material in an analysis case by linking to a material that exists in the **Materials Library** and then unlinking the material.

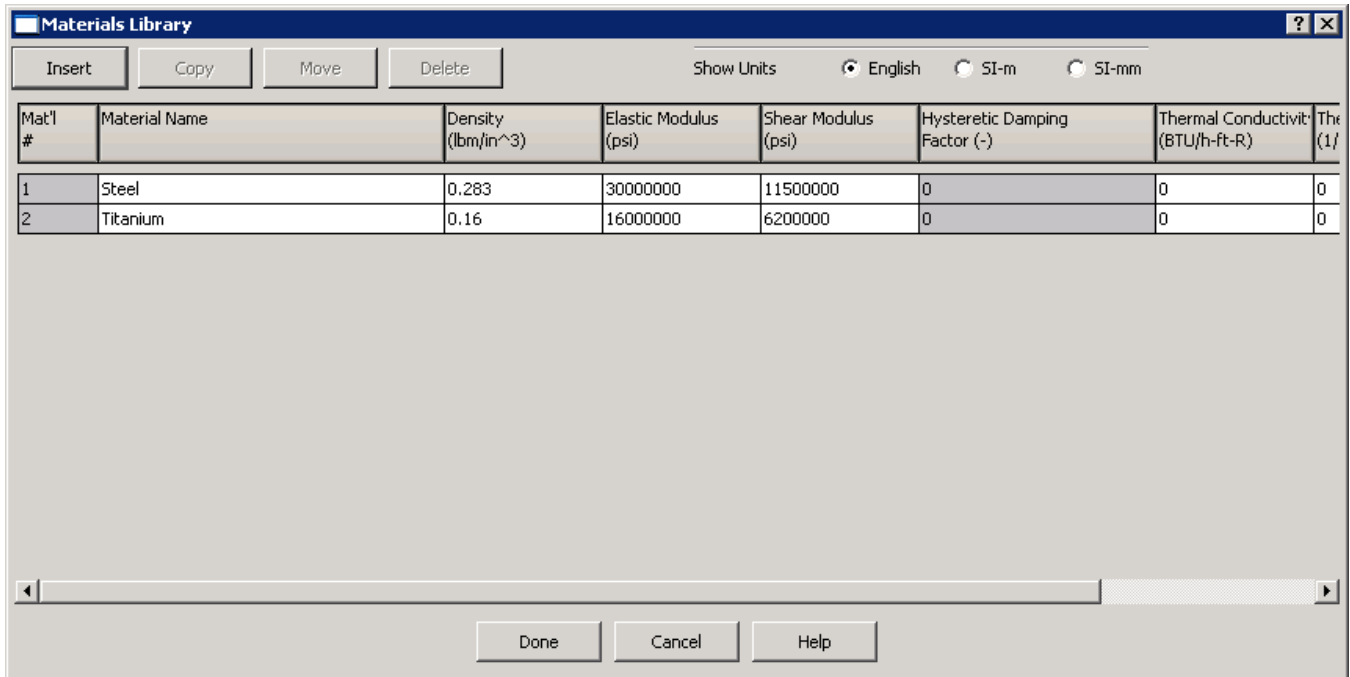
See **Linking to a material from the Materials Library** on page 86 and **Breaking a link to a material in the Materials Library** on page 88.

**NOTE:** Adding a material using this method is practical if you want to define a material similar to an existing material. You can obtain the values of the properties from the **Materials Library** by linking to the material. Once the values are imported onto the **Materials** page from the **Materials Library**, you can unlink the material, allowing you to change the values.

# RAPPID User's Guide

## Using the Materials Library

Material properties for an analysis case can be defined manually within the case on the **Materials** page, or you can use a material defined and stored in the **Materials Library**, therefore using the properties of the material in the library.

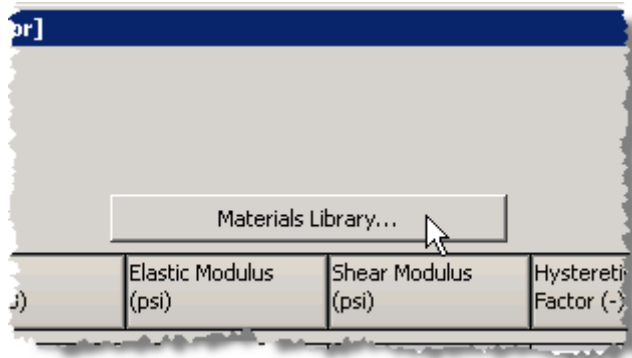


Materials defined in the **Materials Library** are available to any analysis case that uses materials in any project listed on the **Active Projects** tab on the **Analysis Manager** window.

**NOTE:** Enter as much information in the **Materials Library** for the properties of the material as possible for each material even if the case you are working on does not use all the values. This ensures that other cases that may require the properties have access to them.

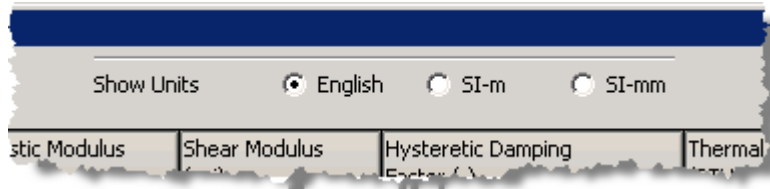
## Accessing the Materials Library

Use the **Materials Library** button on the **Materials** page of any analysis case that uses materials to access the **Materials Library**.



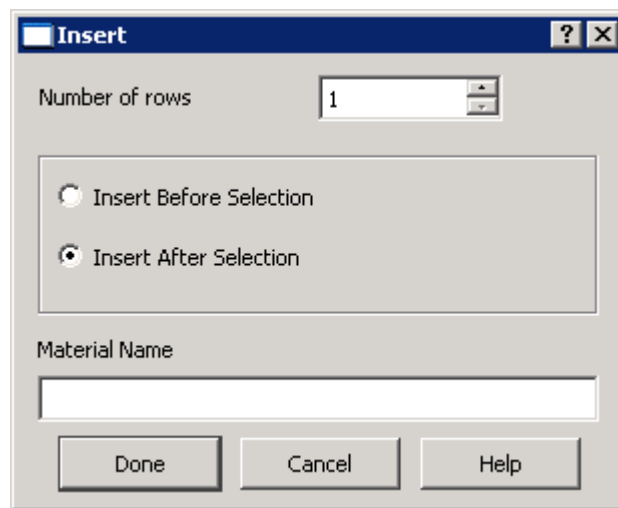
## Choosing the unit system to apply to all material properties

Use the **Show Units** radio buttons to select the unit system in which you want to enter the values of the properties for the materials.



## Adding a new material to the Materials Library

Use the **Insert** button to add a new material to the **Materials Library**. If there are existing materials in the **Materials Library**, select one before or after which the new material will be added.



Indicate the number of materials you want to add by entering or selecting the amount in the **Number of rows** spin box.

If this is not the first material added to the **Materials Library**, use the **Insert Before Selection** and **Insert After Selection** radio buttons to specify the location of the new materials in the list.

If you are only adding one material, enter the name of the material in the **Material Name** edit box. This field is not available when adding more than one material.

Once the material has been inserted into the list in the **Materials Library**, use the columns of the grid to enter the name of the material and applicable properties for the material. Properties may include the following, depending on the current analysis case type:

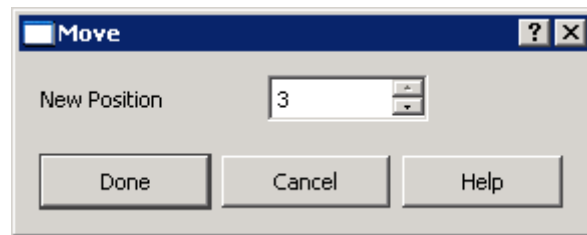
- Density
- Elastic Modulus
- Shear Modulus

- Hysteretic Damping (currently not available)
- Thermal Conductivity
- Thermal Conductivity
- Thermal Expansion Coefficient

### Moving a material in the Materials Library

Use the **Move** button to move a selected material to any other location in the list of materials in the **Materials Library**.

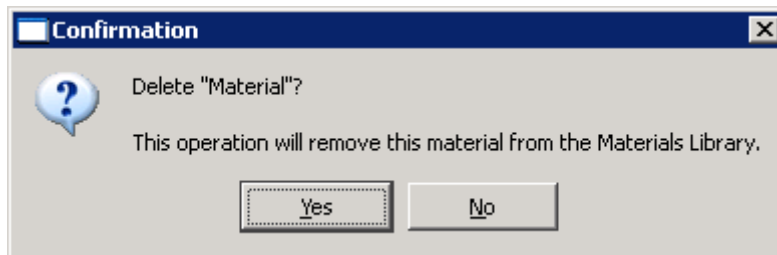
On the **Move** dialog box, specify the **New Position** for the selected analysis case by typing in a new number or using the up and down arrows to select a value.



**NOTE:** The **Move** button is only available when there are multiple materials in the **Materials Library**.

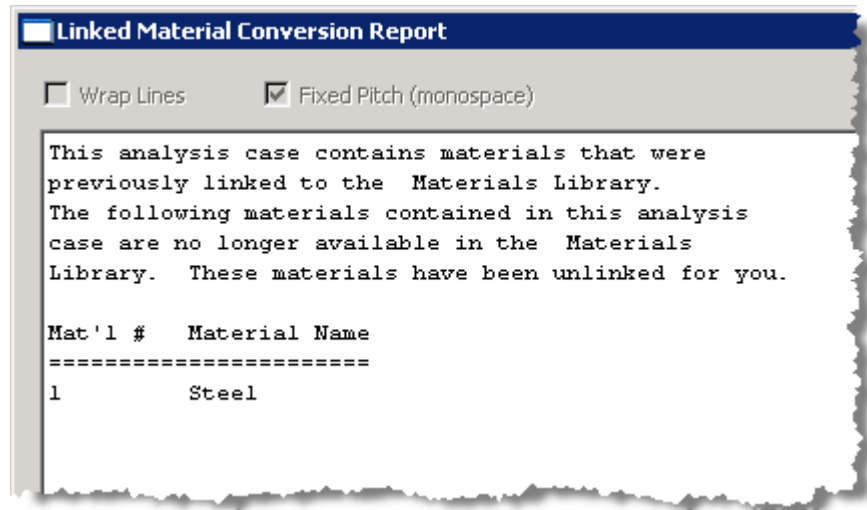
### Deleting a material from the Materials Library

Use the **Delete** button to delete a selected material from the **Materials Library**.



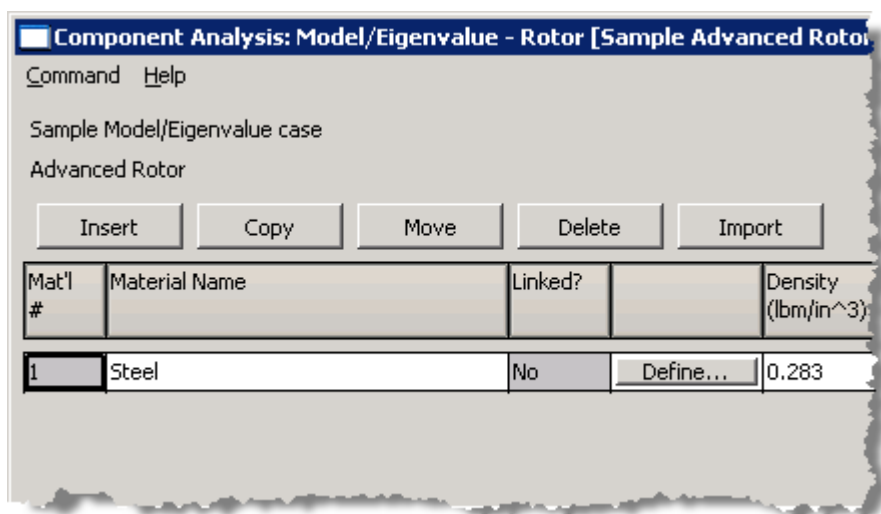
## RAPPID User's Guide

If you delete a material from or change a material's properties in the **Materials Library**, all cases that link to that material are affected. If you are currently in an analysis case when you delete or modify a material or if you open an analysis case that references a material you deleted or modified, the **Linked Material Conversion Report** displays listing the material that was deleted and informing you that the material has been unlinked for you.



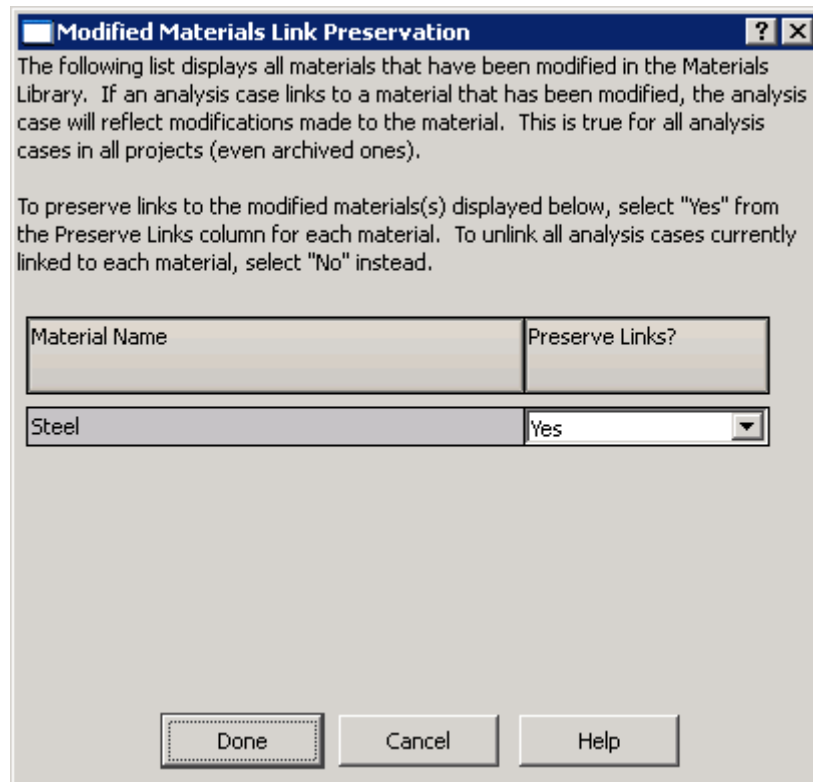
If multiple materials are deleted, all materials that are deleted are unlinked in all analysis cases that linked to the materials. All materials that are deleted are listed in the **Linked Material Conversion Report**.

When you delete a material, the **Linked?** column on the **Materials** page in any analysis cases that linked to the deleted material displays **No** for the material that was deleted. The material properties contained in the material when it was in the **Materials Library** are available on the **Materials** page and can now be edited. The material can still be used in the analysis cases in which it was linked, but it will now be embedded in those analysis cases.



## Modifying a material in the Materials Library

When you change the name of a material or any of the properties for a material in the **Materials Library**, you have the choice of propagating the changes to every analysis case that contains a link to that material.

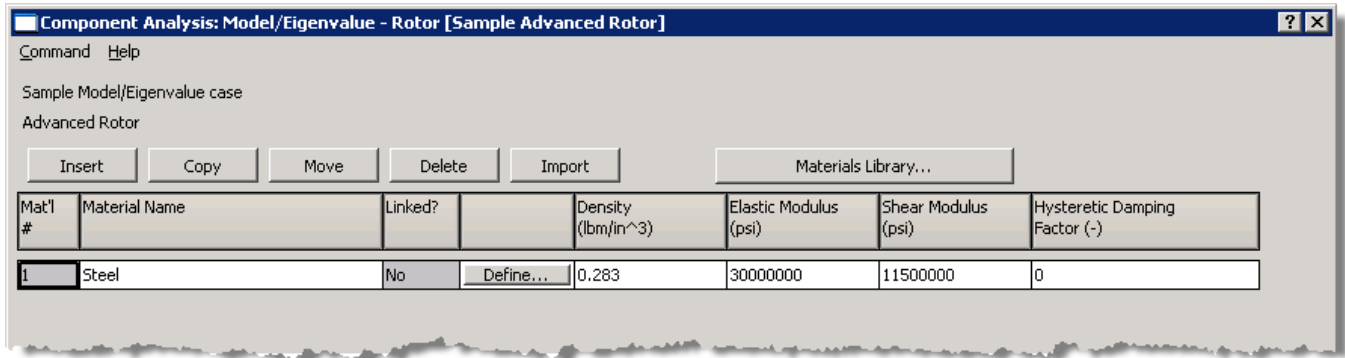


Each material you changed in the **Materials Library** is listed on the dialog box. For each material, select **Yes** or **No** from the drop-down menu in the **Preserve Links?** column to indicate whether all links should be preserved or not.

**NOTE:** You cannot preserve links to one material in some analysis cases and not other analysis cases. If you decide to not preserve links to a material in the list on the **Modified Materials Link Preservation** dialog box, all links to that material in all relevant analysis cases will be broken. This rule also applies to analysis cases inside archived projects.

## Using the Materials page

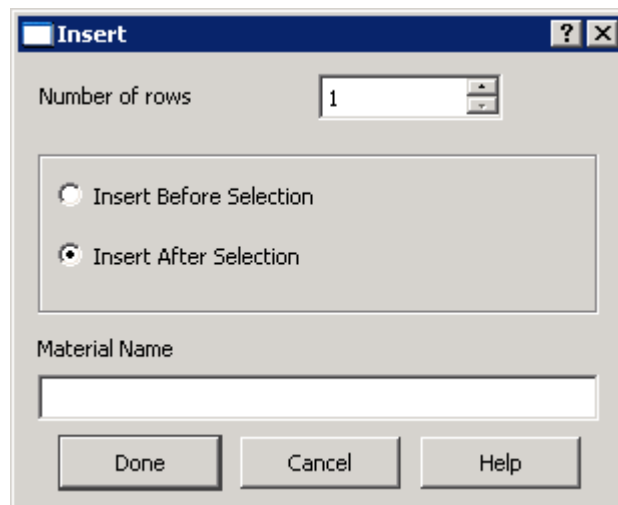
The **Materials** page is common for all analysis cases that use materials. Use the **Materials** page to define the material properties list that is available for developing a finite element model for the analysis case. The following figure displays an example of a **Materials** page from a *Model/Eigenvalue Rotor* analysis case.



The units specified for the current analysis case on the **Options** page are also applied to the material properties on the **Materials** page.

## Adding a material directly into an analysis case

Use the **Insert** button to add a new material to the **Materials** page.



If you want to save the new material in the current analysis case, use the columns of the grid to enter the name of the material and applicable properties for the material. Properties may include the following, depending on the current analysis case type:

- Density
- Elastic Modulus
- Shear Modulus



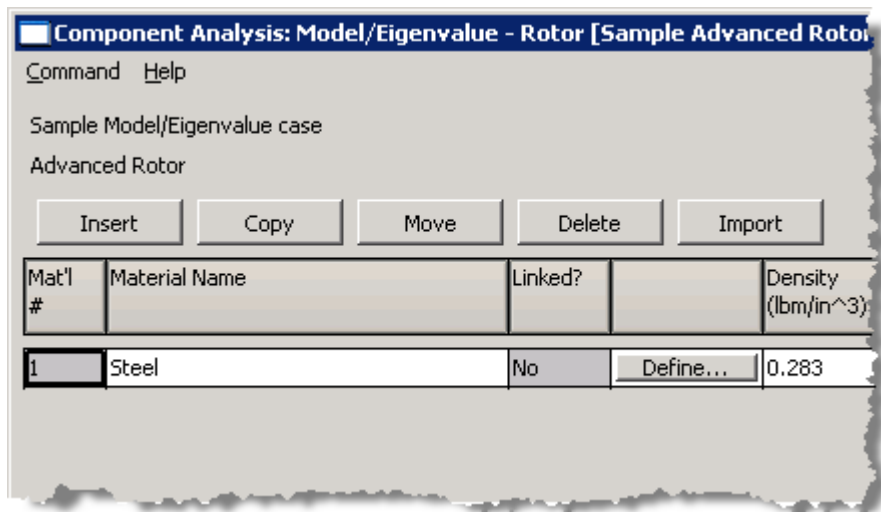
- Hysteretic Damping (currently not available)
- Thermal Conductivity
- Thermal Conductivity
- Thermal Expansion Coefficient

**WARNING:** When you embed a material in an analysis case, the material is only available to that analysis case. It cannot be used in other analysis cases unless you manually add and define it in other analysis cases. If you need to create materials for use in multiple analysis cases, add these materials to the Materials Library and link to them. See **Using the Materials Library** on page 80 and **Linking to a material from the Materials Library** on page 86.

## Linking to a material from the Materials Library

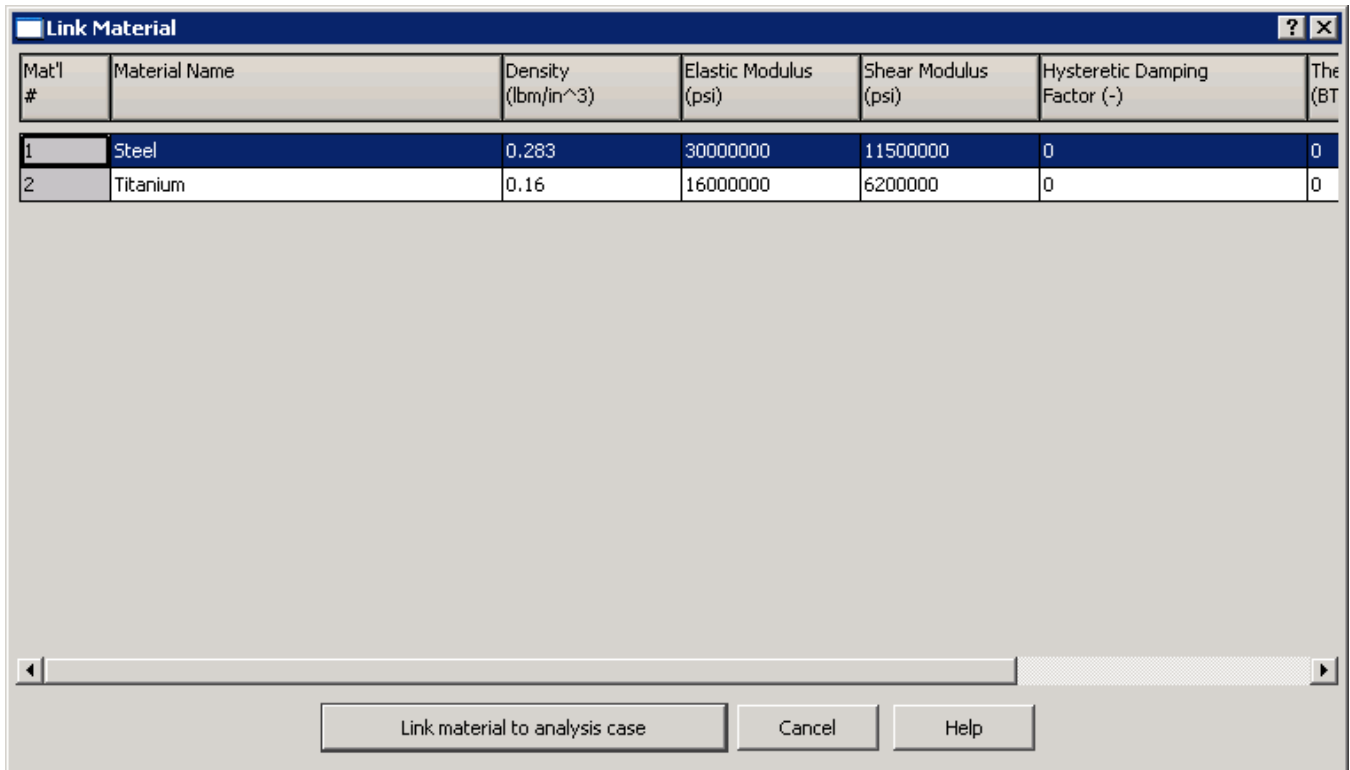
Use the **Linked?** column on the **Materials** page to use a material defined in the **Materials Library** within the current analysis case. When you link to a material, the values for the properties of that material are inserted into the data grid on the **Materials** page and cannot be edited on the **Materials** page. You must edit the material properties of a linked material in the **Materials Library**.

The **Linked?** column on the **Materials** page indicates whether the material is linked to a material in the **Materials Library**.



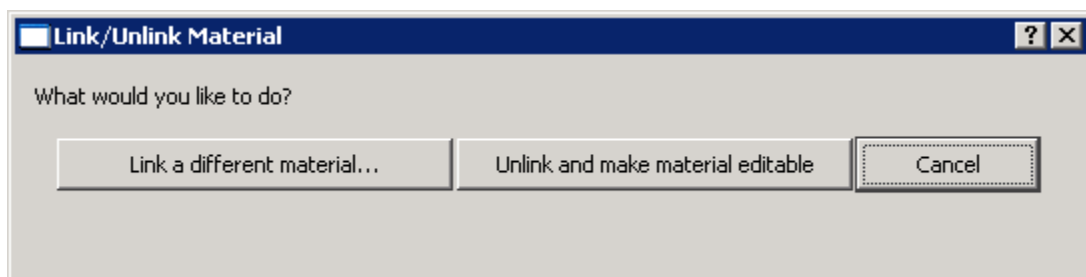
# RAPPID User's Guide

Use the **Define** button next to the **Linked?** column to select a material to which to link in the current analysis case. Select a material in the read-only list of materials on the **Link Material** dialog box and click the **Link material to analysis case** button.



## Linking to a different material in the Materials Library

To link to a different material for any material on the **Materials** page in an analysis case, click the **Define** button next to the **Linked?** column for the material for which you want to change the link.

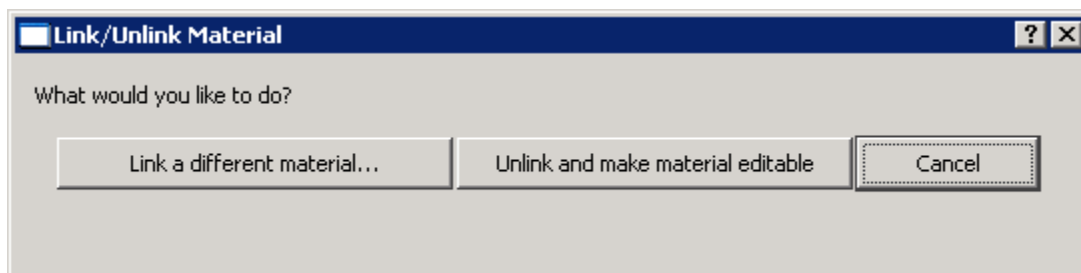


On the **Link/Unlink Material** dialog box, click the **Link a different material** button to bring up the **Link Material** dialog box and select another material. See **Linking to a material from the Materials Library** on page 86 for more information.

**NOTE:** When you link to a different material, the material name and the material properties for the selected material on the **Materials** page change to the values from the new material.

### Breaking a link to a material in the Materials Library

It may be desirable to break the link to, or unlink, a material within an analysis case. You may want to embed the material in the analysis case and/or change some properties of the material just for the current analysis case. To do this, click the **Define** button next to the **Linked?** column for the material you want to unlink.



Click the **Unlink and make material editable** button. When you unlink a material, the values obtained from the **Materials Library** when you linked the material are embedded in the analysis case. These values can be changed and are not affected by changes made to the same material in the **Materials Library**.

### Modifying a material on the Materials page

To modify a material on the **Materials** page, the material cannot be linked. If you want to edit a material that is linked to a material in the **Materials Library**, open the **Materials Library** to edit the material (see **Accessing the Materials Library** on page 80) or unlink the material to make it editable on the **Materials** page in the current analysis case (see **Breaking a link to a material in the Materials Library** on page 88).

If you unlink the material to make it editable on the **Materials** page, use the grid to edit the name and/or the material properties as applicable.

## Chapter 6: Working with fluids

Use the **Fluids Library** and the **Fluids** page to add fluids to analysis cases. Use one of the following methods to add a fluid to an analysis case. Some analysis cases can use more than one fluid.

- Use the **Fluids** page in an analysis case to add new fluids and store the fluid properties within the analysis case. The properties available on the **Fluids** page depends on the type of fluid you are adding to the analysis case.

You may add the following types of fluids to analysis cases:

- Incompressible
- Compressible
- Real Properties

**NOTE:** The **Fluids** page allows you to only add one fluid in some analysis case types and multiple fluids in other types.

See x-ref.

- Use the **Fluids Library** to add and store fluids for use in multiple analysis cases in any active project. On the **Fluids** page in an analysis case, link to a fluid from the **Fluids Library**.

See x-ref.

- Embed an existing fluid in an analysis case by linking to a material that exists in the **Fluids Library** and then unlinking the material.

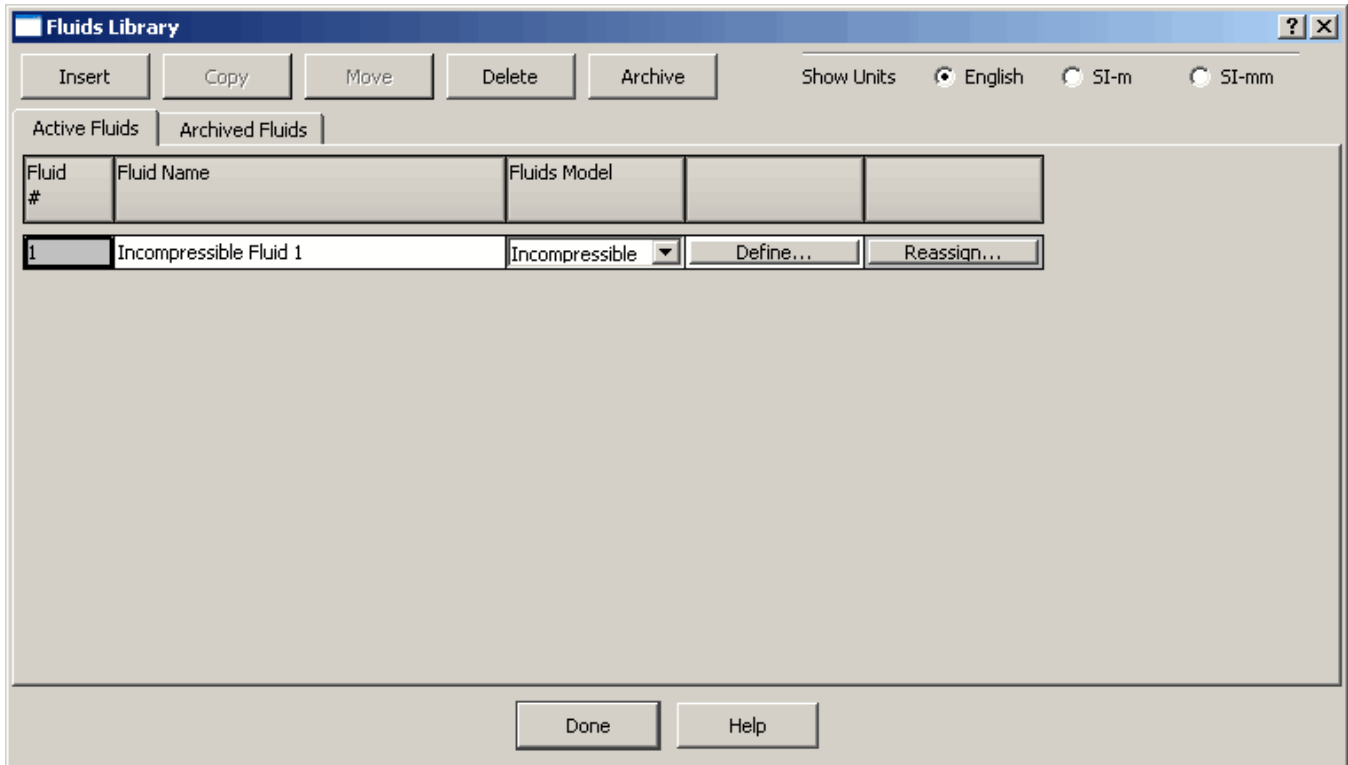
See x-ref.

**NOTE:** Real Properties fluids must be linked to analysis cases. They cannot be embedded.

**TIP:** Adding a fluid using this method is practical if you want to define a fluid similar to an existing fluid. You can obtain the values of the properties for the incompressible or compressible fluid from the **Fluids Library** by linking to the fluid. Once the values are imported onto the **Fluids** page from the **Fluids Library**, you can unlink the fluid, allowing you to change the values.

## Using the Fluids Library

Use the **Fluids Library** to define and store fluids for any fluid model (**Incompressible**, **Compressible**, **Real Properties**).



It is similar to the **Project Manager** (see **Chapter 2: Working with projects** on page 14) in that it has two tabs: **Active Fluids** and **Archived Fluids**.

There are five buttons above the two tabs: **Insert/Refresh**, **Copy**, **Move**, **Delete**, and **Archive**. Only the **Insert** button is active until a fluid has been added or un-archived.

## Accessing the Fluids Library

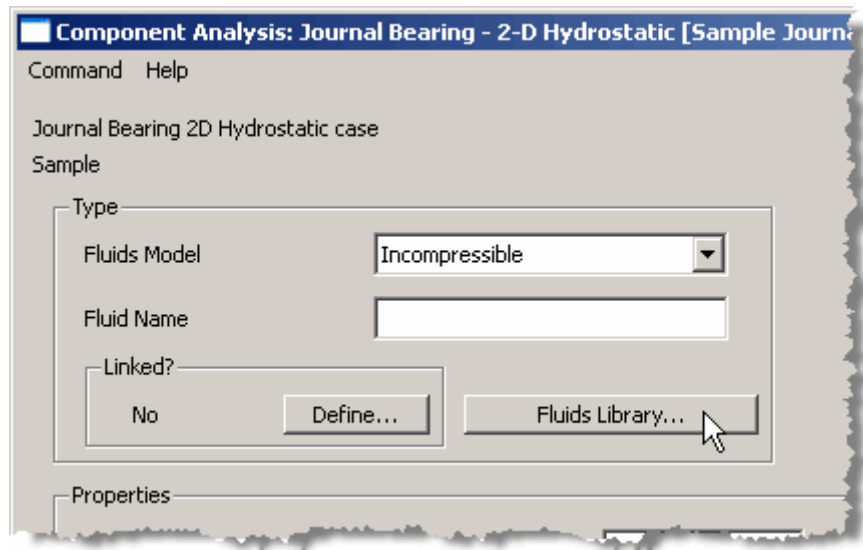
To access the **Fluids Library**, open an analysis case for a case type that uses fluids (such as *Bearings* or *Seals*).

**NOTE:** Make sure a case exists that uses fluids. If you do not, see **Creating a new analysis case** on page 22.

# RAPPID User's Guide

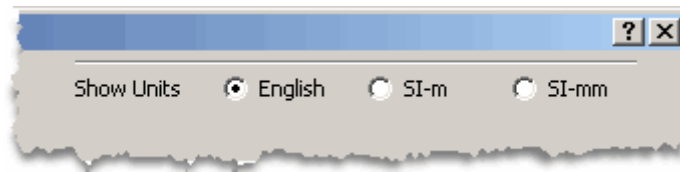
---

Navigate to the **Fluids** page and click the **Fluids Library** button.



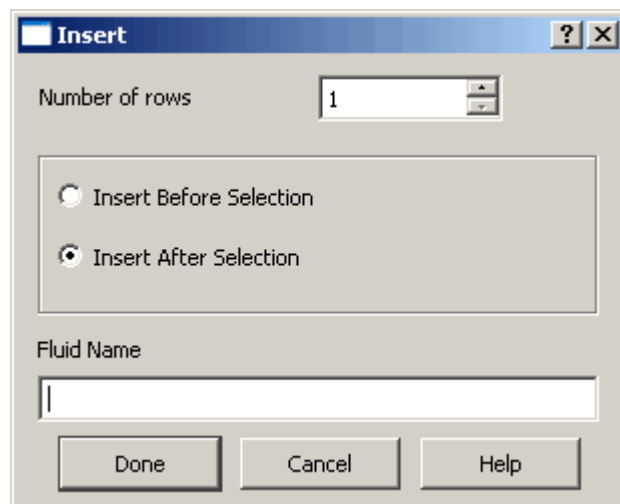
## Choosing the unit system to apply to all fluid properties

Use the **Show Units** radio buttons to select the unit system in which you want to enter the values of the properties for the materials.



## Adding a new fluid to the Fluids Library

Click the **Active Fluids** tab and then click the **Insert** button to add new fluids to the **Fluids Library** for use in multiple analysis cases.



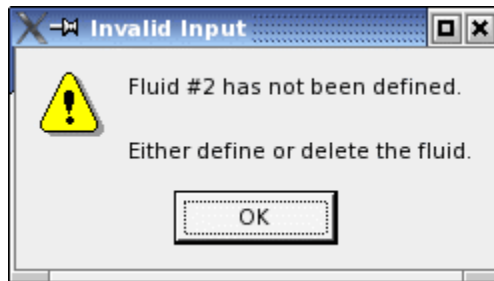
Use the **Fluid Name** edit box to specify a unique name for the new fluid and click **Done** to add the fluid.

In the **Fluids Library**, in the grid row that corresponds to the new fluid you added, use the **Fluid Model** drop-down menu to specify the type of fluid you are adding.

### Defining fluids in the Fluids Library

Use the **Define** button in the row corresponding to that fluid to enter values for the fluid properties or to select a pre-defined table for generating the fluid in the case of **Real Properties** fluids.

**NOTE:** When you add a new fluid, you must define it. You cannot reassign a fluid, close the **Fluids Library**, or archive a fluid, until you define or delete the added fluid. A dialog box displays informing you that you must define or delete the new fluid.



See **Linking a fluid** on the Fluids page for multiple fluids

The **Linked?** column displays the status indicating whether the selected fluid is linked to a fluid in the **Fluids Library**.

For **Incompressible** and **Compressible** fluids, the value in the **Linked?** column is 'Yes' or 'No.' For **Real Properties** fluids, the **Linked?** column displays 'Undefined' until you define the fluid.

**NOTE:** You cannot run a *Transient Flow* network case if you have any **Real Properties** fluids on the **Fluids** page that are not defined.

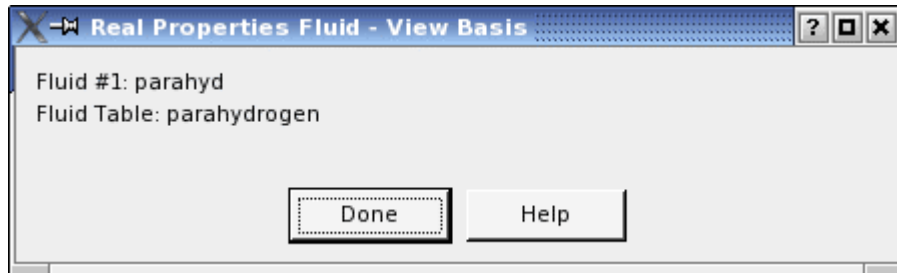
Use the **Define** button in the column next to the **Linked?** column in the row for the fluid to be defined to link to a fluid in the **Fluids Library**.

**NOTE:** The Link Fluid dialog box displays all fluids for all types of fluids. However, you can only link to a fluid in the **Fluids Library** of the same type that you have selected in the **Fluid Model** column. If you select a different type of fluid, the selection in the **Fluid Model** column changes to match the type of fluid you selected in the **Fluids Library**.

**Defining** fluid properties on page 101 for information about the fluid properties available to define for each type of fluid model.

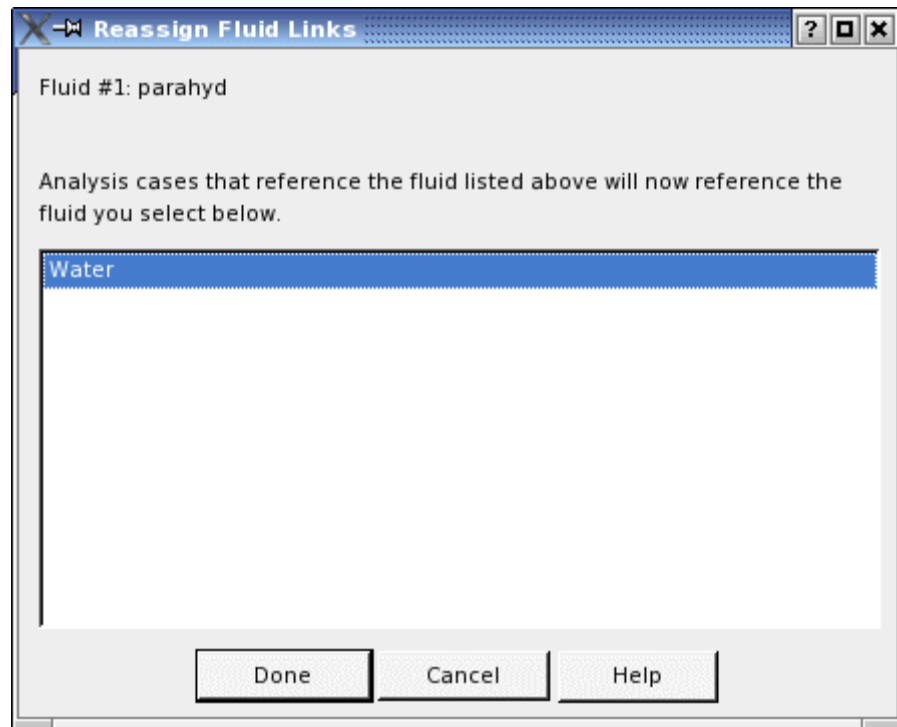
## Viewing the basis of a generated Real Properties fluid

Once you have defined (generated) a **Real Properties** fluid, it cannot be redefined. The **Define** button for the generated fluid becomes the **View Basis** button. The **View Basis** dialog box displays the name of the fluid table being used for the selected fluid.



## Reassigning a Real Properties fluid

Use the **Reassign** button to change the all links to a specific fluid to link to a different fluid.



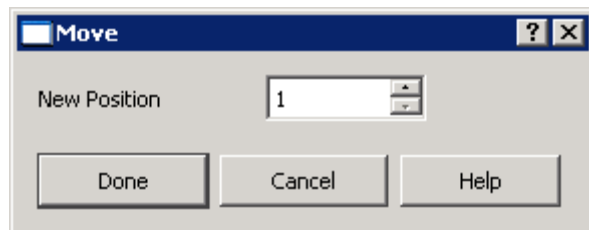
For example, you have 2 analysis cases that reference Fluid A and you add Fluid B to the **Fluids Library**. Fluid B is very similar to Fluid A and you want the 2 cases that currently link to Fluid A to now link to Fluid B. Use the **Reassign** button for Fluid A to change all links currently linking to Fluid A to link to Fluid B.

**WARNING:** When you reassign the link to a fluid, ALL analysis cases that reference the fluid are modified. Reassigning a fluid link applies to ALL your active projects.



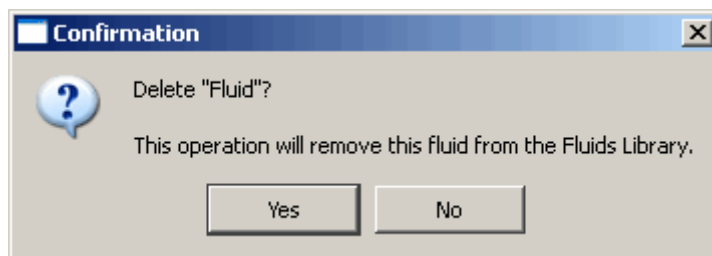
## Moving a fluid in the Fluids Library

Click the **Active Fluids** tab and click the **Move** button to move an active fluid to another position in the list.

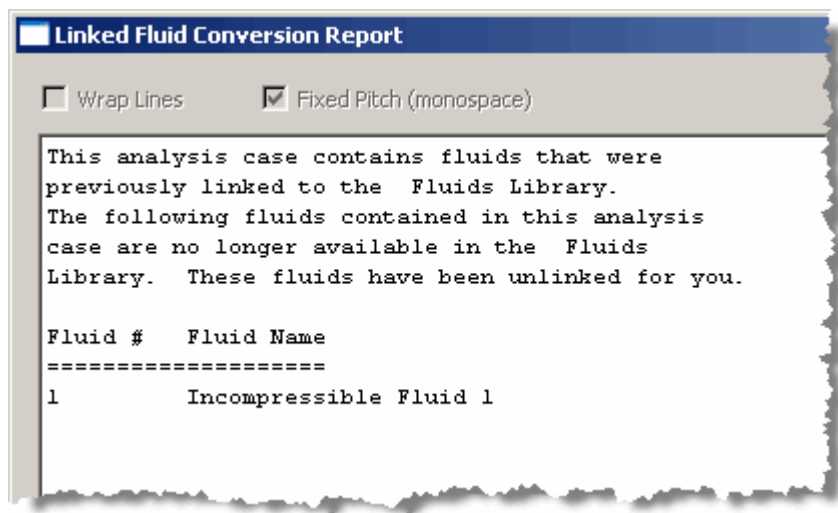


## Deleting a fluid in the Fluids Library

Use the **Delete** button to delete a selected fluid from the **Fluids Library**.



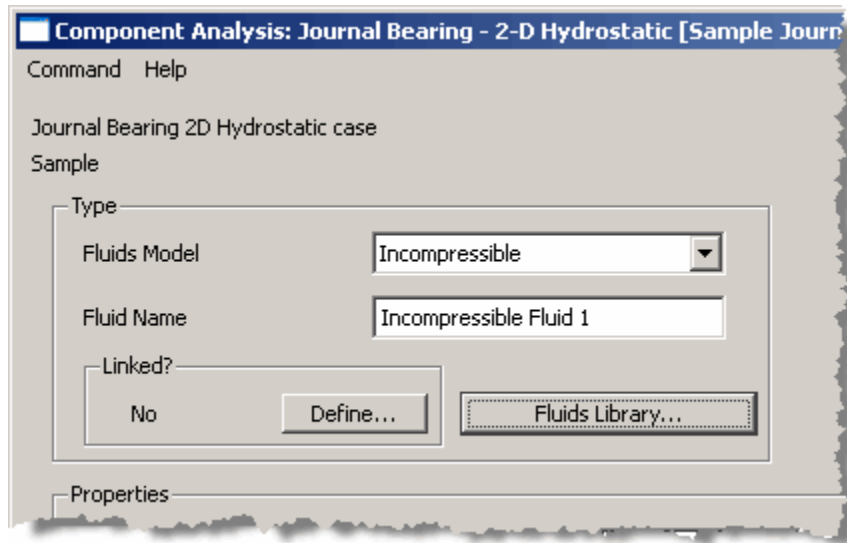
If you delete a fluid from or change a fluid's properties in the **Fluids Library**, all cases that link to that fluid are affected. If you are currently in an analysis case when you delete or modify a fluid or if you open an analysis case that references a fluid you deleted or modified, the **Linked Fluid Conversion Report** displays listing the material that was deleted and informing you that the material has been unlinked for you.



If multiple fluids are deleted, all fluids that are deleted are unlinked in all analysis cases that linked to the fluids. All fluids that are deleted are listed in the **Linked Fluid Conversion Report**.

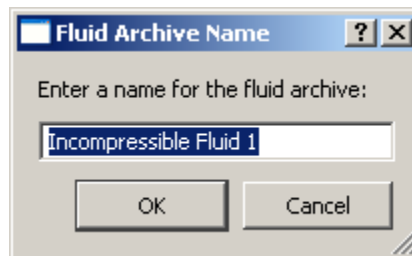
# RAPPID User's Guide

When you delete a fluid, the **Linked?** column on the **Fluids** page in any analysis cases that linked to the deleted fluid displays **No** for the fluid that was deleted. The fluid properties contained in the fluid when it was in the **Fluids Library** are available on the **fluids** page and can now be edited. The fluid can still be used in the analysis cases in which it was linked, but it will now be embedded in those analysis cases.

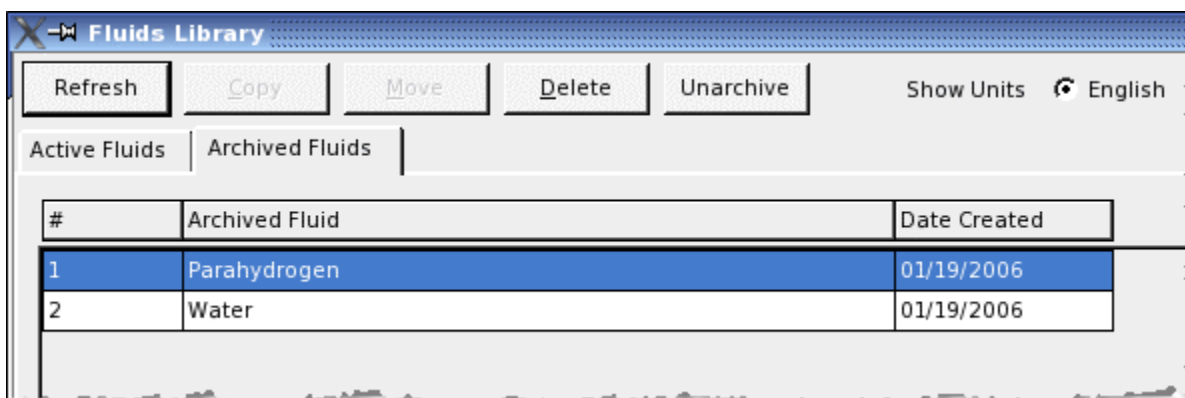


## Archiving a fluid in the Fluids Library

Click the **Archived Fluids** tab and click the **Archive** button to archive an active fluid.



RAPPID™ provides the ability to archive a fluid. The **Archived Fluids** tab displays all fluids that have been archived. The name of the fluid displays with the date the archive was created.



---

## RAPPID User's Guide

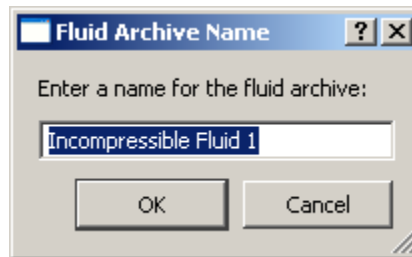
---

**NOTE:** Only one fluid may be archived at a time. You cannot make one archive of all your fluids.

You may want to archive a fluid for various reasons such as:

- You want to have a backup of a fluid.
- You want to move a copy of a fluid to another system.
- You want to preserve a copy of a fluid before you make substantial changes to the fluid.

To archive a fluid, click the **Archived Fluids** tab to make it active and click the **Archive** button. Enter a unique name for the fluid archive.



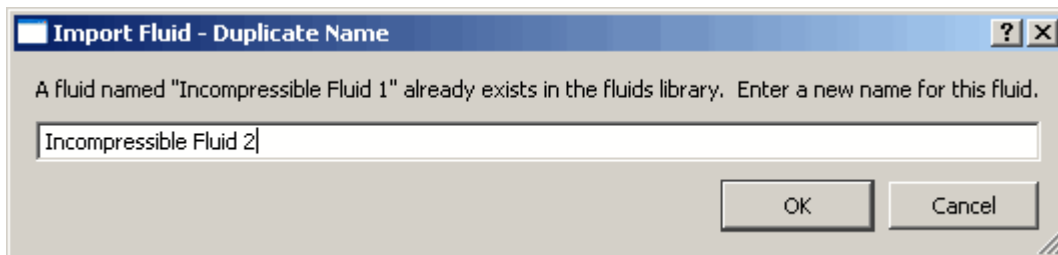
**NOTE:** Archiving a fluid DOES NOT remove the fluid from the **Active Projects** tab. You must delete a fluid manually. See **Deleting a fluid in the Fluids Library** on page 94.

To use an archived fluid, use the **Unarchive** button to unarchive the project and place it on the **Active Fluids** tab. If there are already fluids listed on the **Active Fluids** tab, the fluid is added to the end of the list of fluids. See **Unarchiving a fluid in the Fluids Library** on page 96 for more information.

If you have an archived fluid from another system or another location on your hard drive outside of your specified archive directory, you can add it to the **Archived Fluids** tab manually while the Program is open. Add it to your specified archive directory and use the **Refresh** button to access in the list on the **Archived Projects** tab.

### Unarchiving a fluid in the Fluids Library

Click the **Archived Fluids** tab and click the **Unarchive** button to unarchive an archived fluid and add it to the **Active Fluids** tab.



If a fluid currently exists in the **Fluids Library** with the same name as the archived fluid, you must enter a new name for the fluid being unarchived.

## Using the Fluids page for adding a single fluid

The **Fluids** page for a single fluid is a dynamic page with entries that vary according to the selected **Fluid Model: Incompressible, Compressible, or Real Properties**.

**NOTE:** The only available fluid type in *Bearings - 2-D Tilting Pad* and *Bearings - 2-D Fixed Geometry* cases (*Journal* and *Thrust*) is **Incompressible** and you cannot link to a fluid.

Use the **Fluids** page to manually define fluid properties to use only for the current case, define properties for a fluid in the **Fluids Library**, or retrieve a previously defined fluid from the **Fluids Library**.

**NOTE:** Currently, you can only add one fluid to component analysis cases.

The following figure displays the **Fluids** page in a component analysis case with **Compressible** selected as the **Fluid Model**.

The screenshot shows the 'Component Analysis: Journal Bearing - 2-D Hydrostatic [Sample Journal Bearing 2D Hydrostatic case]' window. The 'Type' section has 'Fluids Model' set to 'Incompressible' and 'Fluid Name' set to 'Incompressible Fluid 1'. The 'Linked?' section has 'No' selected. The 'Properties' section contains input fields for Density (lbm/ft^3), Thermal Expansion Coef. (1/R), Bulk Modulus (psi), Specific Heat (BTU/lbm-R), Thermal Conductivity (BTU/h-ft-R), and Cavitation Pressure (psi), all set to 0. The 'Temperature (F)' and 'Viscosity (Reyns)' section has input fields for '1st Temp./Viscosity Point' and '2nd Temp./Viscosity Point', both set to 0. The 'Page' dropdown is set to 'Fluids'. The bottom navigation bar includes buttons for '< Previous', 'Next >', 'Done', 'Run', 'View', 'Plot', 'Cancel', and 'Help'.

## Specifying the type of fluid and the fluid name

Use the **Type** box to specify the type of fluid you are defining for the current analysis case.

**NOTE:** The only available fluid type in *Bearings - 2-D Tilting Pad* and *Bearings - 2-D Fixed Geometry* cases (*Journal* and *Thrust*) is **Incompressible** and you cannot link to a fluid. Therefore, the **Type** box is not available in these types of cases.

# RAPPID User's Guide

Use the **Fluids Model** drop-down menu to specify the model (**Incompressible**, **Compressible**, **Real Properties**) for the fluid to be used in the current analysis case. Selecting a fluid model displays the appropriate fields for input in the **Properties** box on the **Fluids** page.

Use the **Fluid Name** edit box to enter a unique name for the group of properties entered. The **Fluid Name** is optional; however, it is strongly recommended that you enter a name for your fluid. This name is used within the analysis case for identifying the fluid and also displays in the viewed output after the analysis case has been run.

Valid entries for the **Fluid Name** are text strings less than 40 characters.

## Defining a fluid on the Fluids page for a single fluid

**Incompressible** and **Compressible** fluids can be defined manually within an analysis case in the **Properties** box on the **Fluids** page. However, **Real Properties** fluids must be linked. See **Defining a Real Properties fluid** on page 107 and for more information about defining **Real Properties** fluids.

To define a fluid directly in an analysis case on the **Fluids** page, enter the fluid properties into the edit boxes in the **Properties** box.

Component Analysis: Journal Bearing - 2-D Hydrostatic [Sample Journal Bearing 2D Hydrostatic case]

Command Help

Journal Bearing 2D Hydrostatic case

Sample

Type

Fluids Model: Incompressible

Fluid Name: Incompressible Fluid 1

Linked?

No Define... Fluids Library...

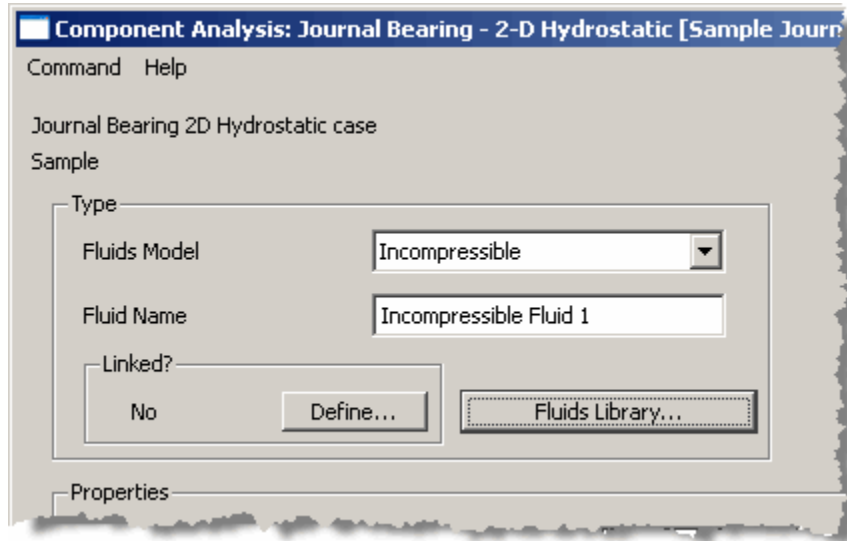
Properties

		Temperature (F)	Viscosity (Reyns)
Density (lbm/ft <sup>3</sup> )	0		
Thermal Expansion Coef. (1/R)	0	1st Temp./Viscosity Point: 0	0
Bulk Modulus (psi)	0	2nd Temp./Viscosity Point: 0	0
Specific Heat (BTU/lbm-R)	0		
Thermal Conductivity (BTU/h-ft-R)	0		
Cavitation Pressure (psi)	0		

Page: Fluids < Previous Next > Done Run View Plot Cancel Help

## Linking a fluid on the Fluids page for single fluids

The **Linked?** box displays the status indicating whether the selected fluid is linked to a fluid in the **Fluids Library**.



For **Incompressible** and **Compressible** fluids, the value in the **Linked?** box is 'Yes' or 'No.' For **Real Properties** fluids, the **Linked?** box displays 'Undefined' until you define the fluid.

Use the **Define** button in the **Linked?** box to link to a fluid in the **Fluids Library**.

**NOTE:** The **Link Fluid** dialog box displays all fluids for all types of fluids. However, you can only link to a fluid in the **Fluids Library** of the same type that you have selected in the **Fluid Model** column. If you select a different type of fluid, the selection in the **Fluid Model** column changes to match the type of fluid you selected in the **Fluids Library**.

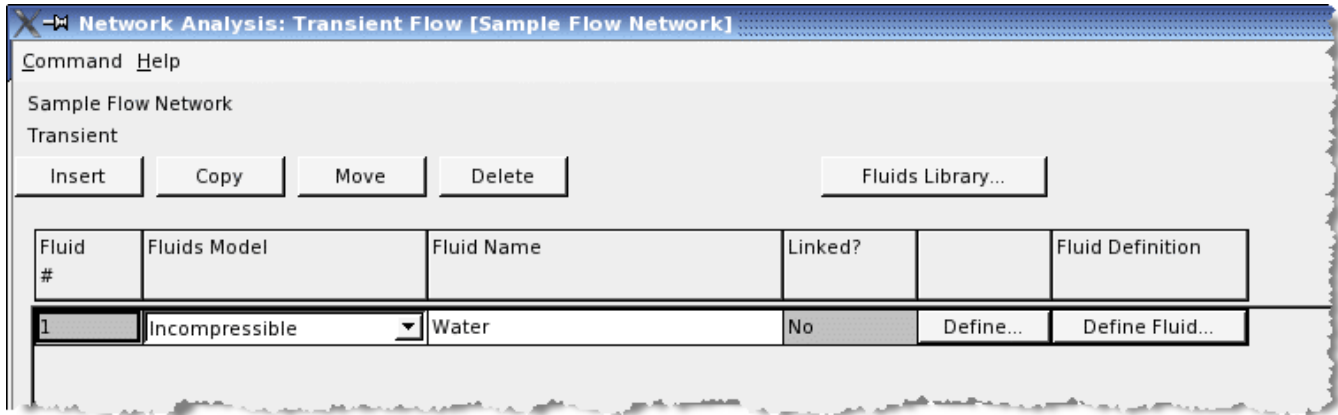
## Using the Fluids page for adding multiple fluids

Network analysis cases that use fluids allow you to add multiple fluids. Use the **Fluids** page to manually add one or more fluids to use only for the current case, define properties for one or more fluids in the **Fluids Library**, or retrieve previously defined fluids from the **Fluids Library**. The Fluids page for adding multiple fluids contains a data grid for defining one or more fluids.

**NOTE:** Currently, the **Fluids** page for adding multiple fluids is only available for *Transient Flow Network*.

# RAPPID User's Guide

The following figure displays the **Fluids** page in a *Transient Flow Network* case with **Incompressible** selected as the **Fluid Model**.



**NOTE:** See **Using a grid to enter data** on page 40 for information on entering data using a data grid.

## Specifying the type of fluid and the fluid name

Use the drop-down menu in the **Fluids Model** column to select the model (**Incompressible**, **Compressible**, **Real Properties**) for the fluid to be used in the current analysis case.

**NOTE:** When you choose **Real Properties** as the **Fluids Model**, the **Fluid Name** column for that fluid becomes unavailable, as well as the **Define Fluid** button.

Use **Fluid Name** to enter a name for the fluid. The **Fluid Name** column is only available for **Incompressible** and **Compressible** fluids, and is optional. However, it is strongly recommended that you enter a name for your fluid. This name is used within the analysis case for identifying the fluid and also displays in the viewed output after the analysis case has been run.

Valid entries are text strings less than 40 characters.

## Defining a fluid on the Fluids page for multiple fluids

Use the **Define Fluid** button in the **Fluid Definition** column to define fluid properties to be stored within the analysis case.

**NOTE:** The **Define Fluid** button is only available for **Incompressible** and **Compressible** fluids. A **Real Properties** fluid in an analysis case must be linked to a **Real Properties** fluid in the **Fluids Library**.

## RAPPID User's Guide

The **Define Fluid** dialog box displays input fields for the properties applicable to the type of fluid you selected. The following figure displays the **Define Fluid** dialog box for an **Incompressible** fluid.

Properties		
Density (kg/m <sup>3</sup> )	997.14	
Thermal Expansion Coef. (1/K)	0.00025751	
Bulk Modulus (Bar)	22112.6	
Specific Heat (kJ/kg-K)	4.1807	
Thermal Conductivity (W/m-K)	0.6728	
Cavitation Pressure (Bar)	0	
	Temperature (C)	Viscosity (cP)
1st Temp./Viscosity Point	20	0.89004
2nd Temp./Viscosity Point	40	1.14

Buttons: Done, Cancel, Help

**NOTE:** For information about the fields available on the **Define Fluid** dialog box for **Incompressible** and **Compressible** fluids, see Error! Reference source not found. on page **Error! Bookmark not defined.**

### Linking a fluid on the Fluids page for multiple fluids

The **Linked?** column displays the status indicating whether the selected fluid is linked to a fluid in the **Fluids Library**.

For **Incompressible** and **Compressible** fluids, the value in the **Linked?** column is 'Yes' or 'No.' For **Real Properties** fluids, the **Linked?** column displays 'Undefined' until you define the fluid.

**NOTE:** You cannot run a *Transient Flow* network case if you have any **Real Properties** fluids on the **Fluids** page that are not defined.

Use the **Define** button in the column next to the **Linked?** column in the row for the fluid to be defined to link to a fluid in the **Fluids Library**.

**NOTE:** The **Link Fluid** dialog box displays all fluids for all types of fluids. However, you can only link to a fluid in the **Fluids Library** of the same type that you have selected in the **Fluid Model** column. If you select a different type of fluid, the selection in the **Fluid Model** column changes to match the type of fluid you selected in the **Fluids Library**.



## Defining fluid properties

Use the **Define** button either on the **Fluids** page in an analysis case or in the **Fluids Library** to define the fluid properties for a fluid.

## Defining properties for an Incompressible fluid

The following properties are available when defining Incompressible fluids in the **Fluids Library** or on the **Fluids** page:

### Density

The following rule applies to the **Density** field:

- Valid entries are positive real numbers.

**NOTE:** **Density** is only available for **Incompressible** fluids.

### Thermal Expansion Coef.

The following rule applies to the **Thermal Expansion Coef.** field:

- Valid entries are positive real numbers.

**NOTE:** **Thermal Expansion Coef.** is only available for **Incompressible** fluids.

### Bulk Modulus

Use **Bulk Modulus** to specify the bulk compressibility modulus of the fluid for the supply pressure and temperature. This is also known as the modulus of elasticity of the fluid. Values of the bulk modulus for several fluids are given in the following table:

Liquid	Isentropic Bulk Modulus* (GN/m <sup>2</sup> )	Specific Gravity (-)
Benzene	1.48	0.879
Carbon tetrachloride	1.36	1.595
Castor oil	2.11	0.969
Gasoline	--	0.72
Glycerin	4.59	1.26

## RAPPID User's Guide

Heptane	0.886	0.684
Kerosine	1.43	0.82
Lubricating oil	1.44	0.88
Mercury	28.5	13.55
Octane	0.963	0.702
Sea-water	2.42	1.025
Water	2.24	0.998

\* Calculated from speed of sound; 1 GN/m<sup>2</sup> = 10<sup>9</sup> N/m<sup>2</sup> (1 N/m<sup>2</sup> = 1.45 x 10<sup>-4</sup> lbf/in.<sup>2</sup>)

If the fluid of interest is not shown in the table, the bulk modulus can be calculated from the definition:

$$\text{Bulk Modulus} = \frac{\rho dp}{d\rho}$$

where:

$p$  = pressure

$\rho$  = fluid density

Values for the equation can be obtained from property tables.

### Thermal Conductivity

The following rule applies to the **Thermal Conductivity** field:

- Valid entries are positive real numbers.

**NOTE:** **Thermal Conductivity** is available for both **Incompressible** and **Compressible** fluids.

### Cavitation

Use **Cavitation Pressure** to define the minimum pressure that may exist in the fluid. When the Program's steady state solution predicts a pressure lower than the

# RAPPID User's Guide

---

## Pressure

specified cavitation pressure, the pressure at the subject angular location will automatically be set to the specified cavitation pressure.

**NOTE: Cavitation Pressure** is only available for **Incompressible** fluids.

The following rule applies to the **Cavitation Pressure** field:

- Valid entries are real numbers.

Generally, the cavitation pressure should be set to the same value as the ambient pressure. However, lower values are possible and may be utilized if the user is in possession of appropriate data.

**NOTE:** The **Cavitation Pressure**, ambient pressure (**Left Boundary Pressure** and **Right Boundary Pressure**), and **Supply Pressure** are relative to each other. As such, they may be entered in either gauge or absolute values as long as all three are entered in consistent terms (do not specify some pressures in gauge and others in absolute).

## Defining properties for a Compressible fluid

The following properties are available when defining Compressible fluids in the **Fluids Library** or on the **Fluids** page:

### Gas Constant

Use **Gas Constant** to specify the gas constant for the working fluid. The gas constant for any gas can be calculated using the following formula:

$$R_g = \frac{\bar{R}}{MW}$$

where:

$$\bar{R} = \text{Universal gas constant} = 1.986 \text{ BTU/lb-mol/R} = 8314 \text{ kJ/kg-mol/K}$$

MW = Molecular weight of the gas

Molar mass and ratio of specific heats for various gases are listed in the following table:

Gas	Molar Mass	$\gamma$
Air	28.96	1.400

## RAPPID User's Guide

Argon	39.94	1.667
Carbon Dioxide	44.01	1.285
Carbon Monoxide	28.01	1.399
Helium	4.003	1.667
Hydrogen	2.018	1.404
Methane	16.04	1.320
Nitrogen	28.008	1.400
Oxygen	32.00	1.395
Steam	18.02	1.329

**NOTE: Cavitation Pressure** if only available for **Compressible** fluids.

### Compressibility Factor

Use **Compressibility Factor** to measure how closely the gas adheres to the ideal gas law. It is a dimensionless number that is determined from the following equation:

$$Z = \frac{P}{\rho R_g T}$$

where:

P = static pressure

$\rho$  = density

T = static temperature

A generalized compressibility factor chart which covers most gases can be found in most textbooks on thermodynamics. To use the chart the reduced pressure and reduced temperature of the gas must be calculated. The reduced value is defined as the value divided by the critical value. The critical pressure and temperatures

---

## RAPPID User's Guide

---

for various gases are given in the following table:

Gas	Pressure (Bar)	Temperature (K)
Air	37.70	132.5
CO <sub>2</sub>	73.80	304.2
CO	35.00	133.0
Cl <sub>2</sub>	77.10	417.0
CH <sub>4</sub>	46.00	190.6
Helium	2.30	5.300
Hydrogen	13.00	33.3
Nitrogen	33.90	126.2
Steam	220.90	647.3

**NOTE: Compressibility Factor** is only available for **Compressible** fluids.

### Ratio of Specific Heats

Use **Ratio of Specific Heats** to specify the specific heats for the gas. This value is dimensionless and is defined by:

$$\gamma = \frac{C_p}{C_v}$$

where  $C_p$  and  $C_v$  are the constant pressure and constant volume specific heats, respectively. The specific heats and the gas constant have the following relationships:

$$C_p = \frac{\gamma R g}{\gamma - 1}$$
$$C_v = \frac{R g}{\gamma - 1}$$
$$C_p - C_v = R g$$

**NOTE: Ratio of Specific Heats** is only available for **Compressible** fluids.

### Thermal Conductivity

The following rule applies to the **Thermal Conductivity** field:

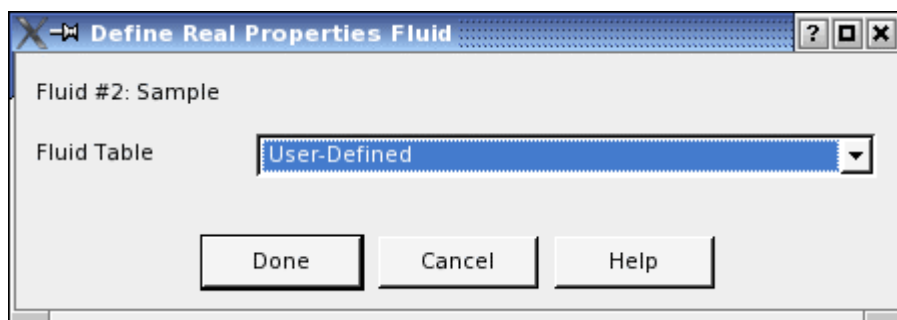
- Valid entries are positive real numbers.

**NOTE: Thermal Conductivity** is available for both **Incompressible** and **Compressible** fluids.

## Defining a Real Properties fluid

There are no properties to define when creating a Real Properties fluid.

When you choose **Real Properties** in the **Fluid Model** dialog box and click the **Define** button, the **Define Real Properties Fluid** dialog box displays allowing you to select the fluid table to be used to generate the fluid.



The **Fluid Table** drop-down menu contains pre-defined fluid tables available for use in generating Real Properties fluids in addition to a **User-Defined** option.

### *Pre-defined fluid table files*

If you want to use a pre-defined fluid table to generate a **Real Properties** fluid, use the **Fluid Table** drop-down menu to select one of the following pre-defined fluids:

- Ammonia
- Argon
- Benzene

- Butane (n-butane)
- C<sub>4</sub>F<sub>10</sub> (decafluorobutane)
- C<sub>5</sub>F<sub>12</sub> (dodecafluoropentane)
- Carbon Monoxide
- Carbon Dioxide
- Carbonyl Sulfide (carbon oxide sulfide)
- Cyclohexane
- Cyclopropane
- Deuterium
- Heavy Water (deuterium oxide)
- Decane
- Dimethylether
- Ethane
- Ethanol (ethyl alcohol)
- Ethylene (ethane)
- Fluorine
- Hydrogen Sulfide
- Helium (helium-4)
- Heptane
- Hexane
- Hydrogen (normal)
- Isopentane (2-methylbutane)
- Isobutane (2-methylpropane)
- Krypton
- Methane

- Methanol
- Nitrous Oxide (dinitrogen monoxide)
- Neon
- Neopentane (2,2-dimethylpropane)
- Nitrogen Trifluoride
- Nitrogen
- Nonane
- Octane
- Oxygen
- Parahydrogen
- Pentane
- Propane
- Propylene (propene)
- Propyne
- R11 (trichlorofluoromethane)
- R12 (dichlorodifluoromethane)
- R13 (chlorotrifluoromethane)
- R14 (tetrafluoromethane)
- R21 (dichlorofluoromethane)
- R22 (chlorodifluoromethane)
- R23 (trifluoromethane)
- R32 (difluoromethane)
- R41 (fluoromethane)
- R113 (1,1,2-trichloro-1,2,2-trifluoroethane)
- R114 (1,2-dichloro-1,1,2,2-tetrafluoroethane)



- R115 (chloropentafluoroethane)
- R116 (hexafluoroethane)
- R123 (2,2-dichloro-1,1,1-trifluoroethane)
- R124 (1-chloro-1,2,2,2-tetrafluoroethane)
- R125 (pentafluoroethane)
- R134a (1,1,1,2-tetrafluoroethane)
- R141b (1,1-dichloro-1-fluoroethane)
- R142b (1-chloro-1,1-difluoroethane)
- R143a (1,1,1-trifluoroethane)
- R152a (1,1-difluoroethane)
- R218 (octafluoropropane)
- R227ea (1,1,1,2,3,3,3-heptafluoropropane)
- R236ea (1,1,1,2,3,3-hexafluoropropane)
- R236fa (1,1,1,3,3,3-hexafluoropropane)
- R245ca (1,1,2,2,3-pentafluoropropane)
- R245fa (1,1,1,3,3-pentafluoropropane)
- RC318 (octafluorocyclobutane)
- Sulfur Hexafluoride
- Sulfur Dioxide
- Toluene (methylbenzene)
- Water
- Xenon

### *User-Defined fluid table files*

If you want to define a fluid yourself, select User-Defined from the Fluid table drop-down menu. Create four ASCII text files formatted using the following four structures of data, separating values with one space:

## RAPPID User's Guide

---

**Pressure/Temperature (PT)** Use the following format for your user-defined **PT** fluid tables:

```
Pressure Increment | Temperature Increment
Min Pressure | Max Pressure | Min Temperature |
Max Temperature
12 columns of values, separated by spaces - see
list below
```

The values for the 12 columns should be entered from left to right in the order listed below, with the required units:

Value	Units
Enthalpy	$\frac{J}{Kg}$
Density	$\frac{Kg}{m^3}$
Internal Energy	$\frac{J}{Kg}$
Entropy	$\frac{J}{Kg K}$
Specific Heat at Constant Pressure	$\frac{J}{Kg K}$
Specific Heat at Constant Volume	$\frac{J}{Kg K}$
Quality	Enter 999 for this value; it is not currently used, but a value needs to be entered.

## RAPPID User's Guide

---

Speed of Sound	$\frac{m}{s}$
Compressibility	$\frac{1}{Pa}$
Conductivity	$\frac{W}{mK}$
Dynamic Viscosity	$Pa-s$
Thermal Expansion Coefficient	$\frac{1}{K}$

### Pressure/Enthalpy (PH)

Use the following format for your user-defined **PH** fluid tables:

```
Pressure Increment | Enthalpy Increment
Min Pressure | Max Pressure | Min Enthalpy |
Max Enthalpy
12 columns of values, separated by spaces - see
list below
```

The values for the 12 columns should be entered from left to right in the order listed below, with the required units:

Value	Units
Temperature	$K$
Density	$\frac{Kg}{m^3}$
Internal Energy	$\frac{J}{Kg}$

## RAPPID User's Guide

---

Entropy	$\frac{J}{KgK}$
Specific Heat at Constant Pressure	$\frac{J}{KgK}$
Specific Heat at Constant Volume	$\frac{J}{KgK}$
Quality	Enter 999 for this value; it is not currently used, but a value needs to be entered.
Speed of Sound	$\frac{m}{s}$
Compressibility	$\frac{1}{Pa}$
Conductivity	$\frac{W}{mK}$
Dynamic Viscosity	$Pa-s$
Thermal Expansion Coefficient	$\frac{1}{K}$

### Pressure/Entropy (PS)

Use the following format for your user-defined **PS** fluid tables:

```
Pressure Increment | Entropy Increment
Min Pressure | Max Pressure | Min Enthalpy |
Max Enthalpy
12 columns of values, separated by spaces - see
list below
```

The values for the 12 columns should be entered from left to right in the order listed below, with the required units:

## RAPPID User's Guide

---

Value	Units
Temperature	$K$
Density	$\frac{Kg}{m^3}$
Internal Energy	$\frac{J}{Kg}$
Enthalpy	$\frac{J}{Kg}$
Specific Heat at Constant Pressure	$\frac{J}{Kg K}$
Specific Heat at Constant Volume	$\frac{J}{Kg K}$
Quality	Enter 999 for this value; it is not currently used, but a value needs to be entered.
Speed of Sound	$\frac{m}{s}$
Compressibility	$\frac{1}{Pa}$
Conductivity	$\frac{W}{mK}$
Dynamic Viscosity	$Pa - s$

## RAPPID User's Guide

Thermal Expansion Coefficient	$\frac{1}{K}$
-------------------------------	---------------

### Density/Internal Energy (DE)

Use the following format for your user-defined **DE** fluid tables:

Density Increment | Internal Energy Increment  
 Min Density | Max Density | Min Internal Energy  
 | Max Internal Energy  
 12 columns of values, separated by spaces - see list below

The values for the 12 columns should be entered from left to right in the order listed below, with the required units:

Value	Units
Temperature	$K$
Pressure	$Pa$
Enthalpy	$\frac{J}{Kg}$
Entropy	$\frac{J}{Kg K}$
Specific Heat at Constant Pressure	$\frac{J}{Kg K}$
Specific Heat at Constant Volume	$\frac{J}{Kg K}$
Quality	Enter 999 for this value; it is not currently used, but a value needs to be entered.

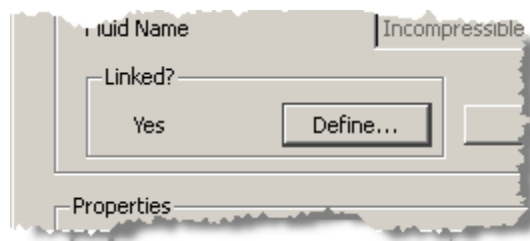
## RAPPID User's Guide

Speed of Sound	$\frac{m}{s}$
Compressibility	$\frac{1}{Pa}$
Conductivity	$\frac{W}{mK}$
Dynamic Viscosity	$Pa-s$
Thermal Expansion Coefficient	$\frac{1}{K}$

### Linking to fluids in the Fluids Library

To use a fluid defined in the **Fluids Library**, link to the fluid on the **Fluids** page within the current analysis case. When you link to a fluid, the values for the properties of that fluid are inserted into the data grid on the **Fluids** page.

The **Linked?** box or column on the **Fluids** page indicates whether the fluid is linked to a fluid in the **Fluids Library** and allows you to define a link to a fluid. The following figure shows an example of the **Linked?** box in a component analysis case.

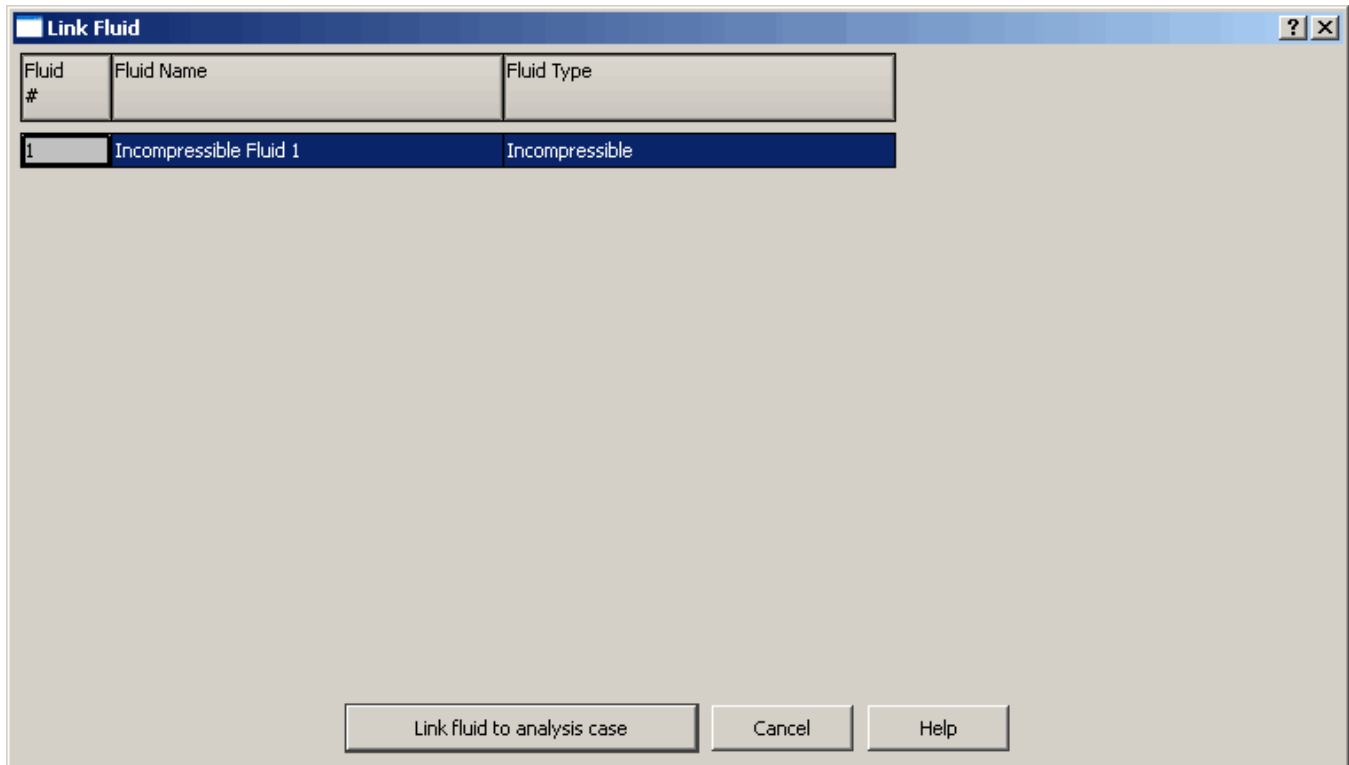


**NOTE:** In component analysis cases, you can only link to one fluid.

## RAPPID User's Guide

---

To link to a fluid from the **Fluids Library**, click the **Define** button in the **Linked?** box. The **Link Fluid** dialog box displays.

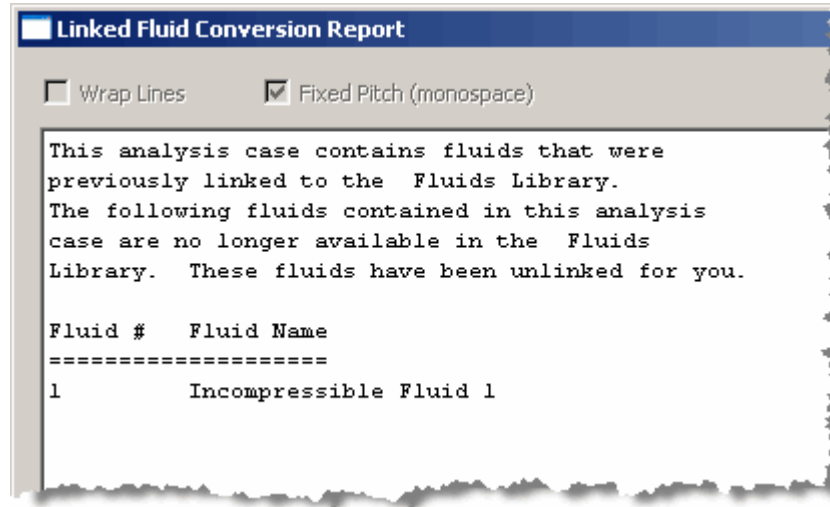


**NOTE:** The **Link Fluid** dialog box displays all fluids for all types of fluids. However, you can only link to a fluid in the **Fluids Library** of the same type that you have selected in the **Fluid Model** column. If you select a different type of fluid, the selection in the **Fluid Model** column changes to match the type of fluid you selected in the **Fluids Library**.

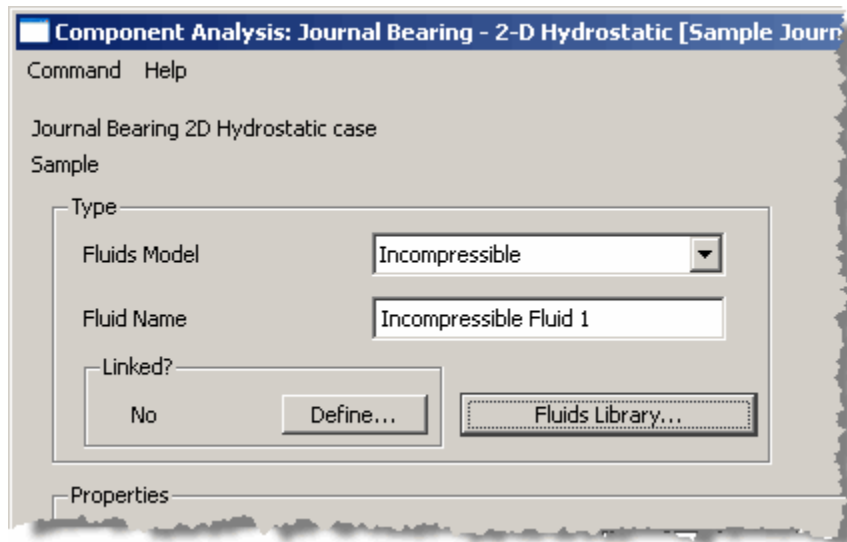


## RAPPID User's Guide

If you delete a fluid from or change a fluid's properties in the **Fluids Library**, this affects all cases that link to that fluid. If you are currently in an analysis case or if you open an analysis case that references that fluid, the **Linked Fluid Conversion Report** displays listing the fluid that was deleted and informing you that the material has been unlinked for you.



The **Linked?** box or column displays **No** for any **Incompressible** or **Compressible** fluid that was deleted. If the fluid was a **Real Properties** fluid, the **Linked?** box or column displays **Undefined**.



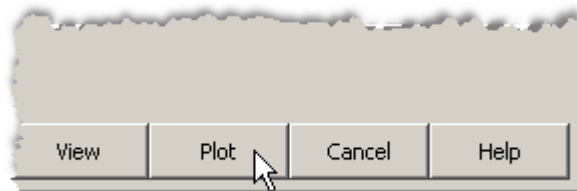
**NOTE:** If multiple fluids are deleted, all materials that are fluids are unlinked in all analysis cases that linked to the deleted fluids. All fluids that are deleted are listed in the **Linked Fluid Conversion Report**. If the fluid(s) you deleted are **Real Properties** fluids, you must redefine the link before you can run the analysis case.

## Chapter 7: Plotting results

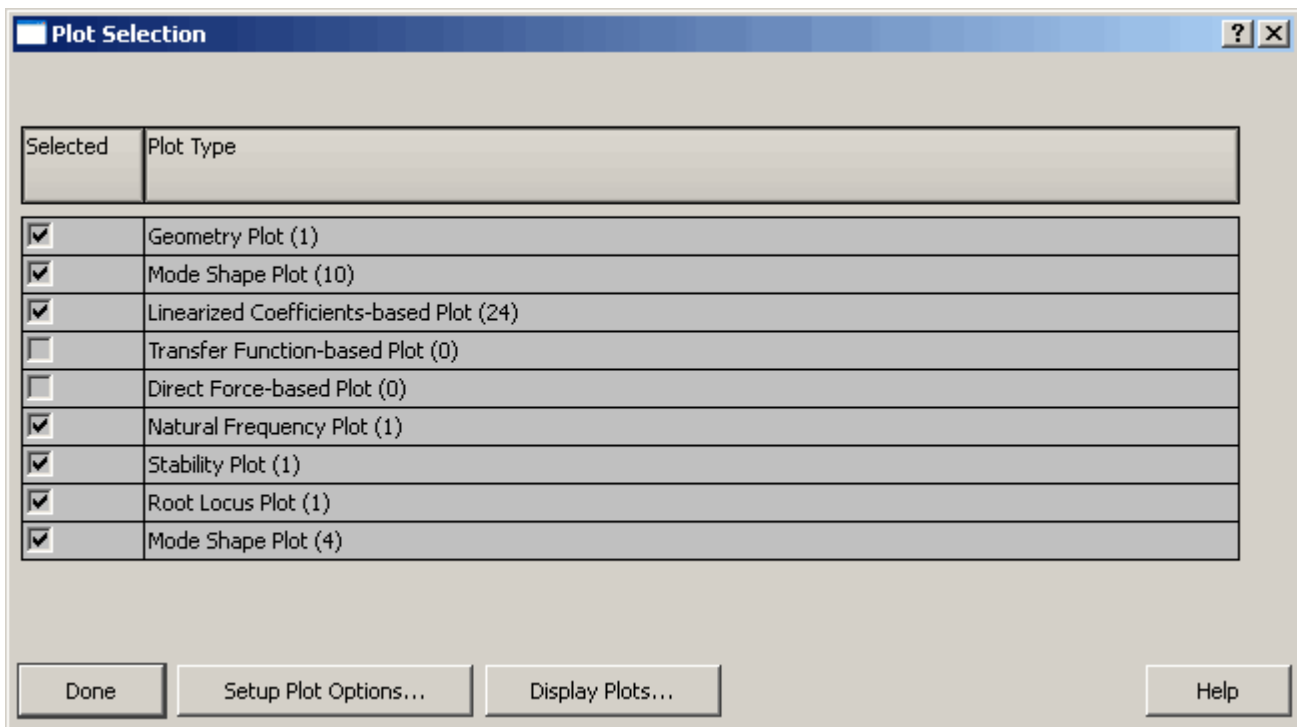
When you run an analysis case and produce output, plot data is also generated. This plot data is available for viewing in graphical form.

### Accessing plots

Use the **Plot** button on the **case manager** window to access the plots available for viewing in each case that can be run. Some component cases cannot be run; they are only for inclusion in network analysis cases. The **Plot** button in *Tabular Input* analysis cases plots the coefficients entered into the analysis case.

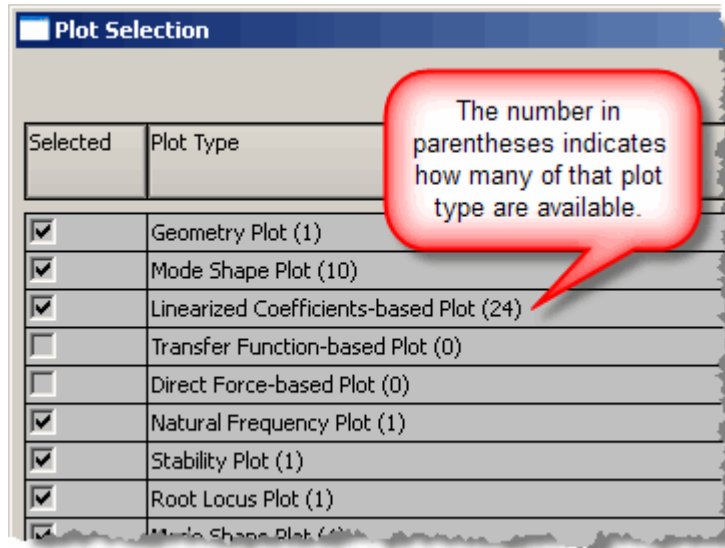


All the plot types available for the selected analysis case type display in the **Plot Type** column on the **Plot Selection** dialog box. The list in the **Plot Type** column differs for each type of analysis case.



# RAPPID User's Guide

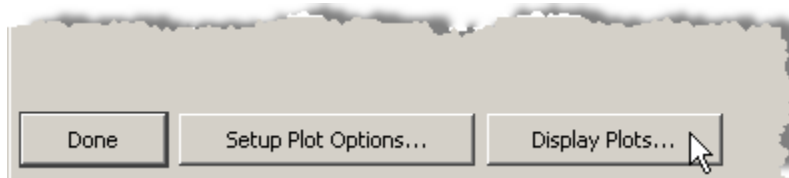
For each plot type, the number of plots available of that type for the current analysis case displays in parentheses.



**NOTE:** All the plot types on the **Plot Selection** dialog box are selected by default.

## Displaying plots

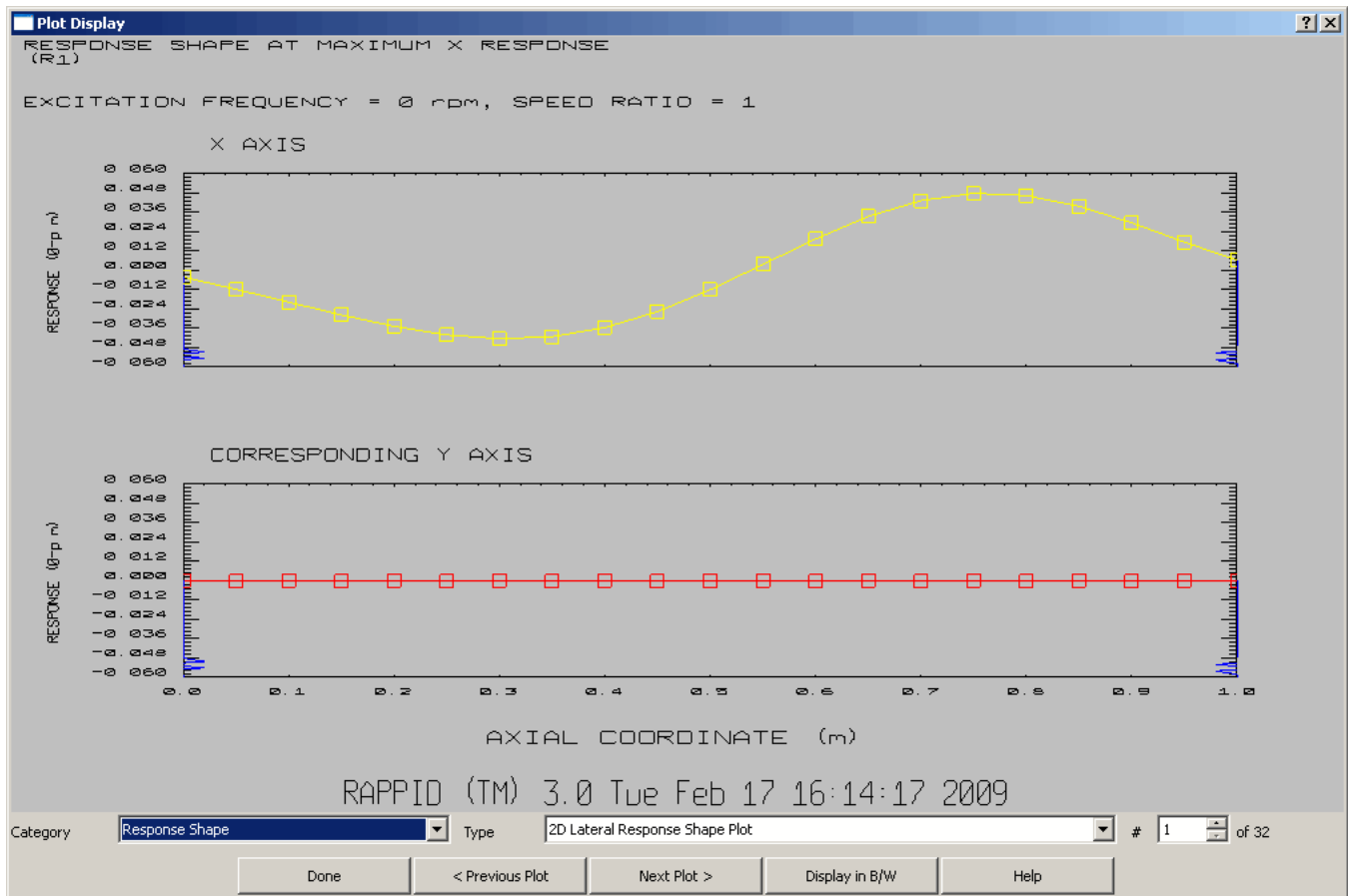
Use the **Display Plots** button to display the available plots.



See **Customizing plots** on page 121 for information about specifying additional settings using the **Customize/Setup Plot Options** button.

# RAPPID User's Guide

The **Plot Display** dialog box displays.



Use the **Category** and **Type** drop-down lists and the # spin edit box to select a plot to view.

You may also use the **Previous Plot** and **Next Plot** buttons to navigate among the plots.

Use the **Display in B/W** button or the **Display in Color** button to select the color scheme to be used when viewing the plots. If you want to print a screenshot (see **Printing and saving plots** on page 123), it is recommended that you set the view on the **Plot Display** dialog box to black and white.

## Customizing plots

Some analysis cases have additional settings available to customize the plots. The options available for setting options to customize the plots differ for each module. The options available for each module are discussed in the sections for the respective modules.

# RAPPID User's Guide

Use the **Customize/Setup Plot Options** button to specify the additional settings for the current analysis case, if available. The following picture shows an example of a **Plot Options** dialog box for a *Dynamics Network Eigenvalue* analysis case.

**Network Analysis: Dynamics Eigenvalue Plot Options**

Select Stability Parameter: Log Dec

☒ Limit Eigenvalue Data to Display (Root Pre-sort)

	Minimum	Maximum	Minimum	Maximum
Natural Frequency Range (cpm)	0	0	2e+004	9e+004
Stability Parameter (-)	0	0	-1.68e-008	1.68e-008
Rotor Speed (rpm)	0	0	1	1.146e+005

☒ Sort Roots on Natural Frequency and Stability Plots

Modes to Sort: Full Modes Only

Modes to Display: Forward Whirl Only

☐ Label Operating Range and Interferences

Operating Range: Start: 0 Stop: 0

☒ Eliminate Mode Shape Plots With Deflections < 0 % of Maximum Deflection

☒ Add Harmonic Lines to Natural Frequency Plot

#	Harmonic Line (e.g. 1x, 2x, ...)
1	
2	
3	
4	
5	
6	

Done Initialize Settings... Cancel Help

See the following topics for information about the settings available in each module:

- **Plot Options – Eigenvalue** (page 304)
- **Plot Options – Steady State Forced Response** (page 306)

- **Plot Options – Transient Forced Response** (page 311)

### Printing and saving plots

Currently, there are no buttons available for printing and saving the plots. In order to print and/or save a plot, you need to take a screen capture of the **Plot Display** dialog box and paste it into a program that allows you to print the graphic and save it as a file.

**NOTE:** You may choose to use specialized screen capture software to take the screen capture, print it, and save the graphic file.

See **Accessing plots** on page 119 and **Displaying plots** on page 120 for more information on opening the **Plot Display** dialog box.

Once the **Plot Display** dialog box is open, press **ALT+PRINT SCR** to take a screen capture of the current active window. This puts the screen capture on the clipboard, ready to be pasted into a program.

To print the plot, do one of the following:

- Print the graphic from within the screen capture program used to take the screen capture or the graphics software program used into which the screen capture was pasted.
- Import or copy and paste the captured window into a program, such as OpenOffice Writer (Linux) or Microsoft Word (Windows), that allows you to print.

To save the plot, save the file in the graphics or word processing program into which you pasted the screen capture or save the graphic file within the screen capture program.

## Chapter 8: Backing up

RAPPID™ provides the ability to archive projects. The **Archived Projects** tab displays all projects that have been archived. The name of the project displays with the date the archive was created.

For information about archiving selected analysis cases rather than a whole project, see **Exporting analysis cases** on page 26.

You may want to archive a project for various reasons such as:

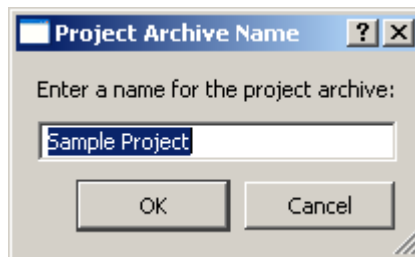
- You want to have a backup of a project.
- You want to move a copy of a project to another system.
- You want to preserve a copy of a project before you make substantial changes to the project.

### Archiving a project

To archive one or more projects, the **Active Projects** tab must be active. When the **Active Projects** tab is active, the **Archive** button is available. Select the project you want to back up and do one of the following:

- Click the **Archive** button.
- Right-click on the project and select **Archive** from the popup menu.

The **Project Archive Name** dialog box displays.

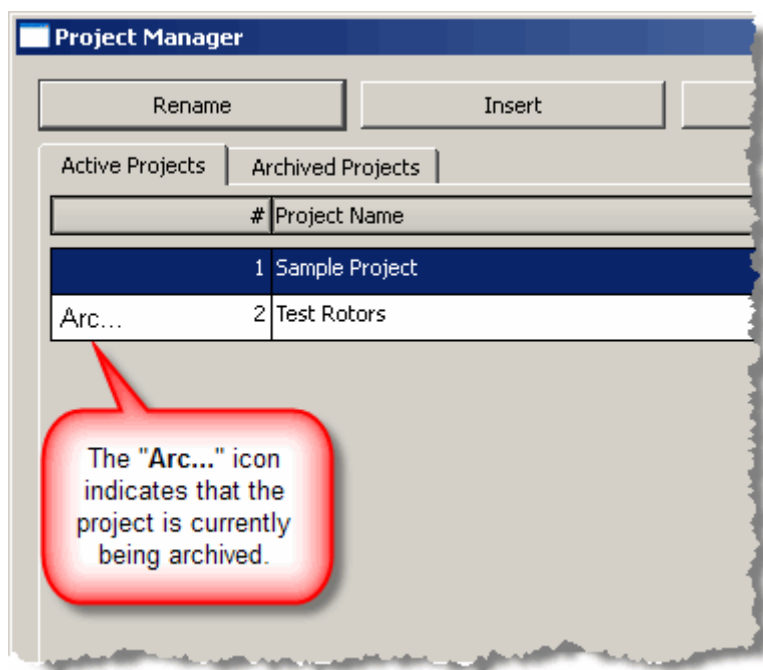


Specify a unique name for the archive in the edit box and click **OK**. The archive is created and added immediately in alphabetical order on the **Archived Projects** tab even during the archiving process.

## RAPPID User's Guide

---

You may archive more than one project while another project is still being archived. If a project is still being archived, there is an "Arc..." icon next to the project name on the **Active Projects** tab and next to the archive name the **Archived Projects** tab.



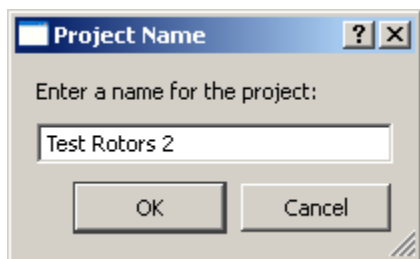
The "Arc..." icon remains on both tabs as long as the archive file is being created. It goes away when the archiving process is complete.

**NOTE:** Archiving a project DOES NOT remove the project from the **Active Projects** tab. You must delete a project manually. See **Deleting a project** on page 17.

### Unarchiving a project

To unarchive a project, make sure the archive file for that project is in the archive directory specified in the preferences. See **Specifying the location** of projects and archives on page 8 for more information. If you need to add the archive file to the archive directory, see **Refreshing the list of archived projects** on page 126.

Select the archive file on the **Archived Projects** tab and click the **Unarchive** button.



Specify a name for the new project to be created from the archive. The name must be unique among the list of projects on the **Active Projects** tab; you cannot use a name from an existing active project.



The new project is added to the end of the list of projects on the **Active Projects** tab. If you want to move the project to a different location in the list, see **Moving an active project** on page 16 for more information.

### Refreshing the list of archived projects

When the **Archived Projects** tab is active, the **Insert** button becomes the **Refresh** button. Use the **Refresh** button if you have added a RAPPID™ archive to the archive directory while RAPPID™ is open. This displays the newly-added file on the **Archived Projects** tab in alphabetical order.

## Part II – Analysis case types

This part of the user's guide is a reference providing information about the analysis case types you purchased for the RAPPID™ system. For information about how to use the RAPPID™ system, see **Part I: The RAPPID™ system** on page 6.

### Chapter 9: Model/Eigenvalue components

#### Overview

This section describes the *Model/Eigenvalue* component modules available in RAPPID™ and the input fields specific to each. Currently, the following categories of *Model/Eigenvalue* components are supported in RAPPID™:

- *Housing* – defined as non-rotating entities that are fully constrained (i.e., no rigid body modes).
- *Advanced Rotor* – defined as rotating entities that are unconstrained (i.e., includes rigid body modes).

The input fields available for each type of *Model/Eigenvalue* component module are mostly the same. The information in this section applies to both types, except where noted.

Structural components are the rotating and stationary entities that can represent rotors, housings, piping, etc. The objective of the *Model/Eigenvalue* component solvers is to calculate the free-free natural frequencies (i.e., eigenvalues) and mode shapes (i.e., eigenvectors) for use in the component modal synthesis (CMS)-based dynamic analysis modules. There are two methods for obtaining the natural frequencies and mode shapes associated with a *Model/Eigenvalue* component:

- calculation with RAPPID™ analysis module
- importing mode shapes from another solver.

For this version of RAPPID™, the calculated mode shapes are restricted to 2-D axisymmetric while imported mode shapes can be either 2-D or 3-D.

RAPPID™ uses a general finite element solver, based on a beam element formulation, to determine undamped normal modes and natural frequencies of rotor and housing entities. The program can create models incorporating axial, torsional, or lateral degrees of freedom. The element used by the program is restricted to a two-node tapered beam element, which is sufficient for most rotordynamic modeling purposes. A general assortment of boundary conditions and releases are provided for in the finite element solver.

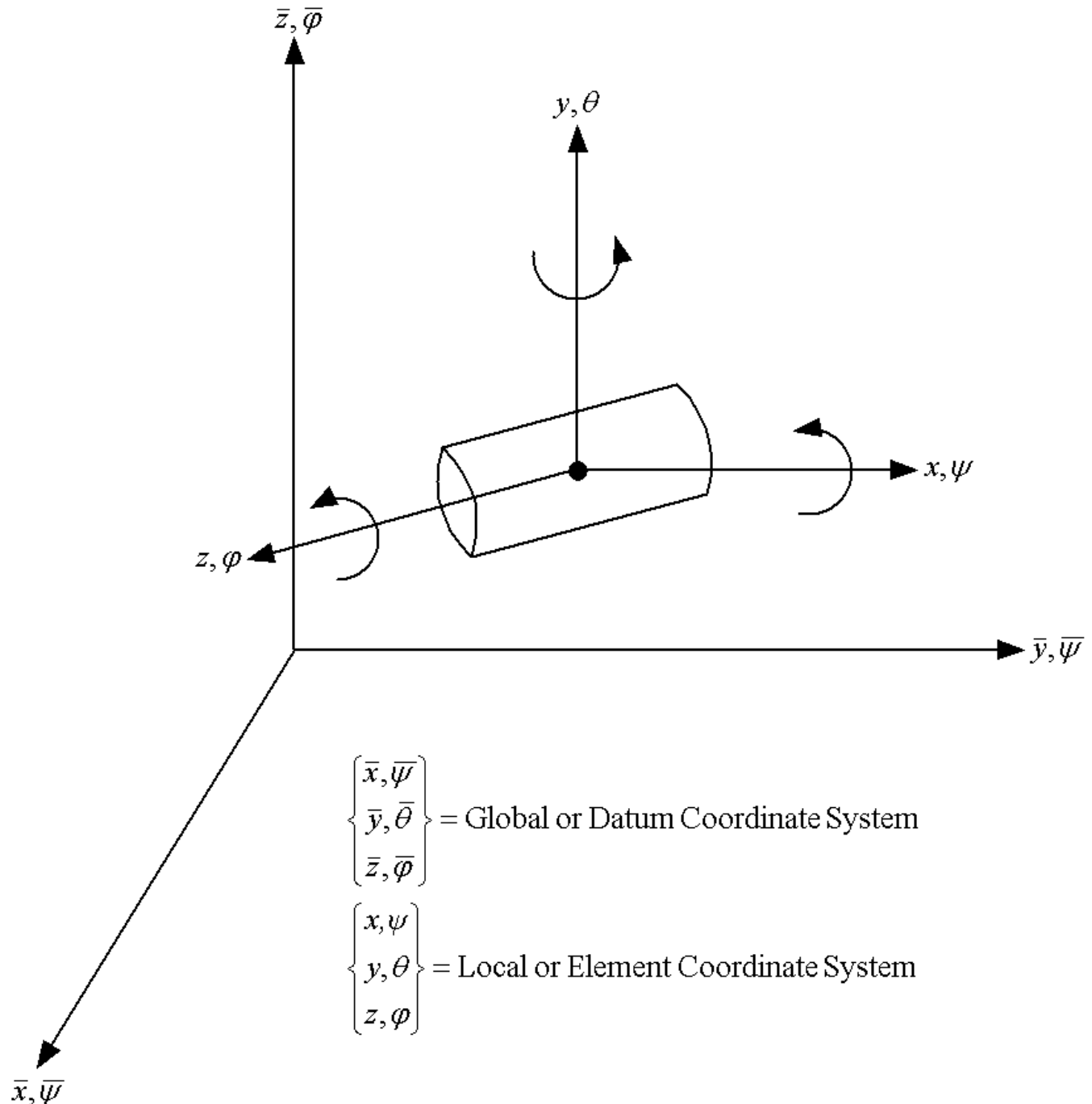
The beam element is a two-node element with the nodes at its endpoints. In general, the beam is straight, has a hollow circular cross section, and permits a linear taper for the inside and outside diameters (i.e., conical). The elastic formulation of the element comes directly from Euler bending and Timoshenko shear formulae. A consistent stiffness matrix and non-consistent (lumped) inertia matrix are used. Inertias are computed directly from beam element input geometries. Additional mass and rotational inertia values can be added explicitly for each degree of freedom in the model.

The inertial characteristics of each rotor and housing entity are determined by summing

- the element mass and inertia properties of the conical beam cross-section, and
- user-defined non-structural mass and inertia that may exist at various nodes.

## RAPPID User's Guide

When defining beam element properties, displacement boundary conditions can be applied at the element ends. It is noted that boundary conditions and end release conditions are in the local reference frame, shown in the following figure. In addition, the module currently only supports rotation about the local z-axis.



*Figure 1. Global-local coordinate systems*

## Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

## Active Degrees of Freedom box

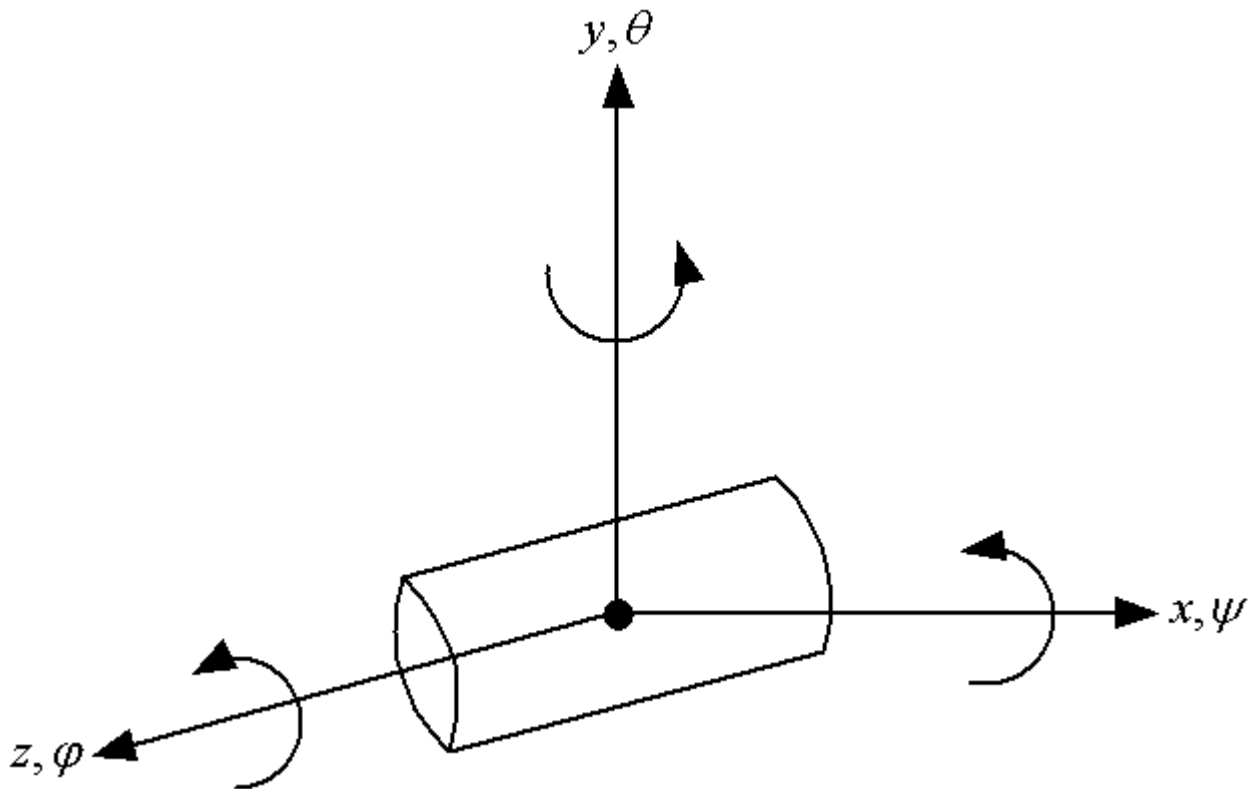
Use the **Active Degrees of Freedom** box to specify the coordinate axes along and/or about which the movement occurs.

The following options are available:

- **Lateral** - movement along and about the lateral (x, y) plane.
- **Torsional** - movement about the z-axis
- **Axial** - movement along the z-axis

**NOTE:** Use the “right-hand rule” when determining the direction of the movement about an axis.

The following figure is a representation of the coordinates and movements.



*Figure 2. Coordinates and Degrees of Freedom for masses*

## Load Path box

Use the **Load Path** box to break up rotors or housings into different sections. The logical sections are load paths and thus the name. The use of **Load Path** definitions is restricted to plotting functions. The analysis results from complex multi-load path entities can be difficult to decipher. Therefore, the option is given to define different load paths to simplify the display of the analysis results.

At least one load path must be defined. The descriptive **Name** for the **Load Path** displays in the plots of analysis results. The load path definitions do not alter the calculations.

## Other box

Use the **Other** box to specify information about the beams, modes, and mode shapes.

**NOTE:** You may choose either **Enter Beams Using** or **Import Mode Shapes**, but not both.

### Enter Beams Using

Use the **Enter Beams Using** radio button to specify whether the beam geometry will be entered using **Diameters** or **Radii**.

### Import Mode Shapes

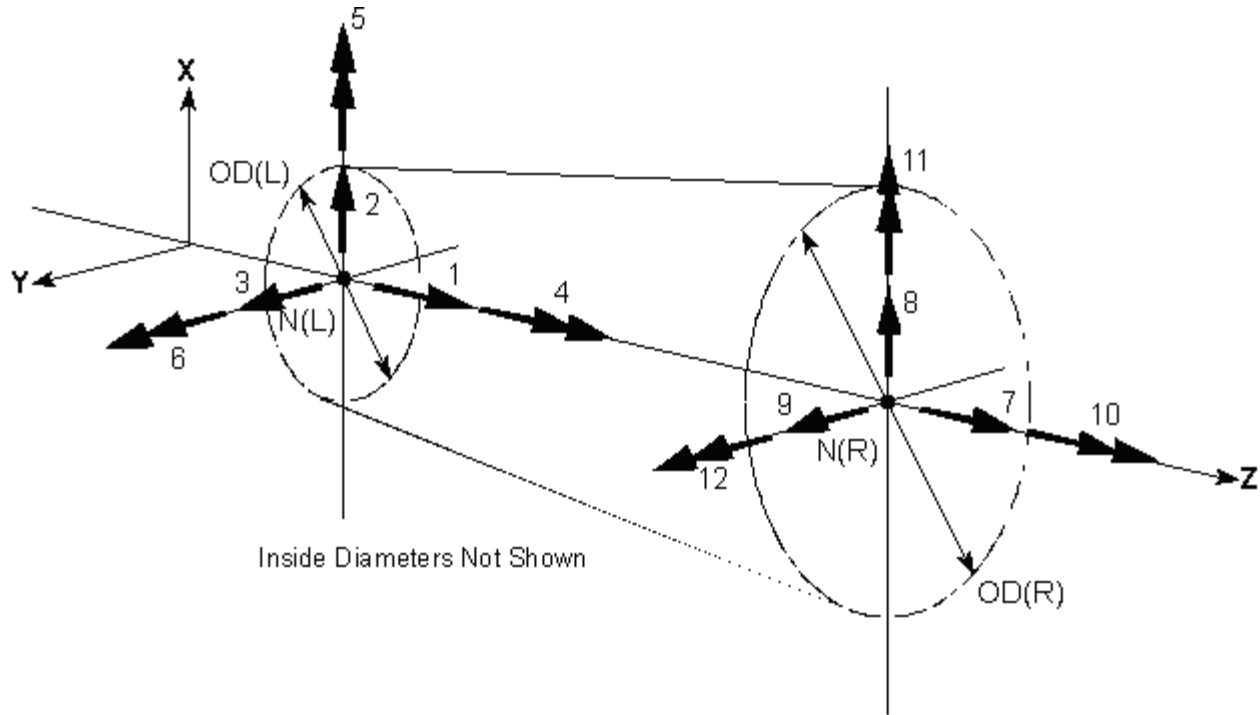
Use the **Import Mode Shapes** radio button to import mode shapes calculated using another finite element solver.

If you select the **Import Mode Shapes** radio button, use the **Import** button to import the mode shapes (**.mde**) file. Use the **View** button to view the contents of the mode shapes file.

**NOTE:** The text in the dialog box displays in read-only mode. You cannot edit the text. To change a mode shapes file, edit the original text file and then re-import the file again into the analysis case.

# RAPPID User's Guide

In order to understand the format of the **.mde** file, the definition of the beam geometry and numbering of the degrees of freedom (DOFs) must be reviewed, as illustrated in the following figure:



**Figure 3. Beam element local coordinate system**

There are six degrees of freedom for each node. For the two-node beam element in the figure, this translates into 12 degrees of freedom for the beam. Regardless of the element used, there will always be 6 degrees of freedom for a node.

The degrees of freedom numbering scheme used for the RAPPID™ **.mde** file is as follows:

1. Translation Z-axis
2. Translation X-axis
3. Translation Y-axis
4. Rotation Z-axis
5. Rotation X-axis
6. Rotation Y-axis

The **.mde** file format for a *Structural Dynamics* component is as follows:

```
TOTALS
Number of DOF
Number of Mode Shapes
Number of Nodes
```

---

## RAPPID User's Guide

---

Numbering of Active DOF for each node (0 indicates inactive DOF)  
...  
Frequency (The next lines repeated for Number of Mode Shapes)  
Modal displacement for each DOF (This line repeated for Number of DOF)  
...  
Center of Gravity Coordinates (X,Y,Z)

### **Number of Modes to Retain**

The *Structural Dynamics* component analysis is based on component mode synthesis (CMS). As such, it requires eigenvalues (modes) and eigenvectors (mode shapes) for each structural entity. Use the **Number of Modes to Retain** spin edit box to specify the number of mode shapes to print both in the output file and in the normal mode file at the conclusion of the calculations.

The *Structural Dynamics* component analysis employs all modes specified by the entry in the **Number of Modes to Retain** edit box in the modal solution for rotordynamic stability and forced response, including all rigid body modes (zero frequency modes).

**NOTE:** If the number of modes specified is more that the total degrees of freedom, the number of modes will be set equal to the number of degrees of freedom.

Valid entry for the **Number of Modes to Retain** edit box is greater than or equal to 1.

### **Print Mode Shape Tables**

Use the **Print Mode Shape Tables** check box to print mode shape tables to the viewed output available using the **View** button at the bottom of the **Analysis case manager** window.

## **Materials page**

The **Materials** page is common for all analysis cases that use material data and is used to define the material properties list that will be available for selection in this analysis case.

Once you insert a new material as a row in the table, there are two options for assigning the material properties:

- Define the properties manually, directly on the **Materials** page in the analysis case.
- Link to a material from the **Materials Library** by clicking the **Materials Library** button, therefore using the properties from the library.

See **Chapter 5: Working with materials** on page 79 for more information.

## **Nodes page**

Use the **Nodes** page to specify node numbers, locations of model nodes (in global space), added mass, added inertia, and load path indicators.



Nodes are defined using x, y, and z coordinates located with respect to the local, datum or system coordinate system. See **Figure 3. Beam element local coordinate system** on page 132.

During rotordynamic analysis, all interconnect stiffness, damping and mass coefficients are relative to the local- or rotor-based coordinate system, as are all analysis results (displacements, lateral interconnection forces, accelerations and relative displacements). If modal information is read in, added mass, added transverse (It), and added polar inertia (Ip) should be eliminated from this input.

## Initialize Using Shaft Sections

Use the **Initialize Using Shaft Section** button to access a dialog box allowing you to enter nodes and beams into your *Advanced Rotor* or *Housing* case by sections to automatically calculate the values for the **Nodes** and **Beams** pages.

Specify the number of beams in each section, the start position, length, left and right outer and inner diameters, and start and end transition angles for each section, and the material and the beam type to apply to all the beams in each section.

Use the **Generate** button at the top of the **Define Shaft Sections** dialog box to generate the values automatically entered into the grids on the **Nodes** and **Beams** pages.

## Node #

The **Node #** column is automatically sequentially incremented as additional nodes (rows) are inserted. This number is used when specifying left and right nodes on the **Beams** page.

Use the **ID Text** column to enter a descriptor used for node identification at the network and integrated network levels.

**NOTES:** Entering data in the **ID Text** is optional, but strongly recommended, at least for nodes that are going to be used in a *Dynamics* network as interconnections.

The text length is limited to 40 characters.

## Interconnection

Use the **Interconnection** check box to define a nodal location for connection of an interconnection (e.g., bearing, seal, damper, etc.).

When selected, this option enables access of the nodal location from within network cases for defining connectivity to interconnection component cases.

## Stiffness to Ground

As mentioned in the Error! Reference source not found. on page **Error! Bookmark not defined.**, housing modes for use in network analyses are constrained modes. To apply constraints, use the **Stiffness to Ground** check box.

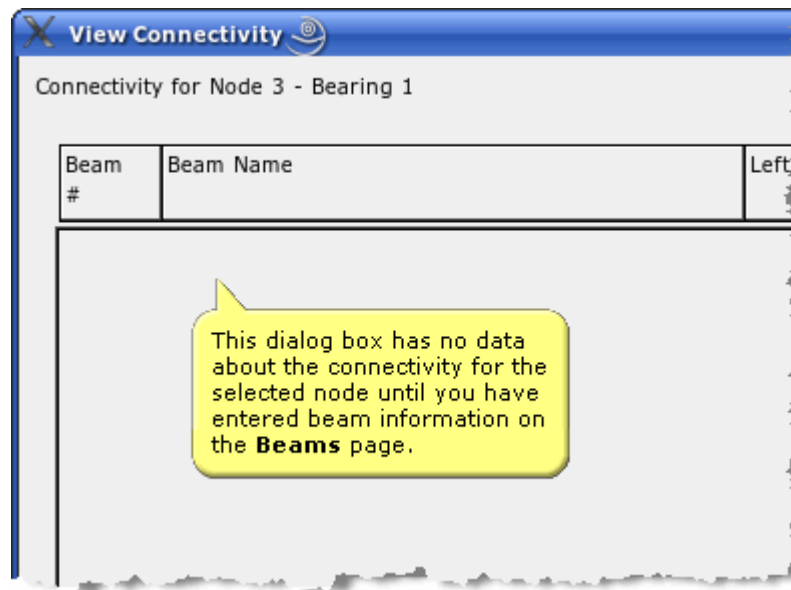
**NOTE:** **Stiffness to Ground** is only available for *Structural Dynamics Housing* cases.

Once you select **Stiffness to Ground** for a specific node, use the **Define** button in the next column to enter translational and rotational stiffness values for the connection between the selected housing node and ground on the **Stiffness to Ground** dialog box. Any combination of values may be entered.

### Connectivity

Click the **View** button in the **Connectivity** column to view a listing of all of the beams to which the selected node connects.

The **View Connectivity** dialog box is empty during the creation of a new case until beams have been defined on the **Beams** page.



*Figure 4. View Connectivity dialog accessed from the Nodes page*

**NOTE:** The data in the **View Connectivity** dialog box is read-only.

### XYZ Coordinates

Use the XYZ coordinate inputs (the **X**, **Y**, and **Z** columns in the data grid) to define the location of the node in the local coordinate system.

**NOTES:** The **X** and **Y** columns are not currently available.

The axis of rotation is always the Z-axis, as illustrated in the *Global-local coordinate systems* figure in the Error! Reference source not found. on page **Error! Bookmark not defined.**

Valid inputs are any real number.

### Added Mass

Use **Added Mass** to enter additional mass to lump at this node.

**NOTE:** The program calculates the mass of the beam based on subsequent input. The input required in this column refers to additional mass for lumped entities such as impellers or coupling hubs.

### Added Ip

Use **Added Ip** to enter the additional polar inertia to lump at this node.

**NOTE:** The program calculates the inertia of the beam based on subsequent input. The input required in this column refers to additional inertia for lumped entities such as impellers or coupling hubs.

### Added It

Use **Added It** to enter the additional transverse inertia to lump at this node.

**NOTE:** The program calculates the inertia of the beam based on subsequent input. The input required in this column refers to additional inertia for lumped entities such as impellers or coupling hubs.

### Active Load Paths

Use the **Active Load Paths** columns to indicate that a node is included in the selected load path. A load path is selected when a check mark displays in the corresponding check box for that node and load path.

The **Active Load Paths** column contains as many sub-columns containing check boxes as **Load Paths** defined on the **Options** page (see **Load Path box** on page 131). Load paths are shown in the order that they were entered on the **Options** page. A node can be included in multiple load paths.

### Beams page

Use the **Beams** page to specify beam elements defined to span the distance between a “left” and “right” node labeled N(L) and N(R) in the **Figure 3. Beam element local coordinate system** on page 132.

The finite element solution module uses a right hand local coordinate system, as illustrated in **Figure 2. Coordinates and Degrees of Freedom for masses** on page 130. The axial coordinate is denoted by Z, and the two transverse axes are labeled as X and Y. The positive Z direction points in the direction of increasing node number. At each node, 6 DOF are permissible, thus each element can have up to 12 DOF. *Note that more than one element may be defined for the same pair of nodes.* In this fashion, the structural contributions of pressed-on/piloted components may be represented. If modal information is read in, the information contained on this page is not necessary.

### Beam #

The **Beam #** column is sequentially incremented as additional beams (rows) are inserted and displays in the **View Connectivity** dialog box accessed using the **Connectivity** column on the **Nodes** page. See **Connectivity** on page 135.

### Left Node

Use the **Left Node** column to enter the integer node number used as the “left” end node for this element.

## Right Node

Use the **Right Node** column to enter the integer node number used as the “right” end node for this element.

## Material

Use the drop-down menu in the **Material** column to select a material for each beam entered. The materials available in the drop-down menu include all materials added on the **Materials** page in the current analysis case.

See **Chapter 5: Working with materials** on page 79 for more information about adding materials to analysis cases.

## Beam Type

At this time, the only **Beam Type** available is the **Conical** beam.

## Left O.D.

Use the **Left O.D.** column to enter the floating point outside diameter of the element at the “left” end node.

## Left I.D.

Use the **Left I.D.** column to enter the floating point inside diameter of the element at the “left” end node.

## Right O.D.

Use the **Right O.D.** column to enter the floating point outside diameter of the element at the “right” end node.

## Right I.D.

Use the **Right I.D.** column to enter the floating point inside diameter of the element at the “right” end node.

## ID Text

Use the **ID Text** column to enter a description of for each beam.

**NOTE:** Entering text in the **ID Text** column is optional; however, it is strongly recommended.

## Chapter 10: 0-D Flow Elements – Fitting

### Overview

Common fittings have losses proportional to the Reynolds number of the flow. A standard form of this relationship is used.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

The **Analysis Parameter** drop-down menu allows you to select the input value by which you want to vary the values of other inputs.

Currently, **Time** is the only available **Analysis Parameter**.

The **Time** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting time (**Start**), the increment of time (**Delta**), and the ending time (**Stop**) for the calculation. The **Time** fields must define a constant or increasing time. Therefore, the **Stop** time must be greater than the **Start** time and the **Delta** time must be positive. Field entries must be non-negative floating point values.

### Other box

#### K1

Use **K1** to specify the loss coefficient for the fitting at a Reynolds number of 1.0. When included with **Kinf**, the loss coefficient, as a function of Reynolds number for the fitting, is calculated with the following formula:

$$K_{fitting} = \frac{K1}{Re} + K_{inf} \left( 1 + \frac{1}{D} \right)$$

where,

- **Re** = Reynolds number
- **D** = Flow Diameter of the fitting
- **Kinf** = see **Kinf** on page 138

#### Kinf

Use **Kinf** to specify the loss coefficient for the fitting at a Reynolds number of infinity.

## **Additional Heat Input**

Use **Additional Heat Input** to specify heat added or subtracted from a branch.

## **Flow Diameter**

Use **Flow Diameter** to specify the effective flow area of a branch.

## **Rotating Component Parameters**

Use the **Define** button next to **Rotating Component Parameters** to define the inlet and exit radii the rotational speed, and the loss factor on the **Define Rotating Component Parameters** dialog box.

- Use **Inlet Radius** to define the radius at the entrance to the rotating component.
- Use **Exit Radius** to define the radius at the exit of the rotating component.
- Use **Rotational Speed** to define the speed at which the rotating component rotates.
- Use **Loss Factor** to ... ???

## Chapter 11: 0-D Flow Elements – Misc.

### Overview

Miscellaneous cases allow you to define a flow resistance in the system. This option can be used for seals, bearings, and other components that do not have models.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

The **Analysis Parameter** drop-down menu allows you to select the input value by which you want to vary the values of other inputs.

Currently, **Time** is the only available **Analysis Parameter**.

The **Time** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting time (**Start**), the increment of time (**Delta**), and the ending time (**Stop**) for the calculation. The **Time** fields must define a constant or increasing time. Therefore, the **Stop** time must be greater than the **Start** time and the **Delta** time must be positive. Field entries must be non-negative floating point values.

### Other box

#### Loss Factor

Use **Loss Factor** to specify the secondary losses proportional to the dynamic head (see the equation below) of the flow.

$$\frac{\rho V^2}{2}$$

#### Additional Heat Input

Use **Additional Heat Input** to specify heat added or subtracted from a branch.

#### Flow Diameter

Enter description here ???

#### Rotating Component Parameters

Use the **Define** button next to **Rotating Component Parameters** to define the inlet and exit radii the rotational speed, and the loss factor on the **Define Rotating Component Parameters** dialog box.

- Use **Inlet Radius** to define the radius at the entrance to the rotating component.

## RAPPID User's Guide

---

- Use **Exit Radius** to define the radius at the exit of the rotating component.
- Use **Rotational Speed** to define the speed at which the rotating component rotates.
- Use **Loss Factor** to ... ???



## Chapter 12: 0-D Flow Elements – Pipe

### Overview

Pipes allow you to define a pipe with roughness, an entrance loss, and an exit loss.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

The **Analysis Parameter** drop-down menu allows you to select the input value by which you want to vary the values of other inputs.

Currently, **Time** is the only available **Analysis Parameter**.

The **Time** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting time (**Start**), the increment of time (**Delta**), and the ending time (**Stop**) for the calculation. The **Time** fields must define a constant or increasing time. Therefore, the **Stop** time must be greater than the **Start** time and the **Delta** time must be positive. Field entries must be non-negative floating point values.

### Type

Use the **Type** drop-down menu to specify the type of pipe. The following options are available.

- Basic
- Abrupt Contraction
- Abrupt Expansion

**NOTE:** When either **Abrupt Contraction** or **Abrupt Expansion** is selected, the **Length** edit box in the **Geometry** box is not available.

### Geometry box

#### Inlet Diameter

Use **Inlet Diameter** to specify the diameter at the entrance to the pipe.

#### Exit Diameter

Use **Exit Diameter** to specify the diameter at the exit of the pipe.

#### Length

Use **Length** to specify the length of the pipe.

**NOTE:** **Length** is only available when **Basic** is selected as the **Type**.

## **Roughness**

Use **Roughness** to specify the relative roughness of the surface of the pipe.

## **Other box**

## **Inlet Loss Factor**

Use **Inlet Loss Factor** in *Pipe* cases only to specify the entrance loss factor for the pipe.

## **Exit Loss Factor**

Use **Exit Loss Factor** in *Pipe* cases only to specify the exit loss factor for the pipe.

## **Additional Heat Input**

Use **Additional Heat Input** to specify heat added or subtracted from a branch.

## **Rotating Component Parameters**

Use the **Define** button next to **Rotating Component Parameters** to define the inlet and exit radii the rotational speed, and the loss factor on the **Define Rotating Component Parameters** dialog box.

- Use **Inlet Radius** to define the radius at the entrance to the rotating component.
- Use **Exit Radius** to define the radius at the exit of the rotating component.
- Use **Rotational Speed** to define the speed at which the rotating component rotates.
- Use **Loss Factor** to ... ???

## Chapter 13: 0-D Flow Elements – Centrifugal Pump and Turbine

### Overview

Pump and turbine cases allow you to add the flow pressure/energy from a pump and subtract the flow pressure/energy from a turbine into a *Transient Flow Network* case. The characteristics can be described as a function of mass flowrate or as a function of power.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

The **Analysis Parameter** drop-down menu allows you to select the input value by which you want to vary the values of other inputs.

Currently, **Time** is the only available **Analysis Parameter**.

The **Time** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting time (**Start**), the increment of time (**Delta**), and the ending time (**Stop**) for the calculation. The **Time** fields must define a constant or increasing time. Therefore, the **Stop** time must be greater than the **Start** time and the **Delta** time must be positive. Field entries must be non-negative floating point values.

### Other box

#### Pressure Drop Characteristic

Use **Pressure Drop Characteristic** to specify whether the pressure drop is **Flow-Based** or **Power-Based**.

Use...	to specify...	when the Pressure Drop Characteristic is...
A0	Constant Term of pressure drop equation	Flow-Based
B0	Term proportional to the square of mass flowrate	Flow-Based
Efficiency	Efficiency term for pressure drop as a function of Power	Power-Based

## RAPPID User's Guide

Use...	to specify...	when the Pressure Drop Characteristic is...
Power	Power term for pressure drop as a function of Power	Power-Based

### Pressure drop equations:

Mass Flowrate  $\Delta P = A_0 + B_0 \dot{m}^2$

Power  $\Delta P = \frac{\rho Pow \eta}{\dot{m}}$

where:

- $\Delta P$  = pressure drop
- $A_0$  = constant term
- $B_0$  = square term
- $\dot{m}$  = mass flowrate
- $\rho$  = density
- $Pow$  = power
- $\eta$  = efficiency

### **Rotational Speed**

Use **Rotational Speed** to specify the speed at which the pump or turbine is rotating.

### **Additional Heat Input**

Use **Rotational Speed** to specify the speed at which the pump or turbine is rotating.

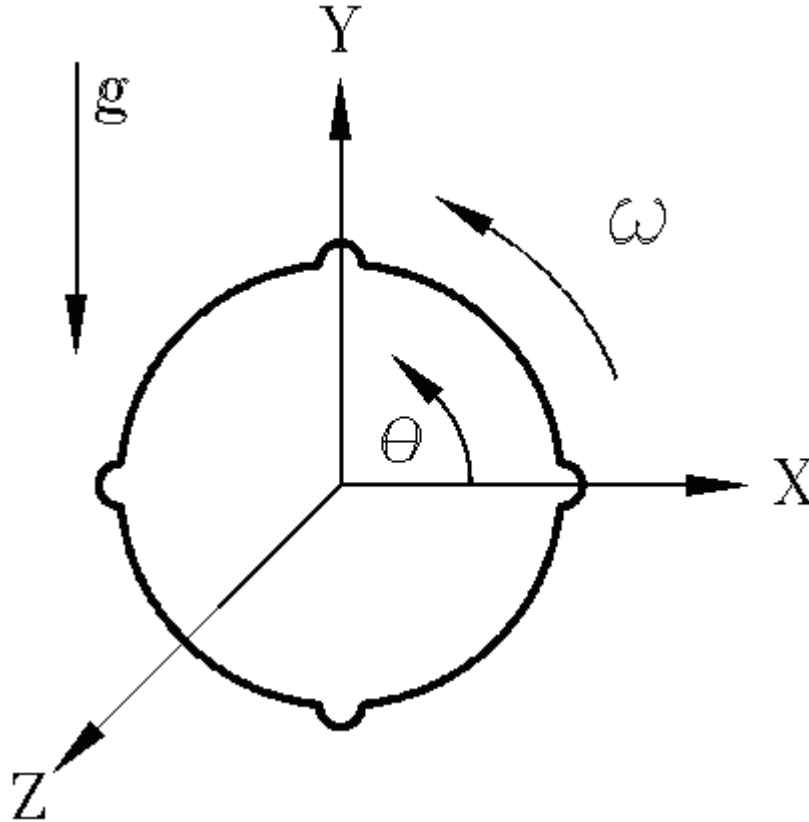
### **Flow Diameter**

Use **Flow Diameter** to specify the effective flow area of a branch.

## Chapter 14: Journal Bearing – Hydrostatic

### Overview

Journal bearing components are the interconnections that transfer loads and forces between the rotating and stationary entities. The objective of the Journal Bearing component solvers is to calculate the steady-state and dynamic performance characteristics of the bearing that represent the flow, heat generation, and transfer function data that is required by the network solvers. The coordinate system used for inputs is illustrated in the following figure.



*Figure 5. Coordinate system*

Journal Bearings is an analysis for radial orifice compensated hydrostatic bearings. For rotordynamics, bearing analysis has the goal of determining the coefficients of the following linearized equations of motion:

$$-\begin{Bmatrix} F_x \\ F_y \end{Bmatrix} = \begin{bmatrix} K_{xx} & K_{xy} \\ K_{yx} & K_{yy} \end{bmatrix} \begin{Bmatrix} X \\ Y \end{Bmatrix} + \begin{bmatrix} C_{xx} & C_{xy} \\ C_{yx} & C_{yy} \end{bmatrix} \begin{Bmatrix} \dot{X} \\ \dot{Y} \end{Bmatrix} + \begin{bmatrix} M_{xx} & M_{xy} \\ M_{yx} & M_{yy} \end{bmatrix} \begin{Bmatrix} \ddot{X} \\ \ddot{Y} \end{Bmatrix}$$

In this equation,  $(X, Y)$  define the motion of the rotor relative to its steady-state arbitrary position;  $(F_x, F_y)$  are the components of the reactive force acting on the rotor; and  $(K, C, M)$  are stiffness, damping and inertia (mass) coefficients, respectively, which are functions of the eccentricity ratio,  $E_0$ .

Journal Bearings predicts the leakage, torque and rotordynamic coefficients for centered, incompressible flow, orifice compensated, hydrostatic journal bearings. It is valuable for the determination of bearing characteristics for bearings which are nominally centered (eccentricity ratio  $< 0.50$ ). It has been shown experimentally that bearing characteristics do not vary significantly below this value.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

The **Analysis Parameter** drop-down menu allows you to select the input value by which you want to vary the values of other inputs. Currently, **Rotor Speed** is the only available **Analysis Parameter**.

The **Rotor Speed** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting speed (**Start**), the increment of speed (**Delta**), and the final speed (**Stop**) for the calculation. The **Rotor Speed** fields must define a constant or increasing speed. Therefore, the **Stop** speed must be greater than or equal to the **Start** speed and the **Delta** speed must be positive. Field entries must be real numbers greater than zero.

### Output Format

The **Output Format** drop-down menu allows you to select whether to generate a **Short** or a **Long** output file.

- The **Short** output format includes a listing of the input variables and output consisting of basic flow parameters and the rotordynamic coefficients.
- The **Long** output format includes a listing of the input variables, and output consisting of an extensive listing of the mass and volumetric flowrate, load capacity, friction power loss, recess pressure, circumferential and axial Reynolds numbers, maximum values in the bearing, pertinent thermodynamic and transport values for every orifice and recess, and rotordynamic coefficients.

### Grid

Use the **Grid** drop-down menu to indicate whether a **Coarse**, **Normal**, or **Fine** grid is used in the solution.

**NOTE:** **Grid** is only available for *2-D Hydrostatic*.

### Dynamic Solution box

Use the **Dynamic Solution** box to obtain rotordynamic coefficients at different frequency points and for translation.

Currently, only the **Linearized Coefficient** radio button is active. The **Transfer Function** radio button is not available.

Use the **Order** edit box to obtain linearized coefficients at frequencies other than rotor speed. The value entered can be any positive floating point value and is multiplied by the rotor speed to obtain the frequency at which the rotordynamic coefficients are calculated.

The **Moment Coefficients** check box and the **Center** edit box are only available for certain cases. **Moment Coefficients** are the rotordynamic characteristics that relate forces to angular displacements. The **Center** is the center point of the perturbation for the moment coefficients.

## Materials page

The **Materials** page is common for all analysis cases that use material data and is used to define the material properties list that will be available for selection in this analysis case.

Once you insert a new material as a row in the table, there are two options for assigning the material properties:

- Define the properties manually, directly on the **Materials** page in the analysis case.
- Link to a material from the **Materials Library** by clicking the **Materials Library** button, therefore using the properties from the library.

See **Chapter 5: Working with materials** on page 79 for more information.

## Geometry page

The fields on the **Geometry** page are used to define the overall bearing geometry, the detailed internal geometry, and the bearing housing geometry for heat conduction. The bearing geometry can be defined as constant or dependent on the analysis parameter (e.g., rotor speed). See **Entering variable data** on page 47 for more information.

## Bearing Geometry box

### Left Diameter

Use the **Left Diameter** edit box to specify the diameter of the rotor at the left end of the bearing at the exit plane.

Valid entries are positive floating point values.

### Right Diameter

Use the **Right Diameter** edit box to specify the diameter of the rotor at the right end of the bearing at the exit plane.

Valid entries are positive floating point values.

## **Rotor Roughness**

Use the **Rotor Roughness** edit box to specify the absolute roughness of the rotor surface for *2-D Hydrostatic* analysis cases only. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer. The program constrains the input value to be greater than or equal to zero but less than or equal to 256 micro inches, 6.5 microns, or 0.0065 mm. If the rotor is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

**NOTES:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the rotor is hydraulically smooth with a surface roughness of 8-32 microinches. This can change if the rotor has been damaged or corroded.

## **Length**

Use the **Length** edit box to specify the length of the bearing as measured along the local z-axis of the bearing.

Valid entries are positive floating point values.

## **Recesses**

Use the **Recesses** drop-down menu to turn on (**Yes**) and off (**No**) the **Recess** entries on the **Supply Ports Definition** dialog box in *2-D Hydrostatic* analysis cases. For *1-D Hydrostatic* analysis cases, **Yes** is the only option available.

## **Supply Ports Define button**

Use the **Define** button next to **Supply Ports** to define all of the detailed geometry related to the recess and orifice.

See **Supply Ports Definition dialog box** on page 151.

## **Clearance**

The **Clearance** represents the gap formed between the surface of the rotor and the surface of the bearing (stator).

There are two types of clearance, indicating whether or not the clearance varies axially:

- **Axially Constant**
- **Axially Varying** (currently not available)

The following rules apply to the **Axially Constant Clearance** field:

- Valid entries are positive floating point values.



---

## RAPPID User's Guide

---

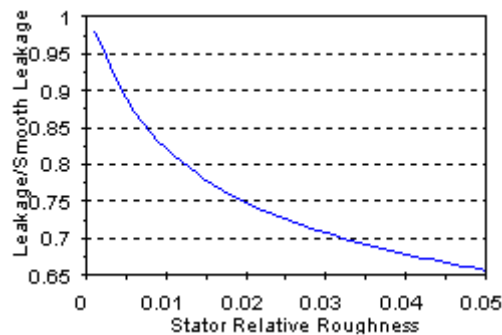
- The current version of the software restricts the **Axially Constant Clearance** entry to a constant value along the length of the bearing.

### Stator Roughness

The **Stator Roughness** represents the absolute roughness of the stator surface and is only available for *2-D Hydrostatic* analysis cases. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer.

**NOTE:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the stator is hydraulically smooth with a surface roughness of 8-32 micro inches. This can change if the stator has been damaged or corroded.

The main factor, other than clearance, which controls cylindrical seal leakage is the stator roughness. The effect of stator roughness on leakage is illustrated the following figure.



**Figure 6. Stator Roughness**

The large values of stator roughness ( $>.02$ ) shown in the figure are only achievable by using patterns such as round holes and diamond knurls. While rotor roughness may also reduce leakage, it has a detrimental effect on rotordynamics.

The following rules apply to the **Stator Roughness** field:

- The program constrains the value to no greater than 256 micro inches, 6.5 microns, or 0.0065 mm.
- If the bearing is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

### Material 1/Material 2

Use **Material 1** and **Material 2** to specify the materials used in the duct.

### Radial Thickness

Use **Radial Thickness** to specify the radial thickness of each corresponding material on the duct.

## *Supply Ports Definition dialog box*

When you click the **Define** button next to **Supply Ports** on the **Geometry** page, the **Supply Ports Definition** dialog box displays.

The following differences exist between *1-D* and *2-D Hydrostatic* analysis cases:

- If the analysis case is a *1-D Hydrostatic* case, edit boxes are provided for entry of the data.
- If the analysis case is a *2-D Hydrostatic*, a grid is provided. The top row of the grid is not editable if you selected **Yes** in the **Recesses** drop-down menu on the **Geometry** page.

The following fields are available depending on the type of analysis case and on the selection in the **Recesses** drop-down menu.

### **Number of Ports**

For *1-D Hydrostatic* analysis cases only, enter the number of ports. The values entered into the remaining edit boxes on the dialog box will be applied to each port.

### **Axial Length**

The **Axial Length** represents the length of the recess along the bearing path coordinate system. For a cylindrical bearing the path coordinate system is coincident with the rotor local coordinate system (i.e., Z-axis).

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu.

The following rules apply to the **Axial Length** field:

- Valid entries are positive floating point values.
- The recess **Axial Length** must be less than the bearing **Length** entered on the **Geometry** page.

### **Axial Location**

The **Axial Location** represents the location of the recess centerline relative to the bearing path length.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu.

The following rules apply to the **Axial Location** field:

- Valid entries are positive floating point values.
- The recess **Axial Location** must be less than (bearing **Length** – recess **Axial Length**/2) and greater than (recess **Axial Length**/2).

---

## RAPPID User's Guide

---

- The recess **Axial Location** must be the same for all ports in a bearing.

### Arc Length

The **Arc Length** represents the angular extent of the recess in the circumferential direction.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu.

The following rules apply to the **Arc Length** field:

- Valid entries are positive floating point values.
- The recess **Arc Length** must be less than (360/number of recesses).

### Arc Location

The **Arc Location** represents the angular location of the recess centerline in the circumferential direction.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu.

The following rules apply to the **Arc Location** field:

- Valid entries are non-negative floating point values.
- The recess **Arc Location** must be set such that the recesses do not overlap.

### Depth

The **Depth** represents the recess depth as measured radially outward from the bearing surface.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu.

The following rules apply to the **Depth** field:

- Valid entries are positive floating point values.
- If a non-uniform recess geometry exists, then an equivalent recess depth, which yields the same volume as the nonuniform geometry should be used as input to the program.

### Orifice Diameter

The **Orifice Diameter** represents the diameter of the orifice in the recess.

The following rules apply to the **Orifice Diameter** field:

- Valid entries include positive floating point values.

- If a non-circular orifice exists, then the flow diameter must be calculated and used as input.
- The **Orifice Diameter** must be less than the **Radial Length**.

### **Relief Diameter**

The **Relief Diameter** represents the diameter of the cylindrical area defined between the exit plane of the orifice and the shaft (also known as the **Curtain Diameter**).

**NOTE:** This field is not available for *I-D Hydrostatic* analysis cases.

The following rules apply to the **Relief Diameter** field:

- Valid entries are positive floating point values.
- The **Relief Diameter** must be less than or equal to the **Axial Length** and greater than or equal to the **Orifice Diameter**.
- The **Relief Diameter** must be the same for all ports in a bearing.

### **Orifice Angle**

The **Orifice Angle** represents the angle of the orifice from the vertical in the circumferential plane.

The following rules apply to the **Orifice Angle** field:

- Valid entries are floating point values in the range –90 to +90 degrees.
- A positive orifice angle would mean that the flow through the orifice exits the orifice in the direction opposite to the rotation of the shaft.

### **Orifice Axial Location**

The **Orifice Axial Location** represents the location of the recess centerline relative to the bearing path length.

**NOTE:** This field is not available for *I-D Hydrostatic* analysis cases.

The following rules apply to the **Orifice Axial Location** field:

- Valid entries are positive floating point values.
- The orifice must lie within the recess.
- The **Orifice Axial Location** must be the same for all ports in a bearing.

### **Orifice Angular Location**

The **Orifice Angular Location** represents the angular location of the *orifice centerline at the orifice exit plane* in the circumferential direction.

**NOTE:** This field is not available for *1-D Hydrostatic* analysis cases.

The following rules apply to the **Orifice Angular Location** field:

- Valid entries are non-negative floating point values.
- The orifice must be located within the boundaries of the recess.

### **Orifice Supply Volume**

The **Orifice Supply Line Volume** represents the volume of the flow channel supplying the orifice.

**NOTE:** This field is not available for *1-D Hydrostatic* analysis cases.

The following rule applies to the **Orifice Supply Volume** field:

- Valid entries are positive floating point values.
- The **Orifice Supply Volume** must be the same for all ports in a bearing.

### **Fluids page**

The **Fluids** page is common to all analysis cases that use a single fluid (compound). Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**
- **Real Properties**

For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

### **Operating Conditions page**

Use the **Operating Conditions** page in a *Journal Bearing* analysis case to set the pressure and temperature boundary conditions, the solution method to obtain position and load, and the angular misalignment (if any).

**NOTE:** If there is a ... button next to an edit box, you may enter variable input for that input field. See **Entering variable data** on page 47 for more information.

## Rotor Speed

Currently, **Rotor Speed** is the analysis parameter by default because it is the only input available as the analysis parameter. Input range and increment for **Rotor Speed** is entered on the **Options** page.

## Pressure/Temperature box

### Fluid Supply Pressure

The **Fluid Supply Pressure** represents the pressure of the fluid upstream of the orifice inlet.

The following rule applies to the **Fluid Supply Pressure** field:

- Valid entries are positive floating point values.

### Fluid Supply Temperature

The **Fluid Supply Temperature** represents the temperature of the fluid upstream of the orifice inlet.

The following rule applies to the **Fluid Supply Temperature** field:

- Valid entries are floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### Left Boundary Pressure

The **Left Boundary Pressure** represents the pressure boundary condition applied to the sector exit plane with the highest z coordinate value.

The following rules apply to the **Left Boundary Pressure** field:

- Valid entries are real number greater than zero.

### Left Boundary Temperature

The **Left Boundary Temperature** represents the temperature of the fluid at the left boundary of the bearing exit plane.

The following rules apply to the **Left Boundary Temperature** field:

- Valid entries are positive floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### Right Boundary Pressure

The **Right Boundary Pressure** represents the pressure of the fluid at the right boundary of the bearing exit plane.

The following rules apply to the **Right Boundary Pressure** field:

- Valid entries are positive floating point values.

## **Right Boundary Temperature**

The **Right Boundary Temperature** represents the temperature of the fluid at the right boundary of the bearing exit plane.

The following rules apply to the **Right Boundary Temperature** field:

- Valid entries are positive floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## **Shaft Temperature**

The **Shaft Temperature** specifies the fixed surface temperature of the shaft in the bearing.

**NOTE:** **Shaft Temperature** is not available for *1-D Hydrostatic* analysis cases.

The following rule applies to the **Shaft Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## **Bearing Back Temperature**

Use **Bearing Back Temperature** to specify the fixed temperature to be applied at the outer surface of the bearing's housing (stationary). The radial thickness of the bearing's housing is the sum of the entries for the following inputs: **Radial Thickness** for **Material 1** and **Radial Thickness** for **Material 2**. The housing radial thickness and the fixed back temperature are assumed to be the same for all sectors. The entered temperature is utilized in the conduction model employed to simulate the radial heat transfer through the bearing's housing.

**NOTE:** **Bearing Back Temperature** is not available for *1-D Hydrostatic* analysis cases.

The following rule applies to the **Bearing Back Temperature** field:

- Valid entries are positive floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## **Exit Inflow Conditions box**

**NOTE:** The **Exit Inflow Conditions** box is not available for *1-D Hydrostatic* analysis cases.

## **Inlet Swirl Ratio**

Use the **Fluid Inlet Swirl Ratio** to specify the ratio of the circumferential velocity of the fluid at the inlet to the bearing to the rotor surface velocity. The value of this term is related to its location in the turbomachine. If no good estimate is available use a value between 0.50-0.80.

## Entrance Loss Coefficient

Use the **Entrance [Inlet] Loss Coefficient** to specify the entrance loss coefficient for flow into the bearing. This is a dimensionless number, and is defined as:

$$\text{Entrance Loss} = \frac{2\Delta p}{\rho V^2} - 1$$

This coefficient is used to determine the pressure loss at a sudden contraction. If no similar experimental data is available, pipe flow contraction coefficients can be calculated using an equation from Blevins (Blevins, R.D., 1984, **Applied Fluid Dynamics Handbook**, Van Nostrand Reinhold Co., New York, p. 76) for pressure loss at pipe contractions:

$$\text{Entrance Loss} = 0.5871 + 0.3954\beta - 4.5385\beta^2 + 14.24 \\ - 19.22\beta^4 + 8.54\beta^5$$

where  $\beta^1$  is the ratio of seal clearance area to the projected area at the entrance to the seal.

## **Loading box**

Use the drop-down menu at the top of the **Loading** box to specify the operation mode for the Program's analysis. There are two options:

- **Prescribed Load**

For the **Prescribed Load** option an initial guess for the journal's equilibrium position is required. If you select **Prescribed Load**, a drop-down menu displays with two options available:

- **Auto Guess:** The **Eccentricity** field and the **Eccentricity Angle** field are not available. The initial guess is generated automatically by the Program using an internal guess algorithm.
- **Manual Guess:** The **Eccentricity** field and the **Eccentricity Angle** field are available. Enter an initial guess into each field manually.

If you select **Prescribed Load**, the **Convergence Criteria**, **Journal Weight**, **Applied Load**, and **Applied Load Direction** fields are available.

- **Prescribed Eccentricity**

If you select **Prescribed Eccentricity**, the **Eccentricity** and **Eccentricity Angle** fields are available.

Specify the following values as applicable.



## **Eccentricity**

Use **Eccentricity** to define the magnitude of the displacement vector of the rotor from the concentric shaft centerline.

- For **Prescribed Eccentricity**, this value will set the position of the shaft for the solution.
- For **Prescribed Load**, this value will be the initial guess of the shaft position for the solution.

The following rules apply to the **Eccentricity** field:

- Valid entries are positive floating point values.
- The **Eccentricity** must be less than the bearing **Clearance**.

## **Eccentricity Angle**

Use **Eccentricity Angle** to specify the angle of the displacement vector of the rotor from the shaft centerline as measured from the positive X-axis.

The following rule applies to the **Eccentricity Angle** field:

- Valid entries are positive floating point values that are less than 360°.

## **Convergence Criteria**

Use **Convergence Criteria** to specify the percent error in the calculated load relative to the **Prescribed Load**.

**NOTE: Convergence Criteria** is not available for **Prescribed Eccentricity**.

The following rule applies to **Convergence Criteria**:

- Valid entries are positive floating point values.

## **Journal Weight**

Use **Journal Weight** to define the load applied to the bearing in the direction of gravity (along the negative Y-axis, or at 270°). **Journal Weight** represents the weight of the rotor supported by this bearing.

**NOTE: Journal Weight** is not available for **Prescribed Eccentricity**.

The following rule applies to **Journal Weight**:

- Valid entries are non-negative floating point values.

## **Applied Load**

Use **Applied Load** to define the magnitude of the load vector applied to the bearing in addition to the user input **Journal Weight**.

**NOTE:** **Applied Load** is not available for **Prescribed Eccentricity**.

The following rule applies to **Applied Load**:

- Valid entries are non-negative floating point values.

**NOTE:** The applied load vector is intended to be used to represent arbitrarily orientated loads (i.e. impeller side loads) that are applied to the bearing. The load applied to the bearing during the program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in **Applied Load** and **Applied Load Direction**).

## **Applied Load Direction**

Use **Applied Load Direction** to define the angular orientation of the applied load vector. This angle is relative to the positive X-axis and is positive in the counter-clockwise direction. The magnitude of the applied load vector is set by the value entered for the **Applied Load**.

**NOTE:** **Applied Load Direction** is not available for **Prescribed Eccentricity**.

The following rule applies to **Applied Load Direction**:

- Valid entries are real numbers between -360° and 360°, inclusive.

## **Misalignment box**

### **Magnitude**

Use **Magnitude** in the **Misalignment** box to define the magnitude of the angular misalignment of the shaft relative to the shaft centerline.

**NOTE:** The **Misalignment** box is not available for *1-D Hydrostatic* analysis cases.

The following rule applies to **Magnitude**:

- Valid entries are positive floating point values.

### **Angle**

Use **Angle** in the **Misalignment** box to define the angular orientation of the angular misalignment vector relative to the X-axis.

**NOTE:** The **Misalignment** box is not available for *1-D Hydrostatic* analysis cases.

The following rule applies to **Angle**:

- Valid entries are real numbers between  $-360^\circ$  and  $360^\circ$ , inclusive.

### **Center**

Use **Center** in the **Misalignment** box to define the center of rotation of the angular misalignment relative to the bearing axial centerline.

**NOTE:** The **Misalignment** box is not available for *1-D Hydrostatic* analysis cases.

The following rule applies to **Center**:

- Valid entries are floating point values

## **Chapter 15: Journal Bearing – 2-D Fixed Geometry**

### **Overview**

The *Journal Bearings - 2-D Fixed Geometry* module is for analysis of radially fed sleeve, lobe, partial arc, pressure dam, and tapered land fluid film journal bearings with axial and/or circumferential grooves. Its primary function is to allow the designer to determine rotordynamic coefficients for various fixed geometry bearing styles in an effort to select a suitable bearing for the machinery application under consideration.

The *Journal Bearings - 2-D Fixed Geometry* module calculates stiffness, damping, and mass coefficients by taking your input and solving a full set of film averaged Navier-Stokes based governing equations: momentum, continuity, and energy. The flow in the bearing may be laminar, transitional, or turbulent. Rotordynamic coefficients are determined using a perturbation analysis which assumes the journal is exhibiting small amplitude cylindrical motion about its static equilibrium position. The *Journal Bearings - 2-D Fixed Geometry* module also reports whirl frequency ratio, operating eccentricity, eccentricity angle, maximum film pressure, fluid temperature rise, maximum bearing temperature, minimum film thickness, required fluid flow rate, and power consumption.

### **Bearing Geometry**

Use *Journal Bearings - 2-D Fixed Geometry* to analyze the following bearing types:

- Sleeve Bearings
- Lobe Bearings
- Partial Arc Bearings
- Pressure Dam Bearings
- Tapered Land Bearings

All bearings may be defined with axial and/or circumferential grooves. The grooves divide the bearing's surface into sectors consisting of a bearing surface followed by an axial groove. The Program currently allows the user to define two different types of sectors: single arc and recessed. Single arc sectors are used to define bearing surfaces which can be described using a single circular arc. Such bearings include sleeve, lobe and partial arc bearings. Recessed sectors are used to describe bearing surfaces which are comprised of two separate circular arcs. Such bearings include pressure dam and tapered land bearings.

For example, a conventional 3 lobe bearing with 3 axial grooves, would be defined using 3 sectors. However, a 3 lobe bearing with 3 axial grooves and 1 circumferential groove would be defined using 6 sectors. An unlimited number of sectors may be used to define a bearing. A conventional pressure dam bearing with two axial grooves would be defined with 1 single arc sector and 1 recessed sector. An unlimited number of sectors can be used to define a bearing.

Each sector can be defined with individual clearance, preload, offset, arc length, axial length, axial location, axial groove depth parameters. In addition, recessed sectors also have inputs to define the axial length, axial location, arc length, pad diameter, eccentricity, and eccentricity angle to define the recess.

### Thermal model

The heat generated in the bearing is dissipated via convection to the shaft, convection to the housing, and hot fluid exiting each sector. Hot fluid exiting a sector circumferentially enters the sector's axial feed groove and will either be completely drained from the bearing or carried over into the next (circumferentially) sector. The heat convected to the bearing surface is conducted through the bearing's housing to a constant temperature at the surface. Radiation and convection from the bearing housing surface to the atmosphere is not considered. This thermal model requires you to enter the following information:

- Shaft Surface Temperature
- Bearing Housing Temperature at Outside Diameter
- Bearing Housing Radial Thickness
- Bearing Housing Conductivity

The housing conduction model allows you to define the bearing housing using two separate materials acting in series. This modeling capability allows you to account for the different conduction rates between the bearing surface material (i.e. babbitt) and the housing material (i.e. steel).

### Thermal and centrifugal growth

Under operating conditions the stationary bearing surface may experience a change in diameter due to thermal growth and/or pressure on its outer diameter (i.e. from a press fit). As these effects are installation dependent, you must account for them by providing the **actual bearing surface diameter under the specified operating conditions** to *Journal Bearings - 2-D Fixed Geometry* (either directly or through the clearance input).

Under operating conditions the rotating bearing surface (journal) may experience a change in diameter due to thermal and centrifugal growth, and/or the presence of a press fitted shaft sleeve. As these effects are installation dependent, you must account for them by providing the **actual journal diameter under the specified operating conditions** to *Journal Bearings - 2-D Fixed Geometry*.

To determine the actual bearing surface and journal diameters under operating conditions you may utilize a separate RSR program called FIT. FIT produces output consistent with that required for entry into *Journal Bearings - 2-D Fixed Geometry*.

### Groove model

Each sector is comprised of a bearing surface followed by an axial groove. The sector's circumferential sides are either bordered by ambient pressure or a circumferential groove. If the sector is not bordered circumferentially by another sector (over any fraction of its arc length), fluid is allowed to leak from the

entire arc length of the bearing surface axially to the user-defined ambient pressure. If the sector is bordered circumferentially by a groove (over any fraction of its arc length), fluid is allowed to leak from the entire arc length of the bearing surface axially to the user-defined supply pressure. All grooves, axial and circumferential, are fed with fluid at the same temperature and pressure.

For bearings without circumferential grooves, the Program includes a mixing model for the axial grooves to account for hot fluid carry over from the previous sector (circumferentially) mixing with new fluid supplied at the user-defined supply temperature. This model assumes the grooves are sealed axially. You have the option of considering hot fluid carry over (**Iterate** on sector inlet temperature) or performing an analysis assuming each sector is supplied with fluid at a **Fixed** user-defined supply temperature.

For bearings without circumferential grooves, the Program includes a model to account for the ram effect of the fluid entering the sector with an initial circumferential velocity. The groove mixing model, if activated, accounts for the loss in circumferential velocity due to the introduction of fresh fluid radially.

For bearings with circumferential grooves, the current version of the Program assumes that each sector is supplied with fresh fluid continuously from the axial grooves, i.e. there is no hot fluid carry over or ram effect from adjoining sectors. To simulate the effect of hot fluid carry over, you can artificially raise the supply temperature of the fluid.

### Applied Load/Prescribed Eccentricity

After the bearing sectors are defined, you have the option of prescribing an applied load vector or enforcing a prescribed eccentricity. If you enter an applied load, the Program will iteratively determine the static equilibrium position of the journal within the bearing. This iterative solution requires you to enter an initial guess for the shaft equilibrium position. To assist with this task, this module has an automated initial guess generation algorithm which is suitable for most cases. However, in the event the program does not converge to an appropriate solution with the automatic initial position guess, you may enter an initial position guess manually.

In the event you elect to enforce an eccentricity, the Program will solve the fluid film model and report the reaction load vector produced by the fluid film when the journal occupies the prescribed position.

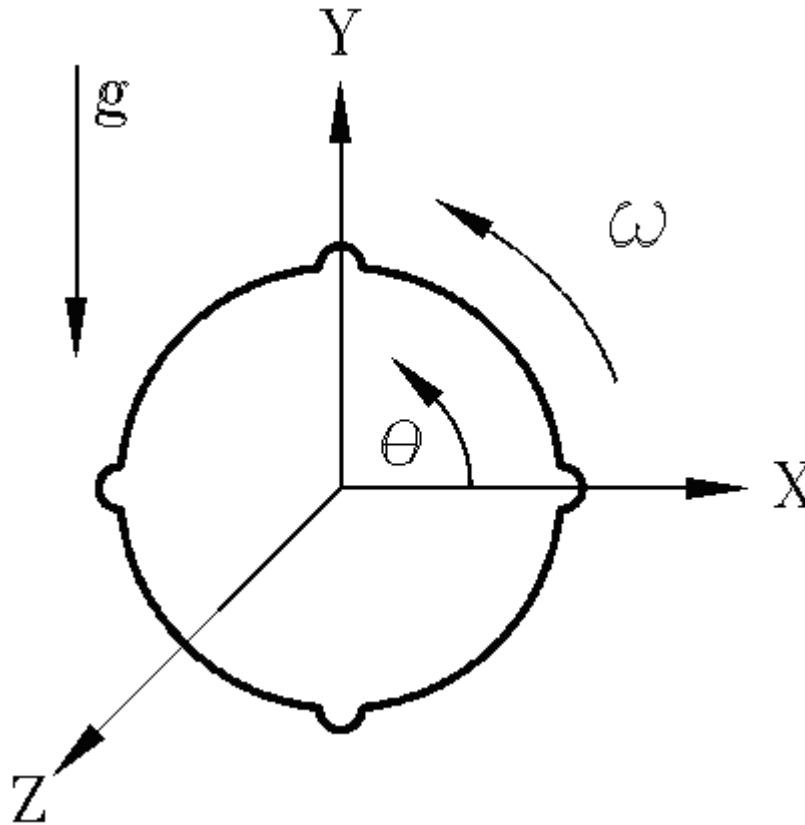
### Fluid Film gridding

*Journal Bearings - 2-D Fixed Geometry* contains an automatic grid generation algorithm for the fluid film solution. You have three options for the grid setting: **Course**, **Normal**, and **Fine**.

- Use the **Normal** grid for most cases.
- Use the **Course** grid to speed up the program during the process of qualitatively comparing numerous bearing designs.
- Use the **Fine** grid when the bearing's operating eccentricity ( $e/C_p$ ) is greater than 0.7, as these high eccentricities may require additional grid points to capture the high gradients present in the fluid film.

## Coordinate system

All angles utilized in *Journal Bearings - 2-D Fixed Geometry* (input and output) are referenced to a right handed coordinate system with the positive X-axis orientated at standard position. This mandates that shaft rotation is counter-clockwise and that gravity acts in the direction of the negative Y-axis (see the following figure).



*Figure 7. Coordinate system*

## Conventions for defining sectors

The following rules must be followed when defining sectors in *Journal Bearings - 2-D Fixed Geometry*:

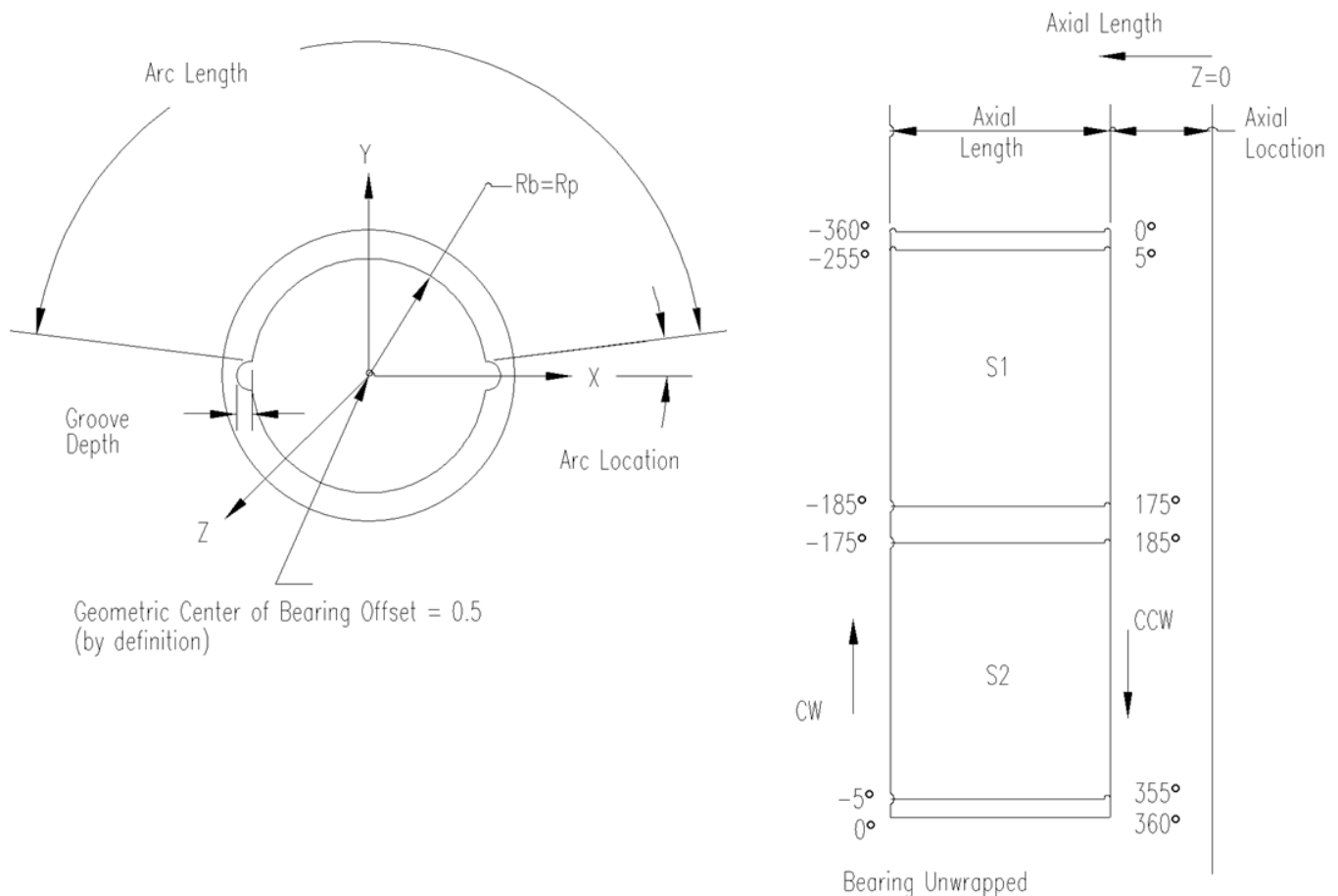
1. The first full axial groove encountered while rotating counter-clockwise from the positive X-axis must be associated with the first sector defined (remember the load bearing surface of a sector precedes it circumferentially). If a bearing has a groove which traverses the positive X-axis said groove is not the first full groove encountered (because part of the groove resides in the fourth quadrant).
2. The Program's mixing model assumes that fluid exiting sector 1 enters sector 2, fluid exiting sector 2 enters sector 3, ....., fluid exiting the last sector enters sector 1. This convention defines the order in which sectors must be defined.
3. No sector's bearing surface can have a end angle (start angle + arc length) greater than 360°. For example, if the first axial groove occurs at 45° and is associated with a 110° (Arc Length)

bearing surface, the sector's starting angular orientation (Arc Location) must be entered as  $-65^\circ$  (not  $295^\circ$ ).

## ***Sector Definition - Single Arc Sectors***

The single arc sector is used to model bearings which feature bearing surfaces that can be defined by a single circular arc segment. Such bearings include sleeve, lobed, and partial arc configurations. The Program requires you to enter the following values to define each single arc sector: axial length, axial position, arc length, arc location, groove depth, offset, bearing clearance (or bearing diameter), pad clearance (or pad diameter), and/or preload.

The values entered are illustrated in the following figure for a circular bore bearing.

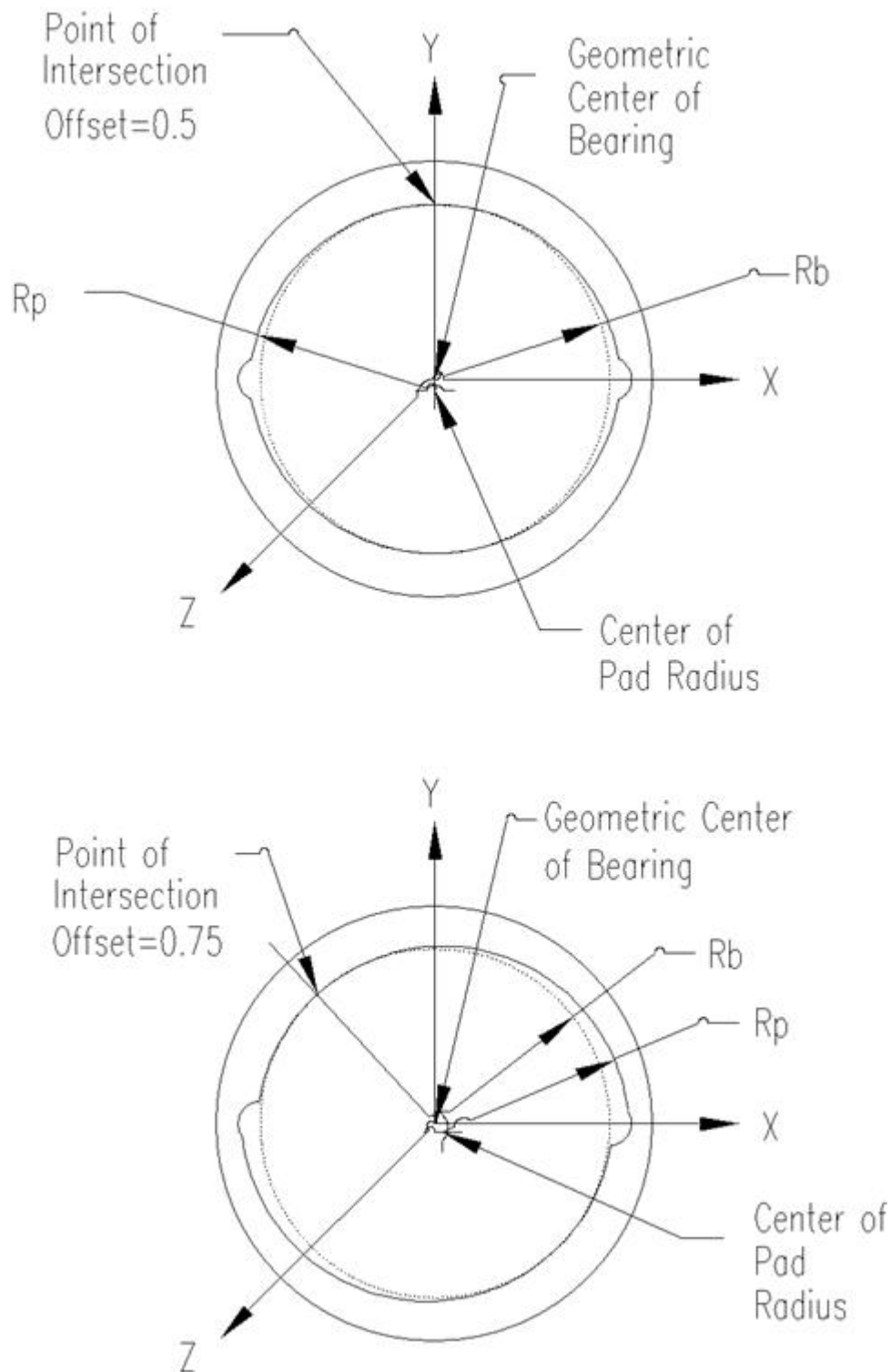


***Figure 8. Circular bore bearing***



## RAPPID User's Guide

The values entered are illustrated in the following figure for a preloaded bore bearing.



**Figure 9. Preloaded bore bearing**

For further discussion of the options for defining the bearing clearance/diameter, see **Clearance Input** on page 179.

Offset is the non-dimensional angular location where the bearing surface (described by the pad diameter/radius [ $R_p$ ]) intersects the imaginary circular bearing bore (described by the bearing diameter/radius [ $R_b$ ]). Offset is equal to the **Angular Location** of this intersection (relative to the start of the sector) divided by the **Arc Length** of the load bearing portion of the sector (excluding the arc length of the groove). For a bearing with a circular bore ( $R_b = R_p$ ), the offset is defined to be 0.5. For a bearing that is preloaded ( $R_p > R_b$ ), the offset may vary between 0 and 1. Figure 9 illustrates a two lobe bearing with an offset of 0.5 (upper) and an offset of 0.75 (lower).

## Sector Definition - Recessed Sectors

The recessed sector is used to model bearings with bearing surfaces that require two circular arc segments to define the geometry. Such bearings include pressure dam and tapered land configurations. In addition to the values required to define a single arc sector, the Program requires you to enter the following values to define the recessed portion of each recessed sector: **Axial Length**, **Axial Position** (**Axial Location**), **Arc Length**, **Pad Diameter**, **Eccentricity**, and **Eccentricity Angle**. These values are illustrated in the following figure for a generic recessed bearing. Note that the recess is constrained to start at the angular location prescribed by you for the first arc.

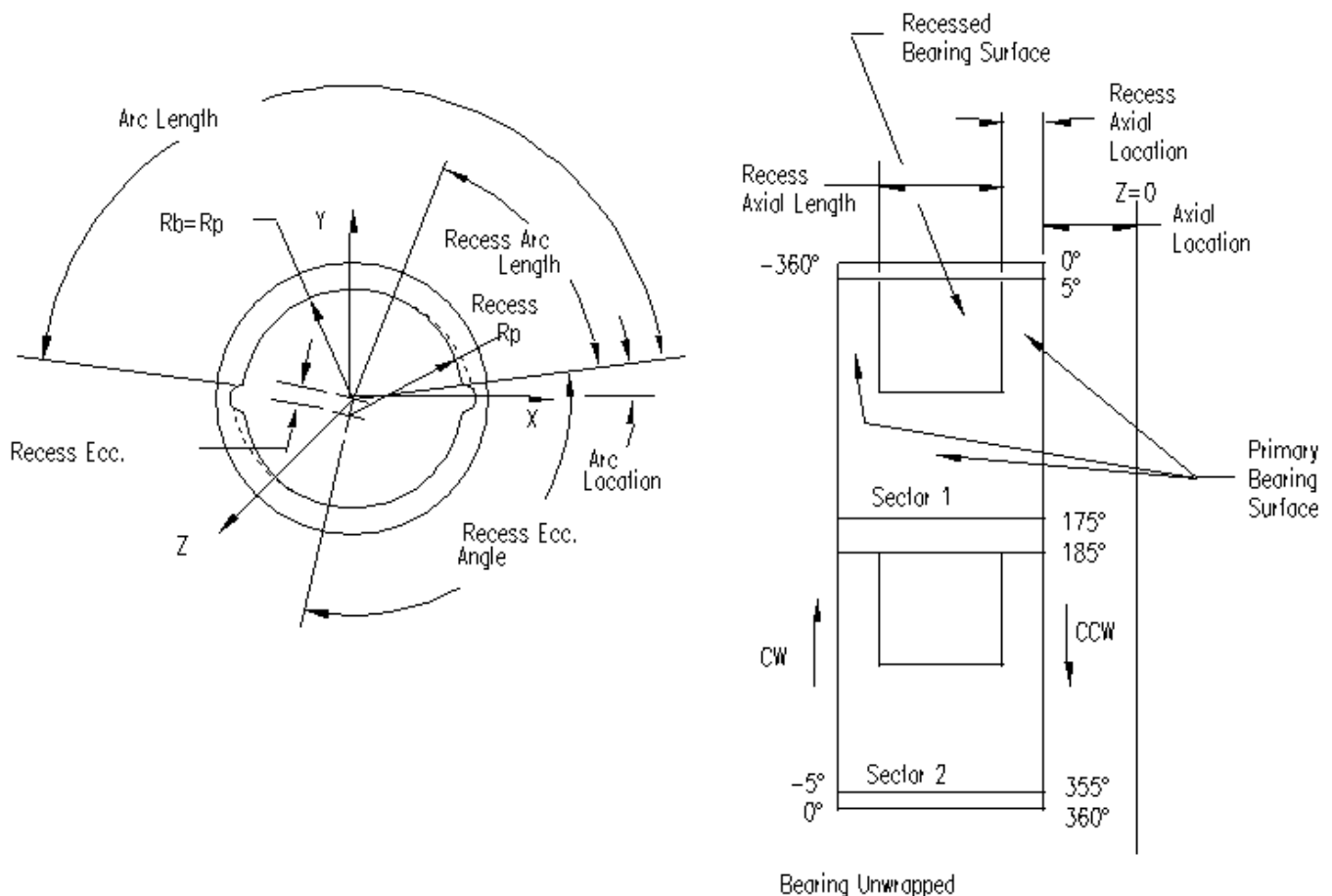


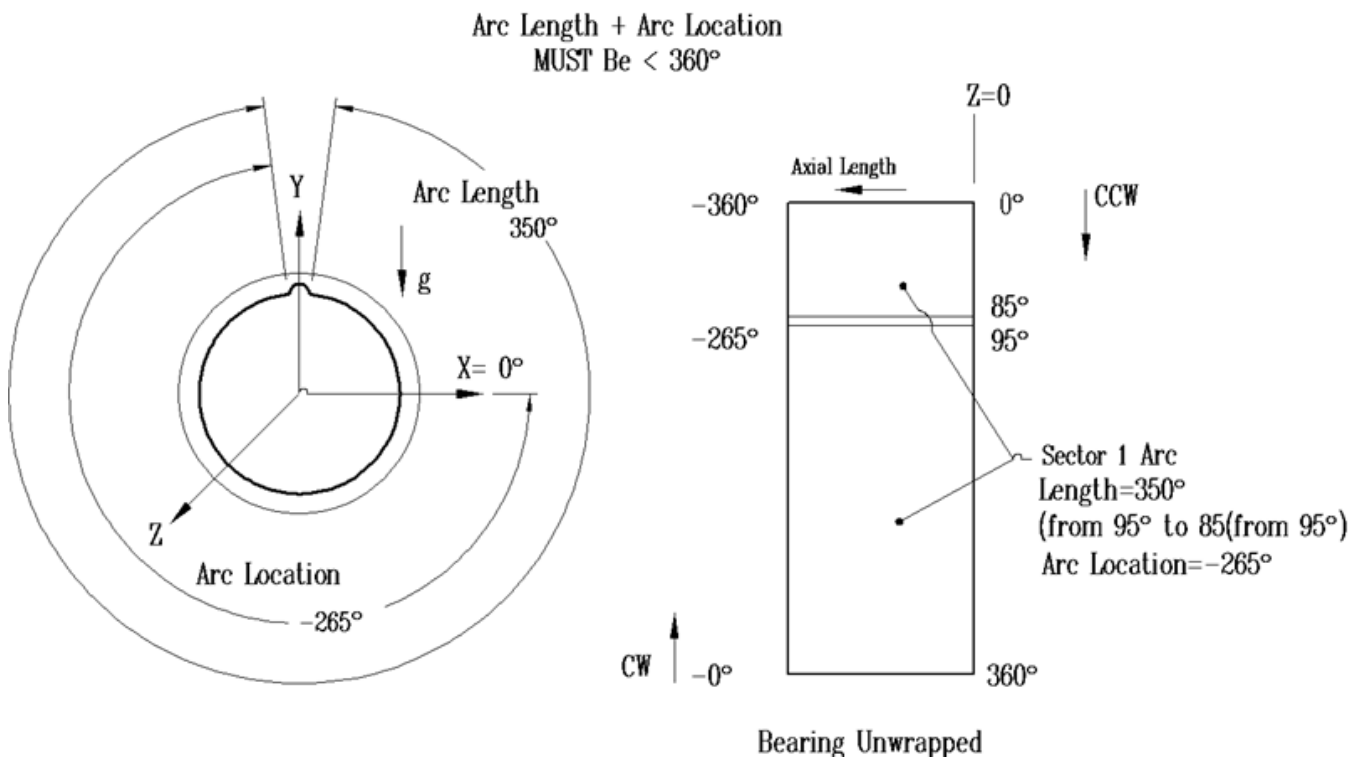
Figure 10. Recessed sector geometry

## *Conventional Bearing Configurations (Axial Grooves only, Single Arc Bearing Surface)*

This section presents several examples of how typical bearing configurations are modeled in the Program. The bearings described in this section have axial grooves (for circumferential grooves, see **Bearing Configurations with Axial and Circumferential Grooves** on page 173) and bearing surfaces that can be defined by one circular arc (for bearing surfaces defined with two circular arcs see **Bearing Configurations with Recessed Sectors (Pressure Dam and Tapered Land)** on page 176).

### 360° Journal Bearing

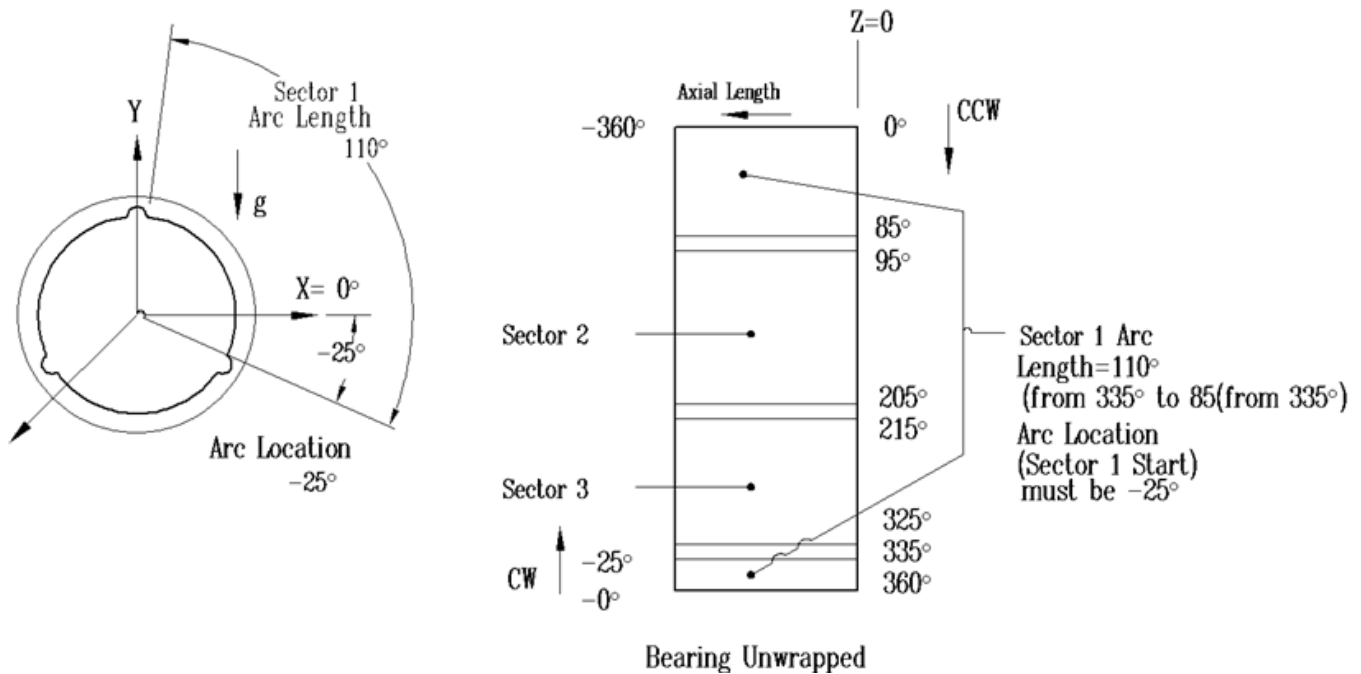
This type of bearing is defined using 1 rectangular sector. As the Program assumes the bearing is fed radially, a groove is required (as small as desired), i.e., the sector's arc length must be less than 360° (see the following figure).



*Figure 11. 360° Journal Bearing configuration*

### Multi-Lobe Bearings

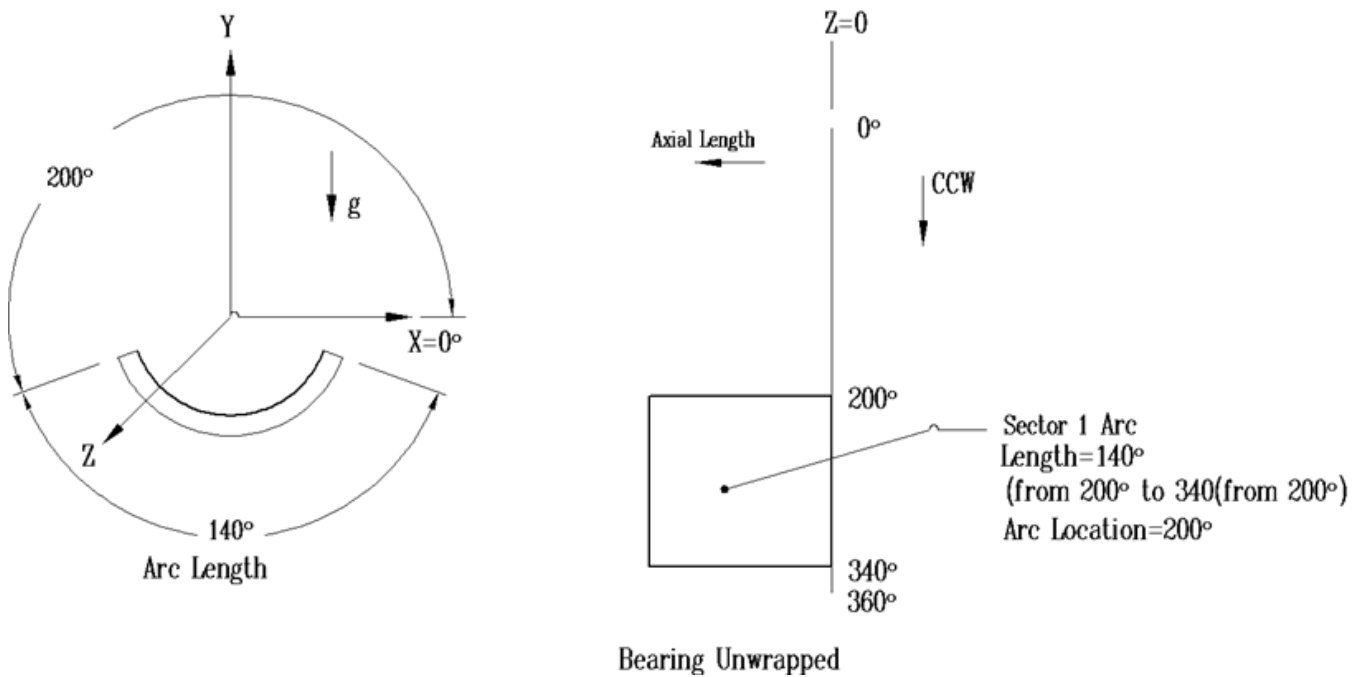
This type of bearing is defined using X rectangular sectors, where X is the number of lobes. A typical three lobe bearing is shown in the following figure.



**Figure 12. Three lobe bearing configuration**

## Partial Arc Bearings

This type of bearing is defined using 1 rectangular sector. A typical partial arc bearing is illustrated in the following figure.

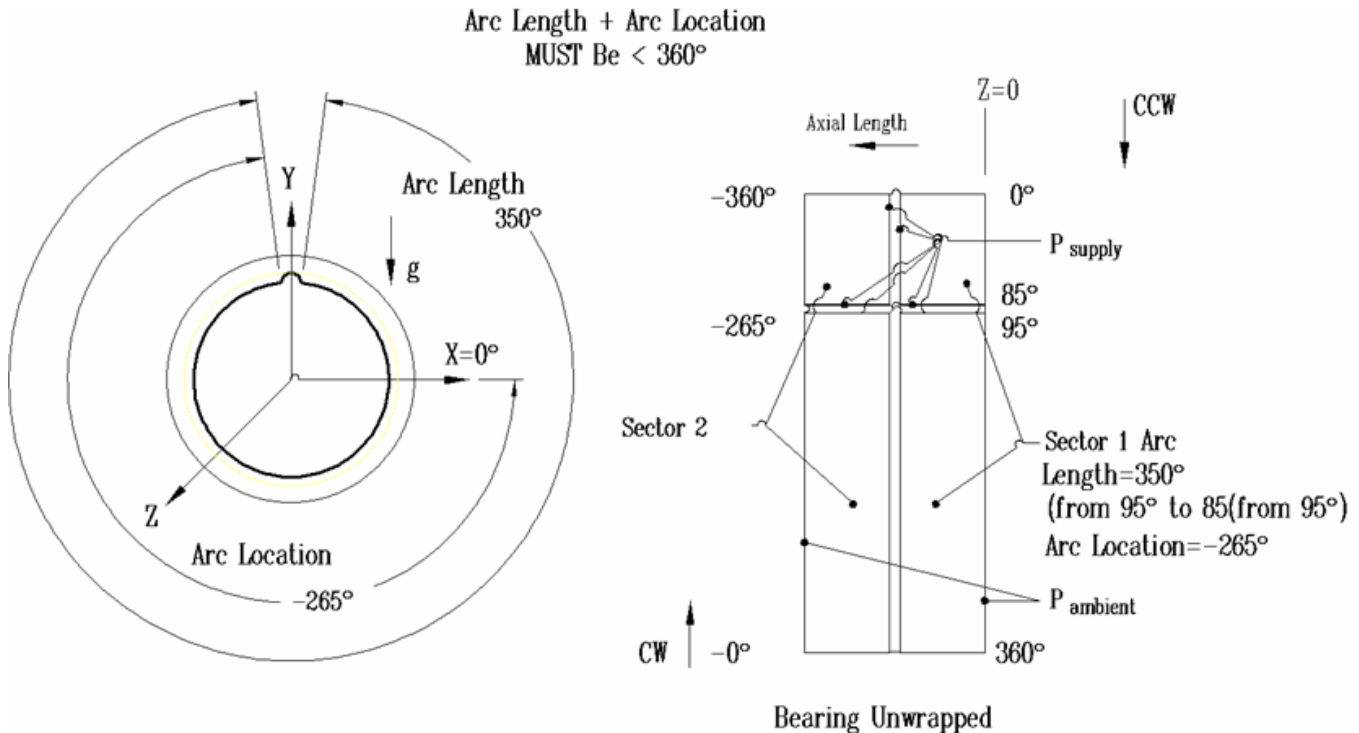


**Figure 13. Partial arc bearing configuration**

# RAPPID User's Guide

## 360° Journal Bearing – Circumferential Groove

To model a 360° bearing with a circumferential groove, you must utilize two sectors. Since the program requires each sector to have an axial groove, the actual bearing analyzed has 1 axial groove and 1 circumferential groove as shown in the figure below. Note how the pressure boundary conditions are applied to the individual sector boundaries. Note that there is no requirement for the axial grooves of sectors 1 and 2 to be aligned (circumferentially).



*Figure 14. 360° Journal Bearing configuration with a circumferential groove*

## Multi-Lobe Bearings – Circumferential Groove

These bearings may also have circumferential grooves. Consider the following selected examples:

**Example 1: 3-lobe bearing with two circumferential grooves:** This bearing requires use of 9 sectors, as shown in the following figure.

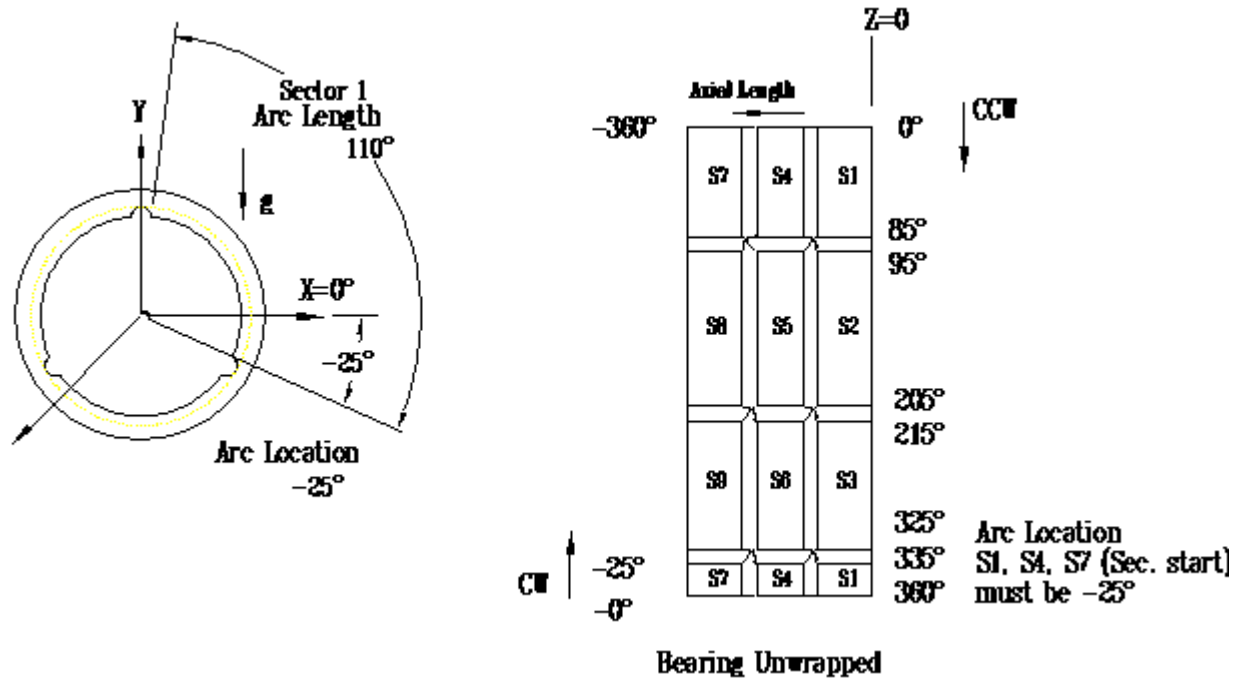


Figure 15. 3-lobe bearing with two circumferential grooves

**Example 2: 2-lobe bearing with one partial circumferential groove:** This bearing requires use of 3 sectors, as shown in the following figure.

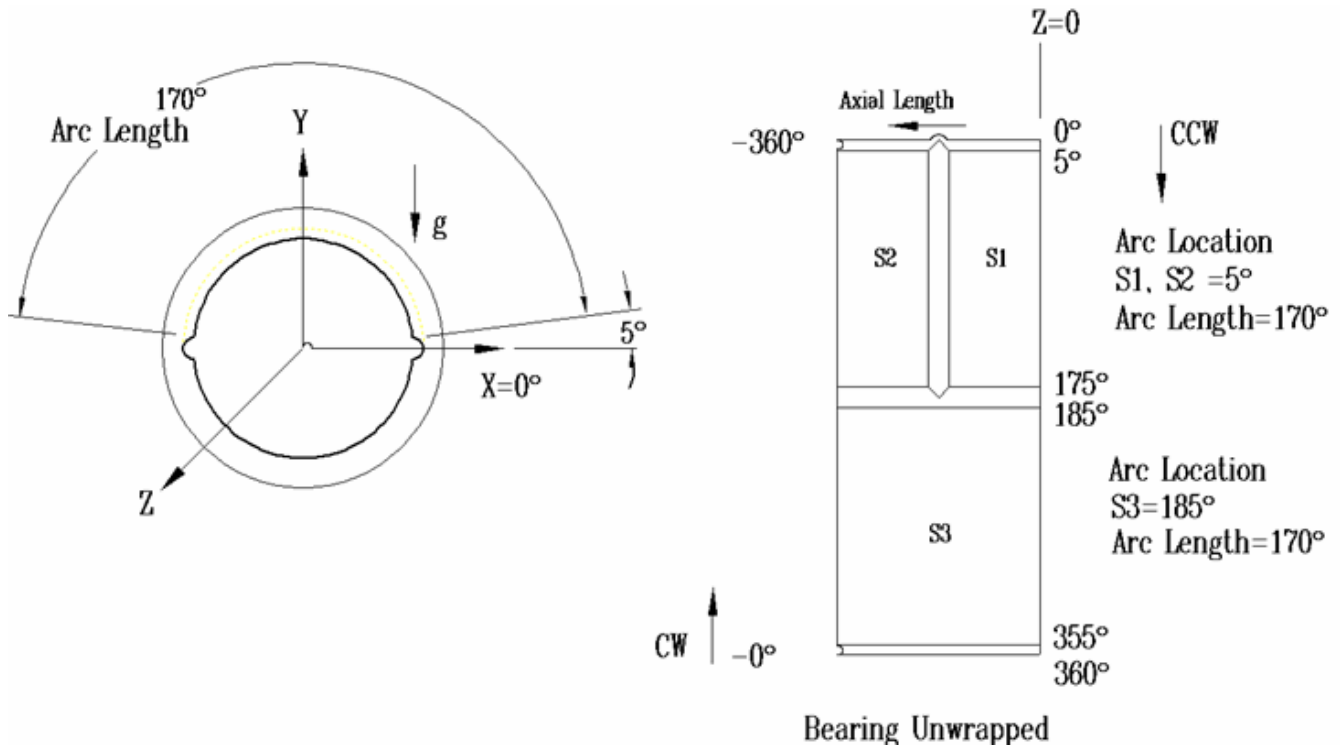
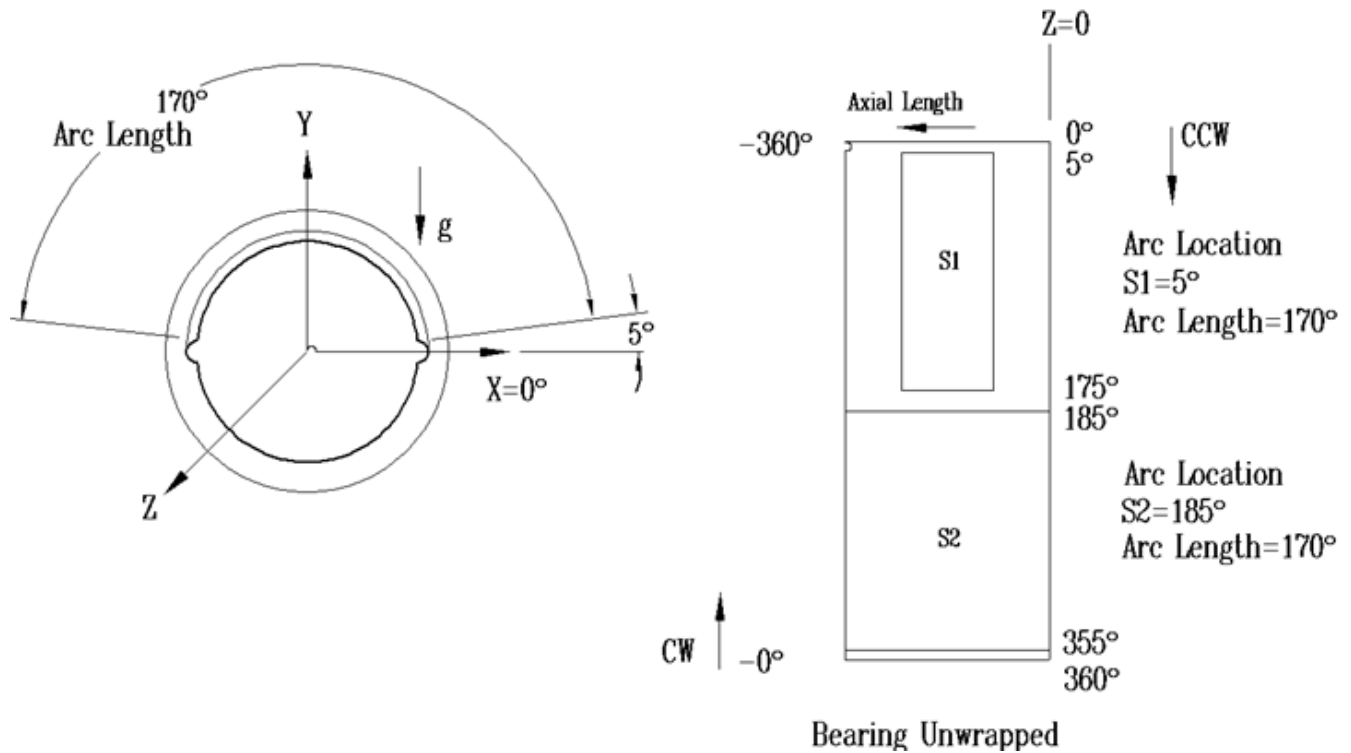


Figure 16. 2-lobe bearing with one partial circumferential groove

**Example 3: 2-lobe bearing with a partial axial length lobe:** This bearing requires use of only two sectors, as shown in the following figure.

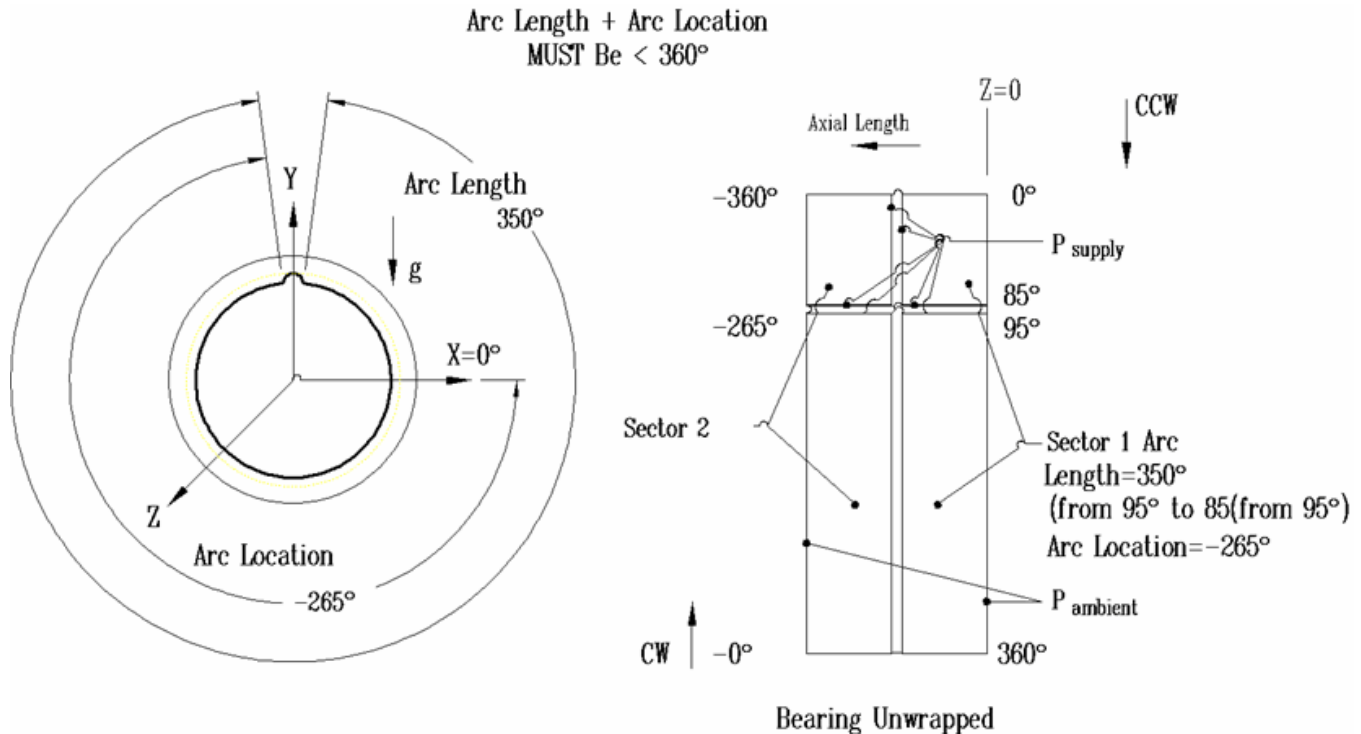


*Figure 17. 2-lobe bearing with a partial axial length lobe*

## *Bearing Configurations with Axial and Circumferential Grooves*

### **360° Journal Bearing**

To model a 360° bearing with a circumferential groove, you must utilize two sectors. Since the Program requires each sector to have an axial groove, the actual bearing analyzed has 1 axial groove and 1 circumferential groove as shown in the figure below. Note how the pressure boundary conditions are applied to the individual sector boundaries. Also, note that there is no requirement for the axial grooves of sectors 1 and 2 to be aligned (circumferentially).



**Figure 18. 360° Journal Bearing configuration**

### **Multi-Lobe Bearings**

These bearings may also have circumferential grooves. Consider the following selected examples:



# RAPPID User's Guide

**Example 1: 3-lobe bearing with two circumferential grooves:** This bearing requires use of 9 sectors, as shown in the following figure.

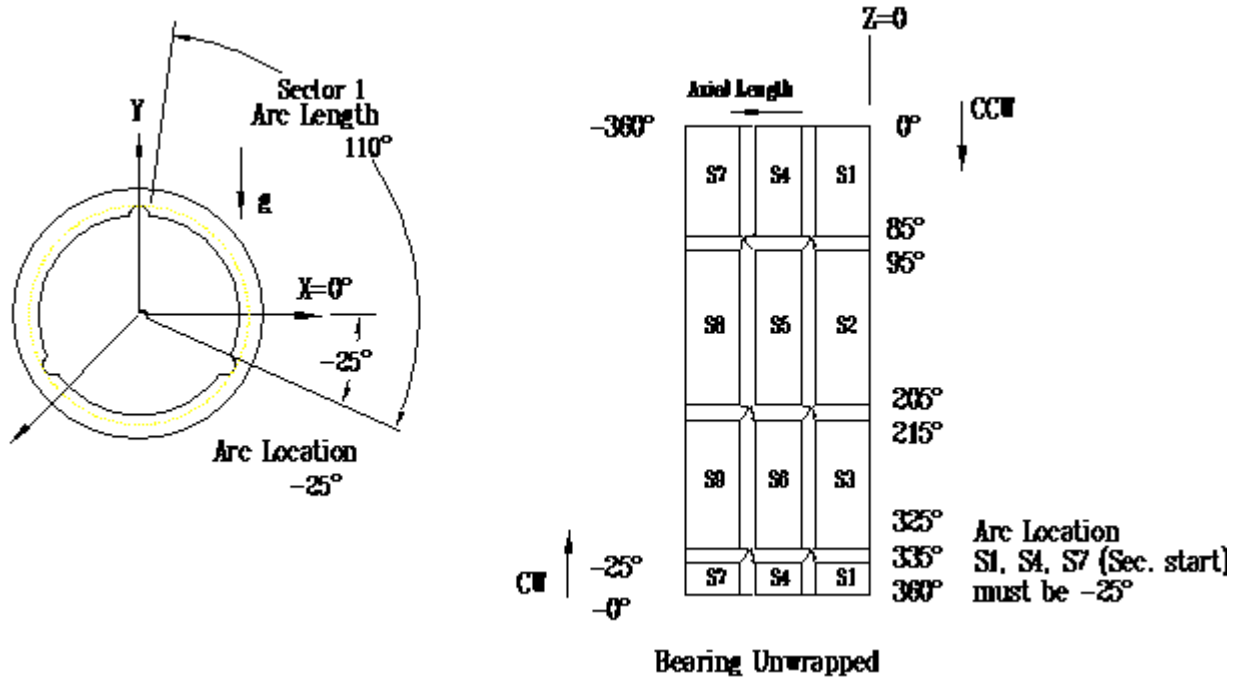
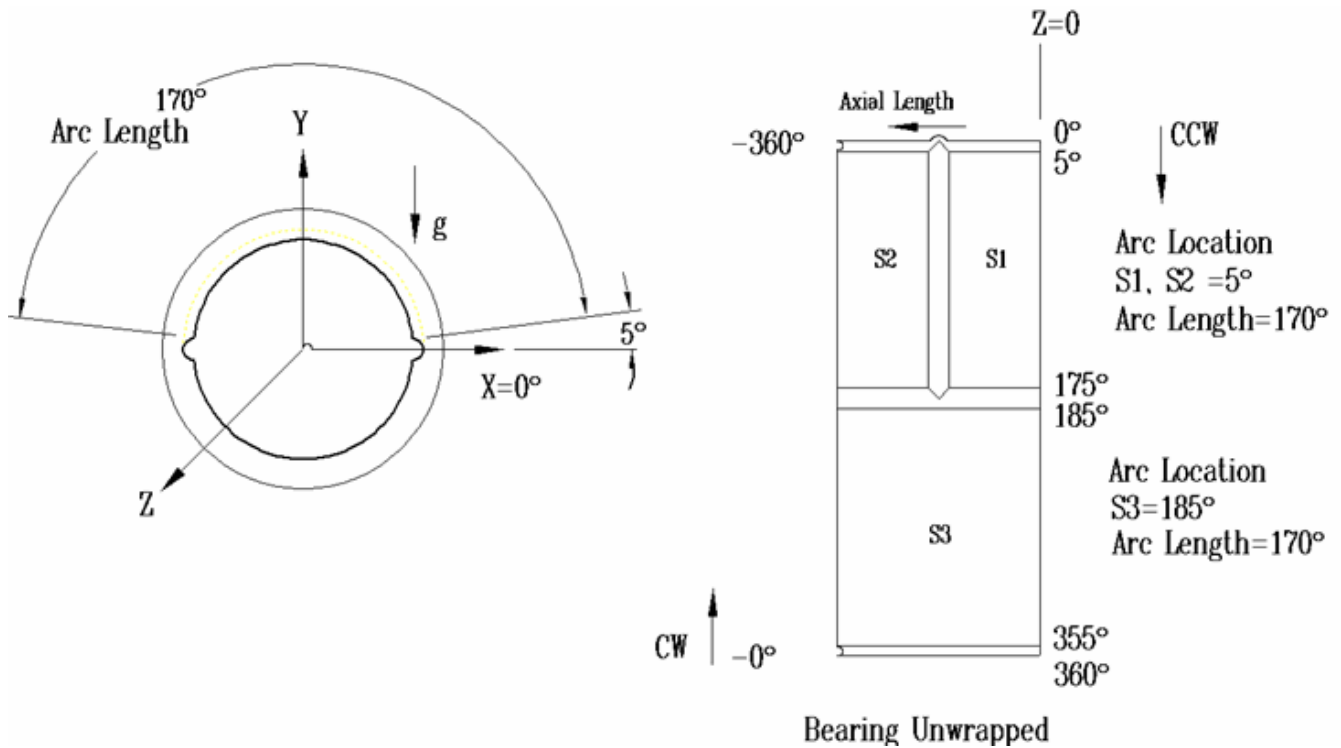


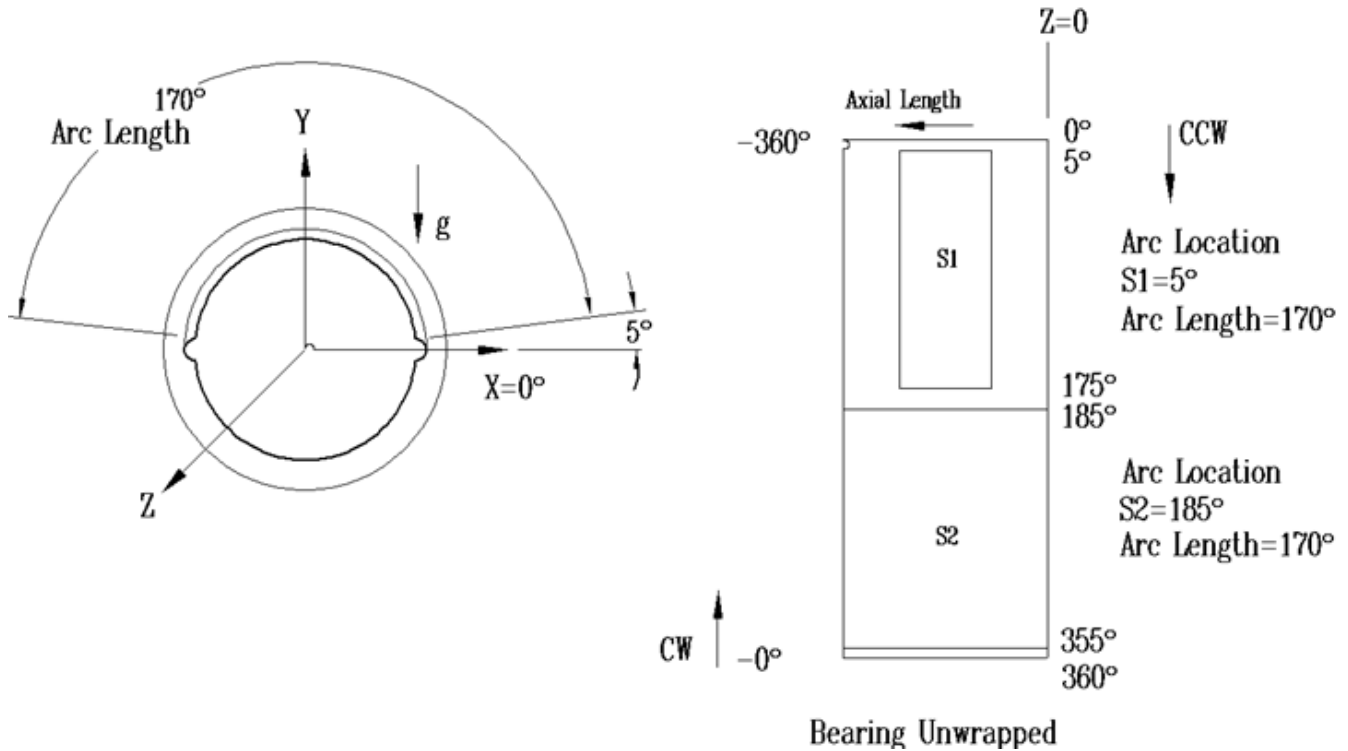
Figure 19. 3-lobe bearing with two circumferential grooves

**Example 2: 2-lobe bearing with one partial circumferential groove:** This bearing requires use of 3 sectors, as shown in the following figure.



*Figure 20. 2-lobe bearing with one partial circumferential groove*

**Example 3: 2-lobe bearing with a partial axial length lobe:** This bearing requires use of only two sectors, as shown in the following figure.



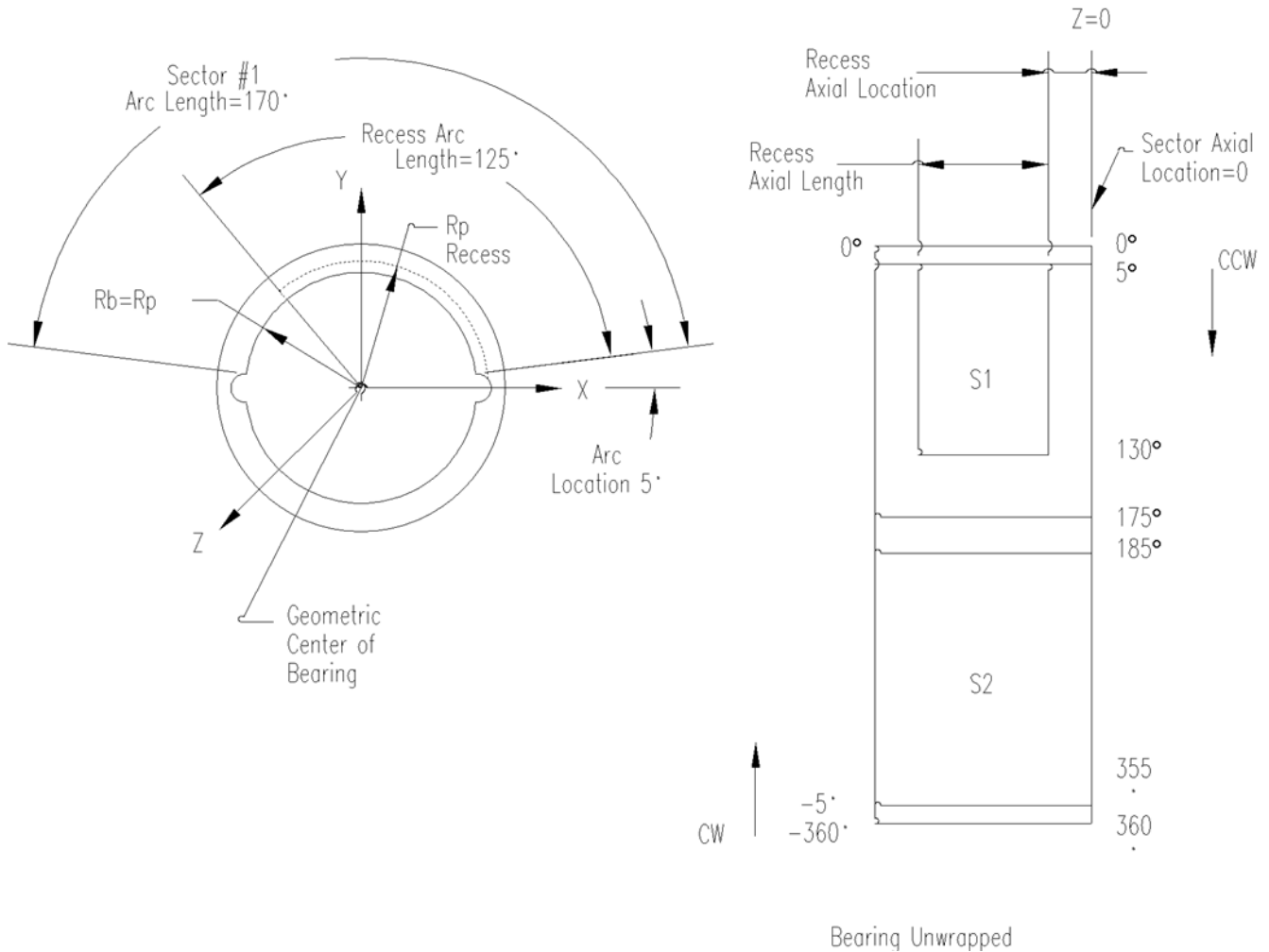
**Figure 21. 2-lobe bearing with a partial axial length lobe**

## ***Bearing Configurations with Recessed Sectors (Pressure Dam and Tapered Land)***

The Program employs a second type of sector, referred to as a recessed sector, to define bearing surfaces that require two separate circular arcs to define the clearance profile. Typical bearings that fall into this category are pressure dam and tapered land bearings.

## Pressure Dam Bearings

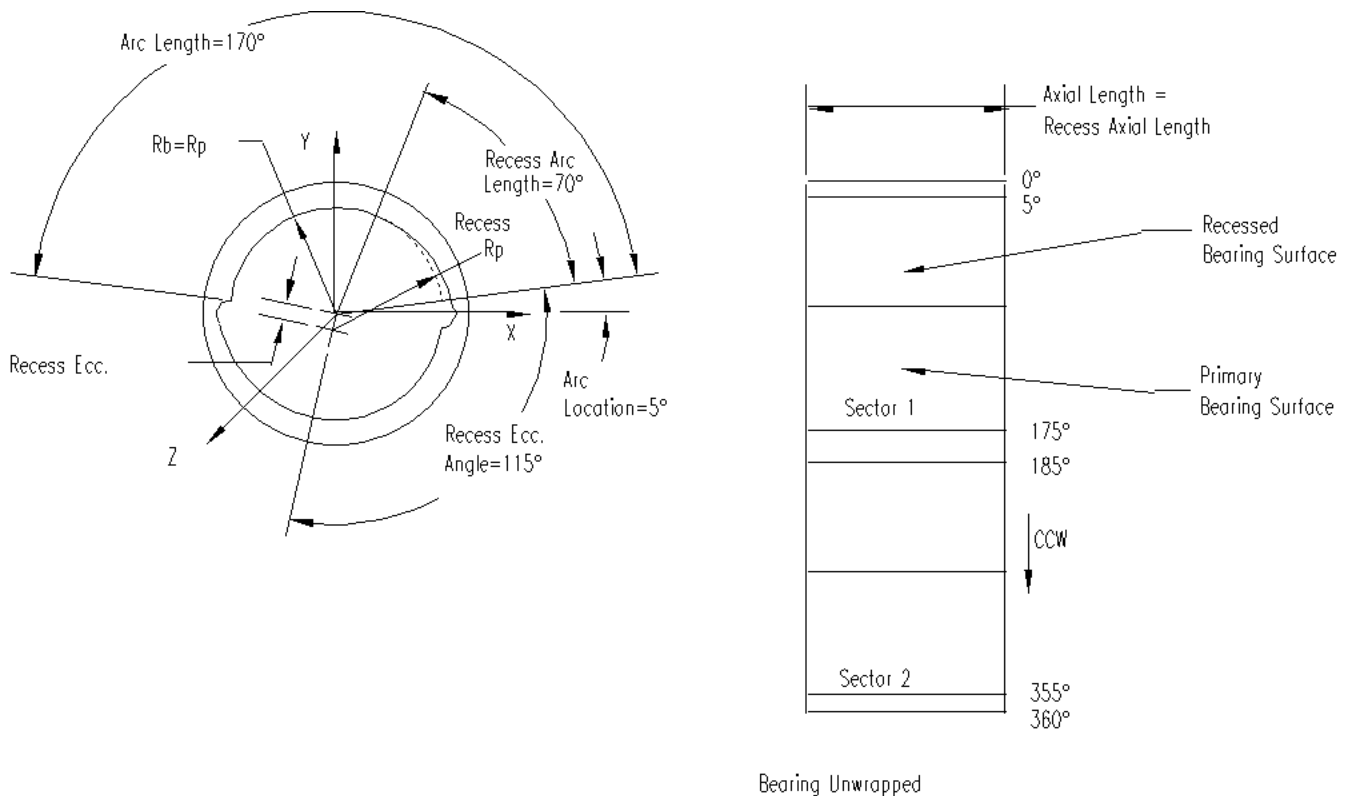
This type of bearing is defined using one single arc sector and one recessed sector. A typical pressure dam bearing (constant clearance recess) is illustrated in the following figure.



**Figure 22. Pressure dam bearing**

## Tapered Land Bearings

This type of bearing is defined using  $N$  recessed sectors, where  $N$  is the number of lobes in the bearing. A typical two lobe tapered land bearing is shown in the following figure.



**Figure 23. Tapered land bearing**

## Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

## Analysis Parameter box

Use the **Analysis Parameter** drop-down menu to select the input value by which you want to vary the values of other inputs. Currently, **Rotor Speed** is the only available **Analysis Parameter**.

The **Rotor Speed** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting speed (**Start**), the increment of speed (**Delta**), and the final speed (**Stop**) for the calculation. The **Rotor Speed** fields must define a constant or increasing speed. Therefore, the **Stop** speed must be greater than or equal to the **Start** speed and the **Delta** speed must be positive. Field entries must be real numbers greater than zero.

## Output Format

Use the **Output Format** drop-down menu to select whether to generate a **Short** or a **Long** output file.

- The **Short** output format includes a listing of the input variables and the calculated rotordynamic coefficients.
- The **Long** output format includes whirl frequency ratio, operating eccentricity, eccentricity angle, maximum film pressure, fluid temperature rise, maximum bearing temperature, minimum film thickness, required fluid flow rate, and power consumption, in addition to a listing of the input variables and the calculated rotordynamic coefficients.

## Grid

Use the **Grid** drop-down menu to indicate whether a **Coarse**, **Normal**, or **Fine** grid is used in the solution.

See **Fluid Film gridding** on page 163 for more information.

## Groove Temp

Use the **Groove Temp** drop-down menu to specify whether hot fluid carryover is considered in the solution.

- If you select **Fixed (Supply)**, the Program sets the inlet temperature for each sector equal to the specified **Fluid Supply Temperature**.
- If you select **Iterate**, the Program iteratively determines the temperature of the fluid entering the each sector based on the flow velocities and temperatures of the fluid leaving the previous sector(circumferentially) and the fluid injected from the external supply.

## Clearance Input

Use the **Clearance Input** drop-down menu to define the methodology to be used to define the surface profile for each sector. You may select entry of a diametrical clearance and preload or entry of two diametrical clearances.

Currently, the following 3 methodologies are available:

- Cb & m: Diametrical Bearing Clearance and Preload
- Cp & m: Diametrical Pad Clearance and Preload
- Cb & Cp: Diametrical Bearing Clearance and Diametrical Pad Clearance

Whichever two entries are selected, the third is calculated by the interface using the following equation:

$$m = 1 - (C_b / C_p)$$

## Materials page

The **Materials** page is common for all analysis cases that use material data and is used to define the material properties list that will be available for selection in this analysis case.

Once you insert a new material as a row in the table, there are two options for assigning the material properties:

- Define the properties manually, directly on the **Materials** page in the analysis case.
- Link to a material from the **Materials Library** by clicking the **Materials Library** button, therefore using the properties from the library.

See **Chapter 5: Working with materials** on page 79 for more information.

## Geometry page

Use the following fields on the **Geometry** page to specify the geometry of the sectors. See **Bearing Geometry** on page 161 for more information about the fixed geometry bearing types.

### Bearing Geometry box

#### Rotor Diameter

Use **Rotor Diameter** to define the rotor outside diameter under the operating conditions specified in the current case (speed and temperature). Valid inputs are real numbers greater than zero.

#### Rotor Roughness

The **Rotor Roughness** represents the absolute roughness of the rotor surface. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer. The program constrains the input value to be greater than or equal to zero but less than or equal to 256 micro inches, 6.5 microns, or 0.0065 mm. If the rotor is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

**NOTE:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the rotor is hydraulically smooth with a surface roughness of 8-32 microinches. This can change if the rotor has been damaged or corroded.

#### Symmetry

Use the **Symmetry** drop-down menu to specify whether all sectors are the same or whether each sector differs:

- **Symmetric:** All the sectors are the same and the sectors are equally spaced. One set of input fields are available on the **Sector Definition** dialog box.

- **Asymmetric:** Each sector may be different and the sectors need not be equally spaced. A grid displays on the **Sector Definition** dialog box allowing you to enter different values for each sector.

### **Sector Definition Define button**

Use the **Sector Definition Define** button to specify geometry for each of the sectors. Indicate whether the values for each sector are the same or different using the **Symmetry** drop-down menu.

### **Housing Material 1/Housing Material 2**

Use **Housing Material 1** and **Housing Material 2** to specify the materials used to fabricate the bearing housing. **Housing Material 1** is the material in contact with the fluid film.

### **Radial Thickness**

Use the **Radial Thickness** fields to specify the radial thickness of each material used to fabricate the bearing housing.

### ***Sector Definition dialog box***

Use the **Sector Definition** dialog box to specify geometry for each of the sectors.

The type of fields available on the **Sector Definition** dialog box depend on the option selected in the **Symmetry** drop-down menu on the **Geometry** page.

- **Symmetric:** A set of fields displays allowing you to enter the number of sectors and the geometry to be applied to every sector.

**NOTE:** The values entered on the **Sector Definition** dialog box when **Symmetric** is selected on the **Geometry** page are applied to each sector. The exception is the **Arc Location** field. Enter the **Arc Location** for the first sector. The Program calculates the **Arc Location** for each of the remaining sectors by dividing 360 degrees by the number entered into the **Number of Sectors** edit box and incrementing the entered **Arc Location**.

- **Asymmetric:** A grid displays allowing you to add sectors and specify different values for each sector.

The following fields are available on the **Sector Definition** dialog box, depending on the **Symmetry** type selected.

### **Number of Sectors**

If you selected **Symmetric** in the **Symmetry** drop-down menu on the **Geometry** page, use the **Number of Sectors** edit box to specify the number of sectors you want to define the bearing. Enter values in the remaining input fields on the **Sector Definition** dialog box.

**NOTE:** This field is not available if you selected **Asymmetric** in the **Symmetry** drop-down menu on the **Geometry** page.



---

## RAPPID User's Guide

---

See also **Conventions for defining sectors** on page 164.

### Axial Length

Use **Axial Length** to define the axial length (along Z-axis) of the sector load bearing surface.

The following rule applies to the **Axial Length** field:

- Valid entries are real numbers greater than zero.

### Axial Location

Use **Axial Location** to define the axial starting position (along the Z-axis) for the sector.

The following rule applies to the **Axial Location** field:

- Valid inputs are real numbers greater than or equal to zero.

The axial distance between two sectors occupying the same circumferential angular location (**Arc Location**), but not the same axial location (**Axial Location**), is used in conjunction with the axial length (**Axial Length**) of the two sectors to define the width of the circumferential groove between the sectors.

### Arc Length

Use **Arc Length** to define the arc length of the sector's load bearing surface.

The following rules apply to the **Arc Length** field:

- Valid entries are real numbers greater than zero but less than 360°.
- The end angle for each pad must always be less than 360° (**Arc Location** + **Arc Length** < 360°).

**NOTE:** The **Arc Length** does not include the arc length of the groove associated with the sector. The Program automatically sets the groove arc length by subtracting the end angle of this sector from the start angle of the next sector.

### Arc Location

Use **Arc Location** to define the starting angular location for the sector's load bearing surface. The load bearing portion of the sector extends from the **Arc Location** to the **Arc Location** + **Arc Length**.

The following rules apply to the **Arc Location** field:

- Valid entries are real numbers between -360° and 360°, exclusive.
- The end angle for each sector must always be less than 360° (**Arc Location** + **Arc Length** < 360°). If the end angle is greater than 360°, simply define the **Arc Location** with a negative angle.

## **Groove Depth**

Use **Groove Depth** to define the radial depth of the axial groove which follows the load bearing surface of the sector.

The following rule applies to the **Groove Depth** field:

- Valid entries are real numbers greater than or equal to zero. If set to zero, there will be no ram pressure effect (due to the initial fluid circumferential velocity of the fluid) included in the solution.

## **Clearance Circumferential Offset**

Use **Clearance Circumferential Offset** to define the angular location of the minimum clearance point (when the journal is centered) on a preloaded sector.

The following rule applies to the **Clearance Circumferential Offset** field:

- Valid entries are real numbers between zero and 1, inclusive.

The offset input is a non-dimensional value determined using the following equation:

$$\text{Offset} = \text{Offset Angle} / \text{Arc Length}$$

where the 'Offset Angle' is the angle from the leading edge of the sector to the location of the minimum clearance point.

For a sector with a preload of zero, the **Clearance Circumferential Offset** is 0.5 by definition.

## **Dia. Brg. Clearance (Cb)**

If the **Clearance Input** option selected on the **Options** page contains 'Cb', use **Dia. Brg. Clearance** to specify the diametrical bearing clearance (Cb) for the sector.

The following rule applies to the **Dia. Brg. Clearance** field:

- Valid entries are real numbers greater than 0 but less than or equal to the value entered for the pad clearance.

**NOTE:** If the **Clearance Input** option selected on the **Options** page does not contain 'Cb', then this field is non-editable and the automatically calculated value for 'Cb' displays.

## **Dia. Pad Clearance (Cp)**

If the **Clearance Input** option selected on the **Options** page contains 'Cp', use **Dia. Pad Clearance** to specify the diametrical pad clearance (Cp) of the sector.

The following rule applies to the **Dia. Pad Clearance** field:

- Valid entries are real numbers greater than or equal to the bore clearance (Cb).

---

## RAPPID User's Guide

---

**NOTE:** If the **Clearance Input** option selected on the **Options** page does not contain 'Cp', then this field is non-editable and the automatically calculated value for 'Cp' displays.

### Preload (m)

If the **Clearance Input** option selected on the **Options** page contains 'm', use **Preload** to enter the preload for the sector.

The following rule applies to the **Preload** field:

- Valid entries are real numbers greater than or equal to zero but less than 1.

**NOTE:** If the **Clearance Input** option selected on the **Options** page does not contain 'm', then this field is non-editable and the automatically calculated value for 'm' displays.

### Recessed Sector

Use the **Recessed Sector** drop-down menu to specify whether you want to define a recess on the sector by entering **Recess Geometry** values. If you select **No**, the fields in the **Recess Geometry** box (if you selected **Symmetric**) or the **Recess** cells in the second row of the grid (if you selected **Asymmetric**) are not available.

**NOTE:** If you selected **Asymmetric** to enter different values for each sector, the **Recessed Sector** drop-down menu is at the far right end of the top row of each pair of rows.

### Recess Axial Length

Use **Recess Axial Length** to define the axial length of the recess (the portion of the sector's bearing surface that is defined using a second arc).

The following rule applies to the **Recess Axial Length** field:

- Valid entries are real numbers greater than zero but less than the **Axial Length** of the sector.

The Program solves the fluid film on both the recessed portion of the bearing surface and the primary bearing surface. If you define a very narrow recess (say less than 10% of the **Axial Length** of the sector), numerical problems may occur during the solution. If a solution cannot be obtained, you should increase the axial length of the recess and/or increase the density of the designated grid.

### Recess Arc Length

Use **Recess Arc Length** to define the arc length of the recess (the portion of the bearing surface that is defined by using a second arc).

The following rules apply to the **Recess Arc Length** field:

- Valid inputs are real numbers that fulfill the following requirements:
  - are greater than zero but less than the **Arc Length** of the sector

---

## RAPPID User's Guide

---

- yield a recess height greater than or equal to zero for the entire included angle of the recess (see the equation below)

The **Recess Arc Length** is measured from the beginning of the sector's load bearing surface (not the positive X axis).

$$H_{\text{recess}} = R_{\text{recess}} - R_{\text{pad}} - \text{ecc} * \cos (180 - A_{\text{ecc}} - A)$$

where

$H_{\text{recess}}$  = the height of the recess at angle A

$R_{\text{recess}}$  = the radius of curvature of the recess surface (0.5 \* **Recess Pad Diameter**)

$R_{\text{pad}}$  = the radius of curvature of the primary bearing surface (0.5 \* **Pad Diameter**)

$\text{ecc}$  = the **Recess Eccentricity**

$A_{\text{ecc}}$  = the **Recess Eccentricity Angle**

A = any angular location within the recess (varies between 0 and **Recess Arc Length**)

If the recess ends at an angular location that corresponds to an intersection of the two arcs defining the primary and recess load bearing surfaces (i.e.,  $H_{\text{recess}}$  at **Recess Arc Length** is zero, within a numerical tolerance), then there will be no step change in the clearance profile for the sector (this is generally the case for tapered land bearings). If the **Recess Arc Length** is set to a location that does not correspond to an intersection of the two arcs defining the primary and recess load bearing surfaces, then a step change in the bearing clearance is considered (this is generally the case for pressure dam bearings). The height of the step change ( $H_{\text{recess}}$  evaluated at  $A = \text{Recess Arc Length}$ ) is determined by the radial distance between the arc defining the primary load bearing surface and the arc defining the recess at the angular position corresponding to **Recess Arc Length**.

### Recess Axial Location

Use **Recess Axial Location** to define the recess (the portion of the bearing surface which is defined using a second arc) edge located closest to the Z axis.

The following rule applies to the **Recess Axial Location** field:

- Valid inputs are real numbers greater than or equal to zero but less than or equal to the following quantity:
  - **Axial Length** of the sector - **Recess Axial Length**

The input locates the edge of the recess closest to the Z axis. The recess may be the entire **Axial Length** of the sector or a fraction of the **Axial Length** of the sector. If the **Recess Axial Length** is less than the **Axial Length** of the sector, the Program does not allow either axial edge of the recess to overlay an axial edge of the sector (i.e., the recess must be seated axially by the primary bearing surfaces). However, the recess does not need to be centrally located (axially) within the sector. The Program solves the fluid film on the portions of the primary bearing surface that bounds the recess axially (referred to as side walls). If you define a very narrow side wall(s) (say less than 10% of the **Axial**

**Length** of the sector), numerical problems may occur during the solution. If a solution cannot be obtained, you should increase the axial length of the side wall and/or increase the density of the designated grid.

### **Recess Pad Diameter**

Use **Pad Diameter** to define the curvature of the bearing surface in the recess (the portion of the bearing surface which is defined using a second arc).

The following rule applies to the **Pad Diameter** field:

- Valid entries are real numbers that:
  - are greater than zero
  - define a recess height (radially) greater than or equal to zero at all angular locations (see the description for **Recess Arc Length** for the equation)

### **Recess Eccentricity**

Use **Recess Eccentricity** to set the distance between center of the **Pad Diameter** that defines the curvature of the primary bearing surface and the center of the pad diameter that defines the curvature of the recess (**Recess Pad Diameter**).

The following rule applies to the **Recess Eccentricity** field:

- Valid entries are real numbers greater than or equal to zero.

This value is used in conjunction with the **Recess Eccentricity Angle** to determine the location of the center of the arc used to define the bearing surface's profile in the recess.

### **Recess Eccentricity Angle**

Use **Recess Eccentricity Angle** to set the angular location of the center of the arc used to define the curvature of the recess.

The following rule applies to the **Eccentricity Angle** field:

- Valid inputs are real numbers that:
  - are greater than or equal to 0 and less than or equal to 360°
  - define a recess height (radially) greater than or equal to zero at all recess angular locations (see description for **Recess Arc Length** for equation)

The **Recess Eccentricity Angle** prescribes the angle between the leading edge of the sector and a line drawn between the center of the **Pad Diameter** that defines the primary bearing surface and the center of the arc that defines the curvature of the recess (**Recess Pad Diameter**). This angle is measured clockwise from the leading edge of the sector.

## Fluids page

The **Fluids** page is common to all analysis cases that use a single fluid (compound). Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**
- **Real Properties**

For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

## Operating Conditions page

Use the **Operating Conditions** page in a *Journal Bearings - 2-D Fixed Geometry* analysis case to set the pressure and temperature boundary conditions and the solution method to obtain eccentricity/load results.

**NOTE:** If there is a ... button next to an edit box, you may enter variable input for that input field. See **Entering variable data** on page 47 for more information.

## Rotor Speed

Currently, **Rotor Speed** is the analysis parameter by default because it is the only input available as the analysis parameter. Input range and increment for **Rotor Speed** is entered on the **Options** page (see page 178).

## Pressure/Temperature box

### Fluid Supply Pressure

The **Fluid Supply Pressure** represents the pressure of the fluid entering all grooves from the external supply.

The following rule applies to the **Fluid Supply Pressure** field:

- Valid entries are real numbers greater than or equal to zero.

---

## RAPPID User's Guide

---

**NOTE:** The **Cavitation Pressure** (???), ambient pressure (**Left Boundary Pressure** and **Right Boundary Pressure**), and **Supply Pressure** are relative to each other. As such, they may be entered in either gauge or absolute values as long as all three are entered in consistent terms (do not specify some pressures in gauge and others in absolute).

### **Fluid Supply Temperature**

The **Fluid Supply Temperature** represents the temperature of the fluid entering all grooves in the bearing from the external supply. This temperature is considered constant and is used, in conjunction with the Program's mixing model, to determine the fluid temperature at the pad's inlet (first row of nodes circumferentially).

The following rule applies to the **Fluid Supply Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### **Left Boundary Pressure**

The **Left Boundary Pressure** represents the pressure boundary condition applied on all circumferential sector lengths not bordered by a circumferential groove. If one or both circumferential lengths of a sector are bordered by a circumferential groove, the supply fluid pressure is applied as the pressure boundary condition.

The following rules apply to the **Left Boundary Pressure** field:

- Valid entries are real numbers greater than zero.

**NOTE:** The **Cavitation Pressure** (???), ambient pressure (**Left Boundary Pressure** and **Right Boundary Pressure**), and **Supply Pressure** are relative to each other. As such, they may be entered in either gauge or absolute values as long as all three are entered in consistent terms (do not specify some pressures in gauge and others in absolute).

### **Right Boundary Pressure**

This field is not editable. The value entered into the **Left Boundary Pressure** applies to both left and right boundary pressures in this version of the Program and is displayed in this field.

### **Shaft Temperature**

The **Shaft Temperature** specifies the fixed surface temperature of the shaft in the bearing.

The following rule applies to the **Shaft Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### **Sector Back Temperature**

Use **Sector Back Temperature** to specify the fixed temperature to be applied at the outer surface of the bearing's housing (stationary). The radial thickness of the bearing's housing is the sum of the entries for

---

## RAPPID User's Guide

---

the following inputs: **Radial Thickness** for **Material 1** and **Radial Thickness** for **Material 2**. The housing radial thickness and the fixed back temperature are assumed to be the same for all sectors. The entered temperature is utilized in the conduction model employed to simulate the radial heat transfer through the bearing's housing.

The following rule applies to the **Sector Back Temperature** field:

- Valid entries are any real number greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### Loading box

Use the drop-down menu at the top of the **Loading** box to specify the operation mode for the Program's analysis. There are two options:

- **Prescribed Load**

For the **Prescribed Load** option an initial guess for the journal's equilibrium position is required. If you select **Prescribed Load**, a drop-down menu displays with two options available:

- **Auto Guess:** The **Eccentricity** and **Eccentricity Angle** fields are not available. The initial guess is generated automatically by the Program using an internal guess algorithm.
- **Manual Guess:** The **Eccentricity** and the **Eccentricity Angle** fields are available. Enter an initial guess into each field manually.

If you select **Prescribed Load**, the **Convergence Criteria**, **Journal Weight**, **Applied Load**, and **Applied Load Direction** fields are available.

- **Prescribed Eccentricity**

If you select **Prescribed Eccentricity**, the **Eccentricity** and **Eccentricity Angle** fields are available.

Specify the following values as applicable.

### Eccentricity

Use **Eccentricity** to define the magnitude of the eccentricity vector.

- For **Prescribed Eccentricity**, this value defines the position of the journal to be used in the solution.
- For **Prescribed Load**, this value defines the initial guess for the equilibrium position of the journal.

Use the **Guess** drop-down menu to specify whether the initial guess is automatically generated (**Auto Guess**) or entered manually (**Manual Guess**).

**NOTE:** This field is not available when **Auto Guess** is selected.



---

## RAPPID User's Guide

---

The following rules apply to the **Eccentricity** field:

- Valid entries are real numbers greater than or equal to zero.
- The **Eccentricity** must be less than half of the **Diametrical Pad Clearance** (see **Sector Definition dialog box** on page 181 for more information) of the bearing at the **Eccentricity Angle**.

### Eccentricity Angle

Use **Eccentricity Angle** to define the angular orientation of the eccentricity vector.

- For **Prescribed Eccentricity**, this value defines the position of the journal to be used in the solution.
- For **Prescribed Load**, this value defines the orientation of the eccentricity vector to be used as the initial guess for the equilibrium position of the journal.

Use the **Guess** drop-down menu to specify whether the initial guess is automatically generated (**Auto Guess**) or entered manually (**Manual Guess**).

**NOTE:** This field is not available when **Auto Guess** is selected.

The following rule applies to the **Eccentricity Angle** field:

- Valid entries are real numbers between -360° and 360°, inclusive.
- This angle is relative to the positive X-axis and is positive in the counter-clockwise direction.

### Convergence Criteria

Use **Convergence Criteria** to define the convergence criteria, as a percentage of the user specified load, used by the Program to determine when to cease iteration on reacted load.

**NOTE:** **Convergence Criteria** is not available for **Prescribed Eccentricity**.

The following rule applies to **Convergence Criteria**:

- Valid entries are real numbers between 0.5 and 5, inclusive. The valid range of entries corresponds to 0.5-5% of the user specified load for the case.

**NOTE:** The execution time of the Program is very sensitive to the value entered in this field. The lower the convergence criteria, the longer the Program takes to obtain the desired solution. Generally, a convergence criteria of 1-2% is sufficient. However, when running numerous cases for initial evaluation purposes, higher convergence criteria may be desirable to reduce computational time.

## **Journal Weight**

Use **Journal Weight** to define the load applied to the bearing in the direction of gravity (along the negative Y-axis, or 270°). **Journal Weight** represents the weight of the rotor supported by this bearing.

The actual load applied to the bearing during the Program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in the **Applied Load** and **Applied Load Direction** fields). The applied load vector is intended to be used to represent arbitrarily orientated forces (i.e. gear load) which act in conjunction with the rotor weight.

**NOTE: Journal Weight** is not available for **Prescribed Eccentricity**.

The following rule applies to **Journal Weight**:

- Valid entries are real numbers greater than or equal to zero. Zero is only valid if a non-zero value is entered in the **Applied Load** field.

## **Applied Load**

Use **Applied Load** to define the magnitude of the load vector applied to the bearing in addition to the user input **Journal Weight**. The direction of the applied load vector is set by the value entered in the **Applied Load Direction** field.

The applied load vector is intended to be used to represent arbitrarily orientated loads (i.e. gear loads) which are applied to the bearing. The load applied to the bearing during the Program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in the **Applied Load** and **Applied Load Direction** fields).

**NOTE: Applied Load** is not available for **Prescribed Eccentricity**.

The following rule applies to **Applied Load**:

- Valid entries are real numbers greater than or equal to zero. Zero is only valid if a non-zero value is entered in the **Journal Weight** field.

## **Applied Load Direction**

Use **Applied Load Direction** to define the angular orientation of the applied load vector. The magnitude of the applied load vector is set by the value entered for the **Applied Load**.

The applied load vector is intended to be used to represent arbitrarily orientated loads (i.e. gear loads) which are applied to the bearing. The load applied to the bearing during the Program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in the **Applied Load** and **Applied Load Direction** fields).

**NOTE: Applied Load Direction** is not available for **Prescribed Eccentricity**.

The following rules apply to **Applied Load Direction**:

- Valid entries are real numbers between  $-360^\circ$  and  $360^\circ$ , inclusive.
- This angle is relative to the positive X-axis and is positive in the counter-clockwise direction.

## Sample analysis cases

This section contains a series of examples to illustrate the types of bearings which may be analyzed using *Journal Bearing – 2-D Fixed Geometry* analysis cases within RAPPID™. The following sample cases are included (the names listed before the colon are the actual names of the cases in the Program).

### Manual Example 1: $360^\circ$ bearing at three speed points

A common bearing type is the simple  $360^\circ$  bearing.

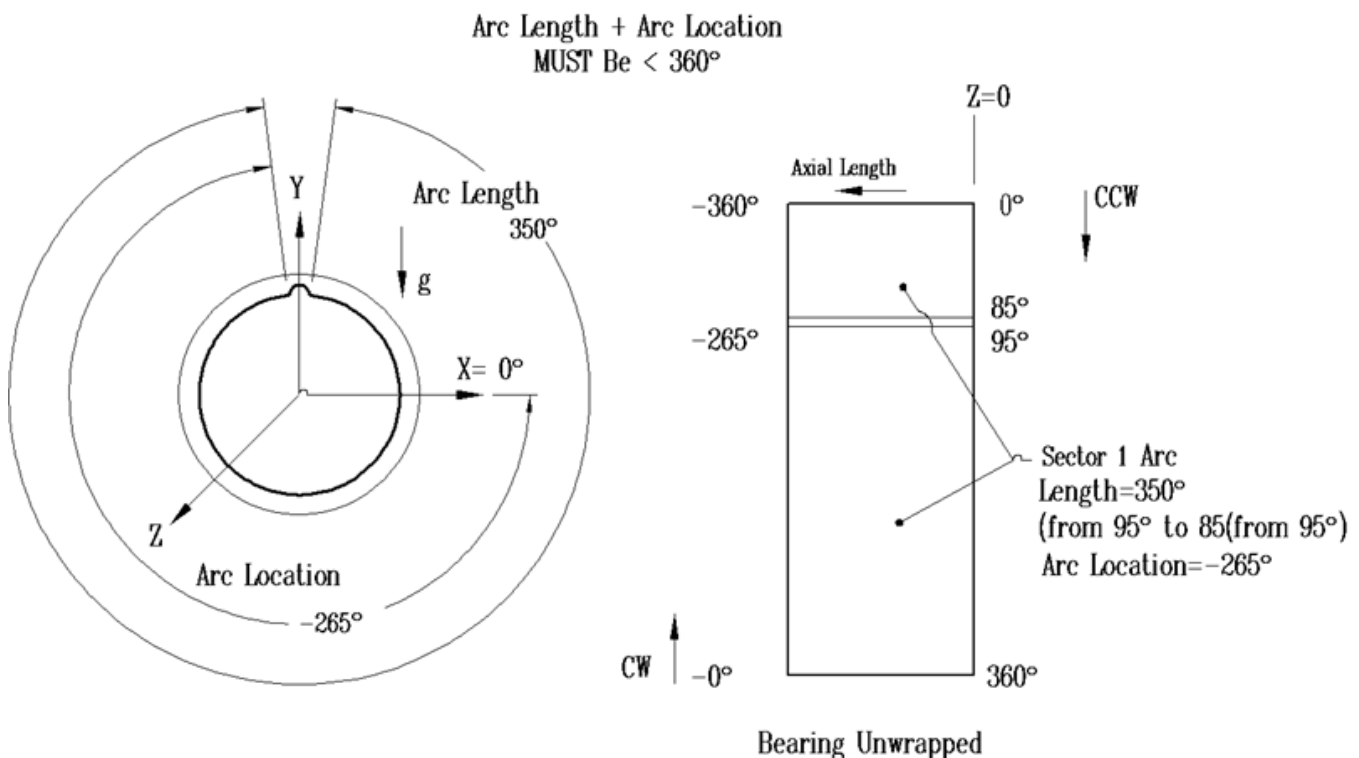


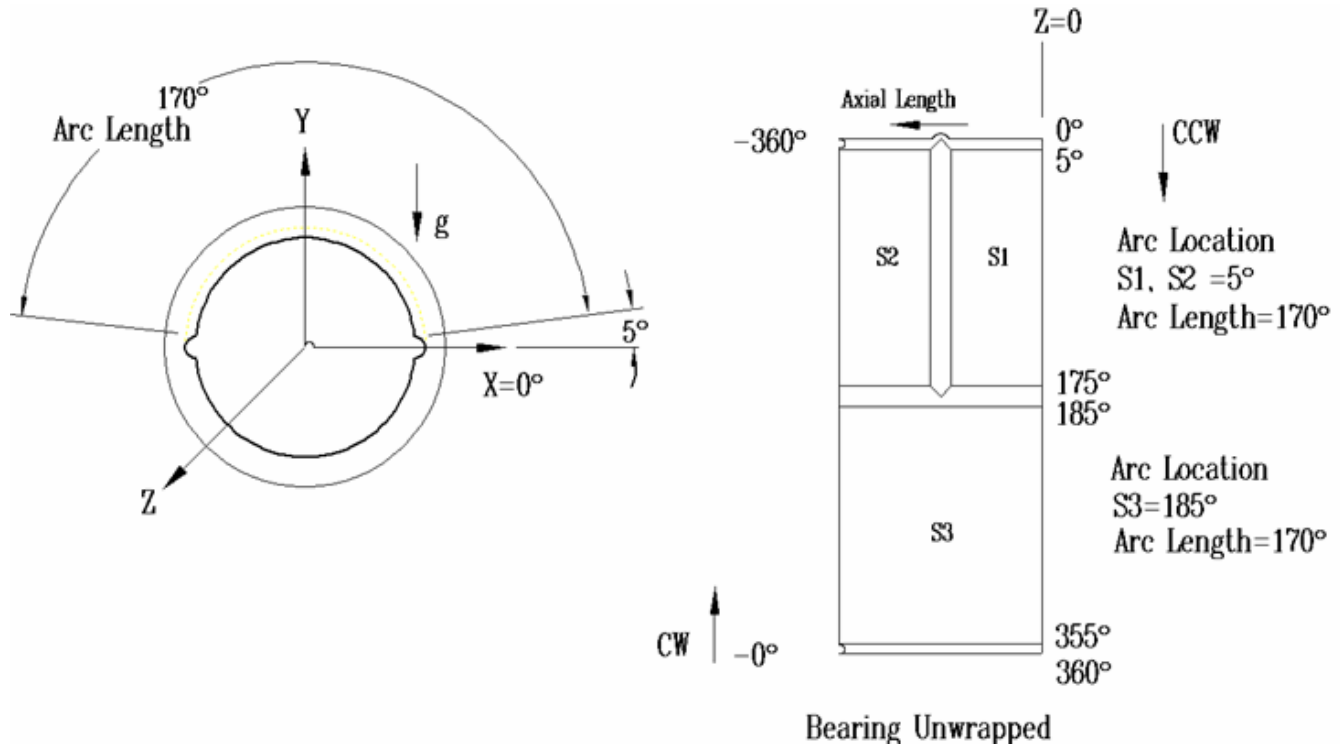
Figure 24.  $360^\circ$  bearing at three speed points

To model this type of bearing in the Program, only one sector is needed. As the Program assumes the bearing is radially fed, the arc length for the bearing surface must be less than  $360^\circ$ . For this example, a  $359^\circ$  arc length is utilized ( $1^\circ$  axial groove). The axial groove has been centrally located on the positive Y-axis by utilizing an arc location angle of  $-269.5^\circ$  (Remember, the end angle of the sector cannot be greater than  $360^\circ$ , in this case  $-269.5^\circ + 359^\circ = 89.5^\circ$ ). The fluid lubricant has been set to the preprogrammed ISO 32 Oil. A journal weight of 1000 lbf has been defined. The automatic initial guess algorithm is being employed. Iteration on groove temperature has been selected. The bearing housing is being modeled with two materials, a 0.005-inch thick layer of tin based babbitt and a 2-inch thick layer of low alloy steel. Three separate cases have been defined which will generate rotordynamic coefficients at 1000, 3000, and 6000 rpm. The load convergence criteria has been set to 0.5%. Note that

the only difference between these three cases for this sample analysis is the rotor speed. *However, for an actual analysis, you must account for differences in operating conditions at different speeds. In particular, this means changes in operating clearance (due to centrifugal and thermal growth) and supply temperature.*

## Manual Example 2a/2b:2- lobe bearing with circumferential groove in upper half

To demonstrate analysis of bearings with circumferential grooves, this sample case considers a 2-lobe bearing with a circumferential groove in the upper half.

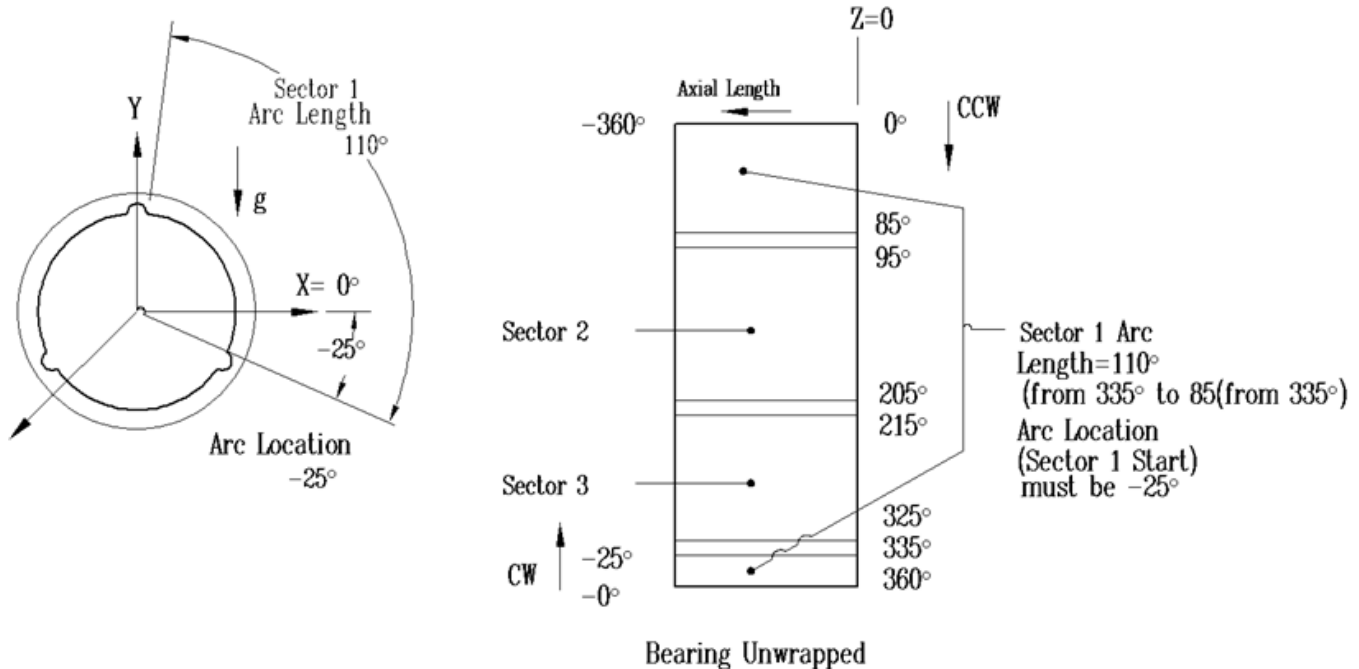


**Figure 25. 2-lobe bearing with circumferential groove in upper half**

To model this type of bearing in the Program, three sectors are required. All of the sectors have an arc length of 170° (10° axial groove). The axial grooves have been centered on the positive and negative X axes. The bearing has also been defined with the lower half preloaded (0.3), and the upper half sectors not preloaded. Note that the first groove in the bearing is the groove centered on the negative X-axis (since the groove centered on the positive Y-axis starts in the fourth quadrant); as such, the sectors in the upper half of the bearing must be defined first. The fluid lubricant has been set to the preprogrammed ISO 32 Oil. A journal weight of 1000 lbf has been defined. The initial guess is set to manual. Since this bearing has a circumferential groove, the axial groove mixing model **cannot** be used. To account for the effect of hot fluid carry over, we must artificially increase the fluid supply temperature. To estimate the hot fluid carry over, an initial analysis case has been defined utilizing the input values of interest, but eliminating the circumferential groove. This analysis indicates the inlet temperature is approximately 132° F for sector 2 (the primary load bearing sector), when the supply temperature is 124° F (in **Manual Example 2a**). The fluid supply temperature was then set to 132° F (in **Manual Example 2b**), 8 degrees higher than the actual supply temperature for the bearing, to account for the oil fluid carry over.

## Manual Example 3: 3-lobe bearing with gear loading

To demonstrate use of the journal weight and applied load vector, this sample case analyzes a 3 lobe bearing subjected to gear loading.



**Figure 26. 3-lobe bearing with gear loading**

To model this type of bearing in the Program, three sectors are required. All of the sectors have an arc length of 110° (10° axial groove). One of the axial grooves is centered upon the positive Y-axis. The fluid lubricant has been set to the preprogrammed ISO 32 Oil. A journal weight of 100 lbf has been defined. The automatic initial guess algorithm is being employed. Iteration on groove temperature will be performed. In addition to the journal weight, an applied load of 400 lbf at -60° is defined at 1000 rpm and an applied load of 700 lbf at -40° is defined at 3600 rpm.

## Manual Example 4a/4b: 4 lobe bearing with circumferential groove

To demonstrate the analysis of a bearing with a circumferential groove around its entire circumference, this sample case analyzes a four lobe bearing.

To model this type of bearing in the Program, eight sectors are required. All of the sectors have arc lengths of 80° (10° axial groove). The load bearing surfaces are centered on the X and Y axes. The fluid lubricant has been set to the preprogrammed ISO 32 Oil. Rotor speed is set to 15,000 rpm. A journal weight of 400 lbf has been defined in **Manual Example 4b**. The automatic initial guess algorithm is being employed. Due to the presence of the circumferential groove, the groove temperature iteration is not available. To estimate the effect of hot fluid carry over, an initial analysis (**Manual Example 4a**) is performed using half of the bearing with half of the load (a conventional four lobe bearing). Since sectors 1 and 2 are completely cavitated (as evidenced from the results of **Manual Example 4a**), and sector 3 is developing minimal pressure, the supply temperature from the results of



**Example 3: 3-lobe bearing with gear loading** on page 194 and adds a recess with no step height to each sector.

To model this type of bearing in the Program, three recessed sectors are required. All of the sectors have arc lengths of  $110^\circ$  ( $10^\circ$  axial groove) and a recess arc length of  $55^\circ$ . The recess has been defined with an eccentricity angle of  $125^\circ$ , a recess eccentricity of 0.005", and a recess pad diameter of 4.019". These recess settings define a recess with no step height (because  $4.009 = 4.019 - 2 \cdot 0.005$ ) that intersects the circular bore portion of the bearing surface at the mid point of the sector ( $110/2 = 55$  and  $125 + 55 = 180$ ).

## Chapter 16: Journal Bearing – 2-D Tilting Pad

### Overview

The *Journal Bearings - 2-D Tilting Pad* module is for analysis of radially fed tilting pad fluid film journal bearings. Its primary function is to allow the designer to determine rotordynamic coefficients for various tilting pad pivot and bore designs in an effort to select a suitable bearing for the machinery application under consideration.

The *Journal Bearings - 2-D Tilting Pad* module calculates the bearing's non-linear transfer function or linearized stiffness, damping, and mass coefficients by taking the user's input and solving a full set of film averaged Navier-Stokes based governing equations: momentum, continuity, and energy. The flow in the bearing may be laminar, transitional, or turbulent. Rotordynamic coefficients are determined using a perturbation analysis which assumes the journal is exhibiting small amplitude cylindrical motion about its static equilibrium position. The *Journal Bearings - 2-D Tilting Pad* module also reports, operating eccentricity, eccentricity angle, maximum film pressure, fluid temperature rise, maximum bearing temperature, minimum film thickness, required fluid flow rate, and power consumption.

### Bearing Geometry

Use *Journal Bearings - 2-D Tilting Pad* to analyze the following tilt pad bearing pivot types:

- Pin
- Rocker Back
- Ball/Socket

See **Pivot Model** on page 199 for more information.

Each pad in the bearing may be defined with individual clearance, preload, offset, arc length, axial length, inertia, groove depth, and pad material properties. All pads in a bearing must have the same type of pivot, but each pivot may have different values of pivot stiffness, friction, pivot location, and pivot radii.

### Thermal Model

The heat generated in the bearing is dissipated via convection to the shaft, convection through each pad, and hot fluid exiting each pad. Hot fluid exiting a pad circumferentially enters the pad's axial feed groove and will either be completely drained from the bearing or carried over into the next (circumferentially) pad. The heat convected to the pad surface is conducted through the pad to a constant temperature at the pad surface opposite to the fluid film. Conduction through the pivot to the bearing housing and convection from the back of the pad to the surrounding fluid are not considered in the thermal model. This thermal model requires the user to enter the following information:

- Shaft Surface Temperature
- Pad Back Temperature



- Pad Radial Thickness
- Pad Thermal Conductivity

The pad conduction model allows the user to define the each pad using two separate materials acting in series. This modeling capability allows the user to account for the different conduction rates between the bearing surface material (i.e. babbitt) and the base pad material (i.e. steel).

### Thermal and centrifugal growth

Under operating conditions the tilting pads may experience a change in clearance due to thermal growth of the pad, thermal growth of the pivot, and/or pivot diameter change due to pressure on the pivot's outer diameter (i.e. from a press fit). Thermal growth of the pad is automatically calculated by the Program based on the predicted temperature distribution in the pad and the user supplied thermal expansion coefficient. However, the effects of pivot growth are installation dependent and the user must account for them by providing the *actual bearing surface diameter under operating conditions* to *Journal Bearings - 2-D Tilting Pad* (either directly or through the clearance input). Note that *Journal Bearings - 2-D Tilting Pad* does not calculate thermal bow of the pad or pad deformations due to the pressure loading from the fluid film.

Under operating conditions the rotating bearing surface (journal) may experience a change in diameter due to thermal and centrifugal growth, and/or the presence of a press fitted shaft sleeve. As these effects are installation dependent, the user must account for them by providing the actual journal diameter under operating conditions to *Journal Bearings - 2-D Tilting Pad*.

To determine the actual bearing surface and journal diameters under operating conditions you may utilize a separate RSR program called FIT. FIT produces output consistent with that required for entry into *Journal Bearings - 2-D Fixed Geometry*.

### Groove Model

Each pad is comprised of a bearing surface followed by an axial groove. The pad's circumferential sides are bordered by ambient pressure and all axial grooves are fed with fluid at the same supply temperature and pressure. Fluid is allowed to leak from the entire arc length of the pad's bearing surface axially to the user-defined ambient pressure. *Journal Bearings - 2-D Tilting Pad* includes a mixing model for the axial grooves to account for hot fluid carry over from the previous pad (circumferentially) mixing with new fluid supplied at the user-defined supply temperature. This model assumes the grooves are sealed axially. You have the option of considering hot fluid carry over (**Iterate** on pad inlet temperature) or performing an analysis assuming each pad is supplied with fluid at a **Fixed** user-defined supply temperature.

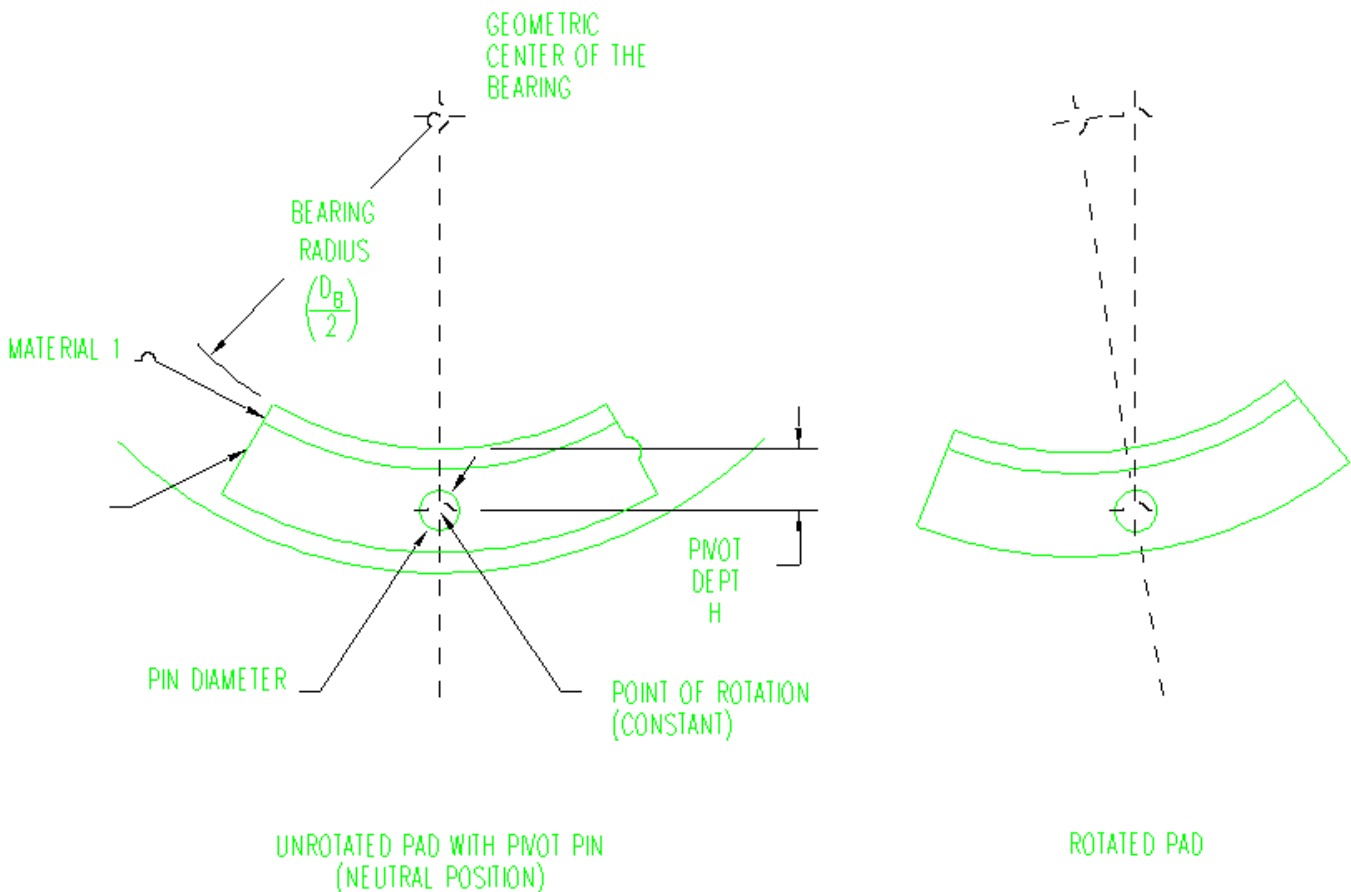
*Journal Bearings - 2-D Tilting Pad* also includes a model to account for the ram effect of the fluid entering the pad with an initial circumferential velocity. The groove mixing model, if activated, accounts for the loss in circumferential velocity due to the introduction of fresh fluid radially.

## Pivot Model

Currently three types of pad pivots can be modeled by *Journal Bearings - 2-D Tilting Pad*. The available pivots types are discussed below.

### Pinned Pivot

To prescribe a pin pivot the user must specify the diameter of the pin, the radial location of the pin, and the circumferential location of the pin (see the following figure). The user may also specify a coefficient of friction to represent the frictional force present on the outside diameter of the pin. This pivot model assumes the pad is free to rotate (it has overcome the static friction) about the center of the pin, and as the pad rotates, the Program automatically adjust the pad's clearance profile.



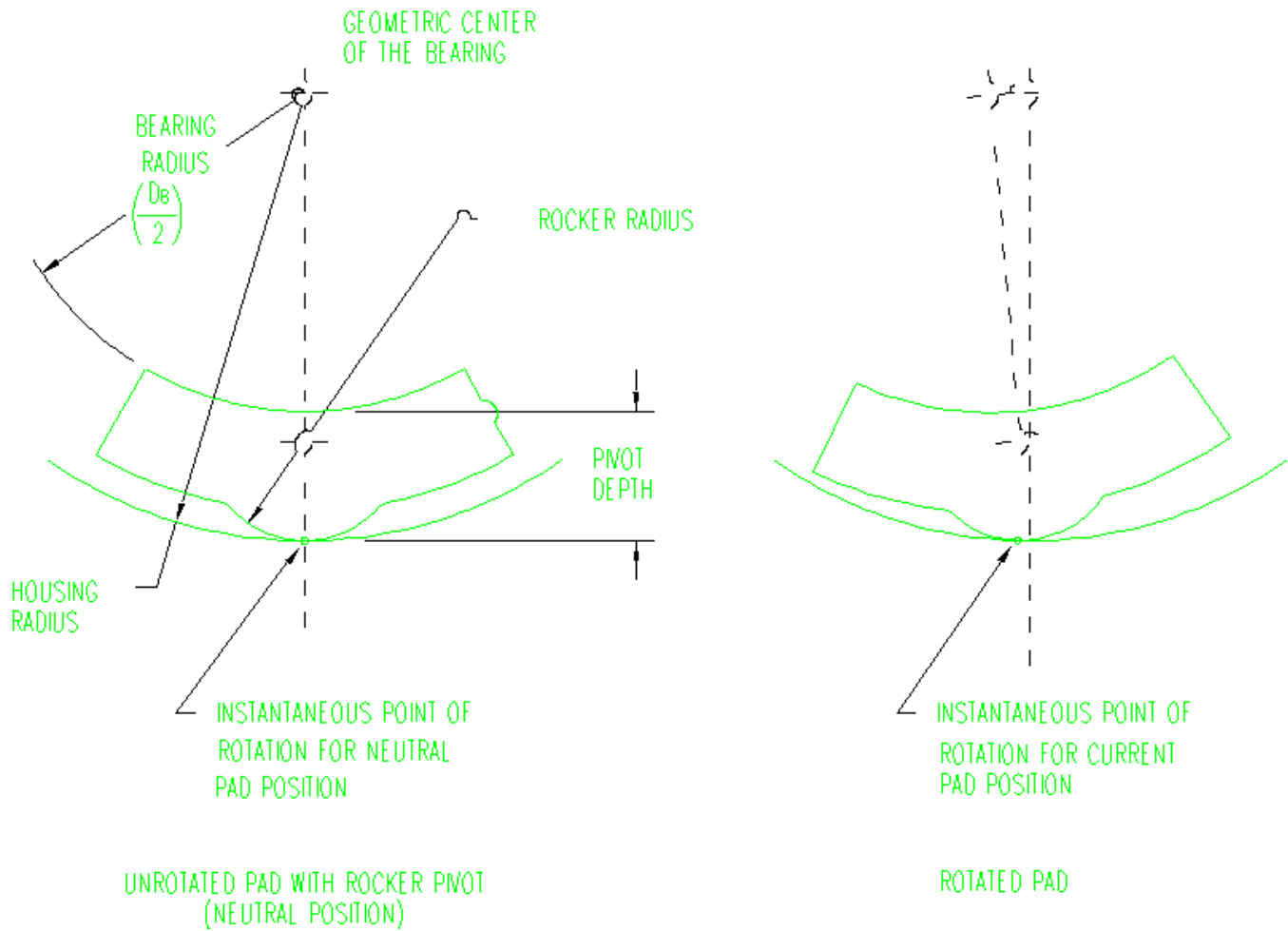
**Figure 28. Pinned pivot**

### Rocker Pivot

To define a rocker back pivot the user defines the radius of the rocker (on the pad), the radius of the mating housing surface (the housing radius must be greater than the rocker radius), the radial location of the pad's pivot point when the pad is in the neutral position, and the circumferential location of the pad's pivot point when the pad is in the neutral position (see the following figure). This pivot model assumes the pad tilts by rolling on the housing surface (no sliding), and as the pad tilts, the Program

## RAPPID User's Guide

automatically adjusts the pad's clearance and pivot offset position. Currently there is no provision to account for friction in this pivot model.

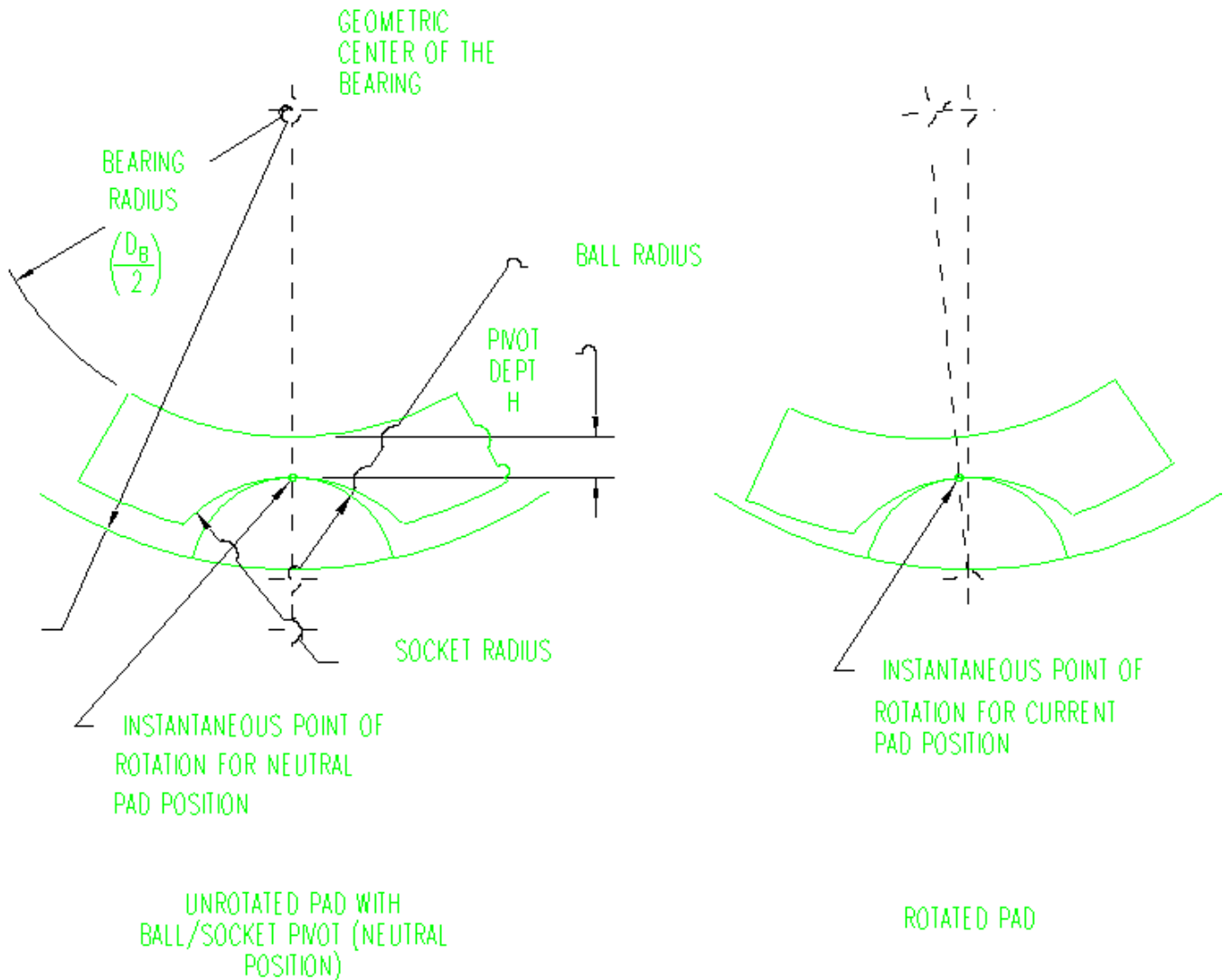


**Figure 29. Rocker pivot**

# RAPPID User's Guide

## Ball/Socket Pivot

Specification of a ball/socket pivot requires the user to define the radius of the socket (in the pad), the radius of the ball (the socket radius must be greater than the ball radius), the radial location of the pad's pivot point when the pad is in the neutral position, and the circumferential location of the pad's pivot point when the pad is in the neutral position (see the following figure). This pivot model assumes the pad tilts by rolling on the ball (no sliding), and as the pad tilts, the Program automatically adjusts the pad's clearance and pivot offset position. Currently there is no provision to account for friction in this pivot model.



**Figure 30. Ball/Socket pivot**

Note that the circumferential position of the pivot (in neutral position) is assumed to be the same angular location as the pad's offset (angular location of the minimum ground in clearance).

To account for the effect of pad inertia, the user may either enter the density of the pad's materials or enter a value for the pad's rotational moment of inertia in the circumferential direction about its pivot point (in neutral position). If the user enters the material density, the Program automatically calculates

the pad inertia by assuming the pad is a uniform cylindrical sector of the user specified arc length. For pads which cannot accurately be modeled as a uniform cylindrical sector the user must calculate the inertia value for the pad and enter it into the Program.

To account for radial flexibility of the pad's pivot, the user is allowed to enter a radial pivot stiffness value. The value for pivot stiffness must be determined by the user outside of the Program.

### Applied Load/Prescribed Eccentricity

After the pads are defined, you have the option of prescribing an applied load vector or enforcing a prescribed eccentricity. If you enter an applied load, the Program will iteratively determine the static equilibrium position of the journal within the bearing. This iterative solution requires entry of an initial guess for the shaft equilibrium position. To assist with this task, this module has an automated initial guess generation algorithm which is suitable for most cases. However, in the event the program does not converge to an appropriate solution with the automatic initial position guess, you may enter an initial position guess manually.

In the event you elect to enforce an eccentricity, the Program will solve the fluid film model and report the reaction load vector produced by the fluid film when the journal occupies the prescribed position.

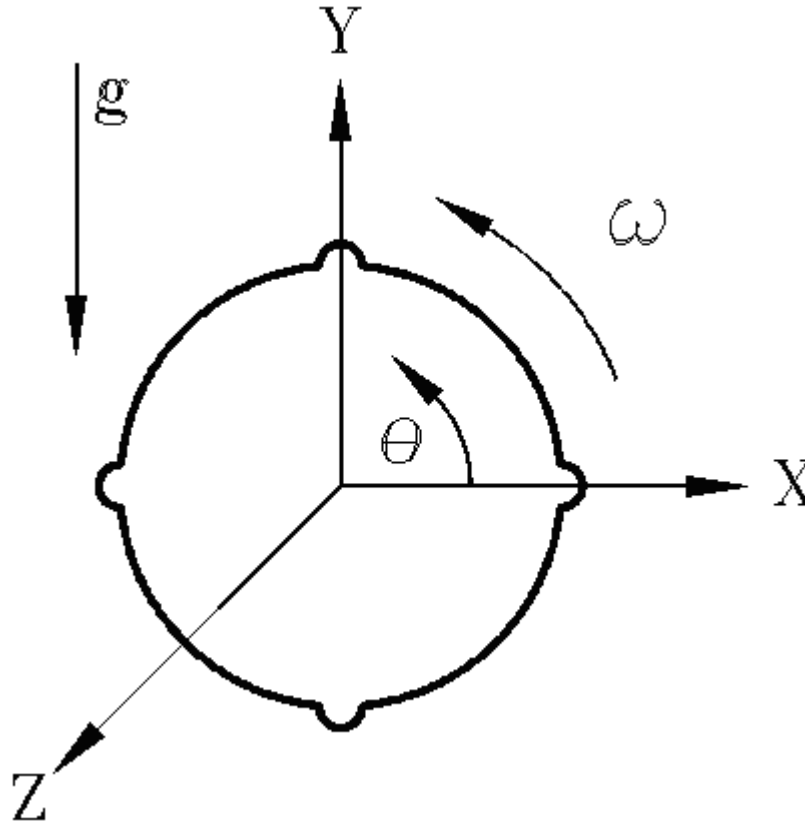
### Fluid Film gridding

*Journal Bearings - 2-D Tilting Pad* contains an automatic grid generation algorithm for the fluid film solution. You have three options for the grid setting: **Course**, **Normal**, and **Fine**.

- Use the **Normal** grid for most cases.
- Use the **Course** grid to speed up the program during the process of qualitatively comparing numerous bearing designs.
- Use the **Fine** grid when the bearing's operating eccentricity ( $e/C_p$ ) is greater than 0.7, as these high eccentricities may require additional grid points to capture the high gradients present in the fluid film.

## Coordinate system

All angles utilized in *Journal Bearings - 2-D Tilting Pad* (input and output) are referenced to a right handed coordinate system with the positive X-axis orientated at standard position. This mandates that shaft rotation is counter-clockwise and that gravity acts in the direction of the negative Y-axis (see the following figure).



*Figure 31. Coordinate system*

## Rotordynamic coefficients

Conventional practice in rotordynamic analysis is to utilize linearized rotordynamic coefficients to represent the transfer function of rotating machinery interconnections (bearings and seals). This practice inherently assumes the frequency dependency of the transfer function's real component can be represented by a constant term **and** a term which is proportional to frequency squared. The model also assumes the frequency dependency of the transfer function's imaginary component can be represented by a term which is proportional to frequency. Considering the transfer function in the Z direction generated by an excitation in the Z direction yields the following equations:

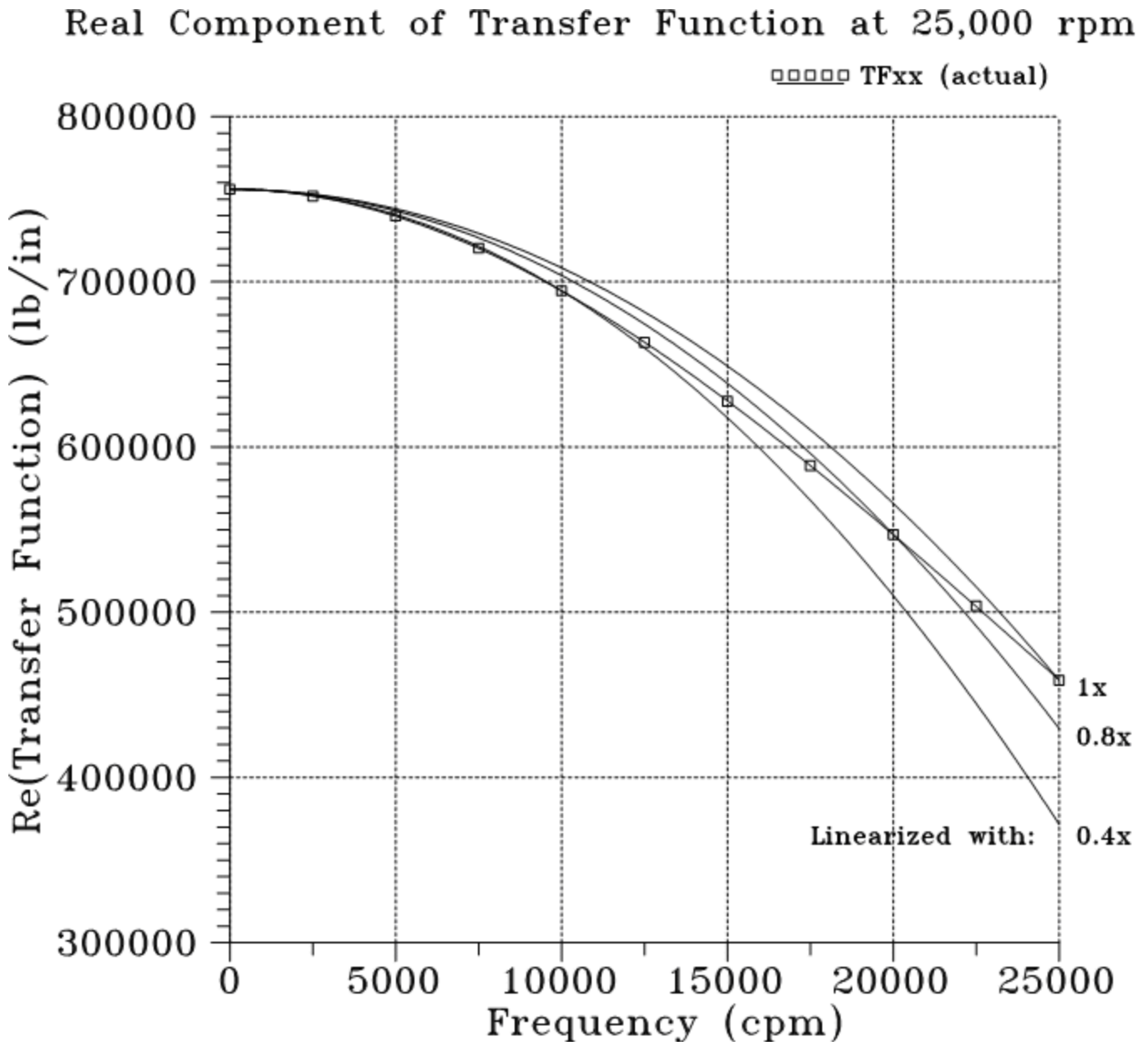
- $\text{real}(TF_z) = a_0 + a_2\omega^2$
- $\text{imaginary}(TF_z) = a_1\omega$

## RAPPID User's Guide

---

The constants  $a_0$ ,  $a_1$ , and  $a_2$ , are commonly referred to as  $K_{zz}$ ,  $C_{zz}$ , and  $M_{zz}$ , respectively. While this assumption is accurate for certain interconnections (incompressible annular seals, and incompressible fixed geometry bearings) it is inaccurate for others (tilt pad, magnetic, and foil bearings). Consider  $TF_{zz}$  for a tilt pad bearing with 6 pads operating at a rotational speed of 25,000 rpm (see the two figures below).

In the following figure, the solid line through the symbols represents a polynomial curve fit of the transfer function calculated by the Program (actual points shown as square symbols). Linearized rotordynamic coefficients can only match two points on the actual transfer function, 0 frequency and a frequency designated by the user. The remaining frequencies are inaccurate. The degree of inaccuracy is dictated by the point selected as the second accurate frequency. The plot shows the curve fits resulting from use of linearized model assuming a second frequency point of 1x, 0.8x, and 0.4x. Note that all of the linearized representations are diverging rapidly from the actual transfer function at 1x, indicating that rotordynamic solutions that utilize linearized coefficients will be highly inaccurate at supersynchronous frequencies.

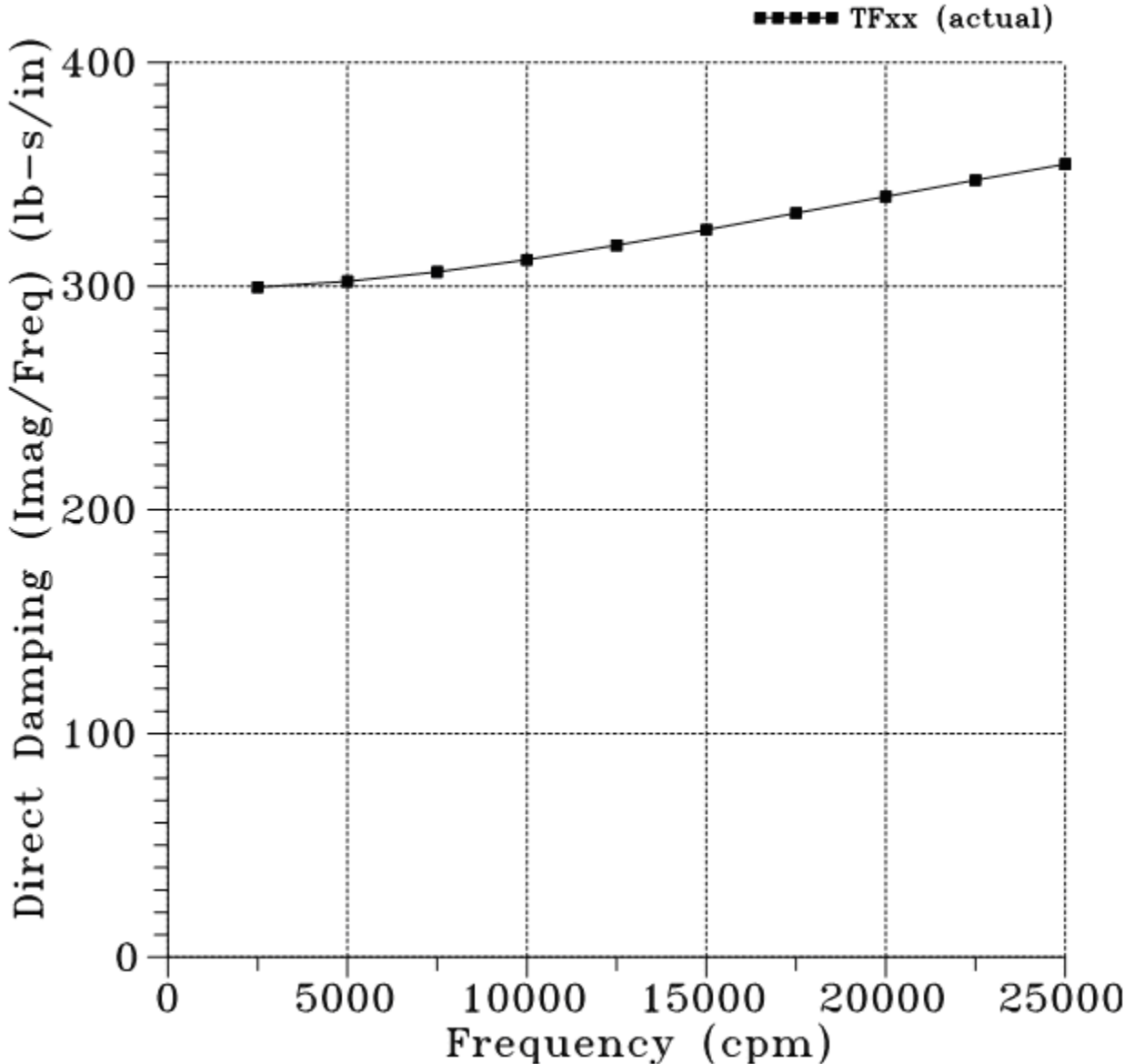


*Figure 32. Sample Tilt Pad Transfer Function with Linearized Representations*



The following figure shows the imaginary part of the transfer function divided by the frequency. The linearized model requires this curve to be horizontal if the model is to be accurate at all frequencies. As such, use of the linearized model can only provide accurate results at a single frequency that you select.

### Imaginary Component of Transfer Function at 25,000 rpm



*Figure 33. Sample Tilt Pad Transfer Function*

Note that the shape of the transfer function is not constant as a function of speed.

### *Implications of Non-Linear Transfer Function on Rotordynamic Analysis*

#### Forced Response Analysis

Since conventional steady state forced response analyses (including the *Rotordynamics Network - Steady State Forced Response* module) provide response data at a single frequency (which is the same as the

excitation frequency) analysis conducted with linearized coefficients can be 100% accurate. As such, use of the *Journal Bearings - 2-D Tilting Pad* output is straight forward with the *Rotordynamics Network - Steady State Forced Response* module, you simply request linearized coefficients at a frequency which is the same order as the analysis to be run. For example, if you intend to run a synchronous response analysis, speed dependent coefficients can be generated using linearized rotordynamic coefficients calculated at the Coefficient Frequency (see input description) equal to the rotor speed (i.e. synchronous or 1x). However, if you intend to run an asynchronous analysis with the *Rotordynamics Network - Steady State Forced Response* module (i.e determine the response at 4x), the Coefficient Frequency must be set to the same order as the desired analysis results (i.e the Coefficient Frequency specified must be 4 times the rotor speed).

### **Eigenvalue**

Conventional, damped, steady-state eigenvalue analyses determine all rotor eigenvalues (natural frequencies and stability parameters) and eigenvectors (mode shapes) at a specified value of rotor speed. At a given speed, the most eigenvalue analyses assume the frequency dependency of the interconnection coefficients adheres to the linearized model to simplify the solution mathematically. This is problematic as we have already illustrated the transfer function for tilt pad bearings deviates substantially from the linearized coefficient model.

The limitations of RSR base *Rotordynamics Network - Steady State Eigenvalue* module constrain you to entering linearized rotordynamic coefficients (you have the ability to specify the second frequency point utilized). By specification of the Coefficient Frequency value you can control which portion of the Natural Frequency Map is fully accurate, which portion is approximate, and which portion is highly inaccurate. Consider the following:

- If you generate linearized coefficients using a Coefficient Frequency equal to the rotor speed (i.e. linearized to synchronous frequency) the only points on the Natural Frequency Map which are fully accurate are those located at intersections between a mode line and the 1x harmonic line. All other locations on the Map are approximate or inaccurate. The degree of inaccuracy is determined by the relationship of the actual transfer function to the linearized fit you select.

It is not possible to generate a fully accurate Natural Frequency Map without the advanced *Rotordynamics Network - Steady State Eigenvalue* module upgrade that can handle non-linear interconnection transfer functions directly. Without the module upgrade, RSR's current recommendation is for you to review the predicted transfer function and select linearized coefficients which represent the frequency section of interest most accurately. Generally speaking the most important result from an eigenvalue analysis is the stability parameter for sub-synchronous modes. Fortunately, review of typical transfer functions indicates that a reasonable representation of the transfer function in the 0x-1x range can be achieved by using a Coefficient Frequency in the range of 0.8-1.0x.

**Warning:** Do not simply implement this recommendation, you must review the actual transfer function for your bearing to determine the most appropriate approximation.

### **Conventions for defining pads**

The following rules must be followed when defining pads in *Journal Bearings - 2-D Tilting Pad*:

- The first full axial groove encountered while rotating counter-clockwise from the positive X-axis must be associated with the first pad defined (remember the load bearing surface of a pad precedes it circumferentially). If a bearing has a groove which traverses the positive X-axis said groove is not the first full groove encountered (because part of the groove resides in the fourth quadrant).
- The Program's mixing model assumes that fluid exiting pad 1 enters pad 2, fluid exiting pad 2 enters pad 3, ....., fluid exiting the last pad enters pad 1. This convention defines the order in which pads must be defined.
- No pad's bearing surface can have a end angle (start angle + arc length) greater than 360°. For example, if the first axial groove occurs at 45° and is associated with a 110° (Arc Length) bearing surface, the pad's starting angular orientation (Arc Location) must be entered as -65° (not 295°).

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

Use the **Analysis Parameter** drop-down menu to select the input value by which you want to vary the values of other inputs. Currently, **Rotor Speed** is the only available **Analysis Parameter**.

The **Rotor Speed** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting speed (**Start**), the increment of speed (**Delta**), and the final speed (**Stop**) for the calculation. The **Rotor Speed** fields must define a constant or increasing speed. Therefore, the **Stop** speed must be greater than or equal to the **Start** speed and the **Delta** speed must be positive. Field entries must be real numbers greater than zero.

### Output Format

Use the **Output Format** drop-down menu to select whether to generate a **Short** or a **Long** output file.

- The **Short** output format includes a listing of the input variables and the calculated rotordynamic coefficients.
- The **Long** output format includes shaft offset, maximum film pressure, fluid temperature rise, maximum bearing temperature, minimum film thickness, required fluid flow rate, and power consumption, in addition to a listing of the input variables and the calculated rotordynamic coefficients.

### Grid

Use the **Grid** drop-down menu to indicate whether a **Coarse**, **Normal**, or **Fine** grid is used in the solution.

See **Fluid Film gridding** on page 202 for more information about the options.

## Groove Temp

Use the **Groove Temp** drop-down menu to specify whether hot fluid carryover is considered in the solution.

- If you select **Fixed (Supply)**, the Program sets the inlet temperature for each sector equal to the specified **Fluid Supply Temperature**.
- If you select **Iterate**, the Program iteratively determines the temperature of the fluid entering the each sector based on the flow velocities and temperatures of the fluid leaving the previous sector(circumferentially) and the fluid injected from the external supply.

## Clearance Input

Use the **Clearance Input** drop-down menu to define the methodology to be used to define the surface profile for each pad. You may select entry of a diametrical clearance and preload or entry of two diametrical clearances.

- **Cb & m:** Diametrical Bearing Clearance and Preload
- **Cp & m:** Diametrical Pad Clearance and Preload
- **Cb & Cp:** Diametrical Bearing Clearance and Diametrical Pad Clearance

Whichever two entries are selected, the third is calculated by the interface using the following equation:

$$m = 1 - (C_b / C_p)$$

## Materials page

The **Materials** page is common for all analysis cases that use material data and is used to define the material properties list that will be available for selection in this analysis case.

Once you insert a new material as a row in the table, there are two options for assigning the material properties:

- Define the properties manually, directly on the **Materials** page in the analysis case.
- Link to a material from the **Materials Library** by clicking the **Materials Library** button, therefore using the properties from the library.

See **Chapter 5: Working with materials** on page 79 for more information.

## Geometry page

Use the fields on the **Geometry** page to specify the geometry of the tilting pad journal bearing. See **Bearing Geometry** on page 197 for more information about the tilt pad pivot types.

## Bearing Geometry box

### Rotor Diameter

Use **Rotor Diameter** to define the rotor outside diameter under the operating conditions specified in the current case (speed and temperature). Valid inputs are real numbers greater than zero.

### Rotor Roughness

The **Rotor Roughness** represents the absolute roughness of the rotor surface. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer. The program constrains the input value to be greater than or equal to zero but less than or equal to 256 micro inches, 6.5 microns, or 0.0065 mm. If the rotor is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

**NOTE:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the rotor is hydraulically smooth with a surface roughness of 8-32 microinches. This can change if the rotor has been damaged or corroded.

### Pivot Type

Use **Pivot Type** to select the type of pivot model to be used in the bearing.

See **Pivot Model** on page 199 for more information about the available options.

### Symmetry

Use the **Symmetry** drop-down menu to specify whether all pads are the same or whether each pad differs:

- **Symmetric:** All the pads are the same and the pads are equally spaced. One set of input fields are available on the **Pad Definition** dialog box.
- **Asymmetric:** Each pad may differ and the pads need not be equally spaced. A grid displays on the **Pad Definition** dialog box allowing you to enter different values for each pad.

### Pad Definition Define button

Use the **Pad Definition Define** button to specify values for each of the pads. Indicate whether the values for each pad on the **Pad Definition** dialog box are the same or different using the **Symmetry** drop-down menu.

## *Pad Definition dialog box*

Use the **Pad Definition** dialog box to specify geometry, materials, and pivot information for each pad.

The fields available on the **Pad Definition** dialog box depend on the option selected in the **Symmetry** drop-down menu on the **Geometry** page.

- **Symmetric:** A set of fields displays allowing you to enter the number of pads and geometry, materials, and pivot information that is applied to every pad.

**NOTE:** The values entered on the **Pad Definition** dialog box when **Symmetric** is selected on the **Geometry** page are applied to each pad. The exception is the **Arc Location** field. Enter the **Arc Location** for the first pad. The Program calculates the **Arc Location** for each of the remaining pads by dividing 360 degrees by the number entered into the **Number of Pads** edit box and incrementing the entered **Arc Location**.

- **Asymmetric:** A grid displays allowing you to add pads and specify different values for each pad.

The following fields are available on the **Pad Definition** dialog box, depending on the **Symmetry** type selected.

### **Number of Pads**

If you selected **Symmetric** in the **Symmetry** drop-down menu on the **Geometry** page, use the **Number of Pads** edit box to specify the number of pads in the bearing.

The following rule applies to the **Number of Pads** field:

- Valid entries are integers greater than zero.

**NOTE:** This field is not available if you selected **Asymmetric** in the **Symmetry** drop-down menu on the **Geometry** page.

### **Axial Length**

Use **Axial Length** to define the axial (along the z-axis) length of the pad's load bearing surface.

The following rule applies to the **Axial Length** field:

- Valid entries are real numbers greater than zero.

### **Arc Length**

Use **Arc Length** to define the circumferential arc length of the pad's load bearing surface.

The following rules apply to the **Arc Length** field:

- Valid entries are real numbers greater than zero but less than 360°.

---

## RAPPID User's Guide

---

- The end angle for each pad must always be less than  $360^\circ$  (**Arc Location** + **Arc Length** <  $360^\circ$ ).

**NOTE:** The **Arc Length** does not include the arc length of the groove associated with the pad. The Program automatically sets the groove arc length by subtracting the end angle of this pad from the start angle of the next pad.

### Arc Location

Use **Arc Location** to define the starting angular location for the pad's load bearing surface. The load bearing portion of the pad extends from the **Arc Location** to the **Arc Location** + **Arc Length**.

The following rules apply to the **Arc Location** field:

- Valid entries are real numbers between  $-360^\circ$  and  $360^\circ$ , exclusive.
- The end angle for each pad must always be less than  $360^\circ$  (**Arc Location** + **Arc Length** <  $360^\circ$ ). If the end angle is greater than  $360^\circ$ , simply define the **Arc Location** with a negative angle.

### Groove Depth

Use **Groove Depth** to define the radial depth of the axial groove which follows the load bearing surface of the pad.

The following rule applies to the **Groove Depth** field:

- Valid entries are real numbers greater than or equal to zero. If set to zero, there will be no ram pressure effect (due to the initial fluid circumferential velocity of the fluid) included in the solution.

### Clearance Circumferential Offset

Use **Clearance Circumferential Offset** to define the angular location of the minimum clearance point (when the journal is centered and the pad is in neutral position) on a preloaded pad and the angular location of the pad's pivot.

The following rule applies to the **Clearance Circumferential Offset** field:

- Valid entries are real numbers between zero and 1, inclusive.

The offset input is a non-dimensional value determined using the following equation:

$$\text{Offset} = \text{Offset Angle} / \text{Arc Length}$$

where the 'Offset Angle' is the angle from the leading edge of the pad to the location of the minimum clearance point.

For a pad with a preload of zero, the Offset is 0.5 by definition.

## **Dia. Brg. Clearance (Cb)**

If the **Clearance Input** option selected on the **Options** page contains 'Cb', use **Dia. Brg. Clearance** to specify the diametrical bearing bore clearance (Cb) of the pad.

The following rule applies to the **Dia. Brg. Clearance** field:

- Valid entries are real numbers greater than 0 but less than or equal to the value entered for the pad clearance.

**NOTE:** If the **Clearance Input** option selected on the **Options** page does not contain 'Cb', then this field is non-editable and the automatically calculated value for 'Cb' displays.

For tilting pad journal bearings, the bore clearance is often referred to as the assembled clearance or the set clearance.

## **Dia. Pad Clearance (Cp)**

If the **Clearance Input** option selected on the **Options** page contains 'Cp', use **Dia. Pad Clearance** to specify the diametrical pad clearance (Cp) of the pad.

The following rule applies to the **Dia. Pad Clearance** field:

- Valid entries are real numbers greater than or equal to the value entered for the bore clearance (Cb).

**NOTE:** If the **Clearance Input** option selected on the **Options** page does not contain 'Cp', then this field is non-editable and the automatically calculated value for 'Cp' displays.

The pad clearance is often referred to as the ground in clearance.

## **Preload (m)**

If the **Clearance Input** option selected on the **Options** page contains 'm', use **Preload** to enter the preload of the pad.

The following rule applies to the **Preload** field:

- Valid entries are real numbers between zero and 1, inclusive.

**NOTE:** If the **Clearance Input** option selected on the **Options** page does not contain 'm', then this field is non-editable and the automatically calculated value for 'm' displays.

## **Material 1/Material 2**

Use **Material 1** and **Material 2** to specify the materials used to fabricate the pad. **Material 1** is the material in contact with the fluid film.



## Axial Thickness

Use the **Axial Thickness** fields to specify the axial thickness of each material used to fabricate the pad.

## Pivot Radial Stiffness

Use **Pivot Radial Stiffness** to define the radial compliance of the pivot. This input allows for modeling the radial flexibility of the pad pivot. The Program automatically includes the pivot's flexibility in the determination of the rotordynamic coefficients. To simulate the effect of a rigid pivot enter a pivot stiffness of 0 and the Program will automatically consider the pivot to be rigid.

The following rule applies to the **Pivot Radial Stiffness** field:

- Valid entries are real numbers greater than or equal to zero.

## Pivot Radial Depth

Use **Pivot Radial Depth** to define the radial location of the pad's effective pivot point, relative to the pad's bearing surface, when the pad is in the neutral position (unrotated).

The following rules apply to the **Pivot Radial Depth** field:

- Valid entries are real numbers greater than or equal to zero.
- The value entered is a function of the type of pivot designated in the **Pivot Type** drop-down menu on the **Geometry** page.

**NOTE:** An entry of zero would indicate that the pad rotates about a point on the pad's bearing surface.

- **Pinned Pivot:** The pivot depth value entered must designate the radial distance from the pad's bearing surface to the center of the pin when the pad is at its neutral position (i.e. unrotated). This entry, in addition to the pad's **Clearance Circumferential Offset** defines the location of the pad's pivot point. Remember for a pinned pivot the pad is assumed to rotate about the center of the pin [sliding friction (dynamic) exists on the outside diameter of the pin].
- **Rocker Back Pivot:** The pivot depth value entered must designate the radial distance from the pad's surface to the point of contact between the rocker surface (on the pad) and the mating surface on the bearing housing when the pad is at its neutral position (i.e. unrotated). Remember for a rocker back pivot the pad's rocker surface is assumed to roll on the bearing housing's mating surface (the pad can not slide).
- **Ball/Socket Pivot:** The pivot depth value entered should designate the radial distance from the pad's surface to the point of contact between the socket (machined into the pad) and the mating surface on the bearing housing (ball) when the pad is at its neutral position (i.e. unrotated). Remember for a ball/socket pivot the pad's socket is assumed to roll on the ball (the pad can not slide).

# RAPPID User's Guide

---

## Pivot Circumferential Offset

The **Pivot Circumferential Offset** edit box is not editable because the **Pivot Circumferential Offset** is constrained to equal the **Clearance Circumferential Offset** in this version of the Program. This field displays the value entered in the **Clearance Circumferential Offset** edit box.

## Coefficient of Friction

Use **Coefficient of Friction** to define the friction coefficient used to determine the magnitude of the friction force between the pad and the pin that resists circumferential pad rotation.

The following rule applies to the **Coefficient of Friction** field:

- Valid entries are real numbers greater than or equal to zero.

**NOTE:** The **Coefficient of Friction** field is only available when **Pinned** is selected as the **Pivot Type**. Since the **Pinned Pivot** model assumes that the pad is always free to rotate, i.e., static friction has been overcome, the appropriate value for the coefficient of friction is the sliding coefficient of dynamic friction.

## Moment of Inertia

Use **Moment of Inertia** to define the circumferential moment of inertia of the pad about its pivot point when the pad is at its neutral position (i.e. it is unrotated). The availability and value of this field is determined by the entries (either on the **Materials** page or in the **Materials Library** – see Chapter 5: Working with materials on page 79) in the **Density** fields for both materials:

If the densities entered for both materials are...	the Moment of Inertia field is...
zero	editable.
non-zero	not editable and 'Auto' displays in the field.

## Pivot surface fields

The two fields below **Moment of Inertia** allow you to specify information about the surface of the pivot. The two fields available depend on the **Pivot Type** selected:

### **Pinned**

#### **Pin Diameter**

Use **Pin Diameter** to specify the outside diameter of the pin.

The following rule applies to the **Pin Diameter** field:

# RAPPID User's Guide

---

- Valid entries are real numbers greater than or equal to zero.

**NOTE:** The pin diameter is used only to determine the magnitude of the moment generated by the dynamic friction force action on the pin's surface.

## Rocker

### Housing Radius

Use **Housing Radius** to specify the radius of the bearing housing's surface upon which the pad rolls.

The following rule applies to the **Housing Radius** field:

- Valid entries are real numbers greater than the radius of the back of the pad.

**NOTE:** The center of the **Housing Radius** is constrained to be the geometric center of the bearing.

### Rocker Radius

Use **Rocker Radius** to specify the radius of the surface machined onto the pad which rolls on the bearing housing.

The following rule applies to the **Rocker Radius** field:

- Valid entries are real numbers greater than zero but less than the **Housing Radius**.

## Ball/Socket

### Ball Radius

Use **Ball Radius** to specify the radius of the bearing housing's ball upon which the pad rolls.

The following rule applies to the **Ball Radius** field:

- Valid entries are real numbers greater than zero.

### Socket Radius

Use **Socket Radius** to specify the radius of the socket machined into the pad which rolls on the ball.

The following rule applies to the **Socket Radius** field:

- Valid entries are real numbers greater than the **Ball Radius**.

**NOTE:** The **Housing Radius**, **Ball Radius**, and **Socket Radius** are all used to determine how the pad's clearance and point of rotation change as the pad rolls in response to the loading from the fluid film.

## Fluids page

The **Fluids** page is common to all analysis cases that use a single fluid (compound). Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**
- **Real Properties**

For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

## Operating Conditions page

Use the **Operating Conditions** page in a *Journal Bearings - 2-D Tilting Pad* analysis case to set the pressure and temperature boundary conditions and the solution method to obtain eccentricity/load results.

**NOTE:** If there is a ... button next to an edit box, you may enter variable input for that input field. See **Constant or variable analysis input** in **Part IV – The Analysis Case Managers of Tutorial 1 – The basics** for more information.

## Rotor Speed

Currently, **Rotor Speed** is the analysis parameter by default because it is the only input available as the analysis parameter. Input range and increment for **Rotor Speed** is entered on the **Options** page (see page 208).

## Pressure/Temperature box

### Fluid Supply Pressure

The **Fluid Supply Pressure** represents the pressure of the fluid entering all grooves from the external supply.

The following rule applies to the **Fluid Supply Pressure** field:

- Valid entries are real numbers greater than or equal to zero.

**NOTE:** The **Cavitation Pressure** (???), ambient pressure (**Left Boundary Pressure** and **Right Boundary Pressure**), and **Supply Pressure** are relative to each other. As such, they may be entered in either gauge or absolute values as long as all three are entered in consistent terms (do not specify some pressures in gauge and others in absolute).

## **Fluid Supply Temperature**

The **Fluid Supply Temperature** represents the temperature of the fluid entering all grooves in the bearing from the external supply. This temperature is considered constant and is used, in conjunction with the Program's mixing model, to determine the fluid temperature at the pad's inlet (first row of nodes circumferentially).

The following rule applies to the **Fluid Supply Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## **Left Boundary Pressure**

The **Left Boundary Pressure** represents the pressure boundary condition applied to the sector exit plane with the highest z coordinate value.

The following rules apply to the **Left Boundary Pressure** field:

- Valid entries are real number greater than zero.

## **Right Boundary Pressure**

The **Right Boundary Pressure** represents the pressure boundary condition applied to the pad exit plane with the lowest z coordinate value.

**NOTE:** This field is not editable. The value entered into the **Left Boundary Pressure** applies to both left and right boundary pressures in this version of the Program and is displayed in this field.

## **Shaft Temperature**

The **Shaft Temperature** specifies the fixed surface temperature of the shaft in the bearing.

The following rule applies to the **Shaft Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## **Pad Back Temperature**

Use **Pad Back Temperature** to specify the fixed temperature to be applied at the outer diameter of the pad. The fixed back of pad temperature is assumed to be the same for all pads. The entered temperature is utilized in the conduction model employed to simulate the radial heat transfer through the pad.

---

## RAPPID User's Guide

---

The following rule applies to the **Pad Back Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### Loading box

Use the drop-down menu at the top of the **Loading** box to specify the operation mode for the Program's analysis. There are two options:

- **Prescribed Load**

For the **Prescribed Load** option an initial guess for the journal's equilibrium position is required. If you select **Prescribed Load**, a drop-down menu displays with two options available:

- **Auto Guess:** The **Eccentricity** field and the **Eccentricity Angle** field are not available. The initial guess is generated automatically by the Program using an internal guess algorithm.
- **Manual Guess:** The **Eccentricity** field and the **Eccentricity Angle** field are available. Enter an initial guess into each field manually.

If you select **Prescribed Load**, the **Convergence Criteria**, **Journal Weight**, **Applied Load**, and **Applied Load Direction** fields are available.

- **Prescribed Eccentricity**

If you select **Prescribed Eccentricity**, the **Eccentricity** and **Eccentricity Angle** fields are available.

Specify the following values as applicable.

### Eccentricity

Use **Eccentricity** to define the magnitude of the eccentricity vector.

- For **Prescribed Eccentricity**, this value defines the position of the journal to be used in the solution.
- For **Prescribed Load**, this value defines the initial guess for the equilibrium position of the journal.

Use the **Guess** drop-down menu to specify whether the initial guess is automatically generated (**Auto Guess**) or entered manually (**Manual Guess**).

The following rules apply to the **Eccentricity** field:

- Valid entries are real numbers greater than or equal to zero.

---

## RAPPID User's Guide

---

- The **Eccentricity** must be less than half of the **Diametrical Pad Clearance** (see **Pad Definition dialog box** on page 211) of the bearing at the **Eccentricity Angle**.

### Eccentricity Angle

Use **Eccentricity Angle** to define the angular orientation of the eccentricity vector.

- For **Prescribed Eccentricity**, this value defines the position of the journal to be used in the solution.
- For **Prescribed Load**, this value defines the orientation of the eccentricity vector to be used as the initial guess for the equilibrium position of the journal.

Use the **Guess** drop-down menu to specify whether the initial guess is automatically generated (**Auto Guess**) or entered manually (**Manual Guess**).

The following rule applies to the **Eccentricity Angle** field:

- Valid entries are real numbers between  $-360^\circ$  and  $360^\circ$ , inclusive.
- This angle is relative to the positive X-axis and is positive in the counter-clockwise direction.

### Convergence Criteria

Use **Convergence Criteria** to define the convergence criteria, as a percentage of the user specified load, used by the Program to determine when to cease iteration on reacted load.

**NOTE: Convergence Criteria** is not available for **Prescribed Eccentricity**.

The following rule applies to **Convergence Criteria**:

- Valid entries are real numbers between 0.5 and 5, inclusive. The valid range of entries corresponds to 0.5-5% of the user specified load for the case.

**NOTE:** The execution time of the Program is very sensitive to the value entered in this field. The lower the convergence criteria, the longer the Program takes to obtain the desired solution. Generally, a convergence criteria of 1-2% is sufficient. However, when running numerous cases for initial evaluation purposes, higher convergence criteria may be desirable to reduce computational time.

### Journal Weight

Use **Journal Weight** to define the load applied to the bearing in the direction of gravity (along the negative Y-axis, or  $270^\circ$ ). **Journal Weight** represents the weight of the rotor supported by this bearing.

The actual load applied to the bearing during the Program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in the **Applied Load** and **Applied Load Direction** fields). The applied load vector is intended to be used to represent arbitrarily orientated forces (i.e. gear load) which act in conjunction with the rotor weight.

# RAPPID User's Guide

---

**NOTE: Journal Weight** is not available for **Prescribed Eccentricity**.

The following rule applies to **Journal Weight**:

- Valid entries are real numbers greater than or equal to zero. Zero is only valid if a non-zero value is entered in the **Applied Load** field.

## **Applied Load**

Use **Applied Load** to define the magnitude of the load vector applied to the bearing in addition to the user input **Journal Weight**. The direction of the applied load vector is set by the value entered in the **Applied Load Direction** field.

The applied load vector is intended to be used to represent arbitrarily orientated loads (i.e. gear loads) which are applied to the bearing. The load applied to the bearing during the Program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in the **Applied Load** and **Applied Load Direction** fields).

**NOTE: Applied Load** is not available for **Prescribed Eccentricity**.

The following rule applies to **Applied Load**:

- Valid entries are real numbers greater than or equal to zero. Zero is only valid if a non-zero value is entered in the **Journal Weight** field.

## **Applied Load Direction**

Use **Applied Load Direction** to define the angular orientation of the applied load vector. The magnitude of the applied load vector is set by the value entered for the **Applied Load**.

The applied load vector is intended to be used to represent arbitrarily orientated loads (i.e. gear loads) which are applied to the bearing. The load applied to the bearing during the Program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in the **Applied Load** and **Applied Load Direction** fields).

**NOTE: Applied Load Direction** is not available for **Prescribed Eccentricity**.

The following rules apply to **Applied Load Direction**:

- Valid entries are real numbers between  $-360^{\circ}$  and  $360^{\circ}$ , inclusive.
- This angle is relative to the positive X-axis and is positive in the counter-clockwise direction.



## Sample analysis cases

### Manual Example #1: 5 pad bearing linearized rotordynamic coefficients

Many tilt pad bearings are designed with 5 pads. This section presents a *Journal Bearings - 2-D Tilting Pad* sample analysis of a 5 pad bearing to illustrate use of the Program to calculate linearized rotordynamic coefficients. The following data summarize the features of the sample bearing:

#### Pad Data

- 5 identical pads
- diametral bearing clearance is 0.004 inches and the preload is 0.4
- first pad is centered on the positive X-axis (start angle is -29 degrees)
- rotor diameter under operating conditions is 2.000 inches
- pad is comprised of 0.008 inch thick layer of babbitt and 0.49 inch thick layer of steel
- pad inertia will be automatically calculated from the entered densities
- pad thermal growth will be automatically calculated from the thermal expansion coefficient

#### Pivot Data

- 5 identical pin pivots
- circumferential pivot location is the center of the pad
- radial pivot location is 0.375 inches below the bearing surface
- sliding coefficient of friction between the pin and pad is 0.1
- diameter of the pin is 0.1875 inches

#### Operating Conditions

- ISO 32 oil, inlet temperature of 120 degrees F
- oil supply pressure is 2 psi above ambient pressure
- rotor speed is 5000 rpm
- journal temperature is 150 degrees F
- back of pad temperature is 120 degrees F
- hot oil carry over is not being considered

- applied load is 300 lbf on the positive X-axis (load on pad)

### Dynamic Coefficients

- The bearing's transfer function is to be linearized using its value at 5000 cpm.

**NOTE:** The Program does not account for growth of the journal (due to rotation, thermal, or press fit effects) or growth of the housing (due to thermal or press fit effects). As such, you must take such changes into account when entering the journal diameter and the bearing clearance.

### Manual Example #2: 4 pad bearing with non-linear transfer function

Many tilt pad bearings are designed with 4 pads. This section presents a *Journal Bearings - 2-D Tilting Pad* sample analysis of a 4 pad bearing to illustrate use of the Program to calculate non-linear transfer functions. The following data summarize the features of this bearing:

#### Pad Data

- 4 identical pads
- diametral bearing clearance is 0.004 inches and the preload is 0.4
- first pad is centered in the first quadrant (start angle is 9 degrees)
- rotor diameter under operating conditions is 2.000 inches
- pad is comprised of 0.008 inch thick layer of babbitt and 0.49 inch thick layer of steel
- pad inertia will be automatically calculated from the entered densities
- pad thermal growth will not be included

#### Pivot Data

- 5 identical rocker back pivots
- circumferential pivot location is the center of the pad
- radial pivot location is 0.498 inches below the bearing surface
- housing radius is 0.010 inches greater than the radius of the rocker

#### Operating Conditions

- oil has user defined properties and an inlet temperature of 120 degrees F
- oil supply pressure is equal to ambient pressure
- rotor speed is 30,000 rpm

- journal temperature is 150 degrees F
- back of pad temperature is 120 degrees F
- hot oil carry over is not being considered
- loading is 100 lbf from journal weight (load between pads)

### **Dynamic Coefficients**

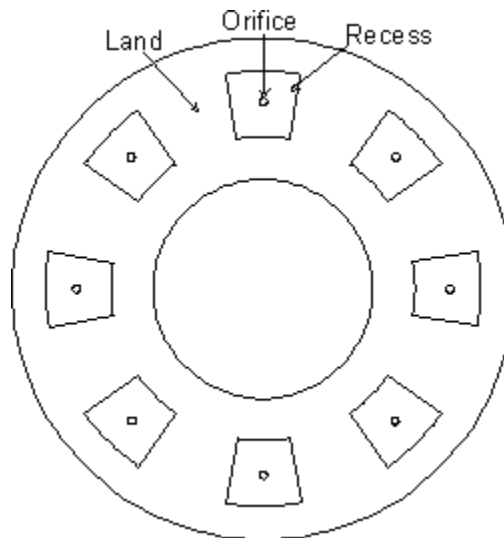
- determine the non-linear transfer function at 0, 15,000, 30,000, 45,000, and 60,000 cpm.

**NOTE:** The Program does not account for growth of the journal (due to rotation, thermal, or press fit effects) or growth of the housing (due to thermal or press fit effects). As such the user must that such changes into account when entering the journal diameter and the bearing clearance.

## Chapter 17: Thrust Bearing – Hydrostatic

### Overview

Hydrostatic bearings are fluid film bearings that utilize the principle of orifices in series to generate a force. As a shaft is displaced away from the runner, the flow through the bearing is primarily restricted by the orifice resulting in a pressure decrease in the bearing recess and a drop in the force. This reduction in force changes the thrust balance enabling the shaft to move closer to the runner. As the shaft moves closer, the flow is primarily restricted by the bearing land resulting in a pressure increase in the bearing recess and an increase in the force. Even if the bearing does not have a recess, this force fluctuation phenomenon occurs. A typical layout of a recessed, orifice compensated hydrostatic bearing is illustrated in the following figure.



*Figure 34. Hydrostatic bearing features*

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

Use the **Analysis Parameter** drop-down menu to select the input value by which you want to vary the values of other inputs. Currently, **Rotor Speed** is the only available **Analysis Parameter**.

The **Rotor Speed** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting speed (**Start**), the increment of speed (**Delta**), and the final speed (**Stop**) for the calculation. The **Rotor Speed** fields must define a constant or increasing speed. Therefore, the **Stop** speed must be greater than or equal to the **Start** speed and the **Delta** speed must be positive. Field entries must be real numbers greater than zero.

## Output Format

Use the **Output Format** drop-down menu to select whether to generate a **Short** or a **Long** output file.

- The **Short** output format includes a listing of the input variables and output consisting of basic flow parameters and the rotordynamic coefficients.
- The **Long** output format includes a listing of the input variables, and output consisting of an extensive listing of the mass and volumetric flowrate, load capacity, friction power loss, recess pressure, circumferential and axial Reynolds numbers, maximum values in the bearing, pertinent thermodynamic and transport values for every orifice and recess, and rotordynamic coefficients.

## Grid

Use the **Grid** drop-down menu to indicate whether a **Coarse**, **Normal**, or **Fine** grid is used in the solution.

## Materials page

The **Materials** page is common for all analysis cases that use material data and is used to define the material properties list that will be available for selection in this analysis case.

Once you insert a new material as a row in the table, there are two options for assigning the material properties:

- Define the properties manually, directly on the **Materials** page in the analysis case.
- Link to a material from the **Materials Library** by clicking the **Materials Library** button, therefore using the properties from the library.

See **Chapter 5: Working with materials** on page 79 for more information.

## Geometry page

The fields on the **Geometry** page are used to define the overall bearing geometry, the detailed internal geometry, and the bearing housing geometry for heat conduction. The bearing geometry can be defined as constant or dependent on the analysis parameter (e.g., rotor speed). See **Entering variable data** on page 47 for more information.

## Bearing Geometry box

### Outside Diameter

The **Outside Diameter** represents the mean outer diameter of the bearing land. It is assumed that the bearing is a circular ring. Valid entries are positive floating point values.

## **Inside Diameter**

The **Inside Diameter** represents the mean inner diameter of the bearing land. It is assumed that the bearing is a circular ring. Valid entries are positive floating point values.

## **Rotor Roughness**

The **Rotor Roughness** represents the absolute roughness of the rotor surface. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer. The program constrains the input value to be greater than or equal to zero but less than or equal to 256 micro inches, 6.5 microns, or 0.0065 mm. If the rotor is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

**NOTE:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the rotor is hydraulically smooth with a surface roughness of 8-32 microinches. This can change if the rotor has been damaged or corroded.

## **Recesses**

Use **Recesses** to turn on (**Yes**) and off (**No**) the **Recess** entries on the **Supply Ports Definition** dialog box in *2-D Hydrostatic* analysis cases. For *1-D Hydrostatic* analysis cases, **Yes** is the only option available.

## **Supply Ports Define button**

Use the **Define** button next to **Supply Ports** to define all of the detailed geometry related to the recess and orifice.

## **Clearance**

The **Clearance** represents the gap formed between the surface of the rotor and the surface of the bearing (stator).

There are two types of clearance, indicating whether or not the clearance varies radially:

- **Radially Constant**
- **Radially Varying** (currently not available)

The following rules apply to the **Radially Constant Clearance** field:

- Valid entries are positive floating point values.
- The current version of the software restricts the **Radially Constant Clearance** entry to a constant value along the length of the bearing.

## *Supply Ports Definition dialog box*

When you click the **Define** button next to **Supply Ports** on the **Geometry** page, the **Supply Ports Definition** dialog box displays.

The following differences exist between *1-D* and *2-D Hydrostatic* analysis cases:

- If the analysis case is a *1-D Hydrostatic* case, edit boxes are provided for entry of the data.
- If the analysis case is a *2-D Hydrostatic*, a grid is provided. The top row of the grid is not editable if you selected **Yes** in the **Recesses** drop-down menu on the **Geometry** page.

The following fields are available depending on the type of analysis case and on the selection in the **Recesses** drop-down menu.

### **Number of Ports**

For *1-D Hydrostatic* analysis cases only, enter the number of ports. The values entered into the remaining edit boxes on the dialog box will be applied to each port.

### **Radial Location**

The **Radial Location** represents the location of the recess centerline relative to the bearing path length.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu. It is not available at all for *1-D Hydrostatic* analysis cases.

The following rules apply to the **Radial Location** field:

- Valid entries are positive floating point values.
- The recess **Radial Location** must be less than (bearing **Length** – recess **Radial Length**/2) and greater than (recess **Radial Length**/2).
- The recess **Radial Location** must be the same for all recesses in a bearing.

### **Arc Length**

The **Arc Length** represents the angular extent of the recess in the circumferential direction.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu.

The following rules apply to the **Arc Length** field:

- Valid entries are positive floating point values.
- The recess **Arc Length** must be less than (360/number of recesses).

## Arc Location

The **Arc Location** represents the angular location of the recess centerline in the circumferential direction.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu. It is not available at all for *1-D Hydrostatic* analysis cases.

The following rules apply to the **Arc Location** field:

- Valid entries are non-negative floating point values.
- The recess **Arc Location** must be set such that the recesses do not overlap.

## Depth

The **Depth** represents the recess depth as measured axially inward from the bearing surface.

**NOTE:** This field is not editable if the current analysis case is a *2-D Hydrostatic* case and you selected **No** in the **Recesses** drop-down menu.

The following rules apply to the **Depth** field:

- Valid entries are positive floating point values.
- If a nonuniform recess geometry exists, then an equivalent recess depth, which yields the same volume as the nonuniform geometry should be used as input to the program.

## Orifice Diameter

The **Orifice Diameter** represents the diameter of the orifice in the recess.

The following rules apply to the **Orifice Diameter** field:

- Valid entries include positive floating point values.
- If a non-circular orifice exists, then the flow diameter must be calculated and used as input.
- The **Orifice Diameter** must be less than the **Radial Length**.

## Relief Diameter

The **Relief Diameter** represents the diameter of the cylindrical area defined between the exit plane of the orifice and the shaft (also known as the **Curtain Diameter**).

**NOTE:** This field is not available for *1-D Hydrostatic* analysis cases.

The following rules apply to the **Relief Diameter** field:

- Valid entries are positive floating point values.



- The **Relief Diameter** must be less than or equal to the **Radial Length** and greater than or equal to the **Orifice Diameter**.
- The **Relief Diameter** must be the same for all ports in a bearing.

### **Orifice Angle**

The **Orifice Angle** represents the angle of the orifice from the vertical in the circumferential plane.

The following rules apply to the **Orifice Angle** field:

- Valid entries are floating point values in the range  $-90$  to  $+90$  degrees.
- A positive orifice angle would mean that the flow through the orifice exits the orifice in the direction opposite to the rotation of the shaft.

### **Orifice Radial Location**

The **Orifice Radial Location** represents the location of the recess centerline relative to the bearing path length.

**NOTE:** This field is not available for *I-D Hydrostatic* analysis cases.

The following rules apply to the **Orifice Radial Location** field:

- Valid entries are positive floating point values.
- The orifice must lie within the recess.
- The **Orifice Radial Location** must be the same for all ports in a bearing.

### **Orifice Angular Location**

The **Orifice Angular Location** represents the angular location of the *orifice centerline at the orifice exit plane* in the circumferential direction.

**NOTE:** This field is not available for *I-D Hydrostatic* analysis cases.

The following rules apply to the **Orifice Angular Location** field:

- Valid entries are non-negative floating point values.
- The orifice must be located within the boundaries of the recess.

### **Orifice Supply Volume**

The **Orifice Supply Volume** represents the volume of the flow channel supplying the orifice.

**NOTE:** This field is not available for *I-D Hydrostatic* analysis cases.

The following rules apply to the **Orifice Supply Volume** field:

- Valid entries are positive floating point values.
- The **Orifice Supply Volume** must be the same for all ports in a bearing.

## Fluids page

The **Fluids** page is common to all analysis cases that use a single fluid (compound). Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**
- **Real Properties**

For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

## Operating Conditions page

Use the **Operating Conditions** page in a *Thrust Bearing* analysis case to set the pressure and temperature boundary conditions, the solution method to obtain position and load, and the angular misalignment (if any).

**NOTE:** If there is a ... button next to an edit box, you may enter variable input for that input field. See **Constant or variable analysis input** in **Part IV – The Analysis Case Managers** of **Tutorial 1 – The basics** for more information.

## Rotor Speed

Currently, **Rotor Speed** is the analysis parameter by default because it is the only input available as the analysis parameter. Input range and increment for **Rotor Speed** is entered on the **Options** page (see page 225).

## **Pressure/Temperature box**

### **Fluid Supply Pressure**

The **Fluid Supply Pressure** represents the pressure of the fluid upstream of the orifice inlet.

The following rule applies to the **Fluid Supply Pressure** field:

- Valid entries are positive floating point values.

### **Fluid Supply Temperature**

The **Fluid Supply Temperature** represents the temperature of the fluid upstream of the orifice inlet.

The following rule applies to the **Fluid Supply Temperature** field:

- Valid entries are floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### **O.D. Boundary Pressure**

The **O.D. Boundary Pressure** represents the pressure of the fluid at the outside diameter Boundary of the bearing exit plane.

The following rule applies to the **O.D. Boundary Pressure** field:

- Valid entries are positive floating point values.

### **O.D Boundary Temperature**

The **O.D. Boundary Temperature** represents the temperature of the fluid at the outside diameter boundary of the bearing exit plane.

The following rule applies to the **O.D. Boundary Temperature** field:

- Valid entries are positive floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### **I.D. Boundary Pressure**

The **I.D. Boundary Pressure** represents the pressure of the fluid at the inside diameter boundary of the bearing exit plane.

The following rule applies to the **I.D. Boundary Pressure** field:

- Valid entries are positive floating point values.

### **I.D. Boundary Temperature**

The **I.D. Boundary Temperature** represents the temperature of the fluid at the inside diameter boundary of the bearing exit plane.

---

## RAPPID User's Guide

---

The following rule applies to the **I.D. Boundary Pressure** field:

- Valid entries are positive floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### **Shaft Temperature**

The **Shaft Temperature** specifies the fixed surface temperature of the shaft in the bearing.

**NOTE:** This field is not available for *I-D Hydrostatic* analysis cases.

The following rule applies to the **Shaft Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### **Bearing Back Temperature**

Use **Bearing Back Temperature** to specify the fixed temperature to be applied at the outer surface of the bearing's housing (stationary). The radial thickness of the bearing's housing is the sum of the entries for the following inputs: **Radial Thickness** for **Material 1** and **Radial Thickness** for **Material 2**. The housing radial thickness and the fixed back temperature are assumed to be the same for all sectors. The entered temperature is utilized in the conduction model employed to simulate the radial heat transfer through the bearing's housing.

**NOTE:** This field is not available for *I-D Hydrostatic* analysis cases.

The following rule applies to the **Bearing Back Temperature** field:

- Valid entries are positive floating point values greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

### **Exit Inflow Conditions box**

**NOTE:** The **Exit Inflow Conditions** box is not available for *I-D Hydrostatic* analysis cases.

### **Inlet Swirl Ratio**

Use **Fluid Inlet Swirl Ratio** to specify the ratio of the circumferential velocity of the fluid at the inlet to the bearing to the rotor surface velocity. The value of this term is related to its location in the turbomachine. If no good estimate is available use a value between 0.50-0.80.

### **Entrance Loss Coefficient**

Use **Entrance [Inlet] Loss Coefficient** to specify the entrance loss coefficient for flow into the bearing. This is a dimensionless number, and is defined as:

$$\text{Entrance Loss} = \frac{2\Delta p}{\rho V^2} - 1$$

---

## RAPPID User's Guide

---

This coefficient is used to determine the pressure loss at a sudden contraction. If no similar experimental data is available, pipe flow contraction coefficients can be calculated using an equation from Blevins (Blevins, R.D., 1984, Applied Fluid Dynamics Handbook, Van Nostrand Reinhold Co., New York, p. 76) for pressure loss at pipe contractions:

$$\text{Entrance Loss} = 0.5871 + 0.3954\beta - 4.5385\beta^2 + 14.24 \\ - 19.22\beta^4 + 8.54\beta^5$$

where  $\beta$  is the ratio of seal clearance area to the projected area at the entrance to the seal.

### Loading box

Use the drop-down menu at the top of the **Loading** box to specify the operation mode for the Program's analysis. There are two options:

- **Prescribed Load**

For the **Prescribed Load** option an initial guess for the journal's equilibrium position is required. If you select **Prescribed Load**, a drop-down menu displays with two options available:

- **Auto Guess:** The **Eccentricity** field and the **Eccentricity Angle** field are not available. The initial guess is generated automatically by the Program using an internal guess algorithm.
- **Manual Guess:** The **Eccentricity** field and the **Eccentricity Angle** field are available. Enter an initial guess into each field manually.

If you select **Prescribed Load**, the **Convergence Criteria**, **Journal Weight**, **Applied Load**, and **Applied Load Direction** fields are available.

- **Prescribed Eccentricity**

If you select **Prescribed Eccentricity**, the **Eccentricity** and **Eccentricity Angle** fields are available.

Specify the following values as applicable.

### Film Thickness

Use **Film Thickness** to specify an additional axial displacement that is added to the defined **Clearance**. This enables you to vary the initial guess of the final bearing **Clearance** without changing the design **Clearance** entered on the **Geometry** page.

The following rule applies to the **Film Thickness** field:

- Valid entries are floating point values.

## Convergence Criteria

Use **Convergence Criteria** to specify the percent error in the calculated load relative to the **Prescribed Load**.

**NOTE:** **Convergence Criteria** is not available for **Prescribed Eccentricity**.

The following rule applies to **Convergence Criteria**:

- Valid entries are positive floating point values.

## Applied Load

Use **Applied Load** to define the magnitude of the load vector applied to the bearing in addition to the user input **Journal Weight**.

**NOTE:** **Applied Load** is not available for **Prescribed Eccentricity**.

The following rule applies to **Applied Load**:

- Valid entries are non-negative floating point values.

**NOTE:** The applied load vector is intended to be used to represent arbitrarily orientated loads (i.e. impeller side loads) that are applied to the bearing. The load applied to the bearing during the program's solution is the vector sum of the **Journal Weight** entry and the applied load vector (defined in **Applied Load** and **Applied Load Direction**).

## **Misalignment box**

### Magnitude

Use **Magnitude** in the **Misalignment** box to define the magnitude of the angular misalignment of the shaft relative to the shaft centerline.

The following rule applies to **Magnitude**:

- Valid entries are positive floating point values.

### Angle

Use **Angle** in the **Misalignment** box to define the angular orientation of the angular misalignment vector relative to the X-axis.

The following rule applies to **Angle**:

- Valid entries are real numbers between -360° and 360°, inclusive.

### **Center**

Use **Center** in the **Misalignment** box to define the center of rotation of the angular misalignment relative to the bearing axial centerline.

The following rule applies to **Center**:

- Valid entries are floating point values.

## Chapter 18: Thrust Bearing – 2-D Tilting Pad

### Overview

The *Thrust Bearings - 2-D Tilting Pad* module is for analysis of axially fed tilting pad fluid film thrust bearings. Its primary function is to allow the designer to determine rotordynamic coefficients for various tilting pad pivot and pad designs in an effort to select a suitable bearing for the machinery application under consideration.

The *Thrust Bearings - 2-D Tilting Pad* module calculates the bearing's non-linear transfer function or linearized stiffness, damping, and mass coefficients by taking the user's input and solving a full set of film averaged Navier-Stokes based governing equations: momentum, continuity, and energy. The flow in the bearing may be laminar, transitional, or turbulent. Rotordynamic coefficients are determined using a perturbation analysis which assumes the journal is exhibiting small amplitude axial motion about its static equilibrium position. The *Thrust Bearings - 2-D Tilting Pad* also reports, rotor operating position, maximum film pressure, fluid temperature rise, maximum bearing temperature, minimum film thickness, required fluid flow rate, and power consumption.

### Bearing Geometry

Use *Thrust Bearings - 2-D Tilting Pad* to analyze the tilting pad thrust bearings with the following pivot types:

- Pin
- Spherical
- 1-D Rocker
- 2-D Rocker

See **Pivot model** on page 238 for more information.

Each pad in the bearing may be defined with individual arc length, inertia, groove depth, and pad material properties. All pads in a bearing must have the same type of pivot, but each pivot may have different values of pivot stiffness, friction, pivot location, and pivot radii.

### Thermal model

The heat generated in the bearing is dissipated via convection to the shaft, convection through each pad, and hot fluid exiting each pad. Hot fluid exiting a pad circumferentially enters the pad's radial feed groove and will either be completely drained from the bearing or carried over into the next (circumferentially) pad. The heat convected to the pad surface is conducted through the pad to a constant temperature at the pad surface opposite to the fluid film. Conduction through the pivot to the bearing housing and convection from the back of the pad to the surrounding fluid are not considered in the thermal model. This thermal model requires the user to enter the following information:

- Shaft Surface Temperature



- Pad Back Temperature
- Pad Axial Thickness
- Pad Thermal Conductivity

The pad conduction model allows the user to define the each pad using two separate materials acting in series. This modeling capability allows the user to account for the different conduction rates between the bearing surface material (i.e. babbitt) and the base pad material (i.e. steel).

### Thermal and centrifugal growth

Under operating conditions the tilting pads may experience a change in clearance due to thermal growth of the pad, pivot, and/or pivot support structure. Thermal growth of the pad is automatically calculated by the Program based on the predicted temperature distribution in the pad and the user supplied thermal expansion coefficient. However, the effects of pivot growth are installation dependent and the user must account for them by providing the *actual axial clearance under operating conditions* to *Thrust Bearings - 2-D Tilting Pad* (through the clearance input). Note that *Thrust Bearings - 2-D Tilting Pad* does not calculate thermal bow of the pad or pad deformations due to the pressure loading from the fluid film.

Under operating conditions the rotating bearing surface may experience a change in dimension due to thermal and centrifugal growth. As these effects are installation dependent, the user must account for them by providing the actual clearance diameter under operating conditions to *Thrust Bearings - 2-D Tilting Pad*.

### Groove model

Each pad is comprised of a bearing surface followed by a radial groove. The pad's circumferential sides are bordered by ambient pressure and all radial grooves are fed with fluid at the same supply temperature and pressure. Fluid is allowed to leak from the entire arc length of the pad's bearing surface radially to the user-defined ambient pressure. *Thrust Bearings - 2-D Tilting Pad* includes a mixing model for the radial grooves to account for hot fluid carry over from the previous pad (circumferentially) mixing with new fluid supplied at the user-defined supply temperature. This model assumes the grooves are sealed radially. You have the option of considering hot fluid carry over (**Iterate** on pad inlet temperature) or performing an analysis assuming each pad is supplied with fluid at a **Fixed** user-defined supply temperature.

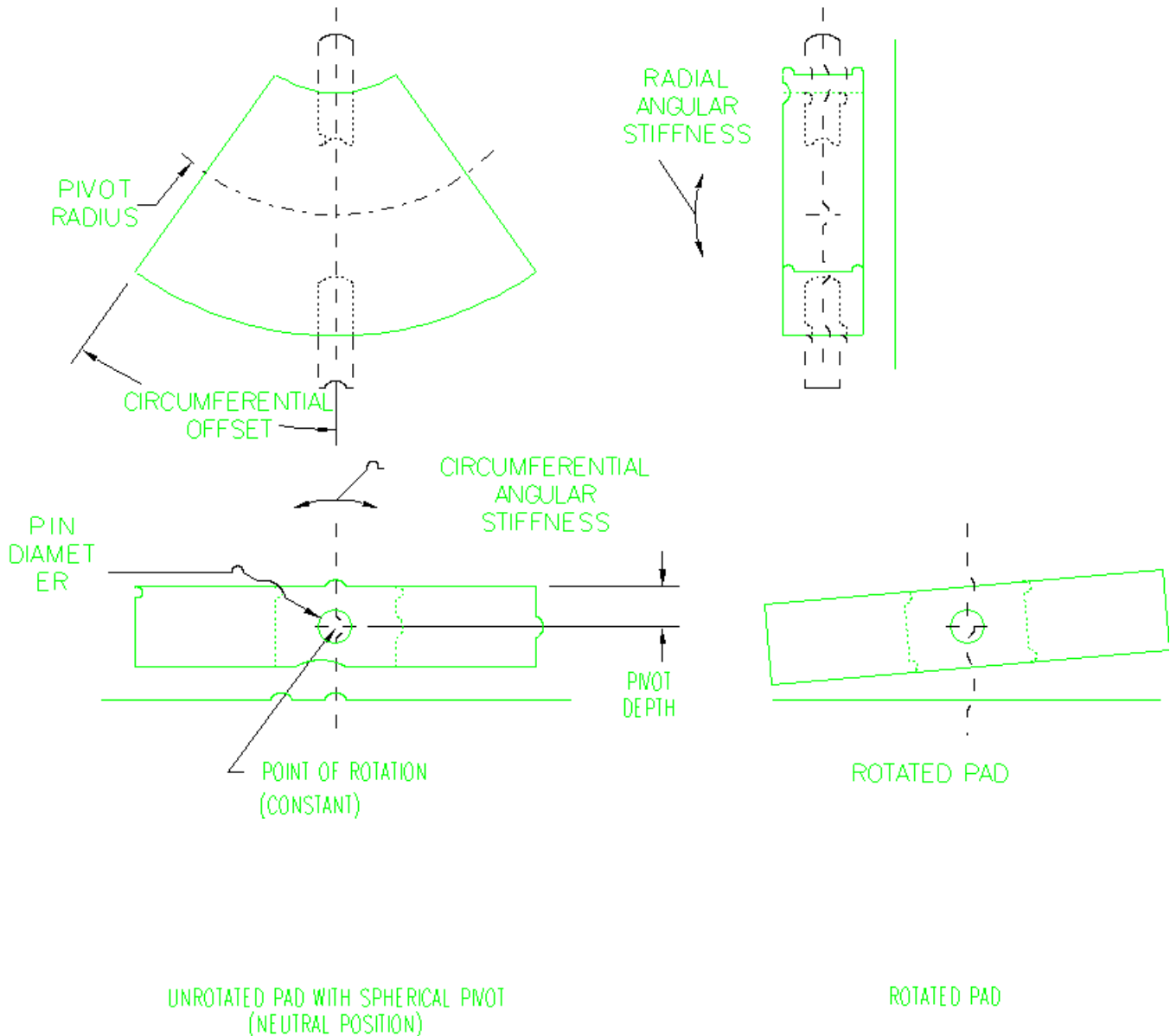
*Thrust Bearings - 2-D Tilting Pad* also includes a model to account for the ram effect of the fluid entering the pad with an initial circumferential velocity. The groove mixing model, if activated, accounts for the loss in circumferential velocity due to the introduction of fresh fluid radially.

### Pivot model

Currently three types of pad pivots can be modeled by *Thrust Bearings - 2-D Tilting Pad*. The available pivots types are discussed below:

## Pinned Pivot

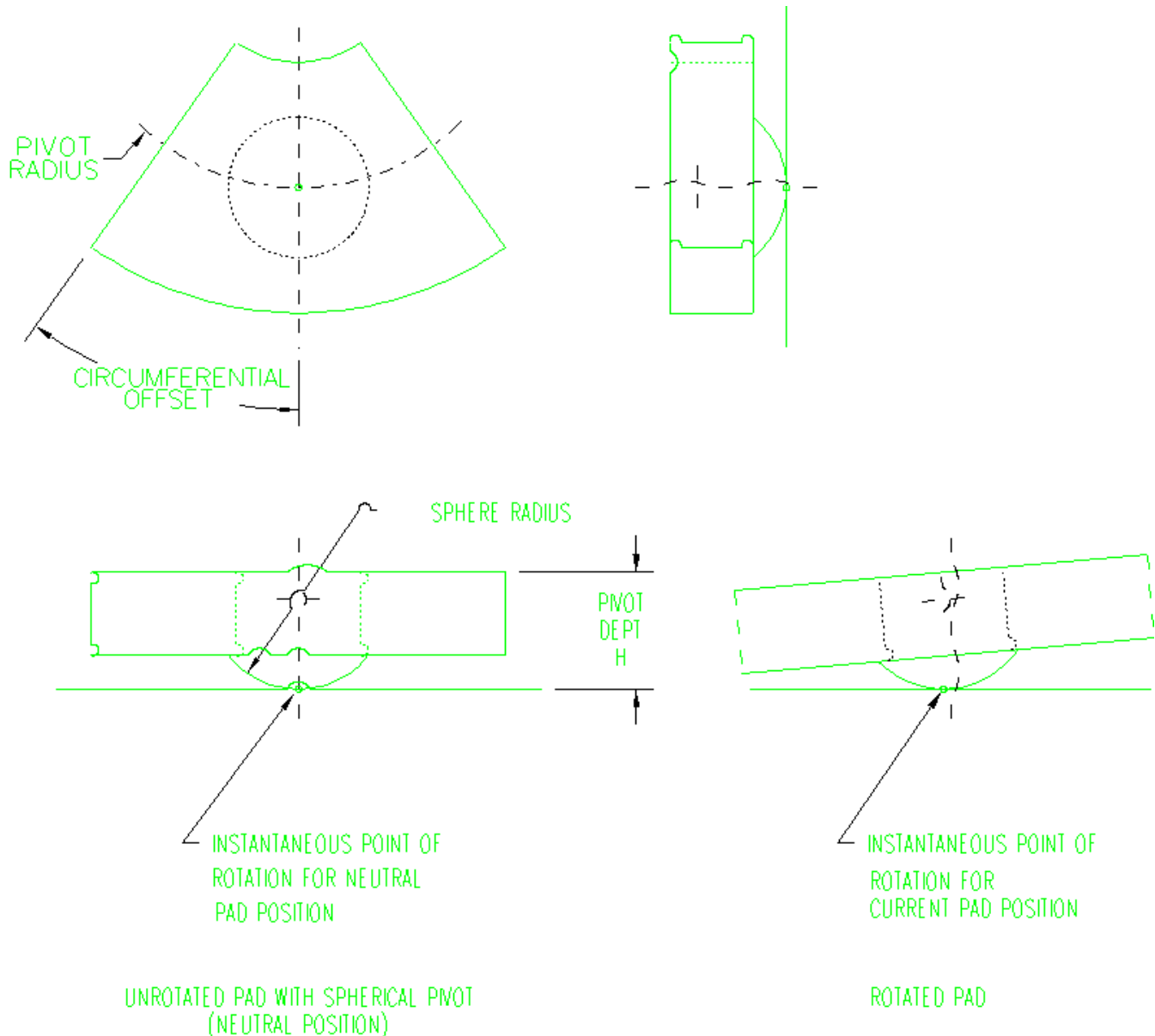
To prescribe a pin pivot the user must specify the diameter of the pin, the axial location of the pin, and the circumferential location of the pin (see the following figure). The user may also specify a coefficient of friction to represent the frictional force present on the outside diameter of the pin and additional angular stiffness values that resist rotation in the circumferential and radial directions. Note that the radial angular stiffness is relative to the specified pivot radius. This pivot model assumes the pad is free to rotate (it has overcome the static friction) about the center of the pin, and as the pad rotates, the Program automatically adjust the pad's clearance profile.



**Figure 35. Pinned pivot**

## Spherical Pivot

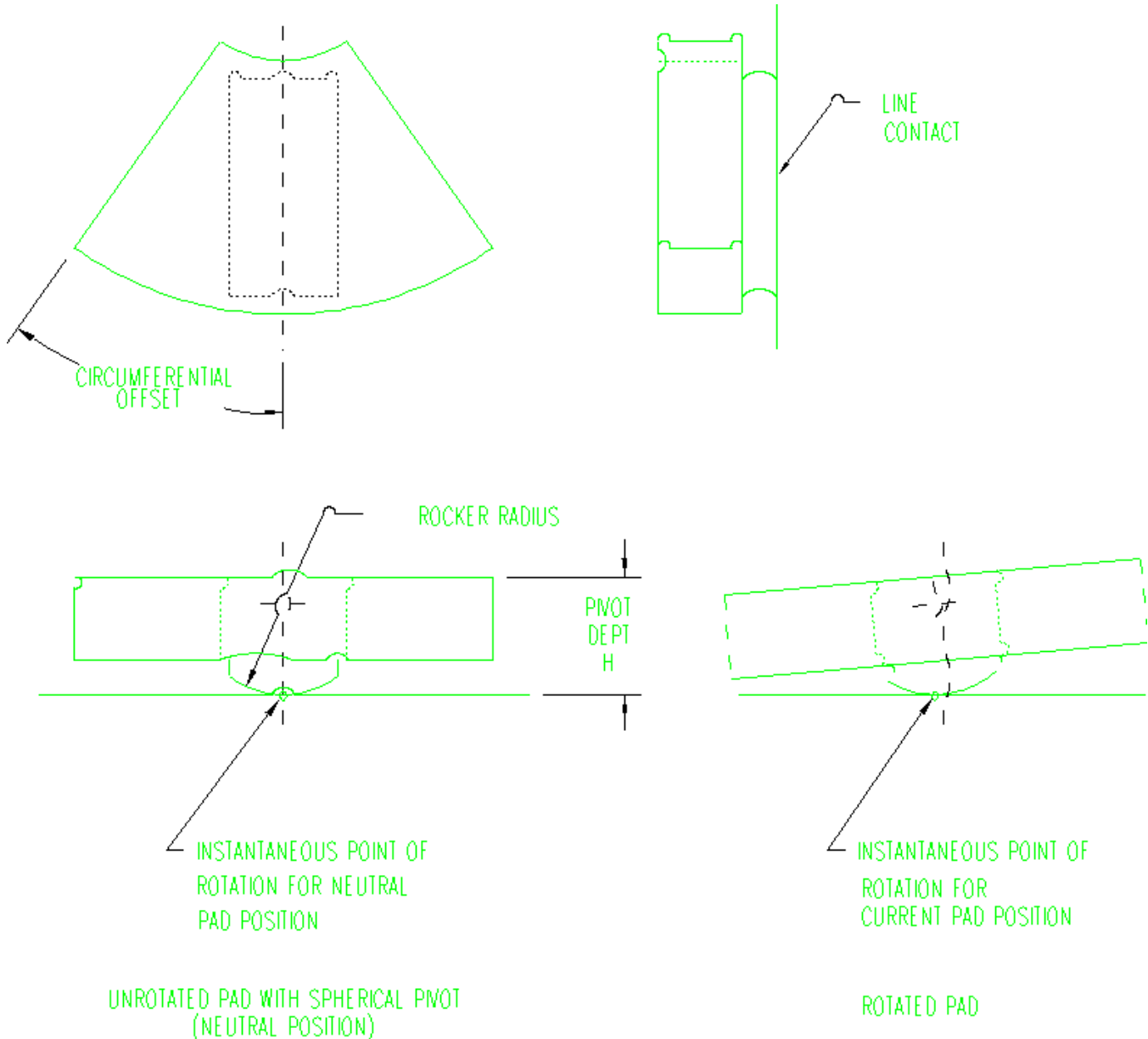
To define a spherical pivot the user defines the radius of the sphere (on the pad), the axial location of the pad's pivot point when the pad is in the neutral position, and the circumferential location of the pad's pivot point when the pad is in the neutral position (see the following figure). This pivot model assumes the pad tilts by rolling on the housing surface (no sliding), and as the pad tilts, the Program automatically adjusts the pad's clearance and pivot offset position. Currently there is no provision to account for friction in this pivot model. This pivot model assumes the housing surface is flat.



**Figure 36. Spherical pivot**

## 1-D Rocker Pivot

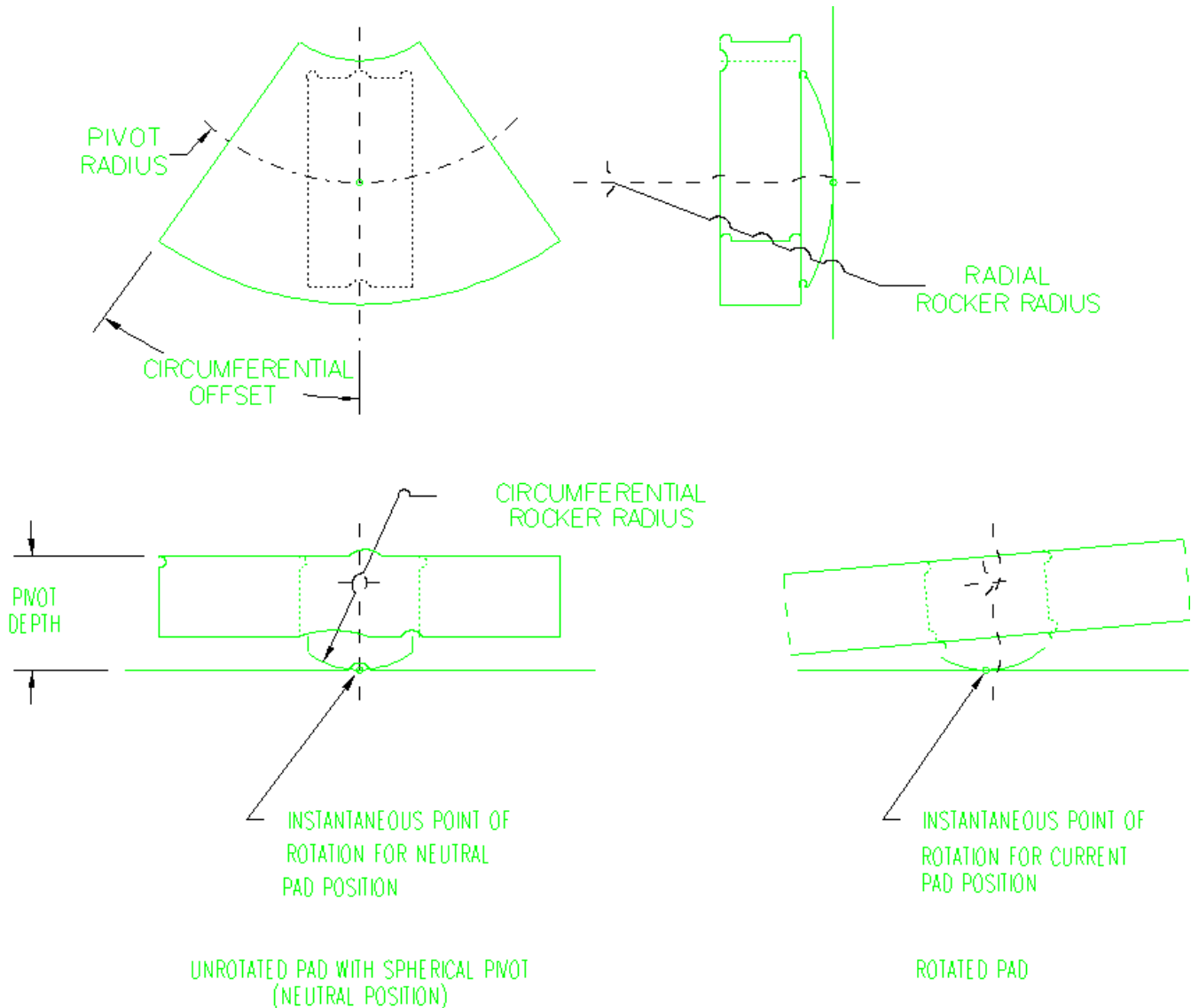
To define a 1-D rocker pivot the user defines the radius of the rocker (on the pad) in the circumferential direction, the axial location of the pad's pivot point when the pad is in neutral position, and the circumferential location of the pad's pivot point when the pad is in neutral position (see the following figure). This pivot model assumes the pad tilts by rolling on the housing surface (no sliding), and as the pad tilts, the Program automatically adjusts the pad's clearance and pivot offset position. Currently there is no provision to account for friction in this pivot model. This pivot model assumes the housing surface is flat.



**Figure 37. 1-D rocker pivot**

## 2-D Rocker Pivot

To define a 2D rocker pivot the user defines the radii for the rocker (on the pad) in both the circumferential and radial directions, the axial location of the pad's pivot point when the pad is in neutral position, and the circumferential location of the pad's pivot point when the pad is in neutral position (see the following figure). This pivot model assumes the pad tilts by rolling on the housing surface (no sliding), and as the pad tilts, the Program automatically adjusts the pad's clearance and pivot offset position. Currently there is no provision to account for friction in this pivot model. This pivot model assumes the housing surface is flat.



**Figure 38. 2-D rocker pivot**

To account for the effect of pad inertia, the user may either enter the density of the pad's materials or enter a value for the pad's rotational moments of inertia about its pivot point (in neutral position). If the user enters the material density, the Program automatically calculates the pad inertia by assuming the pad is a uniform cylindrical sector of the user specified arc length. For pads which can not accurately be

modeled as a uniform sector the user must calculate the inertia values for the pad and enter them into the Program.

To account for axial flexibility of the pad's pivot, the user is allowed to enter a pivot stiffness value. The value for pivot stiffness must be determined by the user outside of the Program.

### Applied Load/Prescribed Eccentricity

After the pads are defined, you have the option of prescribing an applied axial load or enforcing a prescribed rotor offset from the surface defined by the pads when they are all in the neutral position (the neutral plane). If you enter an applied load, the Program will iteratively determine the static equilibrium position of the shaft with respect to the neutral plane. This iterative solution requires entry of an initial guess for the shaft's offset from the neutral plane. Currently no automatic guess algorithm is available in the Program to generate the required initial guess.

In the event you elect to enforce a specific shaft offset from the neutral plane, the Program will solve the fluid film model and report the reaction load produced by the fluid film when the shaft occupies the prescribed offset position.

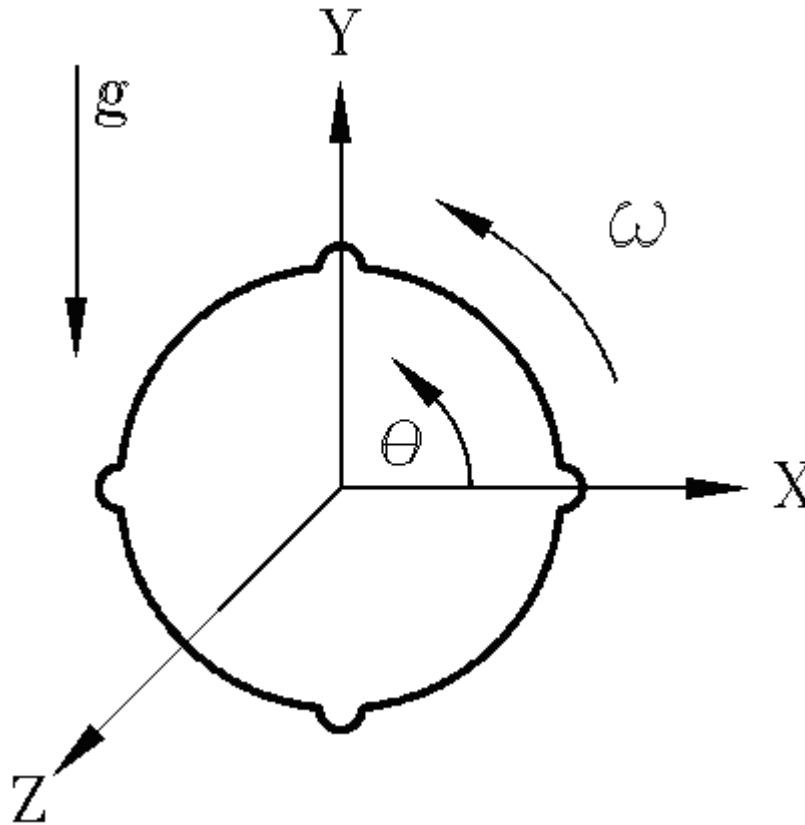
### Fluid Film gridding

*Thrust Bearings - 2-D Tilting Pad* contains an automatic grid generation algorithm for the fluid film solution. You have three options for the grid setting: **Course**, **Normal**, and **Fine**.

- Use the **Normal** grid for most cases.
- Use the **Course** grid to speed up the program during the process of qualitatively comparing numerous bearing designs.
- Use the **Fine** grid for cases where the bearing is heavily loaded as the thin films may require additional grid points to capture the high gradients present in the fluid film.

## Coordinate system

All angles utilized in *Thrust Bearings - 2-D Tilting Pad* (input and output) are referenced to a right handed coordinate system with the positive X-axis orientated at standard position. This mandates that shaft rotation is counter-clockwise and that gravity acts in the direction of the negative Y-axis (see the following figure).



*Figure 39. Coordinate system*

## Conventions for defining pads

The following rules must be followed when defining pads in *Thrust Bearings - 2-D Tilting Pad*:

1. The first full radial groove encountered while rotating counter-clockwise from the positive X-axis must be associated with the first pad defined (remember the load bearing surface of a pad precedes it circumferentially). If a bearing has a groove which traverses the positive X-axis said groove is not the first full groove encountered (because part of the groove resides in the fourth quadrant).
2. The Program's mixing model assumes that fluid exiting pad 1 enters pad 2, fluid exiting pad 2 enters pad 3, ....., fluid exiting the last pad enters pad 1. This convention defines the order in which pads must be defined.
3. No pad's bearing surface can have a end angle (start angle + arc length) greater than 360°. For example, if the first radial groove occurs at 45° and is associated with a 110° (Arc Length)

bearing surface, the pad's starting angular orientation (Arc Location) must be entered as  $-65^{\circ}$  (not  $295^{\circ}$ ).

## Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

## Analysis Parameter box

Use the **Analysis Parameter** drop-down menu to select the input value by which you want to vary the values of other inputs. Currently, **Rotor Speed** is the only available **Analysis Parameter**.

The **Rotor Speed** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting speed (**Start**), the increment of speed (**Delta**), and the final speed (**Stop**) for the calculation. The **Rotor Speed** fields must define a constant or increasing speed. Therefore, the **Stop** speed must be greater than or equal to the **Start** speed and the **Delta** speed must be positive. Field entries must be real numbers greater than zero.

## Output Format

The **Output Format** drop-down menu allows you to select whether to generate a **Short** or a **Long** output file.

- The **Short** output format includes a listing of the input variables and the calculated rotordynamic coefficients.
- The **Long** output format includes shaft offset, maximum film pressure, fluid temperature rise, maximum bearing temperature, minimum film thickness, required fluid flow rate, and power consumption, in addition to a listing of the input variables and the calculated rotordynamic coefficients.

## Grid

Use **Grid** to indicate whether a **Coarse**, **Normal**, or **Fine** grid is used in the solution.

See **Fluid Film gridding** on page 243 for more information.

## Groove Temp

Use the **Groove Temp** drop-down menu to specify whether hot fluid carryover is considered in the solution.

- If you select **Fixed (Supply)**, the Program sets the inlet temperature for each sector equal to the specified **Fluid Supply Temperature**.



- If you select **Iterate**, the Program iteratively determines the temperature of the fluid entering the each sector based on the flow velocities and temperatures of the fluid leaving the previous sector(circumferentially) and the fluid injected from the external supply.

## Materials page

The **Materials** page is common for all analysis cases that use material data and is used to define the material properties list that will be available for selection in this analysis case.

Once you insert a new material as a row in the table, there are two options for assigning the material properties:

- Define the properties manually, directly on the **Materials** page in the analysis case.
- Link to a material from the **Materials Library** by clicking the **Materials Library** button, therefore using the properties from the library.

See **Chapter 5: Working with materials** on page 79 for more information.

## Geometry page

Use the following fields on the **Geometry** page to specify the geometry of the tilting pad thrust bearing.

### Bearing Geometry box

See **Bearing Geometry** on page 237 for more information about bearing geometry.

#### Outer Diameter

The **Outside Diameter** represents the largest diameter of the pad's load bearing surface.

The following rule applies to the **Outside Diameter** field:

- Valid entries are real numbers greater than the **Inner Diameter**.

#### Inner Diameter

The **Inner Diameter** represents the smallest diameter of the pad's load bearing surface.

The following rule applies to the **Inner Diameter** field:

- Valid entries are real numbers greater than zero.

#### Rotor Roughness

The **Rotor Roughness** represents the absolute roughness of the rotor surface. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer. The program constrains the input value to be greater than or equal to zero but less than or equal to 256 micro inches, 6.5 microns, or

0.0065 mm. If the rotor is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

**NOTE:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the rotor is hydraulically smooth with a surface roughness of 8-32 microinches. This can change if the rotor has been damaged or corroded.

### **Pivot Type**

Use **Pivot Type** to select the type of pivot model to be used in the bearing.

See **Pivot model** on page 238 for more information about the available options.

### **Symmetry**

Use the **Symmetry** drop-down menu to specify whether all pads are the same or whether each pad differs:

- **Symmetric:** All the pads are the same and the pads are equally spaced. One set of input fields are available on the **Pad Definition** dialog box.
- **Asymmetric:** Each pad may differ and the pads need not be equally spaced. A grid displays on the **Pad Definition** dialog box allowing you to enter different values for each pad.

### **Pad Definition Define button**

Use the **Pad Definition Define** button to specify values for each of the pads. Indicate whether the values for each pad on the **Pad Definition** dialog box are the same or different using the **Symmetry** drop-down menu.

See for information about the fields available for entering information about the pads.

### ***Pad Definition dialog box***

Use the **Pad Definition** dialog box to specify values for each of the pads.

The type of fields available on the **Pad Definition** dialog box depend on the option selected in the **Symmetry** drop-down menu on the **Geometry** page.

- **Symmetric:** A set of fields displays allowing you to enter the number of pads and the geometry, materials, and pivot information to be applied to all pads.

**NOTE:** The values entered on the **Pad Definition** dialog box when **Symmetric** is selected on the **Geometry** page are applied to each pad. The exception is the **Arc Location** field. Enter the **Arc Location** for the first pad. The Program calculates the **Arc Location** for each of the remaining pads by dividing 360 degrees by the number entered into the **Number of Pads** edit box and incrementing the **Arc Location**.

- **Asymmetric:** A grid displays allowing you to add pads and specify different values for each pad.

---

## RAPPID User's Guide

---

The following fields are available on the **Pad Definition** dialog box, depending on the **Symmetry** type selected.

### **Number of Pads**

If you selected **Symmetric** in the **Symmetry** drop-down menu on the **Geometry** page, use the **Number of Pads** edit box to specify the number of pads in the bearing.

The following rule applies to the **Number of Pads** field:

- Valid entries are integers greater than zero.

**NOTE:** This field is not available if you selected **Asymmetric** in the **Symmetry** drop-down menu on the **Geometry** page.

### **Arc Length**

Use **Arc Length** to define the circumferential arc length of the pad's load bearing surface.

The following rules apply to the **Arc Length** field:

- Valid entries are real numbers greater than zero but less than 360°.
- The end angle for each pad must always be less than 360° (**Arc Location** + **Arc Length** < 360°).

**NOTE:** The **Arc Length** does not include the arc length of the groove associated with the pad. The Program automatically sets the groove arc length by subtracting the end angle of this pad from the start angle of the next pad.

### **Arc Location**

Use **Arc Location** to define the starting angular location for the pad's load bearing surface. The load bearing portion of the pad extends from the **Arc Location** to the **Arc Location** + **Arc Length**.

The following rules apply to the **Arc Location** field:

- Valid entries are real numbers between -360° and 360°, exclusive.
- The end angle for each pad must always be less than 360° (**Arc Location** + **Arc Length** < 360°). If the end angle is greater than 360°, simply define the **Arc Location** with a negative angle.

### **Groove Depth**

Use **Groove Depth** to define the axial depth of the radial groove which follows the load bearing surface of the pad.

### **Material 1/Material 2**

Use **Material 1** and **Material 2** to specify the materials used to fabricate the pad. **Material 1** is the material in contact with the fluid film.

## Axial Thickness

Use the **Axial Thickness** fields to specify the axial thickness of each material used to fabricate the pad.

## Pivot Axial Stiffness

Use **Pivot Axial Stiffness** to define the axial compliance of the pivot. This input allows you to model axial flexibility of the pad pivot. The Program automatically includes the pivot's flexibility in the determination of the rotordynamic coefficients. To simulate the effect of a rigid pivot simply enter a pivot stiffness of 0 and the Program will automatically consider the pivot to be rigid.

The following rule applies to the **Pivot Axial Stiffness** field:

- Valid entries are real numbers greater than or equal to zero.

## Pivot Axial Depth

Use **Pivot Axial Depth** to define the axial location of the pad's effective pivot point relative to the pad's bearing surface.

The following rules apply to the **Pivot Axial Depth** field:

- Valid entries are real numbers greater than or equal to zero.
- The value entered is a function of the type of pivot designated in the **Pivot Type** drop-down menu on the **Geometry** page.

**NOTE:** An entry of zero would indicate that the pad rotates about a point on the pad's bearing surface.

- **Pinned Pivot:** The pivot depth value entered must designate the distance from the pad's bearing surface to the center of the pin when the pad is at its neutral position (i.e. unrotated). This entry, in addition to the pad's **Pivot Circumferential Offset** defines the location of the pad's pivot point. Remember for a pinned pivot the pad is assumed to rotate about the center of the pin (sliding friction (dynamic) exists on the outside diameter of the pin).
- **1-D and 2-D Rocker Pivot:** The pivot depth value entered must designate the distance from the pad's bearing surface to the point of contact between the rocker surface (on the pad) and the mating surface in the bearing housing when the pad is at its neutral position (i.e. unrotated). Remember for a rocker back pivot the pad's rocker surface is assumed to roll on the bearing housing's mating surface (the pad can not slide).
- **Spherical Pivot:** The pivot depth value entered should designate the distance from the pad's bearing surface to the point of contact between the spherical surface machined into the back of the pad and the mating surface in the bearing housing when the pad is at its neutral position (i.e. unrotated). Remember for a spherical pivot the pad's spherical surface is assumed to roll on the bearings housing's mating surface (the pad can not slide).

---

# RAPPID User's Guide

---

## **Pivot Circumferential Offset**

Use **Pivot Circumferential Offset** to define the circumferential location of the pivot point when the pad is in neutral position (i.e. unrotated).

The following rule applies to the **Pivot Circumferential Offset** field:

- Valid entries are real numbers between zero and 1, inclusive.

The offset input is a non-dimensional value determined using the following equation:

$$\text{Offset} = \text{Offset Angle} / \text{Arc Length}$$

where the 'Offset Angle' is the angle from the leading edge of the pad to the location of the pivot point when the pad is in the neutral position.

## **Pivot Radius**

Use **Pivot Radius** field(s) to define the radial location of the pad's pivot point when the pad is in the neutral position.

The following rules apply to the **Pivot Radius** field(s):

- Valid entries are real numbers greater than **Inside Diameter** and less than the **Outside Diameter**, inclusive.
- The value entered is a function of the type of pivot designated in the **Pivot Type** drop-down menu on the **Geometry** page:
  - **Pinned Pivot:** The **Pivot Radius** value should designate the radial location of the attachment point for the rotational spring defined by the **Radial Angular Stiffness** input when the pad is at its neutral position (i.e. unrotated).
  - **1-D and 2-D Rocker Pivot:** The **Pivot Radius** value entered should designate the radial location of the point of contact between the rocker surface (on the pad) and the mating surface in the bearing housing when the pad is at its neutral position (i.e. unrotated).
  - **Spherical Pivot:** The **Pivot Radius** value entered should designate the radial location of the point of contact between the spherical surface on the back face of the pad and the mating surface in the bearing housing when the pad is at its neutral position (i.e. unrotated).

## **Coefficient of Friction**

Use **Coefficient of Friction** to define the friction coefficient used to determine the magnitude of the friction force between the pad and the pin that resists circumferential pad rotation.

The following rule applies to the **Coefficient of Friction** field:

## RAPPID User's Guide

---

- Valid entries are real numbers greater than or equal to zero.

**NOTE:** The **Coefficient of Friction** field is only available when **Pinned** is selected as the **Pivot Type**. Since the **Pinned Pivot** model assumes that the pad is always free to rotate, i.e., static friction has been overcome, the appropriate value for the coefficient of friction is the sliding coefficient of dynamic friction.

### Moment of Inertia Circumferential

Use **Moment of Inertia Circumferential** to define the moment of inertia of the pad that resists circumferential rotation. This moment of inertia is defined about the pivot point when the pad is at its neutral position (i.e. unrotated). The availability and value of this field is determined by the entries (either on the **Materials** page or in the **Materials Library**) in the **Density** fields for both materials:

If the densities entered for both materials are...	the Moment of Inertia Circumferential field is...
zero	editable.
non-zero	non-editable and 'Auto' displays in the field.

### Moment of Inertia Radial

Use **Moment of Inertia Radial** to define the moment of inertia of the pad that resists radial rotation. This moment of inertia is defined about the pivot point when the pad is at its neutral position (i.e. unrotated). The availability and value of this field is determined by the entries (either on the **Materials** page or in the **Materials Library**) in the **Density** fields for both materials:

If the densities entered for both materials are...	the Moment of Inertia Radial field is...
zero	editable.
non-zero	non-editable and 'Auto' displays in the field.

**NOTE:** If **1-D Rocker** is the selected **Pivot Type**, the **Moment of Inertia Radial** field is not editable (because the pad is constrained not to rotate in the radial direction) and 'Auto' displays in the field, even if the densities entered for both materials are zero.

## RAPPID User's Guide

### Pivot Surface fields

The two fields below **Moment of Inertia Radial** allow you to specify additional information to further define the pivot's characteristics. The fields available depend on the **Pivot Type** selected:

Field	Description	Pivot types
<b>Pin Diameter</b>	<p>Use <b>Pin Diameter</b> to specify the outside diameter of the pin.</p> <p>The following rule applies to the <b>Pin Diameter</b> field:</p> <ul style="list-style-type: none"><li>Valid entries are real numbers greater than or equal to zero.</li></ul> <p><b>NOTE:</b> The pin diameter is used only to determine the magnitude of the moment generated by the dynamic friction force action on the pin's surface.</p>	<b>Pinned</b>
<b>Circumferential Angular Stiffness</b>	<p>Use <b>Circumferential Angular Stiffness</b> to specify a rotational spring that resists pad rotation in the circumferential direction. The radial attachment point for the spring is defined by the <b>Pivot Radius</b> and the spring is orientated perpendicular to a radial line at the <b>Pivot Circumferential Offset</b> angle.</p> <p>The following rule applies to the <b>Circumferential Angular Stiffness</b> field:</p> <ul style="list-style-type: none"><li>Valid entries are real numbers greater than or equal to zero.</li></ul> <p><b>NOTE:</b> The if the <b>Circumferential Angular Stiffness</b> and <b>Coefficient of Friction</b> are both set to zero the pad is free to rotate circumferentially in response to the fluid film forces.</p>	<b>Pinned</b> <b>1-D Rocker</b> <b>2-D Rocker</b>
<b>Radial Angular Stiffness</b>	<p>Use <b>Radial Angular Stiffness</b> to specify a rotational spring that resists pad rotation in the radial direction. The radial attachment point for the spring is defined by the <b>Pivot Radius</b> and the spring is orientated parallel to a radial line at the <b>Pivot Circumferential Offset</b> angle.</p> <p>The following rule applies to the <b>Radial Angular</b></p>	<b>Pinned</b>

## RAPPID User's Guide

Field	Description	Pivot types
	<p><b>Stiffness field:</b></p> <ul style="list-style-type: none"><li>Valid entries are real numbers greater than or equal to zero.</li></ul> <p><b>NOTE:</b> The if the <b>Radial Angular Stiffness</b> is set to zero the pad is free to rotate radially in response to the fluid film forces.</p>	
<b>Radial Rocker Radius</b>	<p>Use <b>Radial Rocker Radius</b> to specify the radius of the surface machined onto the back of the pad that rolls on the bearing housing when the pad rotates in the radial direction.</p> <p>The following rule applies to the <b>Radial Rocker Radius</b> field:</p> <ul style="list-style-type: none"><li>Valid entries are real numbers greater than zero.</li></ul>	<b>2-D Rocker</b>
<b>Sphere Radius</b>	<p>Use <b>Sphere Radius</b> to specify the radius of the surface machined onto the back of the pad which rolls on the bearing housing when the pad rotates in the circumferential and/or radial direction.</p> <p>The following rule applies to the <b>Sphere Radius</b> field:</p> <ul style="list-style-type: none"><li>Valid entries are real numbers greater than zero.</li></ul>	<b>Spherical</b>

### Fluids page

The **Fluids** page is common to all analysis cases that use a single fluid (compound). Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**
- **Real Properties**



For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

## Operating Conditions page

Use the **Operating Conditions** page in a *Thrust Bearings - 2-D Tilting Pad* analysis case to set the pressure and temperature boundary conditions and the solution method to obtain position/load results.

**NOTE:** If there is a ... button next to an edit box, you may enter variable input for that input field. See **Entering variable data** on page 47 for more information.

## Rotor Speed

Currently, **Rotor Speed** is the analysis parameter by default because it is the only input available as the analysis parameter. Input range and increment for **Rotor Speed** is entered on the **Options** page.

## Pressure/Temperature box

### Fluid Supply Pressure

The **Fluid Supply Pressure** represents the pressure of the fluid entering all grooves from the external supply.

The following rule applies to the **Fluid Supply Pressure** field:

- Valid entries are real numbers greater than or equal to zero.

**NOTE:** The **Cavitation Pressure**, ambient pressure (**Left Boundary Pressure** and **Right Boundary Pressure**), and **Supply Pressure** are relative to each other. As such, they may be entered in either gauge or absolute values as long as all three are entered in consistent terms (do not specify some pressures in gauge and others in absolute).

### Fluid Supply Temperature

The **Fluid Supply Temperature** represents the temperature of the fluid entering all grooves in the bearing from the external supply. This temperature is considered constant and is used, in conjunction with the Program's mixing model, to determine the fluid temperature at the pad's inlet (first row of nodes circumferentially).

The following rule applies to the **Fluid Supply Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## **Outer Boundary Pressure**

The **Outer Boundary Pressure** represents the pressure boundary condition applied to the pad exit plane defined by the bearing's outside diameter.

The following rules apply to the **Outer Boundary Pressure** field:

- Valid entries are real numbers greater than or equal to zero.

## **Inner Boundary Pressure**

The **Inner Boundary Pressure** represents the pressure boundary condition applied to the pad exit plane defined by the bearing's inside diameter.

The following rules apply to the **Inner Boundary Pressure** field:

- Valid entries are real numbers greater than or equal to zero.

## **Shaft Temperature**

The **Shaft Temperature** specifies the fixed surface temperature of the shaft in the bearing.

The following rule applies to the **Shaft Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## **Pad Back Temperature**

Use **Pad Back Temperature** to specify the fixed temperature to be applied at the back of the pad. The fixed back of pad temperature is assumed to be the same for all pads. The entered temperature is utilized in the conduction model employed to simulate the axial heat transfer through the pad.

The following rule applies to the **Pad Back Temperature** field:

- Valid entries are real numbers greater than absolute zero (i.e., 0 Kelvin or 0° Rankine).

## Loading box

Use the drop-down menu at the top of the **Loading** box to specify the operation mode for the Program's analysis. There are two options:

- **Prescribed Load**

For the **Prescribed Load** option an initial guess for the shaft's equilibrium. Currently, the only **Guess** option available in the right-hand drop-down menu is **Manual Guess**. Enter an initial guess into the **Rotor Position** edit box manually.

If you select **Prescribed Load**, the **Convergence Criteria** and **Axial Load** fields are available.

- **Prescribed Rotor Position**

If you select **Prescribed Rotor Position**, the **Rotor Position** field is available.

Specify the following values as applicable.

### Rotor Position

Use **Rotor Position** to specify the rotor's position relative to the plane defined by the load bearing surfaces of the pad's in the neutral position (i.e. unrotated).

The following rule applies to **Rotor Position**:

- Valid entries are real numbers greater than zero.

**NOTE:** The **Rotor Position** field is used to define the initial guess for the equilibrium shaft position when running in **Prescribed Load** mode of operation.

### Convergence Criteria

Use **Convergence Criteria** to define the convergence criteria, as a percentage of the user specified load, used by the Program to determine when to cease iteration on reacted load.

**NOTE:** **Convergence Criteria** is not available for **Prescribed Eccentricity**.

The following rule applies to **Convergence Criteria**:

- Valid entries are real numbers between 0.5 and 5, inclusive. The valid range of entries corresponds to 0.5-5% of the user specified load for the case.

**NOTE:** The execution time of the Program is very sensitive to the value entered in this field. The lower the convergence criteria, the longer the Program takes to obtain the desired solution. Generally, a convergence criteria of 1-2% is sufficient. However, when running numerous cases for initial evaluation purposes, higher convergence criteria may be desirable to reduce computational time.

### **Axial Load**

Use **Axial Load** to define the magnitude of the axial load applied to the bearing.

**NOTE:** **Axial Load** is not available for **Prescribed Rotor Position**.

The following rule applies to **Axial Load**:

- Valid entries are real numbers greater than zero.

## Chapter 19: Flow Elements components

### Overview

This section describes the *Flow Elements* component modules available in RAPPID™ and the input fields specific to each. Currently, the following categories of *Flow Elements* components are supported in RAPPID™:

- 1-D Circular Branch
- 2-D Circular Branch

The *Circular Branch* components model any passage with an arbitrary path that can be properly analyzed using a hydraulic diameter and flow diameter approach.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

Use the **Analysis Parameter** drop-down menu to select the input value by which you want to vary the values of other inputs.

Currently, **Time** is the only available **Analysis Parameter**.

Use **Start**, **Delta**, and **Stop** under **Time** to represent the starting time (**Start**), the increment of time (**Delta**), and the ending time (**Stop**) for the calculation. The **Time** fields must define a constant or increasing time. Therefore, the **Stop** time must be greater than the **Start** time and the **Delta** time must be positive. Field entries must be non-negative floating point values.

### Output Format

Use the **Output Format** drop-down menu to select whether to generate a **Short** or a **Long** output file.

- The **Short** output format includes only the branch resistance.
- The **Long** output format includes branch resistance and the following:
  - Average mass flow rate
  - Average Reynolds number
  - Average axial velocity
  - Average friction factor
  - Head loss

- Average density
- Average viscosity

### Wall Thermal Model

Use the **Wall Thermal Model** drop-down menu to specify the heat transfer model using one of the following options:

- **Conduction** – specified temperature on the outer wall of the duct
- **Specified Temperature** – specified inner duct temperature along path
- **Specified Heat Rate** – specified heat transfer rate along path

### Stability Criteria

Use the **Stability Criteria** drop-down menu to choose whether to utilize an internally calculated (and variable) time step for the transient simulation based on **CFL** (Courant-Friedrichs-Lewy) stability criteria. To not use CFL, select **None**.

**NOTE:** When you select **CFL** as the **Stability Criteria**, the **Delta** edit box for the **Analysis Parameter** is not available.

### Grid

Use the **Grid** drop-down menu to indicate whether a **Coarse**, **Normal**, or **Fine** grid is used in the solution.

### Solution Type

The **Solution Type** drop-down menu is currently set to **Quasi-Steady** and cannot be changed.

### Materials page

The **Materials** page is common for all analysis cases that use material data and is used to define the material properties list that will be available for selection in this analysis case.

Once you insert a new material as a row in the table, there are two options for assigning the material properties:

- Define the properties manually, directly on the **Materials** page in the analysis case.
- Link to a material from the **Materials Library** by clicking the **Materials Library** button, therefore using the properties from the library.

See **Chapter 5: Working with materials** on page 79 for more information.

## Geometry page

Use the **Geometry** page to specify the design and arrangement of the duct, such as diameter, nodes, and roughness.

### Bearing Geometry box

#### Define Geometry Using

Use **Define Geometry Using** to specify whether to use the **Diameter** or **Area** when specifying the geometry points.

#### Wall Roughness

Use **Wall Roughness** to specify the absolute roughness for the inside surface of the duct.

#### Geometry points data grid

Use the geometry points data grid to specify nodes or points with **X**, **Y**, and **Z** coordinates, a **Flow Diameter** or **Flow Area**, and a **Hydraulic Diameter**, increasing along the path of the duct.

#### Material 1/Material 2

Use **Material 1** and **Material 2** to specify the materials used in the duct.

#### Radial Thickness

Use **Radial Thickness** to specify the radial thickness of each material on the duct.

## Fluids page

The **Fluids** page is common to all analysis cases that use a single fluid (compound). Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**
- **Real Properties**

For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

### Operating Conditions page

Use the **Operating Conditions** page to specify the boundary pressures and temperatures, gravity vector, and rotational speed of the duct as functions of time.

**NOTE:** **Velocity** is not currently available.

### Boundary Conditions box

Use the **Boundary Conditions** box to specify the pressure type, boundary pressures and temperatures, and the boundary loss factors.

**NOTE:** See **Entering variable data** on page 47 for more information about entering a constant value or a set of variable values when a field has a ... button next to it.

### Radial Profile

Use **Radial Profile** to specify boundary conditions values at differing radii at the **Entrance** and/or the **Exit** of the duct. Select the check box to activate this option.

Use the **Define** button to specify the radii values.

**NOTE:** **Radial Profile** is only available for *2-D Circular Branch*.

### Pressure Type

Use **Pressure Type** to specify whether the pressure at the **Entrance** to the duct is **Static** or **Total**. The **Pressure Type** for the pressure at the **Exit** of the duct can only be **Static**.

### Pressure

Use **Pressure** to specify the pressure at the **Entrance** to the duct and at the **Exit** of the duct.

### Temperature

Use **Temperature** to specify the temperature at the Exit of the duct.

**NOTE:** If you do not vary the **Temperature** by time, the constant **Temperature** entered is used for the entire range of time.

### Loss Factor

Use **Entrance Loss Factor** to specify pressure correction expressed as a fraction of the dynamic head at the entrance plane. Typical values are in range 0.1-0.5. Valid inputs are in the range 0-1



**NOTE:** The **Loss Factor** cannot be varied by time.

## **Outer Wall Temp**

Use **Outer Wall Temp** to specify the temperature outside of the duct.

**NOTE:** The **Outer Wall Temp** edit box is only available if **Conduction** is selected as the **Wall Thermal Model** on the **Options** page (see **Wall Thermal Model** on page 259).

## **Inner Wall Temp**

Use **Inner Wall Temp** to specify the inner duct temperature along the path.

**NOTE:** The **Inner Wall Temp** edit box is only available if **Specified Temperature** is selected as the **Wall Thermal Model** on the **Options** page (see **Wall Thermal Model** on page 259).

## **Wall Heat Rate**

Use **Wall Heat Rate** to specify the heat transfer rate along the path.

**NOTE:** The **Wall Heat Rate** edit box is only available for *1-D Circular Branch* analysis cases and only if **Specified Heat Rate** is selected as the **Wall Thermal Model** on the **Options** page (see **Wall Thermal Model** on page 259).

## **Loading box**

Use the **Loading** box to specify the **Gravity Vector** points and **Rotation** vector points.

## **Gravity Vector**

Use **Gravity Vector** to specify the direction and magnitude of the acceleration of the branch. The direction of the gravity vector is given in **X, Y, Z** coordinates in global space. The **Magnitude** is expressed as multiples of g's (the acceleration due to gravity).

Use the **Define** button to either enter constant values for the **X, Y, Z** coordinates and the **Magnitude** or to vary multiple sets of coordinates and magnitude by time. See **To enter gravity vector values** for more information.

## **Rotation**

Use **Rotation** to specify the orientation in global space of the rotational axis of the branch and the speed of rotation. Two sets of **X, Y, Z** coordinates are required to specify the axis of rotation.

Use the **Define** button to either enter constant values for the **X1, Y1, Z1, X2, Y2, and Z2** coordinates and the **Rotational Speed** or to vary multiple sets of coordinates and rotational speed by time. See **To enter rotation values** for more information.

## Output Options page

Use the **Output Options** page to specify what types of plots you wish to view.

### Sample Rate

Use the **Sample Rate** drop-down menu to select the sample rate to be applied to all plots you select on the **Output Options** page.

### Plot Types tab

Use the **Plot Types** tab to specify which types of plots you want available on the **% Analysis Time Plots** and **% Length Plots** tabs for selection.

- Global Mass Flowrate
- Average Reynolds Number
- Total Pressure
- Static Pressure
- Total Temperature (Fluid)
- Static Temperature (Fluid)
- Velocity
- Density
- Viscosity
- Enthalpy
- Mach Number

**NOTE:** Selecting the **Global Mass Flowrate** plot type generates one **Mass Flowrate** plot and does not affect the types of plots available on the **% Analysis Time Plots** and **% Length Plots** tabs.

### % Analysis Time Plots tab

Use the **% Analysis Time Plots** tab to select the types of plots you want to view versus time. All the plots on this tab are calculated as a percentage of the total path length of the duct.

### % Length Plots

Use the **% Length Plots** tab to select the types of plots you want to view versus length, or distance. All the plots on this tab are calculated as a percentage of the total analysis time.

## Chapter 20: Miscellaneous – Tabular Input

### Overview

**Tabular Input** is a module that is common to *Journal Bearings*, *Thrust Bearings*, and *Annular Seals*. It provides a method of either manually entering coefficients or importing them from a text file.

**NOTES:** The input fields for the **Tabular Input** module only enable entry of rotordynamic characteristics.

The **Run** and **View** buttons are not available for *Tabular Input* modules. Data is entered in *Tabular Input* modules to be included in *Network* and *Integrated Network* cases.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

Use the **Analysis Parameter** drop-down menu to select the independent variable (**Rotor Speed** or **Excitation Frequency**) used to define the relationship for the rotordynamic characteristics used in the analysis.

### Other box

#### Type

Use **Type** to specify **Lateral** or **Axial** input.

- In *Journal Bearing Tabular Input* cases, this field is set to **Lateral** and not editable.
- In *Thrust Bearing Tabular Input* cases, this field is set to **Axial** and not editable.

#### Moment Coefficients

Use the **Moment Coefficients** check box to choose to enter moment coefficients. **Moment Coefficients** are the rotordynamic characteristics that relate forces to angular displacements.

#### Characteristic

Currently, the **Characteristic** of the rotordynamic characteristics is limited to **Linearized Coefficients**.

### Dynamic Forces page

Use the **Dynamic Forces** page to enter the coefficients. The columns in the data grid on the **Dynamic Forces** page vary according to the options selected on the **Options** page.

# RAPPID User's Guide

Selecting an **Analysis Parameter** of **Rotor Speed** with **Lateral Type** rotordynamic characteristics will display the window shown in the following figure. The data grid on the **Tabular Input** page enables the creation of a table of lateral rotordynamic coefficients (i.e., stiffness, damping, and added mass) as a function of **Rotor Speed** (off the screen in the last column). A selection of **Excitation Frequency** would result in a similar display with the only difference being the last column reading **Excitation Frequency**.

Component Analysis: Journal Bearing - Tabular Input [Sample Journal Bearing Tabular Input Case]

Command Help

Sample Tabular Input case

Journal Bearing

Insert Copy Move Delete Import

Set #	Kxx (lbf/in)	Kxy (lbf/in)	Kyx (lbf/in)	Kyy (lbf/in)	Cxx (lbf-s/in)	Cxy (lbf-s/in)	Cyx (lbf-s/in)	Cyy (lbf-s/in)	Mxx (lbf-s <sup>2</sup> /in)	Mxy (lbf-s)
1	252721	-60829.7	-581789	801940	870.667	-1411.19	-1410.41	4694.18	0	0
2	275711	-35950.7	-581788	682339	713.398	-994.814	-995.455	3011.71	0	0
3	301055	-18257.2	-594812	622209	636.092	-789.32	-789.832	2243.76	0	0
4	327280	-2470.36	-614175	586925	591.548	-668.686	-668.734	1830.72	0	0
5	353893	13169.3	-637650	564947	564.033	-590.904	-590.558	1586.05	0	0
6	380684	29416	-664281	551328	546.804	-537.97	-537.476	1429.82	0	0
7	407547	46646.7	-693591	543594	536.49	-500.883	-500.573	1322.04	0	0
8	434425	65070.9	-725315	540373	531.217	-474.641	-474.896	1240.5	0	0
9	461281	84814.3	-759293	540838	529.861	-456.244	-457.473	1171.9	0	0
10	488093	105957	-795423	544467	531.708	-443.782	-446.414	1107.74	0	0
11	514847	128552	-833640	550909	536.281	-435.983	-440.457	1042.38	0	0
12	541532	152636	-873895	559922	543.249	-431.964	-438.731	971.904	0	0
13	568141	178237	-916155	571333	552.377	-431.098	-440.611	893.511	0	0
14	594668	205372	-960397	585015	563.492	-432.921	-445.641	805.155	0	0

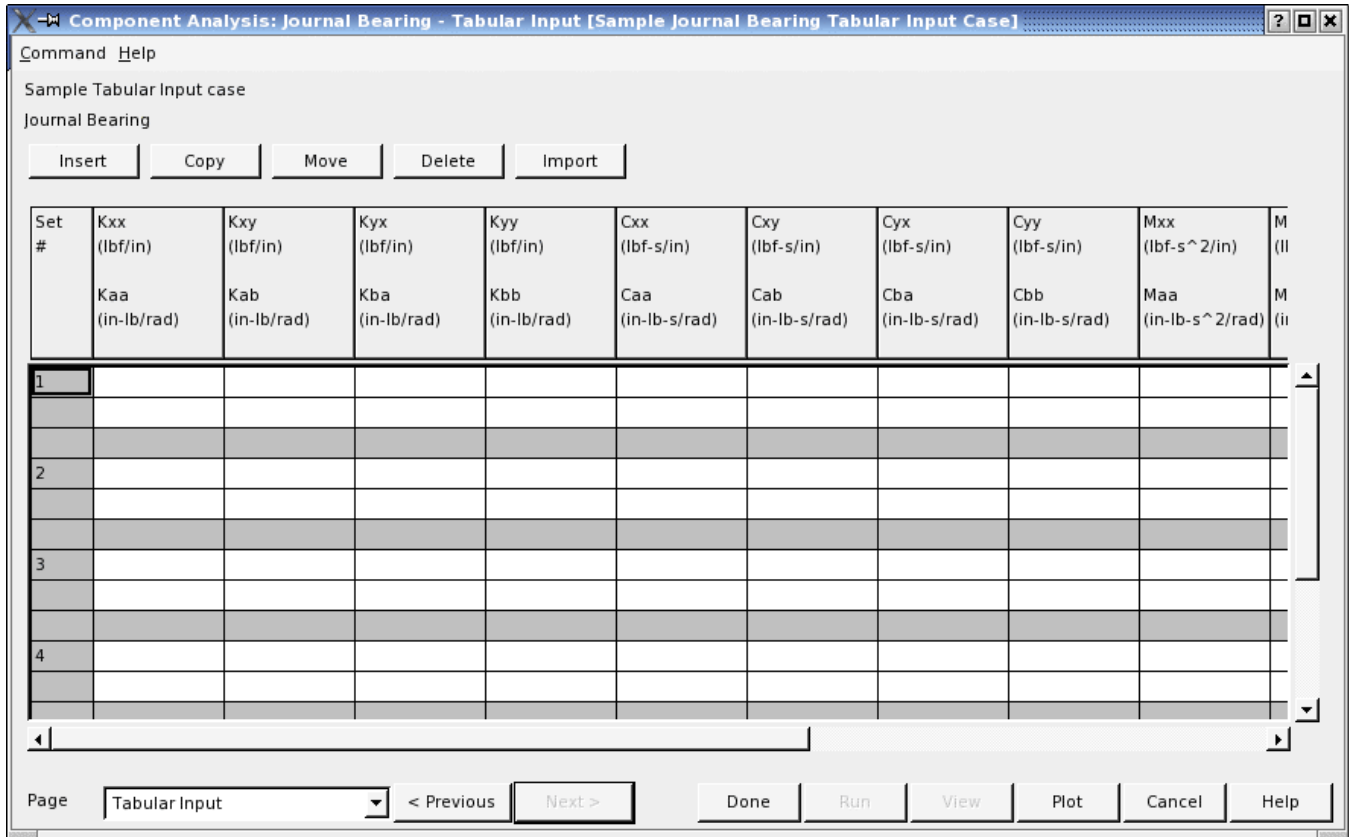
Page Tabular Input < Previous Next > Done Run View Plot Cancel Help

*Figure 40. Tabular Input page for Lateral type with Rotor Speed*

# RAPPID User's Guide

Selecting **Rotor Speed**, **Lateral Type**, and **Moment Coefficients** will result in the display shown in the following figure. The figure shows that the tabular input requires two rows per entry to accommodate the expanded data needed to define the **Moment Coefficients**.

**NOTE:** For *Journal Bearing Tabular Input*, **Lateral** is the only **Type** available.



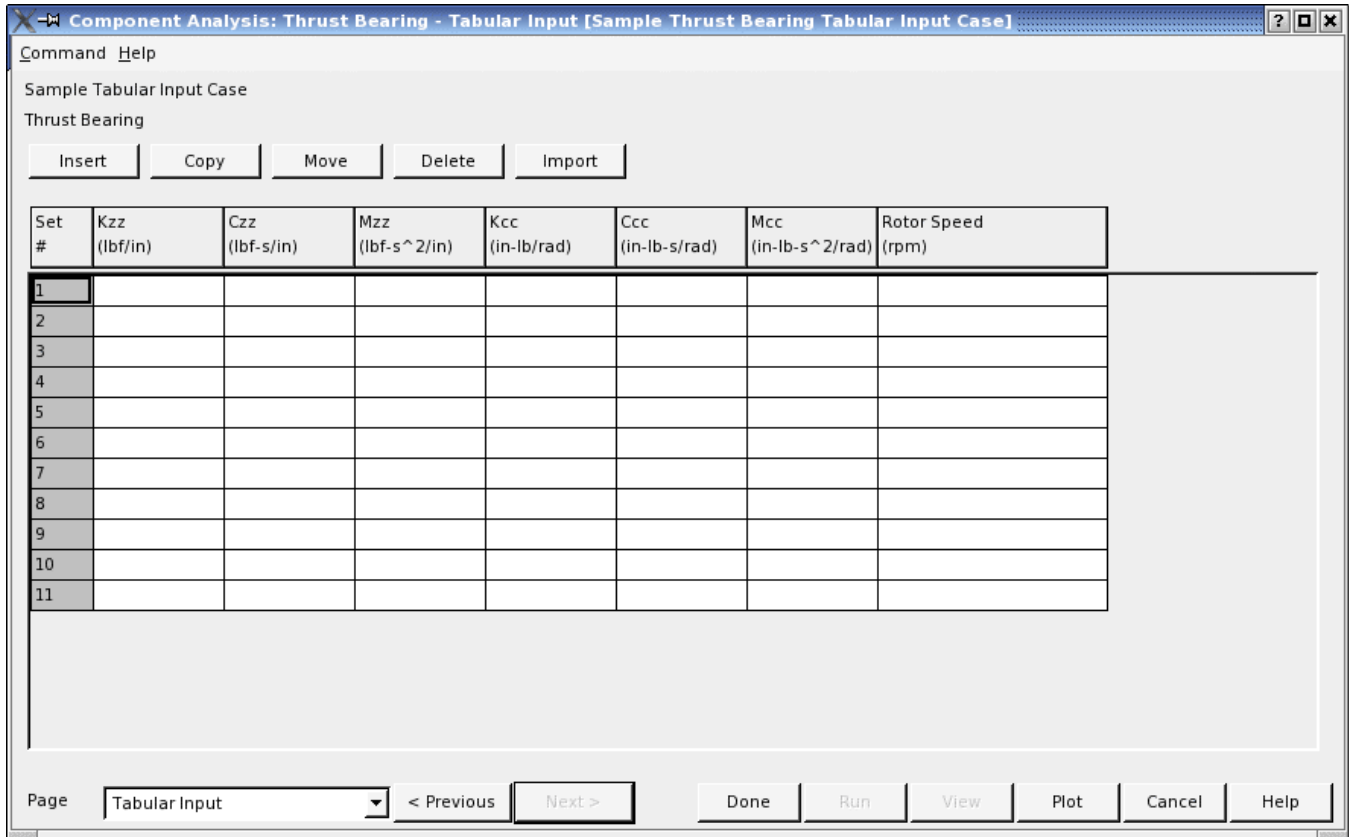
Set #	Kxx (lbf/in)	Kxy (lbf/in)	Kyx (lbf/in)	Kyy (lbf/in)	Cxx (lbf-s/in)	Cxy (lbf-s/in)	Cyx (lbf-s/in)	Cyy (lbf-s/in)	Mxx (lbf-s <sup>2</sup> /in)	M
	Kaa (in-lb/rad)	Kab (in-lb/rad)	Kba (in-lb/rad)	Kbb (in-lb/rad)	Caa (in-lb-s/rad)	Cab (in-lb-s/rad)	Cba (in-lb-s/rad)	Cbb (in-lb-s/rad)	Maa (in-lb-s <sup>2</sup> /rad)	M
1										
2										
3										
4										

**Figure 41.** *Tabular Input page for Lateral type with Rotor Speed and Moment Coefficients*

## RAPPID User's Guide

Selecting **Rotor Speed**, **Axial Type**, and **Moment Coefficients** results in the display shown in the following figure. The figure shows that due to the reduced data requirements for **Axial Type** rotordynamic coefficients, all of the entries will fit on a single row.

**NOTE:** For *Thrust Bearing Tabular Input*, **Axial** is the only **Type** available.



Component Analysis: Thrust Bearing - Tabular Input [Sample Thrust Bearing Tabular Input Case]

Command Help

Sample Tabular Input Case

Thrust Bearing

Insert Copy Move Delete Import

Set #	Kzz (lbf/in)	Czz (lbf-s/in)	Mzz (lbf-s <sup>2</sup> /in)	Kcc (in-lb/rad)	Ccc (in-lb-s/rad)	Mcc (in-lb-s <sup>2</sup> /rad)	Rotor Speed (rpm)
1							
2							
3							
4							
5							
6							
7							
8							
9							
10							
11							

Page Tabular Input < Previous Next > Done Run View Plot Cancel Help

*Figure 42. Tabular Input page for Axial type with Rotor Speed and Moment Coefficients*

## Chapter 21: Annular Seals components

### Overview

The *Annular Seal* modules assist in determining static and dynamic performance characteristics of compressible and incompressible annular seals. *Annular Seal* cases calculate the following parameters:

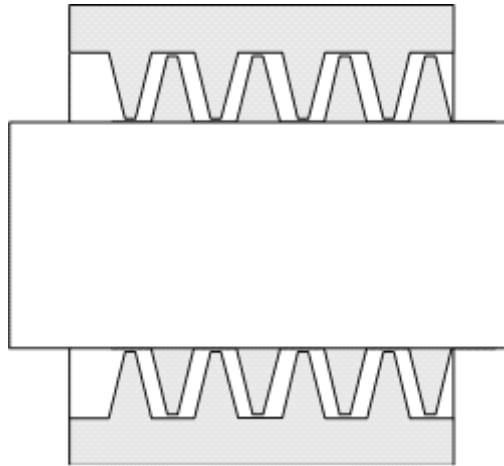
- Seal leakage and torque
- Seal rotordynamics coefficients

### Labyrinth Configurations

For *Annular Seal – Labyrinth* analysis cases the following configurations are available.

#### Interlocking

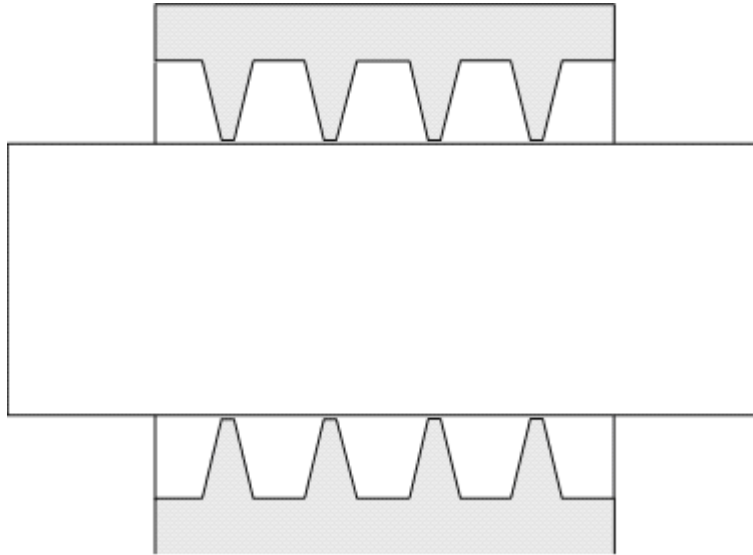
On **Interlocking** labyrinth seals, there are teeth on the rotor and stator. The stator is a half shell which is fitted about the shaft at assembly. As with staggered and stepped seals, great care must be taken to limit axial shaft travel when using this seal. This configuration has been used extensively in the past, however, high cost and design difficulties have caused interest to wane.



*Figure 43. Interlocking labyrinth seal*

## See-through and See-through TOR/Honeycomb Stator

**See-through** labyrinth seals are used extensively in both compressors and turbines. Typically teeth-on-stator seals are used for compressors and teeth-on-rotor seals are used for turbines.

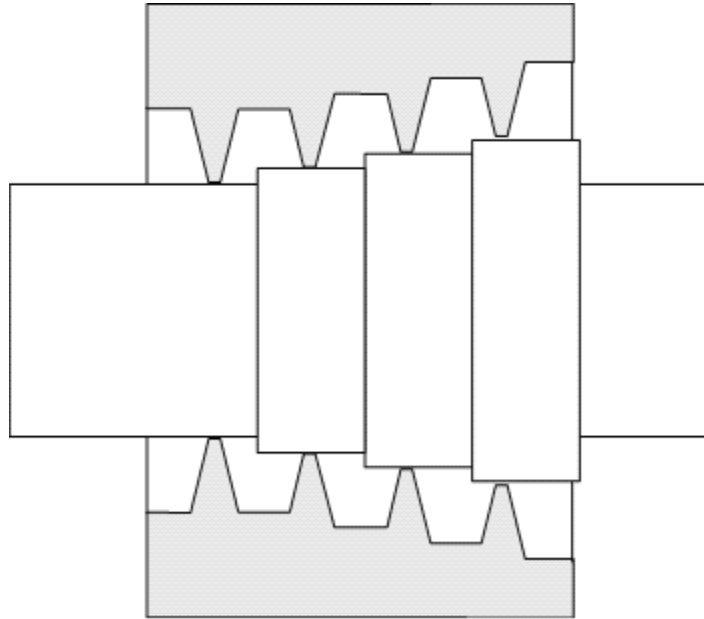


*Figure 44. See-through labyrinth seal*



## Stepped and Stepped TOR/Honeycomb Stator

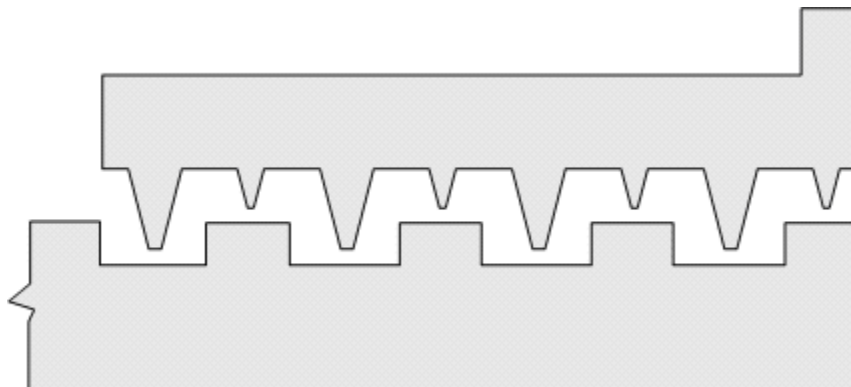
In **Stepped** seal configurations, the rotor can step up or down depending on the flowpath requirements. As with the staggered seal, great care must be taken to limit axial travel when using this seal configuration. This configuration is used extensively in buffer seals, eye seals, and for seals on the back side of impellers.



*Figure 45. Stepped labyrinth seal*

## Staggered and Staggered TOR/Honeycomb Stator

The **Staggered** labyrinth seal has a rotor which steps up and down and teeth on the stator which may or may not be the same length. In many cases, the teeth are made using J-strips. When using a staggered configuration care must be taken to limit axial shaft travel. This configuration is used extensively in steam turbines and compressors.



*Figure 46. Staggered labyrinth seal*

## Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

## Analysis Parameter box

Use the **Analysis Parameter** drop-down menu to select the input value by which you want to vary the values of other inputs. Currently, **Rotor Speed** is the only available **Analysis Parameter**.

Use the three fields for the **Rotor Speed** input (**Start**, **Delta**, and **Stop**) to specify the starting speed (**Start**), the increment of speed (**Delta**), and the final speed (**Stop**) for the calculation.

The **Rotor Speed** fields must define a constant or increasing speed. Therefore, the **Stop** speed must be greater than the **Start** speed and the **Delta** speed must be positive. Entries must be non-negative floating point values.

## Output Format

Use the **Output Format** drop-down menu to select whether to generate a **Short** or a **Long** output file.

- The **Short** output format includes a listing of the input variables and output consisting of basic flow parameters and the rotordynamic coefficients.
- The **Long** output format includes a listing of the input variables, and output consisting of an extensive listing of the mass and volumetric flowrate, load capacity, friction power loss, recess pressure, circumferential and axial Reynolds numbers, maximum values in the bearing, pertinent thermodynamic and transport values for every orifice and recess, and rotordynamic coefficients.

## Grid

For *Annular Seal Smooth* analysis cases only, use **Grid** to indicate whether a **Coarse**, **Normal**, or **Fine** grid is used in the solution.

## Geometry page

Use the **Geometry** page to specify the design and arrangement of the seal, such as length, width, and diameter.

## Settings box – Damping

The following fields are available in the **Settings** box on the **Geometry** page for *Annular Seal – Damping* analysis cases.

### Seal Type

Use **Seal Type** to specify the type of seal:

- **Honeycomb** – stator containing hexagonal holes in a honeycomb pattern

- **Hole Pattern** – stator containing holes in a specific pattern with a specified diameter and depth for the holes.

**NOTE:** The **Seal Type** drop-down menu is only available for Damping cases. If you select **Hole Pattern** from the **Seal Type** drop-down menu, the **Cell Width** edit box in the **Seal Geometry** section is not available.

### **Optimization Point - Rotor Speed**

Use **Rotor Speed** under **Optimization Point** to specify the rotor speed at which there will be an optimal cell width for a honeycomb seal.

**NOTE:** This setting does not affect **Hole Pattern** seals.

### **Settings box – Labyrinth**

The following fields are available in the **Settings** box on the **Geometry** page for *Annular Seal – Labyrinth* analysis cases.

### **Labyrinth Configuration**

Use **Labyrinth Configuration** to specify the configuration for the Labyrinth seal. See **Labyrinth Configurations** on page 268 for information about the options available.

### **Tooth Location**

Use **Tooth Location** to specify the location of the tooth that is the first or closest to the inlet of the seal.

**NOTE:** The **Tooth Location** drop-down menu is available for **See-through**, **Stepped**, and **Staggered** seal configurations.

**TOR** is Tooth-on-Rotor and **TOS** is Tooth-on-Stator.

This field is called **First Tooth Location** for **Interlocking** seal configurations. The options differ but have the same function:

- **Rotor**
- **Stator**

### **First Tooth**

Use **First Tooth** to indicate whether the tooth closest to the inlet of the seal is a **Large** or **Small** tooth.

**NOTE:** The **First Tooth** drop-down menu is only available for the **Staggered** seal configurations.

### **Tooth Input Method**

Use the **Tooth Input Method** to specify whether the values for the seal geometry are constant or whether they vary for each tooth.

---

## RAPPID User's Guide

---

Currently, the only option for the **Tooth Input Method** is **Constant Geometry**. This indicates that there is one value applicable to all the teeth on the rotor or stator. The values do not vary among the teeth.

### **Number of Teeth**

Use **Number of Teeth** to specify the number of teeth or number of sealing points in the seal.

**NOTE:** The **Number of Teeth** edit box is not available for **Stepped** seal configurations.

### **Number of Steps**

Use **Number of Steps** to specify the number of steps for the seal.

The maximum allowable number of user defined steps is 32. Enter a '0' to indicate the default situation where the number of steps is the number of teeth minus one and the step height is constant. The number of steps must be one less than the number of teeth or fewer. At least one tooth must be placed at the seal entrance, before the steps.

**NOTE:** The **Number of Steps** edit box is only available for **Stepped** and **Stepped TOR/Honeycomb Stator** seal configurations.

### **Seal Geometry box – Damping and Smooth**

The following fields are available in the **Seal Geometry** box on the **Geometry** page for *Annular Seal – Damping and/or Annular Seal – Smooth* analysis cases.

### **Diameter**

Use **Diameter** to specify the mean diameter of the rotor. It is assumed that the rotor is cylindrical. If the rotor is tapered, this must be accounted for by altering the radial clearance at the entrance and exit.

**Diameter** is available for both *Annular Seal – Damping* and *Annular Seal – Smooth* analysis cases.

### **Rotor Roughness**

The **Rotor Roughness** represents the absolute roughness of the rotor surface. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer. The program constrains the input value to be greater than or equal to zero but less than or equal to 256 micro inches, 6.5 microns, or 0.0065 mm. If the rotor is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

**NOTE:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the rotor is hydraulically smooth with a surface roughness of 8-32 microinches. This can change if the rotor has been damaged or corroded.

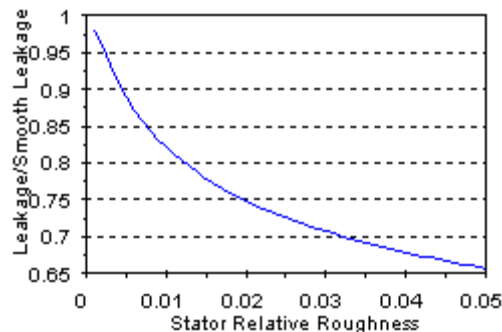
**Rotor Roughness** is available for both *Annular Seal – Damping* and *Annular Seal – Smooth* analysis cases.

## Stator Roughness

The **Stator Roughness** represents the absolute roughness of the stator surface. Generally, this is an rms value, which is defined on the drawing or measured using a profilometer.

**NOTE:** The measured roughness can only be used for surfaces which have not been intentionally roughened. Typically, the stator is hydraulically smooth with a surface roughness of 8-32 micro inches. This can change if the stator has been damaged or corroded.

The main factor, other than clearance, which controls cylindrical seal leakage is the stator roughness. The effect of stator roughness on leakage is illustrated the following figure.



*Figure 47. Stator Roughness*

The large values of stator roughness ( $>.02$ ) shown in the figure are only achievable by using patterns such as round holes and diamond knurls. While rotor roughness may also reduce leakage, it has a detrimental effect on rotordynamics.

The following rules apply to the **Stator Roughness** field:

- The program constrains the value to no greater than 256 micro inches, 6.5 microns, or 0.0065 mm.
- If the bearing is roughened excessively due to rubbing, a value must be used which is based on the effective flow based friction factor and not the measure rms value.

**Stator Roughness** is only available for *Annular Seal – Smooth* analysis cases.

## Axially Constant Clearance

Use **Axially Constant Clearance** to specify the clearance, or gap, formed between the surface of the rotor and the surface of the seal (stator). “Axially Constant” indicates that the clearance does not change axially.

The following rules apply to the **Axially Constant Clearance** field:

- Valid entries are positive floating point values.

---

## RAPPID User's Guide

---

The current version of the software restricts the **Axially Constant Clearance** entry to a constant value along the length of the bearing.

**Axially Constant Clearance** is available for both *Annular Seal – Damping* and *Annular Seal – Smooth* analysis cases.

### **Axially Varying Clearance**

The **Axially Varying Clearance** option is not currently available for either *Annular Seal – Damping* or *Annular Seal – Smooth* analysis cases.

### **Length**

Use **Length** to specify the total axial length of the seal.

**Length** is available for both *Annular Seal – Damping* and *Annular Seal – Smooth* analysis cases.

### **Cell Width**

Use **Cell Width** to specify the width of the cells in the honeycomb seal.

**NOTE:** The **Cell Width** edit box only displays when **Honeycomb** is chosen as the **Seal Type**.

**Cell Width** is only available for *Annular Seal – Damping* analysis cases.

### **Inlet Loss Coefficient**

Use the **Inlet Loss Coefficient** to specify the entrance loss coefficient for the seal inlet geometry. This is a dimensionless number, and is defined as:

$$\text{Entrance Loss} = \frac{2\Delta p}{\rho V^2} - 1$$

This coefficient is used to determine the pressure loss at a sudden contraction. Typical values for water seals are in the range 0.20-0.50. These values were determined from matching experimental flowrate data. If no similar experimental data is available, pipe flow contraction coefficients can be calculated using an equation from Blevins (Blevins, R.D., 1984, **Applied Fluid Dynamics** Handbook, Van Nostrand Reinhold Co., New York, p. 76) for pressure loss at pipe contractions:

$$\text{Entrance Loss} = 0.5871 + 0.3954\beta - 4.5385\beta^2 + 14.24 \\ - 19.22\beta^4 + 8.54\beta^5$$

where  $\beta$  is the ratio of seal clearance area to the projected area at the entrance to the seal.

**Inlet Loss Coefficient** is only available for *Annular Seal – Smooth* analysis cases.

## Seal Geometry box – Labyrinth

The seal configuration selected in the **Labyrinth Configuration** drop-down menu determines which of the fields in the **Seal Geometry** box are available.

For the **Staggered** and **Staggered TOR/Honeycomb Stator** configurations, use the **Small** and **Large** columns of input fields to enter separate values for the small and large teeth, respectively.

### Inlet Rotor Diameter

Use **Inlet Rotor Diameter** to specify the mean diameter of the rotor at the first tooth of the seal. It is assumed that the rotor is cylindrical.

The diameter is at the point of sealing, i.e. the tooth tip diameter for teeth on rotor labyrinths. **DO NOT** enter the diameter at the base of a tooth on the rotor.

**Inlet Rotor Diameter** is available for the following configurations:

- Interlocking
- See-through
- Stepped
- Stepped TOR/Honeycomb Stator
- See-through TOR/Honeycomb Stator

### Radial Clearance

Use **Radial Clearance** to specify the clearance, or gap, formed between the surface of the rotor and the surface of the seal (stator).

The **Radial Clearance** is constant for each tooth for all labyrinth seals except the staggered teeth-on-stator seal. This value should be obtained for the operating condition (full pressure, speed and temperature). Values on the assembly drawing are usually not the same as the operating values.

**Radial Clearance** is available for all configurations.

### Pitch

Use **Pitch** to specify the axial distance between corresponding points on the first two teeth of the labyrinth seal, i.e. the middle of each tooth.

For the interlocking labyrinth seal, one tooth is on the rotor and the next is on the stator or vice versa. The tooth pitch is considered to be constant for the entire seal.

**Pitch** is available for all configurations.

## **Height**

Use **Height** to specify the radial distance from the base to the top of the tooth.

**Height** is available for all configurations.

## **Tip Width**

Use **Tip Width** to specify the flat portion of the tooth at the radial clearance of the seal.

**Tip Width** is available for all configurations.

## **Teeth / Step**

Use **Teeth / Step** to specify the number of teeth per step for each step.

**NOTE:** The number of **Teeth / Step** must be less than or equal to the total number of teeth minus 1 because the entrance must include at least one tooth.

**Teeth / Step** is only available for **Stepped** and **Stepped TOR/Honeycomb Stator** configurations.

## **Step Height**

Use **Step Height** to specify the radial step height for each step.

The first step height is measured from the entrance radius (entered as a diameter) and each subsequent step is measured from the previous step. The entrance must include at least one tooth.

**NOTE:** All step heights must either be positive or negative. An error occurs if a negative or zero radius occurs from negative step heights.

**Step Height** is only available for **Stepped** and **Stepped TOR/Honeycomb Stator** configurations.

## **Diameter**

Use **Diameter** to specify the mean diameter of the rotor. It is assumed that the rotor is cylindrical. If the rotor is tapered, this must be accounted for by altering the radial clearance at the entrance and exit.

**Diameter** is only available for **Staggered** and **Staggered TOR/Honeycomb Stator** configurations.

## **Fluids page**

The **Fluids** page is common to all analysis cases that use a single fluid (compound). Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**



- **Real Properties**

For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

## **Operating Conditions page**

The following fields are available on the **Operating Conditions** page in all *Annular Seals* analysis cases.

### **Fluid Supply Pressure**

Use **Fluid Supply Pressure** to specify the static pressure immediately upstream of the seal entrance. The supply pressure should be determined as close to the seal entrance as possible.

### **Fluid Supply Temperature**

Use **Fluid Supply Pressure** to specify the static temperature immediately upstream of the seal entrance. The supply temperature should be determined as close to the seal entrance as possible.

### **Fluid Discharge Pressure**

Use **Fluid Discharge Pressure** to specify the static pressure immediately downstream of the seal exit. The exit pressure should be determined as close to the seal exit as possible.

### **Rotor Speed**

**Rotor Speed** is currently always set to be the **Analysis Parameter**, so you cannot enter a value in this field.

### **Fluid Inlet Swirl Ratio**

Use **Fluid Inlet Swirl Ratio** to specify the ratio of the circumferential velocity of the fluid at the inlet to the seal to the rotor surface velocity.

This is the most important input in the entire module. The whirl frequency ratio of the seal is directly related to this input.

The value of this term is related to its location in the turbomachine. If no good estimate is available use a value between 0.50-0.80. The lower limit of 0.50 is a good estimate for a seal at the same diameter as

## **RAPPID User's Guide**

---

the primary components (such as a turbine tip or shroud seal). The upper limit of 0.80 is a good estimate for a seal which is at a smaller diameter than the turbine or compressor wheels (such as a turbine interstage seal). The larger the number used, the more conservative the calculation.

## Chapter 22: Dynamics Networks

### Overview

This section describes the *Dynamics* networks modules available in RAPPID™ and the input fields specific to each. Currently, the following types of *Dynamics* networks are supported in RAPPID™:

- Eigenvalue
- Steady State Forced Response
- Transient Forced Response

### Eigenvalue

The *Eigenvalue* network performs complex eigenanalysis for stability studies. For a given rotordynamics model, *Eigenvalue* can be used to perform the following types of analysis:

- Damped Eigenvalue Analysis
- Damped natural frequency map
- Stability map
- Damped mode shapes

### Steady State Forced Response

The *Steady State Forced Response* network performs linear forced response analysis. For a given rotordynamics model, *Steady State Forced Response* can be used to perform the following types of analysis:

- Synchronous Forced Response to Harmonic Forcing Functions
  - Rotor and housing response as a function of rotor speed
  - Rotor response shapes
  - Dynamic loads
- Asynchronous Forced Response to Harmonic Forcing Functions
  - Rotor and housing response as a function of excitation frequency
  - Rotor response shapes
  - Dynamic loads

## Transient Forced Response

*Transient Forced Response* allows the user to couple the rotor and casing with nonlinear elements. Available nonlinear elements include interconnections with discontinuities in rotordynamic coefficients (e.g., bearing with deadband), interconnections with nonlinear load - deflection curves, and interconnections with frequency dependent rotordynamic characteristics (e.g., magnetic bearing). Other common rotating machinery effects which can be simulated include constant loads and either fixed or rotating eccentricities. The available forcing functions include rotor unbalance, rotor/casing sideloads, and rotor/casing sinusoidal forces applied by an external source, rotor/casing impulses and rotor/casing periodic loads. *Transient Forced Response* can be run at a constant rotor speed or with a time varying speed profile.

## Options page – Eigenvalue

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

Use **Analysis Parameter** to calculate rotor and housing response as a function of **Reference Rotor Speed**, which is the only available option for *Eigenvalue* analysis cases.

The **Reference Rotor Speed** input consists of two sets of three fields each: **Start**, **Delta**, and **Stop** for **Eigenvalue Range** and **Mode Shape Range**.

Use the **Eigenvalue Range** and **Mode Shape Range** inputs to set the eigenvalue analysis points. These analysis points are used for all rotors (after scaling by speed ratio) to determine interconnection coefficients. The user should insure that adequate input data is available for all interconnections to cover the range of speeds defined on this input line.

### Other box

The **Other** box below the **Analysis Parameter** box is currently not available.

## Options page – Steady State Forced Response

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

Use **Analysis Parameter** to specify whether to calculate rotor and housing response as a function of **Reference Rotor Speed** or **Excitation Frequency**.

The **Reference Rotor Speed/Excitation Frequency** input consists of two sets of three fields each: **Start**, **Delta**, and **Stop** for **Response Range** and **Response Shape Range**.

# RAPPID User's Guide

Use the **Response Range** and **Response Shape Range** inputs to set the response analysis points. These analysis points are used for all rotors to determine interconnection coefficients. The user should insure that adequate input data is available for all interconnections to cover the range of frequencies defined on this input line. If the interconnection coefficients are a function of relative rotor speed, this speed is a function of the analysis frequencies, the entity speed ratios, and the reference rotor speed.

## Other box

Use the **Other** box to specify **Constant Rotor Speed** and **Constant Order**. When you select **Excitation Frequency** as the **Analysis Parameter**, the options in the **Other** box become available.

## Options page – Transient Forced Response

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

## Analysis Parameter box

Use **Analysis Parameter** to calculate rotor and housing response as a function of **Time**, which is the only available option for *Transient Forced Response* analysis cases.

The **Time** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting time (**Start**), the increment of time (**Delta**), and the ending time (**Stop**) for the calculation. The **Time** fields must define a constant or increasing time. Therefore, the **Stop** time must be greater than the **Start** time and the **Delta** time must be positive. Field entries must be non-negative floating point values.

## Rotors and Housings pages

Use the **Rotors** and **Housings** pages to add rotors and housings to your *Dynamics* network case. Any *Structural Dynamics - Rotor* or *Housing* cases that have been added to the current project are available for inclusion in the current *Dynamics* network case. The following picture is an example of a **Rotors** page in an *Eigenvalue* case.

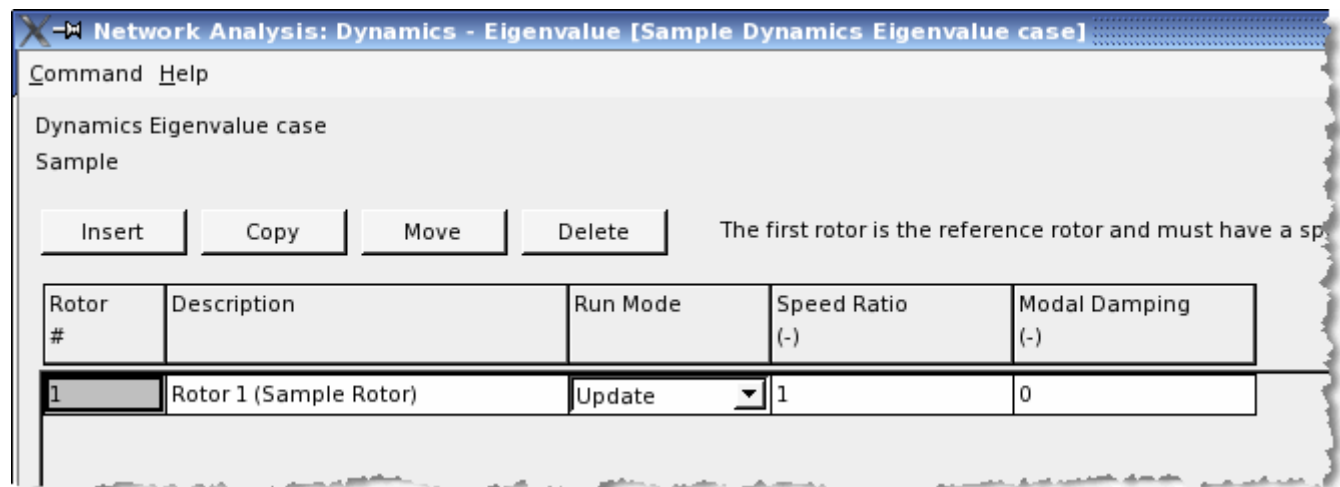


Figure 48. Rotors page for an Eigenvalue analysis case

# RAPPID User's Guide

The following are guidelines for rotors and housings added to a *Dynamics* network case:

- Any number of rotating assemblies (rotors) and stationary assemblies (housings) may be defined and connected to form the rotordynamic model.
- The structural characteristics of each rotor and housing entity are defined using 2-node conical beam elements. The inertial characteristics of each rotor and housing entity are determined by summing a) the element mass and inertia properties of the conical beam cross-section, and b) user-defined non-structural mass and inertia that may exist at various nodes.

Individual rotor and housing entities may be defined to have any relative orientation in 3-D space. As such, each beam element node contains 6 degrees of freedom, translation and rotation about the X-, Y- and Z- axes.

Each rotor is assigned a rotational speed and a rotational direction, relative to a user-defined reference rotor. Co-axial, co-rotating, or counter-rotating rotors may thus be analyzed.

## Description

Use the **Description** column to select a *Structural Dynamics Rotor* or *Housing* case and to add any additional description to the existing case description. The description for the selected rotor or housing (and any additional description) displays in the **Description** column of the data grid.

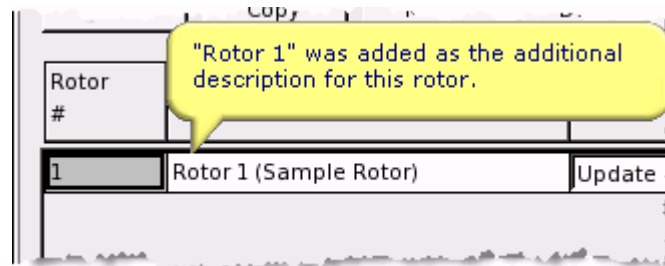


Figure 49. Additional description for a rotor on the Rotors page

## Run Mode

Use **Run Mode** in network and integrated network analysis cases to specify how the data from the component analysis case is associated with the current analysis case.

The following options are available:

**Update** Use the output file from the last run of the component analysis case.

**NOTE:** If the component analysis case has not been run before (if the output file for the component does not exist), this option works exactly like the **Re-Run** option.

**Store** Store a complete copy of the component analysis case at the network level with the

current network analysis case.

**Re-Run** Re-run the component analysis case at the component level and uses the updated results.

To update a case stored in the current network analysis case, follow the steps below.

1. Open the network analysis case in which you want to specify the method for updating an analysis case used in that network case.

**NOTE:** The **Run Mode** setting is available on screens or dialog boxes on which you select component analysis cases to connect to the current network analysis case. See the chapters for network analysis case types in **Part II – Analysis case types** on page 127 for more information.

2. Select either **Update** or **Re-Run** from the drop-down menu in the **Run Mode** column for the component analysis case you wish to update.
3. Refer to the following conditions to determine which step or set of steps to follow:
  - a. If you select **Update** as the **Run Mode**, and the component analysis case has not been run before (no output file exists), proceed to step 4.

- i. Click **Done** on the case manager window for the current network analysis case to save and close the case.

This also deletes the stored component analysis case.

**NOTE:** If the current network refers to a given component as a stored component, the component analysis case at the network level will not be deleted. Make sure you change the **Run Mode** to **Update** or **Re-Run** for all references to the component within the current network analysis case.

- ii. Open the component analysis case you want to update, make any desired changes, and run the case.
- iii. Click **Done** on the case manager window to save and close the component analysis case.
- iv. Open the network analysis case in which you have included the component analysis case.
- v. Select **Store** from the **Run Mode** column for the component analysis case you updated.
- vi. Run the current network analysis case.

The updated case is then stored with the network analysis case.

- b. If you select **Update** and the component analysis case has been run (viewable output exists for the component) or if you select **Re-Run**, do the following:
  - i. Run the current network analysis case.

See **Running analysis cases** on page 49 for more information.

### Speed Ratio/Rotor Speed

Use **Speed Ratio/Rotor Speed** to set the speeds of multiple rotors relative to a reference rotor speed.

**NOTE:** This column is called **Speed Ratio/Rotor Speed** for *Eigenvalue* analysis cases and **Rotor Speed** for *Steady State Forced Response* and *Transient Forced Response* analysis cases.

Apply the following guidelines when adding rotors to a *Dynamics* analysis case:

- The first rotor you insert in the data grid is the reference rotor.
- The **Speed Ratio/Rotor Speed** for the reference rotor (the first rotor) must be **1.0**.
- The **Speed Ratio/Rotor Speed** for all other rotors is entered relative to the **Speed Ratio/Rotor Speed** for the reference rotor.

**NOTES:** The **Speed Ratio/Rotor Speed** can only be changed on the **Rotors** page, not the **Housings** page.

### Modal Damping

Use **Modal Damping** to specify the critical damping value of the entity, which is expressed as a dimensionless decimal fraction.

**Modal Damping** can be included for either **Rotors** or **Housings**. You may enter a single value for the entity, or you may vary the value for each mode in the entity. The number of modes for which you can vary the **Modal Damping Factor** is obtained from the number of modes to retain entered in selected entity case.

Click the cell in the **Modal Damping** column to access the **Modal Damping** dialog box. See **Constant or variable analysis input** in **Part IV – The Analysis Case Managers of Tutorial 1 – The basics** for more information on entering either constant or variable data for **Modal Damping**.

### Interconnection page

Use **Interconnections** page to add interconnections (bearings and seals) to the *Transient Forced Response* case. Any bearing or seals cases that have been added to the current project are available for inclusion in the current *Transient Forced Response* case. The following picture is an example of an **Interconnections** page in a *Dynamics Eigenvalue* case.

The following are guidelines for interconnections added to a *Dynamics* network case:



- **Rotor-Rotor, Rotor-Housing, Housing-Housing, or Rotor-Ground** interconnections may be defined using **Lateral, Axial, or Full** degree-of-freedom Interconnections. It is possible to assign multiple interconnections, of either type, to any rotor or housing node.
- Interconnections allow the user to specify linearized, stiffness, damping, and mass coefficients or nonlinear transfer function information to represent forces transmitted through the interconnection. The linearized force coefficients are specified as a function of rotor speed and may be used to represent the forces generated by bearings, seals, impellers, or any other load-producing device. Nonlinear transfer function information can be used to represent magnetic bearings, gas seals, tilting pad journal bearings or any other force reacting element that can not be modeled using linearized coefficients. Interconnections are generally used to tie rotors to housings or rotors to ground; however, for the special case of co-axially located rotors, they may be used to tie rotors to rotors. In addition, for the case of multiple concentric housings, they may be used to tie housings to housings.

## Component Category

Use **Component Category** to specify whether the interconnect is a bearing or a seal.

## Component Type

Use **Component Type** to specify the type of bearing or seal selected in **Component Category**. The options available depend on the **Component Category** selected.

## Case Name

Use **Case Name** to select the interconnect analysis case to use for the current interconnection.

## Define button

Use the **Define** button to access the **Interconnection Definition** dialog box to define the selected interconnection.

## Interconnection Definition dialog box

Use the **Interconnection Definition** dialog box to specify the details of the current interconnection.

The text at the top of the dialog box (**Type, Moment Coefficients, and Characteristic**) lists the settings from the case selected for the current interconnection.

### Interpolation

Use **Interpolation** to specify the method of interpolation to be used in the calculations.

### Connectivity

Use **Connectivity** to specify the type of interconnection, i.e., what elements are being connected.

Below **Connectivity** there are two rows of drop-down menus. The first row

is for specifying the details for the first part of the connectivity relationship. The second row is for specifying the details for the second part.

The first drop-down menu in each row contains a list of the rotor or housing elements available in the current *Eigenvalue* case. These are *Structural Dynamics* cases you have added on the **Rotors** and **Housings** pages in the current *Eigenvalue* case. Select which element you want to use for each end of the connection.

The second drop-down menu on each row contains a list of the nodes for the selected element. This list of nodes is obtained from the nodes entered on the **Nodes** page for the selected *Structural Dynamics* case. Select the node at which the interconnection will connect to that element.

### Run Mode

Use **Run Mode** to specify how the data from the component analysis case is associated with the current network analysis case.

The following options are available:

- **Update**

Use the output file from the last run of the component analysis case.

**NOTE:** If the component analysis case has not been run before (if the output file for the component does not exist), this option works exactly like the **Re-Run** option.

- **Store**

Store a complete copy of the component analysis case at the network level with the current network analysis case.

- **Re-Run**

Re-run the component analysis case at the component level and uses the updated results.

See **Part VII – Miscellaneous features** in **Tutorial II** for more information. ????

### Rub Model

Use **Rub Model Define** button to select whether the rub model is active or inactive.

#### Define Rub Model dialog box

Use the **Define Rub Model** dialog box to specify **Rub Model** values to model contact between **Rotors** and **Housings**. This simple model considers

only the interface contact stiffness and friction. The radius at which the rub occurs is used to apply frictional reaction forces to the entities in contact. The clearance between the two entities is modeled based on an average radial clearance and an assembled **Housing** eccentricity.

**NOTE:** The **Rub Model** is only available for *Transient Forced Response* network cases.

- **Radial Clearance**

Specify the **Radial Clearance** between rotating and stationary entities. This value must be a non-zero positive value.

- **Housing Eccentricity X**

Specify the initial eccentricity of the **Housing** relative to the **Rotor** along the X-axis in the **Housing Eccentricity X** edit box.

- **Housing Eccentricity Y**

Specify the initial eccentricity of the **Housing** relative to the **Rotor** along the Y-axis in the **Housing Eccentricity Y** edit box.

- **Interface Stiffness**

Use **Interface Stiffness** to specify the effective stiffness used to calculate the restoring forces at the interface of a **Rotor** and **Housing** rub. The value must be a non-zero positive value.

- **Interface Friction**

Use the **Interface Friction** to specify the effect coefficient of sliding friction for the material pairs for the **Rotor** and **Housing** at the interface of a rub. The value must be a non-zero positive value.

- **Interface Radius**

Use the **Interface Radius** to specify the effective radius used to calculate the torsional rubbing forces at the interface of a **Rotor** and **Housing** rub. The value must be a non-zero positive value.

### **Directionality**

Use **Directionality** to define the orientation of an axial rub. When selected, this indicator is used to determine when the axial displacement in the selected direction has exceeded the axial gap thus resulting in a rub.

The following options are available:

- **Left**
- **Right**

**NOTE: Directionality** is only available for axial interconnects (Thrust Bearings).

### Forcing Function page

**Forcing Functions** are used only in *Steady State Forced Response* and *Transient Forced Response* network cases to describe the forces that are applied to **Rotors** or **Housings**. Any **Forcing Function** characteristic can be described using the **Constant Forcing Function** merely by entering tabular data as a function of either **Time** or **Rotor Speed**. For convenience, several widely used **Forcing Function** characteristics have been included.

The following are guidelines for forcing functions added to a *Steady State Forced Response* or *Transient Forced Response* network case:

Harmonic response due to rotating forcing functions may be determined. Forcing functions may only be placed on rotor entities. .

- An unlimited number of rotating forcing functions, with arbitrary phasing, may be specified. Also, no limit on the number of different rotors with an excitation exists.
- The magnitude of each forcing function is defined by a user specified mass-length value or by a force value. If the magnitude is a mass-length, *Steady State Forced Response* internally multiplies this mass-length value by the excitation frequency squared to set the forcing function's magnitude at a given excitation frequency. If the excitation frequency is equal to the rotor speed of a rotor, the mass-length is an unbalance. Therefore, an unbalance response analysis must be for rotors running at the same speed ratio only.

### Type

Use **Type** to specify the type of forcing function, **Rotor** or **Housing**.

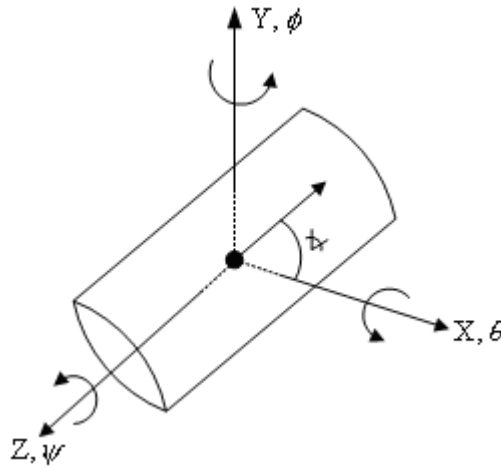
### DOF (Degrees-of-Freedom)

Use **DOF** (degrees-of-freedom) to specify the coordinate axes along which the force/moment acts. The following options are available:

- **Lateral** - radial force in the lateral (x, y) plane using magnitude and angle/phase.
- **Axial** - force along the z-axis
- **Torsional** - moment (torque) about the z-axis
- **Moment** - moment in the lateral (x, y) plane using magnitude and angle/phase

**NOTE:** Use the “right-hand rule” when determining the direction of the force about an axis.

The following picture is a representation of the coordinates and forces/moments.



*Figure 50. Coordinates and Degrees-of-Freedom for masses*

## Forcing Function Name

Use **Forcing Function Name** to select a rotor or housing case for the forcing function. The type of entity selected in the **Type** drop-down menu determines whether you select a rotor or housing for the **Forcing Function Name**.

## Define button

Use the **Define** button to specify the details of the current forcing function.

## Forcing Function Definition dialog box

Use the **Forcing Function Definition** dialog box to specify the details of the current forcing function.

The text at the top of the dialog box displays information about the *Structural Dynamics* component case selected for the current forcing function.

## Characteristic

The **Characteristic** drop-down menu allows you to specify which values (displayed in the table) are varied with respect to either **Rotor Speed** or **Time** (selected in the **Dependency** drop-down menu).

- **Constant** is a forcing function characteristic that has a constant value.
- **Harmonic** is a forcing function characteristic that is linearly proportional to any frequency you select.
- **Impulse** is a forcing function characteristic that is frequency

---

## RAPPID User's Guide

---

independent and consists of a single, short time duration, square wave.

- **Periodic** is a forcing function characteristic that can be a user defined function of time that is repeated according to a user defined period.
- **Unbalance** is a forcing function characteristic that is proportional to frequency squared where the frequency is limited to **Rotor Speed**.

### Dependency

The magnitude of any of the forcing functions can be made to vary with either **Rotor Speed** or **Time**. The **Dependency** drop-down menu allows you to specify whether the values chosen through the **Characteristic** drop-down menu vary versus **Rotor Speed** or **Time**. Note that while the magnitude of the **Forcing Function** can vary, the characteristic remains the same (i.e., **Unbalance** is still proportional to **Rotor Speed** squared).

If **Time** is selected, two additional fields display, **Start Time** and **End Time**. A **Forcing Function** can be started and stopped at any time during the simulation. The user selects the point at which a **Forcing Function** begins using **Start Time** and selects the point during the simulation at which it stops using **End Time**. The **Start Times** and **End Times** for **Forcing Functions** are completely independent of each other.

**NOTE:** When **Periodic** is chosen as the **Characteristic**, the only choice for **Dependency** is **Time**.

### Location

Select the **Location** to specify on which node on the **Rotor** or **Housing** to apply the selected **Forcing Function**.

### Data grid

Use the **Forcing Function Definition** data grid to define the magnitude, frequency, phase, and angle of the **Forcing Function** characteristic. The columns of the table include only those settings that apply to the chosen forcing function **Characteristic** (e.g., **Impulse** does not include frequency).

#### Magnitude

Use **Magnitude** to specify the magnitude of the forcing function expressed in terms of:

- Mass-length if value is positive or,
- Constant force or moment if value is negative.

Only constant moment and force values are valid for torsional and axial forcing functions. All forcing

functions must be either mass-length or force values.

**Magnitude** in the data grid is available for the following characteristics:

- Constant
- Impulse
- Periodic
- Harmonic
- Unbalance (Rotors only)

**NOTE:** The **Unbalance** characteristic is only available when **Rotor** is selected as the **Type** of forcing function.

## Angle

Use **Angle** to define the angular position where the forcing function is applied.

**Angle** in the data grid is available for the following characteristics:

- Constant
- Impulse
- Periodic
- Harmonic (Housings only)

**NOTE:** **Angle** is only available for the **Harmonic** characteristic when the type selected for the forcing function is **Housings**.

## Phase

Use **Phase** to specify the phase of the forcing function relative to other forcing functions in the system model.

**Phase** in the data grid is available for the following characteristics:

- Harmonic
- Unbalance

## Frequency

Use **Frequency** to define the frequency of the forcing function.

**Frequency** in the data grid is available for the following characteristic:

- Harmonic

## Output Options page – Eigenvalue

Use the **Output Options** page to specify the data to be included in the output and the plots and the frequency range to be used when calculating output and displaying plots.

## Other box

Use the **Other** box for *Eigenvalue* analysis cases to specify what to include in the viewed output and to be used when calculating results for plots.

### Print Input Mode Shape Tables

Use **Print Input Mode Shape Tables** to include the mode shape tables from the *Structural Dynamics* components in the viewed output.

### Print Interconnection Data Tables

Use **Print Interconnection Data Tables** to include the interpolated data from the interconnections (i.e., *Bearings*, *Seals*, etc.) in the viewed output.

### Print Output Mode Shape Tables

Use **Print Output Mode Shape Tables** to print mode shapes used for analysis in the viewed output.

**NOTE:** This option is only available for *Eigenvalue* analysis cases.

### Print Response Shape Tables

Use **Print Response Shape Tables** to print response shapes for structural entities in the viewed output.

**NOTE:** This option is only available for *Steady State Forced Response* analysis cases.

### Print/Plot Rotor Displacements

Use **Print/Plot Rotor Displacements** to include rotor displacement information in the output file and in the plots.

When this check box is selected, the **Define** button next to **Rotor Locations to Output Displacements** in the **Output Locations** box becomes available. Use the **Define** button to specify rotor output locations.



See **Output Locations** for more information. ????

**NOTE:** This option is only available for *Steady State Forced Response* and *Transient Forced Response* analysis cases.

## **Print/Plot Housing Displacements and Print/Plot Housing Accelerations**

Use **Print/Plot Housing Displacements** and **Print/Plot Housing Accelerations** to include housing displacement and/or acceleration information in the output file and in the plots.

When either of these two check boxes is selected, the **Define** button next to **Housing Locations to Output Displacements/Accelerations** in the **Output Locations** box becomes available. Click the **Define** button to specify housing output locations for housing displacements and/or accelerations.

See **Output Locations** for more information. ????

## **Print/Plot Interconnection Loads/Displacements**

Use **Print/Plot Interconnection Loads/Displacements** to include rotor displacement information in the output file and in the plots.

When this check box is selected, the **Define** button next to **Interconnections to Output Loads/Displacements** in the **Output Locations** box becomes available. Use the **Define** button to specify interconnection output locations.

See **Output Locations** for more information. ????

## **Print/Plot Forcing Functions**

Use **Print/Plot Forcing Functions** to include rotor displacement information in the output file and in the plots.

**NOTE:** This option is only available for *Transient Forced Response* cases.

When this check box is selected, the **Define** button next to **Forcing Functions to Output Magnitude/Angle** in the **Output Locations** box becomes available. Use the **Define** button to specify forcing function output locations.

See **Output Locations** for more information. ????

## **Frequency Range for Reported Results box**

Specify the minimum and maximum **Natural Frequencies** and rotational frequencies for **Mode Shapes** in the **Frequency Range for Reported Results** box. These frequency limits apply when you view output and display plots.

---

## RAPPID User's Guide

---

**NOTE:** The **Frequency Range for Reported Results** box is only available for *Eigenvalue* cases.

### Output Locations box

Use the **Define** buttons in the **Output Locations** box to specify information about the rotors, housings, interconnects, and forcing functions to be included in the viewed output and to be used in the calculations for the plots.

Use corresponding check boxes in the **Other** box to activate the desired **Define** buttons in the **Output Locations** box.

### Rotor/Housing/Interconnection Output Locations dialog box

Use the **Output Locations** dialog box to specify information about the rotor node locations to be included in the viewed output and to be used when calculating results for plots. The following field descriptions are applicable for **Rotor**, **Housing**, and **Interconnection Output Locations**.

#### Rotor / Housing / Interconnection

Use **Rotor/Housing/Interconnection** to specify a rotor for the current location.

#### Node

Use **Node** to select a node on the selected rotor for the current location.

**NOTE:** The **Node** column is only available for **Rotor** and **Housing Output Locations**.

#### Output Type

Use **Output Type** to specify indicate whether to include this location in the output file and in the plots. You may temporarily not want to include a specific location in a run of the module; however, you may not want to delete the information from your case.

**NOTE:** For **Rotor Output Locations** in *Steady State Forced Response* analysis cases, there is only one type of output, **Displacements**.

#### Mount Angle

Use **Mount Angle** to enable the user to define instrumentation locations relative to the local entity coordinate system. Values must be in the range 0-360 degrees.

#### Rub Limit

Enter a **Rub Limit** value is used to enable the user to define a clearance for plotting purpose to indicate a potential rub.

**NOTES:** The **Rub Limit** is only for plotting purposes and will not affect the **Rub Model**.

---

# RAPPID User's Guide

---

This column is only available for *Transient Forced Response* cases.

## Forcing Function Output dialog box

Use the **Forcing Function Output** dialog box to specify information about the forcing functions to be included in the viewed output and to be used when calculating results for plots.

**NOTE:** The **Forcing Function Output** dialog box is only available for *Transient Forced Response* analysis cases.

### Forcing Function

Use **Forcing Function** to select a forcing function to be plotted.

### Details button

Use the **Details** button to view information about the selected forcing function on the **Forcing Function Definition** dialog box. This allows you to make sure the selected forcing function is the one you want to include in the viewed output and displayed plots.

**NOTE:** The **Forcing Function Definition** dialog box that displays using the **Details** button is the same dialog box used to define the forcing function on the **Forcing Functions** page. However, the fields are read-only when accessed using the **Details** button. You cannot change any settings here. This dialog box displays so you can quickly determine if this forcing function is the one you want to include in the output and the plots. To change the settings for this forcing function, return to the **Forcing Functions** page.

### Sample Rate

Use **Sample Rate** to specify how many samples per second to use when calculating results to be plotted.

### ID Text

Use **ID Text** to enter text to identify the selected forcing function. This text is used when viewing output and displaying forcing function plots.

## Output Options page – Steady State Forced Response

Use the **Output Options** page to specify the data to be included in the output and the plots.

## Other box

Use the **Other** box for *Steady State Forced Response* cases to specify what to include in the viewed output and to be used when calculating results for plots.

## **Print Input Mode Shape Tables**

Use **Print Input Mode Shape Tables** to include the mode shape tables from the *Structural Dynamics* components in the viewed output.

## **Print Interconnection Data Tables**

Use **Print Interconnection Data Tables** to include the interpolated data from the interconnections (i.e., *Bearings, Seals*, etc.) in the viewed output.

## **Print Response Shape Tables**

Use **Print Response Shape Tables** to print response shapes for structural entities in the viewed output.

## **Print Rotor Displacements to File**

Use **Print Rotor Displacements to File** to include rotor displacement information in the output file and in the plots.

When this check box is selected, the **Define** button next to **Rotor Locations to Output Displacements** in the **Output Locations** box becomes available. Use the **Define** button to specify rotor output locations.

See **Output Locations box** on page 298 for more information.

## **Print Housing Displacements to File/Print Housing Accelerations to File**

Use **Print Housing Displacements to File** and **Print Housing Accelerations to File** to include housing displacement and/or acceleration information in the output file and in the plots.

When either of these two check boxes is selected, the **Define** button next to **Housing Locations to Output Displacements/Accelerations** in the **Output Locations** box becomes available. Click one of the **Define** buttons to specify housing output locations.

See **Output Locations box** on page 298 for more information.

## **Print Interconnection Loads/Displacements to File**

Use **Print Interconnection Loads/Displacements to File** to include rotor displacement information in the output file and in the plots.

When this check box is selected, the **Define** button next to **Interconnections to Output Loads/Displacements** in the **Output Locations** box becomes available. Use the **Define** button to specify interconnection output locations.

See **Output Locations box** on page 298 for more information.

## Output Locations box

Use the **Define** buttons in the **Output Locations** box to specify information about the rotors, housings, and interconnects to be included in the viewed output and to be used in the calculations for the plots.

Use corresponding check boxes in the *Other box* (on page 300) to activate the desired **Define** buttons in the **Output Locations** box.

## Rotor Output Locations dialog box

Use the **Rotor Output Locations** dialog box to specify information about the rotor node locations to be included in the viewed output and to be used when calculating results for plots.

### Rotor

Use **Rotor** to specify a rotor for the current location.

### Node

Use **Node** to select a node on the selected rotor for the current location.

### Output Type

Use **Output Type** to specify indicate whether to include this location in the output file and in the plots. You may temporarily not want to include a specific location in a run of the module; however, you may not want to delete the information from your case. For rotors, there is only one type of output, **Displacements**.

### Mount Angle

Use **Mount Angle** to enable the user to define instrumentation locations relative to the local entity coordinate system. Values must be in the range 0-360 degrees.

## Housing Output Locations dialog box

Use the **Housing Output Locations** dialog box to specify information about the housing node locations to be included in the viewed output and to be used when calculating results for plots.

### Housing

Use **Housing** to specify a housing for the current location.

### Node

Use **Node** to select a node on the selected housing for the current location.

## **Output Type**

Use **Output Type** to specify whether to include this location in the output file and in the plots. You may temporarily not want to include a specific location in a run of the module; however, you may not want to delete the information from your case. For housings, there are two types of output, **Displacements** and **Accelerations**.

## **Mount Angle**

Use **Mount Angle** to enable the user to define instrumentation locations relative to the local entity coordinate system. Values must be in the range 0-360 degrees.

## **Limit**

Enter a **Limit** value is used to enable the user to define a clearance for plotting purpose to indicate a potential rub.

**NOTE:** The **Limit** is only for plotting purposes and will not affect the **Rub Model**.

## **Interconnection Output Locations dialog box**

Use the **Interconnection Output Locations** dialog box to specify information about the interconnections to be included in the viewed output and to be used when calculating results for plots.

## **Interconnection**

Use **Interconnection** to specify a housing for the current location.

## **Output Type**

Use **Output Type** to specify whether to include this location in the output file and in the plots. You may temporarily not want to include a specific location in a run of the module; however, you may not want to delete the information from your case. For interconnections, there are two types of output, **Relative Displacements** and **Loads**.

## **Mount Angle**

Use **Mount Angle** to enable the user to define instrumentation locations relative to the local entity coordinate system. Values must be in the range 0-360 degrees.

## **Limit**

Enter a **Limit** value is used to enable the user to define a clearance for plotting purpose to indicate a potential rub.

**NOTE:** The **Limit** is only for plotting purposes and will not affect the **Rub Model**.

## Output Options page – Transient Forced Response

Use the **Output Options** page to specify the data to be included in the output and the plots.

### Other box

Use the **Other** box for *Transient Forced Response* cases to specify what to include in the viewed output and to be used when calculating results for plots.

### Print Input Mode Shape Tables

Use **Print Input Mode Shape Tables** to include the mode shape tables from the *Structural Dynamics* components in the viewed output.

### Print Interconnection Data Tables

Use **Print Interconnection Data Tables** to include the interpolated data from the interconnections (i.e., *Bearings, Seals, etc.*) in the viewed output.

### Print Forcing Functions to File

Use **Print Forcing Functions to File** to include rotor displacement information in the output file and in the plots.

**NOTE:** This option is only available for *Transient Forced Response* cases.

When the **Print Forcing Functions to File** check box is selected, the **Define** button next to **Forcing Functions to Output** in the **Output Locations** box becomes available. Use the **Define** button to specify forcing function output locations.

See **Output Locations** box on page 301 for more information.

### Print Forcing Functions to File

Use **Print Forcing Functions to File** to include rotor displacement information in the output file and in the plots.

**NOTE:** This option is only available for *Transient Forced Response* cases.

When the **Print Forcing Functions to File** check box is selected, the **Define** button next to **Forcing Functions to Output** in the **Output Locations** box becomes available. Use the **Define** button to specify forcing function output locations.

See **Output Locations** box on page 301 for more information.

### Print Interconnection Loads/Displacements to File

Use **Print Interconnection Loads/Displacements to File** to include rotor displacement information in the output file and in the plots.

When this check box is selected, the **Define** button next to **Interconnections to Output Loads/Displacements** in the **Output Locations** box becomes available. Use the **Define** button to specify interconnection output locations.

See **Output Locations box** on page 301 for more information.

## **Print Forcing Functions to File**

Use **Print Forcing Functions to File** to include rotor displacement information in the output file and in the plots.

**NOTE:** This option is only available for *Transient Forced Response* cases.

When the **Print Forcing Functions to File** check box is selected, the **Define** button next to **Forcing Functions to Output** in the **Output Locations** box becomes available. Use the **Define** button to specify forcing function output locations.

See **Output Locations box** on page 301 for more information.

## **Output Locations box**

Use the **Define** buttons in the **Output Locations** box to specify information about the rotors, housings, interconnects, and forcing functions to be included in the viewed output and to be used in the calculations for the plots.

Use corresponding check boxes in the **Other box** on page 300 to activate the desired **Define** buttons in the **Output Locations** box.

## **Rotor Output Locations dialog box**

Use the **Rotor Output Locations** dialog box to specify information about the rotor node locations to be included in the viewed output and to be used when calculating results for plots.

### **Rotor**

Use **Rotor** to specify a rotor for the current location.

### **Node**

Use **Node** to select a node on the selected rotor for the current location.

### **Output Type**

Use **Output Type** to specify indicate whether to include this location in the output file and in the plots. You may temporarily not want to include a specific location in a run of the module; however, you may not want to delete the information from your case. For rotors, there is only one type of output, **Displacements**.



## **Mount Angle**

Use **Mount Angle** to enable the user to define instrumentation locations relative to the local entity coordinate system. Values must be in the range 0-360 degrees.

## **Housing Output Locations dialog box**

Use the **Housing Output Locations** dialog box to specify information about the housing node locations to be included in the viewed output and to be used when calculating results for plots.

## **Housing**

Use **Housing** to specify a housing for the current location.

## **Node**

Use **Node** to select a node on the selected housing for the current location.

## **Output Type**

Use **Output Type** to specify whether to include this location in the output file and in the plots. You may temporarily not want to include a specific location in a run of the module; however, you may not want to delete the information from your case. For housings, there are two types of output, **Displacements** and **Accelerations**.

## **Mount Angle**

Use **Mount Angle** to enable the user to define instrumentation locations relative to the local entity coordinate system. Values must be in the range 0-360 degrees.

## **Limit**

Enter a **Limit** value is used to enable the user to define a clearance for plotting purpose to indicate a potential rub.

---

**NOTE:** The **Limit** is only for plotting purposes and will not affect the **Rub Model**.

---

## **Interconnection Output Locations dialog box**

Use the **Interconnection Output Locations** dialog box to specify information about the interconnections to be included in the viewed output and to be used when calculating results for plots.

## **Interconnection**

Use **Interconnection** to specify a housing for the current location.

## **Output Type**

Use **Output Type** to specify whether to include this location in the output file and in the plots. You may temporarily not want to include a specific location in a run of the module; however, you may not want to delete the information from your case. For interconnections, there are two types of output, **Relative Displacements** and **Loads**.

## **Mount Angle**

Use **Mount Angle** to enable the user to define instrumentation locations relative to the local entity coordinate system. Values must be in the range 0-360 degrees.

## **Limit**

Enter a **Limit** value is used to enable the user to define a clearance for plotting purpose to indicate a potential rub.

**NOTE:** The **Limit** is only for plotting purposes and will not affect the **Rub Model**.

## **Forcing Function Output dialog box**

Use the **Forcing Function Output** dialog box to specify information about the forcing functions to be included in the viewed output and to be used when calculating results for plots.

## **Forcing Function**

Use **Forcing Function** to select a forcing function to be plotted.

## **Details button**

Use the **Details** button to view information about the selected forcing function on the **Forcing Function Definition** dialog box. This allows you to make sure the selected forcing function is the one you want to include in the viewed output and displayed plots.

**NOTE:** The **Forcing Function Definition** dialog box that displays using the **Details** button is the same dialog box used to define the forcing function on the **Forcing Functions** page. However, the fields are read-only when accessed using the **Details** button. You cannot change any settings here. This dialog box displays so you can quickly determine if this forcing function is the one you want to include in the output and the plots. To change the settings for this forcing function, return to the **Forcing Functions** page.

## **Sample Rate**

Use **Sample Rate** to specify how many samples per second to use when calculating results to be plotted.

## **ID Text**

Use **ID Text** to enter text to identify the selected forcing function. This text is used when viewing output and displaying forcing function plots.

## **Plot Options – Eigenvalue**

Each *Dynamics* network contains different options for setting up the plots. These options customize the plot displays.

Use the following settings to customize the available *Eigenvalue* plots.

For general information on plots, see **Chapter 7: Plotting results** on page 119.

## **Select Stability Parameter**

Use **Select Stability Parameter** to specify the stability parameter used on the **Stability Plot**. This setting indicates whether the various rotor modes are stable or unstable.

## **Limit Eigenvalue Data to Display (Root Pre-sort)**

Use **Limit Eigenvalue Data to Display (Root Pre-sort)** to define harmonic lines to be displayed on the natural frequency plot and/or presort parameters to filter roots to be displayed on the **Natural Frequency Plot**, **Stability Plot** and the **Root Locus Plot**.

If you select this option, enter the minimum and maximum values to be displayed on the Eigenvalue plots for the **Natural Frequency Range (cpm)**, **Stability Parameter**, and **Rotor Speed (rpm)**. For the **Natural Frequency Range (cpm)** and **Rotor Speed (rpm)**, minimum and maximum values are displayed next to the edit boxes. The values you enter should be within these respective ranges.

## **Sort Roots on Natural Frequency and Stability Plots**

Use **Sort Roots on Natural Frequency and Stability Plots** to specify whether the roots displayed on the Natural Frequency, Stability and Root Locus plots are sorted and associated points are connected.

The accuracy of the sorting is dependent upon the existence of full and partial modes.

- **Full modes** refers to modes that are complex (non-zero natural frequencies) over the entire analysis range
- **Partial modes** refers to modes that are real (0 rpm natural frequency) over part of the analysis range and complex over the remaining parts. Partial modes by definition, may disappear and reappear throughout the analysis range, which complicates the sorting.

If you select the **Sort Roots on Natural Frequency and Stability Plots** check box, choose options and enter values for the following input fields:

## Modes to Sort

Two sorting options are available to assist the user in obtaining an accurate sort of the analysis results.

- **Full Modes Only**

Only full modes are sorted and connected on the plots. Remaining roots not associated with a full mode are shown as points.

- **Full and Partial Modes**

Both full and partial modes are sorted and connected on the plots.

If neither of the two available sorting routines are able to accurately sort the roots, you should select the **Limit Eigenvalue Data to Display (Root Pre-sort)** option at the top of the **Plot Options** dialog box and try the following:

1. Remove roots associated with heavily damped modes by specifying a maximum value of stability parameter to be displayed.
2. Remove roots associated with low and/or high rotor speed points which are causing sorting problems.
3. Display only a portion of the rotor's natural frequencies by defining a range of natural frequencies that can be sorted correctly.
4. Re-run the analysis and obtain more rotor speed points within the desired speed range.

By using the 6 presorting root filters, virtually all natural frequency plots can be sorted accurately. However, use of these filters may require multiple natural frequency plots to display all of the relevant/desired roots.

## Modes to Display

Use **Modes to Display** to specify whether backward whirling modes are displayed on the **Natural Frequency**, **Stability**, or **Root Locus** plots. Select **Forward Whirling Only** if you want to exclude backward whirling modes from the plots.

If for any mode, all calculated mode shapes for that mode have backward whirl at 80% or more of the requested stations, that mode is considered to be backward whirling.

**NOTE:** If mode shapes are calculated for a particular mode, that mode will never be classified as backward whirling even if it is actually backward whirling. Also, if two or more mode shapes are calculated for a particular mode (at different values of the analysis parameter), and one of these mode shapes does not have 80% or more of the stations backward whirling, then

the mode will not be designated as backward whirling.

## **Label Operating Range and Interferences**

Use **Label Operating Range and Interferences** to specify the operating range that displays on the **Natural Frequency** and **Stability** plots. Use the **Start** and **Stop** edit boxes to specify the range. Additionally, all interferences of harmonic lines and natural frequencies are circled and labeled.

## **Eliminate Mode Shape Plots With Deflections < xx.xx% of Maximum Deflection**

Use **Eliminate Mode Shape Plots With Deflections < xx.xx% of Maximum Deflection** to eliminate modes shapes based on normalized magnitudes. All mode shapes which do not have a deflection greater than the percentage specified of the maximum system orbit will not be plotted. Note that the mode shapes of the system have been normalized, and the mode shape plots are scaled to the maximum.

## **Add Harmonic Lines to Natural Frequency Plot**

Use **Add Harmonic Lines to Natural Frequency Plot** to enter the order of the harmonic lines to be displayed on the **Natural Frequency Plot**.

## **Plot Options – Steady State Forced Response**

Each *Dynamics* network contains different options for setting up the plots. These options customize the plot displays. Use the following settings to customize the available *Steady State Forced Response* plots.

For general information on plots, see **Chapter 7: Plotting results** on page 119.

## **Select coordinate systems for all nodal location plots**

Use **Select coordinate systems for all nodal location plots** to select the coordinate system (associated with the response, acceleration, dynamic load, and relative response ellipses) to be used in calculating the responses, accelerations, loads and relative responses displayed in the plots.

The following options are available:

- **X'-Y'** (user defined **X'-Y'** coordinate system)
- **Major/Minor** (the **Major Orbital Coordinate** system)
- **Both**

## **Display forcing function location and magnitude on plots**

Use **Display forcing function location and magnitude on plots** to display a table listing the station, magnitude of each forcing function applied, and the corresponding phase is shown on all plots except the geometry plot. The geometry plot has an extra symbol showing the locations on the shaft where excitation is applied.

### **Set displacement plots maximum ordinate values to**

Use **Set displacement plots maximum ordinate values to** to set the maximum value for the ordinate (Y-axis) on the displacement plots to the value you specify. This option allows you to control the ordinate scale for easy comparison of multiple runs.

**NOTE:** If the displacement magnitude is greater than the maximum ordinate value for any part of the analysis range, that portion of the displacement curve is not displayed on the plot.

### **Set housing acceleration plots maximum ordinate values to**

Use **Set housing acceleration plots maximum ordinate values to** to set the maximum value for the ordinate (Y-axis) on the housing acceleration plots to the value you specify. This option allows you to control the ordinate scale for easy comparison of multiple runs.

**NOTE:** If the housing acceleration magnitude is greater than the maximum ordinate value for any part of the analysis range, that portion of the housing acceleration curve is not displayed on the plot.

### **Set dynamic load plots maximum ordinate values to**

Use **Set dynamic load plots maximum ordinate values to** to set the maximum value for the ordinate (Y-axis) on the dynamic load plots to the value you specify. This option allows you to control the ordinate scale for easy comparison of multiple runs.

**NOTE:** If the dynamic load magnitude is greater than the maximum ordinate value for any part of the analysis range, that portion of the dynamic load curve is not displayed on the plot.

### **Set relative displacement plots maximum ordinate values to**

Use **Set relative displacement plots maximum ordinate values to** to set the maximum value for the ordinate (Y-axis) on the relative displacement plots to the value you specify. This option allows you to control the ordinate scale for easy comparison of multiple runs.

**NOTE:** If the relative displacement magnitude is greater than the maximum ordinate value for any part of the analysis range, that portion of the relative displacement curve is not displayed on the plot.

### **Eliminate response shapes with maximum < xx.xx% of Maximum Response**

Use **Eliminate response shapes with maximum < xx.xx% of Maximum Response** to eliminate response shapes based on response magnitudes. All response shapes which do not have an orbit greater than the specified percentage of the maximum system orbit are not plotted.

### **Calculate separation margins on X'-Y' Response plots**

Use **Calculate separation margins on X'-Y' Response plots** to specify that all critical speeds within the analysis range with amplification factors higher than the value you specify are labeled with a separation margin. For the purposes of this calculation, separation margin is defined as the ratio of the critical speed to the maximum (or minimum) operating frequency.

**NOTE:** To use this option, you must define an operating frequency range (**Response Range** and **Response Shape Range**) in the **Analysis Parameter** box on the **Options** page. You must also specify that amplification factors are calculated (see Show amplification factors > 'value' on X'-Y' Response plots).

### Show amplification factors > 'value' on X'-Y' Response plots

Use **Show amplification factors > 'value' on X'-Y' Response plots** to specify that the amplification factor is calculated and labeled for each relative maximum having an amplification factor greater than the 'value' you specify.

The point of maximum response and the two other points used to calculate the amplification factor is circled to allow the user to judge the quality of the calculation.

### Remove grid lines from X'-Y' Response plots

Use **Remove grid lines from X'-Y' Response plots** to remove the grid lines from the response plots. This improves the clarity of the plots when separation margins and amplification factors are displayed.

### Plot Multiple curves on the Response plots

Use **Plot Multiple curves on the Response plots** to define multiple curves to be plotted on the same plot. This option may be used to overplot the dynamic load/response from different stations in the same set of plot results on the same graph and/or to overplot stations from different sets of plot results. To define the plot to be created, you must click the **Add** button. Note that when using this overplot option, the only plots that may be created are Response and Dynamic Load plots.

**NOTES:** When using this overplot option, the only plots that may be created are **Response** and **Dynamic Load** plots.

If you select the **Plot Multiple curves on the Response plots** check box, none of the other options on the **Plot Options** dialog box are applicable.

Existing plot curves are listed in the drop-down menu below the Plot multiple curves on Response plots check box.

**NOTE:** Plot curves added in one *Steady State Forced Response* analysis case are available in other *Steady State Forced Response* analysis cases within the same project.

Use the following buttons to manage the plot curves you want to add to the project:

**Add**            Use the **Add** button to add new plot curves to the list of available plot curves.

**Edit**            Use the **Edit** button to change the properties of existing plot curves.

**Copy** Use the **Copy** button to copy an existing plot curve. This is useful if you want to create a new plot curve that is similar to a plot curve that already exists.

**Delete** Use the **Delete** button to delete the selected plot curve.

## Define Plot Curves dialog box

Use the **Define Plot Curves** dialog box to define new plot curves, edit existing plot curves, and to make a copy of an existing plot curve, using the **Add**, **Edit**, and **Copy** buttons, respectively, on the **Plot Options** dialog box.

Specify values for the following fields:

**Name** Use **Name** to specify the name of the plot curve. The **Name** is used for display in the drop-down menu below the **Plot multiple curves on the Response plots** check box on the **Plot Options** dialog box.

**Rublimit** Use **Rublimit** as a visual check to see if the response at a station exceeds the clearance available at that station. To use the **Rublimit**, enter a nonzero value. A horizontal line displays on the major axis response plot. If the value entered for the rublimit exceeds the value of the ordinate on the plot, no horizontal line displays.

**Plot Titles** Use **Plot Titles** to enter up to three (3) lines of alphanumeric comment text to be displayed at the top of each plot. Note that the first line of user-entered text displays immediately to the right of the fixed title of the plot, while the second and third lines display immediately below the fixed title line.

**Response Coordinates** Use **Response Coordinates** to select the coordinate systems (associated with the response and load ellipses) to be used in calculating the responses and dynamic loads displayed in the plots. The options are the user defined X'-Y' coordinate system and/or the Major Orbital Coordinate system.

**Probe Mount Angle** If instrumentation is present on the system, the **Probe Mount Angle** allows you to convert output values to the same angular position as the probe. If **X'-Y'** or **Both** is selected as the **Response Coordinates**, a value for the **Probe Mount Angle** is required. This angle is relative to the fixed X-Y coordinate system. See the *Structural Dynamics components* (on page **Error! Bookmark not defined.**) section for a description of the coordinate



# RAPPID User's Guide

---

system.

<b>Plots to be Generated</b>	Use <b>Plots to be Generated</b> to select whether <b>Response</b> and/or <b>Dynamic Load</b> plots are generated.				
<b>Generate Acceleration</b>	Use <b>Generate Acceleration</b> to determine if housing acceleration plots are to be generated. The response coordinates chosen for <b>Response Coordinates</b> also apply to the acceleration plots.				
<b>Acceleration Probe Mount Angle</b>	<p>Use <b>Acceleration Probe Mount Angle</b> to define the location of the user defined X'-Y' acceleration planes relative to the fixed X-Y coordinate system. See the <i>Structural Dynamics</i> section for a description of the coordinate system.</p> <p><b>NOTE:</b> If X'-Y' or <b>Both</b> is selected as the <b>Response Coordinates</b>, and <b>Yes</b> is selected for <b>Generate Acceleration</b>, a value for the <b>Acceleration Probe Mount Angle</b> is required.</p>				
<b>X-Axis Title</b>	Enter a text string to identify the analysis parameter being plotted on the X-axis.				
<b>Plot curve data selection grid</b>	<p>Use the <b>Plot curve data selection grid</b> to select the network cases from which the data is obtained for the plot curve data set. Insert a row into the data grid (see <b>To insert a row</b> in the <b>Analysis Cases</b> section) and specify the following options:</p> <table><tr><td><b>Network Case Name</b></td><td><p>Select a <i>Steady State Forced Response</i> analysis case from the list of available analysis cases.</p><p>When you click in the box in the <b>Network Case Name</b> column for a network analysis case you are selecting, the <b>Edit Description</b> dialog box displays allowing you to select a network analysis case and to add to the description for that network analysis case.</p></td></tr><tr><td><b>Entity Type</b></td><td><p>Select <b>Rotor</b> or <b>Housing</b> to specify whether the location to plot is on a rotor or housing defined on the <b>Output Options</b> page in the <i>Steady State Forced Response</i> analysis case selected in the <b>Network Case Name</b> column.</p></td></tr></table>	<b>Network Case Name</b>	<p>Select a <i>Steady State Forced Response</i> analysis case from the list of available analysis cases.</p> <p>When you click in the box in the <b>Network Case Name</b> column for a network analysis case you are selecting, the <b>Edit Description</b> dialog box displays allowing you to select a network analysis case and to add to the description for that network analysis case.</p>	<b>Entity Type</b>	<p>Select <b>Rotor</b> or <b>Housing</b> to specify whether the location to plot is on a rotor or housing defined on the <b>Output Options</b> page in the <i>Steady State Forced Response</i> analysis case selected in the <b>Network Case Name</b> column.</p>
<b>Network Case Name</b>	<p>Select a <i>Steady State Forced Response</i> analysis case from the list of available analysis cases.</p> <p>When you click in the box in the <b>Network Case Name</b> column for a network analysis case you are selecting, the <b>Edit Description</b> dialog box displays allowing you to select a network analysis case and to add to the description for that network analysis case.</p>				
<b>Entity Type</b>	<p>Select <b>Rotor</b> or <b>Housing</b> to specify whether the location to plot is on a rotor or housing defined on the <b>Output Options</b> page in the <i>Steady State Forced Response</i> analysis case selected in the <b>Network Case Name</b> column.</p>				

<b>Entity #</b>	Select the number for the rotor or housing on which the node to plot is located.
<b>Node #</b>	Select the node number on the selected rotor or housing to plot. Only nodes defined for the selected rotor or housing are available.
<b>Legend Label</b>	Enter the text to be included in the plot legend to identify the curve.

## Plot Options – Transient Forced Response

The **Plot Options** dialog box for *Transient Forced Response* network analysis cases allows you to select smaller sets of data to be plotted using the following types of user-defined plots:

- Orbit
- Fast Fourier Transform (FFT)
- Frequency Response Function (FRF)
- Power Spectral Density (PSD)
- Waterfall

For general information on plots, see **Chapter 7: Plotting results** on page 119.

The following options are available as columns on the data grid for all of the user-defined plot types listed above:

**NOTE:** The options available for specifying entities to be plotted (**Entity Type**, **Output Type**, **Output Node/Interconnect**) are dependent upon the entities selected and locations specified on the **Output Options** page in the current *Transient Forced Response* analysis case. The entities must be selected to be printed to file in the **Other** box, and defined in the **Output Locations** box.

<b>Plot #</b>	Automatically generated.
<b>Entity Type</b>	The following options are available for <b>Entity Type</b> : <ul style="list-style-type: none"><li>• Rotor</li><li>• Housing</li></ul>

# RAPPID User's Guide

---

- Interconnect

**NOTE:** For **FRF** plots, there is a **Reference Entity Type** and a **Response Entity Type**.

## Output Type

The following options are available for **Output Type**:

- For Rotors: Displacements
- For Housings: Displacements and Accelerations
- For Interconnects: Loads and Relative Displacements

**NOTE:** For **FRF** plots, there is a **Reference Output Type** and a **Response Output Type**.

## Output Node / Interconnect

The nodes selected when you defined each location on the entity to be plotted on the Output Options page are available.

**NOTE:** For **FRF** Plots, there is a **Reference Output Node/Interconnect** and a **Response Output Node/Interconnect**.

## Start Time

The starting time for the subset of data to be plotted.

## End Time

The ending time for the subset of data to be plotted.

## Time points from plot data

### Start Time

The starting time from the selected set of data

### End Time

The ending time from the selected set of data

The following options are available as columns on the data grid for all of the user-defined plot types except for **Orbit** plots:

## Block Size

Use **Block Size** to specify the number of input time points used for each FFT block in the calculation.

## Overlap

Use **Overlap** to specify how much overlap to use between blocks. The value entered must be between 0 (no overlap) and 1 (overlap all but 1

point), inclusive.

## Window

Use **Window** to select a window function option to apply to each point in the time data set. Choose from the following window functions:

- None (no windowing)
- Hanning
- Hamming
- Flat Top
- Parzen
- Blackman
- Exponential

## Averaging

Use **Averaging** to select the method used to average the results of the FFT calculations. Select from the following options:

- **Mean**

The calculated data points are totaled and then each point is divided by the total number of data sets.

- **Peak Hold**

The highest of the last calculated value and the newly calculated value is used for each data point.

## Min Frequency

Use **Min Frequency** to specify the minimum value for the x-axis. Any data points having a frequency less than the minimum value specified are not displayed on the plot.

## Max Frequency

Use **Max Frequency** to specify the maximum value for the x-axis. Any data points having a frequency greater than the maximum value specified are not displayed on the plot.

## Max Amplitude

Use **Max Amplitude** to specify the maximum value for the y-axis. Any data points having an amplitude greater than the maximum value specified are not displayed on the plot.

## FRF Plots

For **FRF** plots, **Entity Type**, **Output Type**, and **Output Node/Interconnect** have two entries per plot.

- Reference value (input)
- Response value (output)

The data grid for **FRF** plots contains one additional column:

- **DOF/Interconnect Type**

Use **DOF/Interconnect Type** to specify the degree of freedom (Rotors and Housings) or interconnect type (Interconnects) from which to obtain the data to be used in the FRF calculations. There are two entries (Reference and Response) in the **DOF/Interconnect Type** column.

## Waterfall plots

The data grid for **Waterfall** plots contains two additional columns beyond the columns listed above for the *Transient Forced Response* **Plot Options** dialog box:

<b>Step</b>	Use <b>Step</b> to specify the amount of time for each subset of data used in the Waterfall calculations.
<b>Axis Angle</b>	Use <b>Axis Angle</b> to specify the value to use to shift each set of data points along the x-axis (frequencies).

## Chapter 23: Transient Flow Network

### Overview

The *Transient Flow Network* analyzes complex flow network transiently. Define components in the network and couple the components at nodes. Specify boundary conditions at the nodes as pressure, temperature and mass flowrate. Mass flow can be injected at internal flow nodes as well. The network structure can easily incorporate new components.

### Options page

See **Introducing a sample case manager** on page 36, for information about the fields on the **Options** page that are common to all analysis cases.

### Analysis Parameter box

The **Analysis Parameter** drop-down menu allows you to select the input value by which you want to vary the values of other inputs.

Currently, **Time** is the only available **Analysis Parameter**.

The **Time** input consists of three fields: **Start**, **Delta**, and **Stop**. These fields represent the starting time (**Start**), the increment of time (**Delta**), and the ending time (**Stop**) for the calculation. The **Time** fields must define a constant or increasing time. Therefore, the **Stop** time must be greater than the **Start** time and the **Delta** time must be positive. Field entries must be non-negative floating point values.

### Loops

Loops allow you to group nodes in the output.

Add loops using the buttons above the data grid in the **Loops** box. Define which nodes are in which loops on the **Nodes** page.

See **Using fields and grids in Part IV – The Analysis case managers of Tutorial I – The basics** for information on entering data using a data grid. Enter a unique name for each loop added.

If you want to import data into the data grid, see **Importing data from external files in Part IV – The Analysis case managers of Tutorial 1 – The basics**.

### Fluids page

The **Fluids** page is common to all analysis cases that use a single fluid (compound) or multiple fluids.

**NOTE:** The **Fluids** page in *Transient Flow* network analysis cases is different from the **Fluids** page available in component analysis cases. You may add multiple fluids using a data grid on the **Fluids** page. See **Using a grid to enter data** on page 40 for information on entering data using a data grid.

Currently, the following three types of fluids are available:

- **Incompressible**
- **Compressible**
- **Real Properties**

For **Incompressible** and **Compressible** fluids, the following two options are available for including a fluid in your case:

- Define the properties manually, directly on the **Fluids** page in the analysis case.
- Link to a fluid from the **Fluids Library** by clicking the **Fluids Library** button, therefore using the properties from the library.

If you select a **Real Properties** fluid, you must link to a fluid from the **Fluids Library**. That fluid can either be a user-defined fluid or a fluid obtained from an existing fluid table.

See **Chapter 6: Working with fluids** on page 89 for more information about using fluids.

## Nodes page

In the *Transient Flow* network, nodes are connected by components. A flow path is comprised of groups of nodes and components. Fluid states are defined at the nodes and components determine the flow and heat transfer of fluids from node to node. For flow balance components and nodes, flow paths feeding them define their flow rate.

The following columns are available in the data grid on the **Nodes** page.

**NOTE:** See **Using a grid to enter data** on page 40 for information on entering data using a data grid.

## ID Text

Enter text to identify the node in the **ID Text** column. This text will identify the node in the output and in any related plots.

## Type

Use **Type** to select the type of node for the currently selected node.

There are three types of node available:

- **Internal**

Nodes on the interior of the network that, in general, connect two branches or elements.

- **Flowrate Boundary**

A boundary node with flowrate specified as the boundary condition.

- **Pressure Boundary**

A boundary node with pressure specified as the boundary condition.

## Define button

Use the **Define** button to define the node using the **Define Internal Node** or **Define Boundary Node** dialog box.

## Define Internal/Boundary Node dialog box

When you click the **Define** button on the **Nodes** page, the **Define Internal Node** or **Define Boundary Node** dialog box displays, depending on which type of node you selected in the **Type** drop-down menu.

**NOTE:** The values in these fields can vary with respect to time. See **Constant or variable analysis input** in **Part IV – The Analysis Case Managers** of **Tutorial 1 – The basics** for more information.

The following fields are available on the dialog box depending on which type of node you selected:

<b>Pressure</b>	Enter the <b>Pressure</b> applicable for the selected node. ??? [more description?]
<b>Temperature</b>	Enter the <b>Temperature</b> applicable for the selected node. ??? [more description?]
<b>Fluid</b>	Use <b>Fluid</b> to select an applicable fluid for the selected node. The fluids available are obtained from the fluids added on the <b>Fluids</b> page. [more description?]
<b>Mass Flowrate</b>	Enter the <b>Temperature</b> applicable for the selected node. ??? [more description]
<b>Enthalpy</b>	Enter the <b>Enthalpy</b> applicable for the selected node. ??? [more description]
<b>Thrust Load Area</b>	Enter the area used to calculate thrust load.

## View button

The **View** button displays the **View Connectivity** dialog box, allowing you to view the connectivity information for the selected node. This information is obtained from the **Component Connectivity** page, which follows the **Nodes** page. The component numbers that are assigned the selected node number on the **Component Connectivity** page are listed in the data grid on the **View Connectivity** dialog box.

**NOTE:** You cannot change any information on the **View Connectivity** dialog box. To make changes to the connectivity, access the **Component Connectivity** page.



## Active Loops

Use the check boxes in the **Active Loops** column to specify which nodes are included in which loops in the output and plots. For example, in the following figure, the first three nodes are included in the first loop and the last three nodes are included in the second loop.

Active loops added on the **Options** page are available for selection in the **Active Loops** column on the data grid for each node added on the **Nodes** page.

Use the check boxes in the column to apply active loops to the selected node.

## Component Connectivity page

Use the **Component Connectivity** page to add bearings, seals, and flow elements to the *Transient Flow Network* case. Specify Inlet and exit nodes of the components and the run mode of the component on this page as well.

The following columns are available in the data grid on the **Component Connectivity** page.

**NOTE:** See **Using a grid to enter data** on page 40 for information on entering data using a data grid.

## Inlet Node

Use **Inlet Node** to specify at which node you predict the flow will enter. If the calculation determines the flow is in the opposite direction, a negative flow displays in the results.

## Exit Node 1

Use **Exit Node 1** to specify the first node at which you predict the flow will exit.

## Exit Node 2

Use **Exit Node 2** to specify an additional node at which you predict the flow will exit. The second exit node is primarily for the hydrostatic bearing components.

**NOTE:** You do not need to specify a second exit node.

## Component Category

Use **Component Category** to select the category for the component case that connects the specified nodes.

## Component Type

Use **Component Type** to select the type of case for the category selected that connects the specified nodes.

## Case Name

Use **Case Name** to select the specific component case that connects the specified nodes.

When you click in the box in the **Case Name** column for an analysis case you are selecting, the **Edit Description** dialog box displays allowing you to select an analysis case and to add to the description for that analysis case.

## Run Mode

Use **Run Mode** in network and integrated network analysis cases to specify how the data from the component analysis case is associated with the current analysis case.

The following options are available:

<b>Update</b>	Use the output file from the last run of the component analysis case.  <b>NOTE:</b> If the component analysis case has not been run before (if the output file for the component does not exist), this option works exactly like the <b>Re-Run</b> option.
<b>Store</b>	Store a complete copy of the component analysis case at the network level with the current network analysis case.
<b>Re-Run</b>	Re-run the component analysis case at the component level and uses the updated results.

To update a case stored in the current network analysis case, follow the steps below.

1. Open the network analysis case in which you want to specify the method for updating an analysis case used in that network case.

**NOTE:** The **Run Mode** setting is available on screens or dialog boxes on which you select component analysis cases to connect to the current network analysis case. See the chapters for network analysis case types in **Part II – Analysis case types** on page 127 for more information.

2. Select either **Update** or **Re-Run** from the drop-down menu in the **Run Mode** column for the component analysis case you wish to update.
3. Refer to the following conditions to determine which step or set of steps to follow:
  - c. If you select **Update** as the **Run Mode**, and the component analysis case has not been run before (no output file exists), proceed to step 4.
    - i. Click **Done** on the case manager window for the current network analysis case to save and close the case.

This also deletes the stored component analysis case.

**NOTE:** If the current network refers to a given component as a stored component, the component analysis case at the network level will not be deleted. Make sure you change the **Run Mode** to **Update** or **Re-Run** for all references to the component within the current network analysis case.

- ii. Open the component analysis case you want to update, make any desired changes, and run the case.
- iii. Click **Done** on the case manager window to save and close the component analysis case.
- iv. Open the network analysis case in which you have included the component analysis case.
- v. Select **Store** from the **Run Mode** column for the component analysis case you updated.
- vi. Run the current network analysis case.

The updated case is then stored with the network analysis case.

- d. If you select **Update** and the component analysis case has been run (viewable output exists for the component) or if you select **Re-Run**, do the following:
  - i. Run the current network analysis case.

See **Running analysis cases** on page 49 for more information.

### Output Options page

Use the **Output Options** page to specify settings that affect the calculations and the results in the output and the plots.

### Sample Rate

Use **Sample Rate** to specify how many samples per second to use when calculating results to be plotted. Select a value from the drop-down menu.

## Chapter 24: Integrated Network

### Overview

The *Integrated Network* allows interconnectivity between a *Dynamics Transient Forced Response* network case and a *Transient Flow Network* case. It is also the only way that components can be interactively shared among network cases.

When a network analysis case is run, the components defined within the network analysis case are the only components that can provide data to the network analysis case. If you include a *Dynamics* network analysis case and a *Flow* network analysis case in an *Integrated Network* analysis case, the components within one network can share data with the other network, and the other way around.

### Options page

#### System Composition box

Use the **System Composition** box to select which network types and cases are included in the *Integrated Network* simulation.

- |                           |  |
|---------------------------|--|
| <b>Include?</b>           | Use the check boxes in the <b>Include?</b> column to select whether to include a case for each of the network types in the <i>Integrated Network</i> simulation.   |
| <b>Network Connectors</b> | Use the drop-down menu in the <b>Network Connectors</b> column for each network you select to include to select the specific network case to include as a connector in the <i>Integrated Network</i> simulation. |

#### Simulation Control box

Use the **Simulation Control** box to specify the settings for the network case that will be the simulation controller.

The **Network Type** and **Case** that are specified as the simulation controller are listed.

- |                      |   |
|----------------------|---|
| <b>Solution Type</b> | Use <b>Solution Type</b> to specify whether the simulation will run through once, normally ( <b>Single Pass</b> ) or whether the simulation will be stopped at each time step to calculate curve fit values ( <b>Two-Pass</b> ). When Two-Pass is selected the curve fit values calculated at each time step are fed back into the simulation when it continues to run. |
|----------------------|---|

Currently, **Single-Pass** is the only available option.

- |                         |  |
|-------------------------|--|
| <b>Component Update</b> | Use <b>Component Update Scheme</b> to select the method for updating the |
|-------------------------|--|

---

# RAPPID User's Guide

---

<b>Scheme</b>	<p>components specified as interactive in the included network analysis cases:</p> <ul style="list-style-type: none"><li>• <b>Uniform Spacing</b>  Interactive components are updated over the entire analysis time range for the specified number of times.</li><li>• <b>Non-uniform Spacing</b>  Interactive components are updated over specified time increments within the analysis time range and for the specified number of times per increment.</li></ul> <p><b>NOTE:</b> If you select <b>Non-uniform Spacing</b>, define the time increments and numbers of updates per increment for each component on the <b>Component Settings</b> page.</p>
<b>Number of Component Updates</b>	<p>If you select <b>Uniform Spacing</b> as the <b>Component Update Scheme</b>, use <b>Number of Component Updates</b> to specify the number of times each interactive component is updated during the run of the <i>Integrated Network</i>.</p>

## Analysis Parameter box

Use the **Analysis Parameter** box to specify the **Start** and **Stop** times of the for the simulation and the increment, or **Delta**.

**NOTE:** The **Start**, **Delta**, and **Stop** time values default to the **Start**, **Delta**, and **Stop** time values in the controlling network case listed in the **Simulation Control** box. These values can be changed.

## Interactions page

Use the **Interactions** page to specify which components in each network are interactive and to define the interactions.

When a component is defined as interactive, that indicates it is receiving data from the output of another component either within the same network analysis case or another network analysis case.

## Network Category

Use **Network Category** to select which network type for which you want to specify interactive components.

**NOTE:** Components from both network types can be specified; however, you can only work with one network category at a time. Select one **Network Category** and make your selections for that network analysis case. Then, select the other **Network Category** and make your selections for the corresponding network analysis case. The selections you made for both network analysis cases are saved and used when running the *Integrated Network* simulation case.

## Network Case

Use **Network Case** to select the network analysis case from which you want to select the components that will be interactive.

**NOTE:** Currently, only one **Network Case** from each **Network Category** can be used within an *Integrated Network* simulation case; therefore, there is only one case available in the **Network Case** drop-down menu.

## Interaction Type

Use **Interaction Type** to display a list of the components available for the currently selected network case.

For *Dynamics* networks, there are three types of interactions available:

- **Forcing Function**
- **Interconnection**
- **Rotor Speed**

For *Flow* networks, there is only one type of interaction available:

- **Component**

## Interactive column

For the **Forcing Function** type of interaction, information about the forcing functions available in the selected *Dynamics* network case display in the data grid. Use the check boxes in the **Interactive** column to specify which forcing functions are interactive.

For the **Interconnection** type of interaction, information about the interconnections available in the selected *Dynamics* network case display in the data grid. Use the check boxes in the **Interactive** column to specify which interconnections are interactive.

**NOTE:** *Tabular Input* interconnections cannot be interactive because they produce no results on their own. Their data is only used as input for network analysis cases.

For the **Rotor Speed** type of interaction, the case name for the rotors available in the *Dynamics* network case displays in the data grid. Use the check boxes in the **Interactive** column to specify which rotors are interactive. See *Flow Network* and *Component* columns for information about additional selections needed when the **Rotor Speed** type of interaction is selected.

For the **Component** type of interaction, information about the interconnections available in the selected *Flow* network case display in the data grid. Use the check boxes in the **Interactive** column to specify which components are interactive.

## Define button

When you specify a **Forcing Function**, **Interconnect**, or **Component** as being interactive (see Interactive column), use the **Define** button to define the interaction.

## Interactions dialog box

The **Interactions** dialog box allows you to define the interaction for the currently selected **Forcing Function**, **Interconnect**, or **Component**.

### View button

Use the **View** button to view a summary of the interactions defined on the **Interactions** dialog box for the currently selected **Forcing Function**, **Interconnect**, or **Component**. The interactions defined for both the **Dynamics Network** category and **Internal Flow Network** category display.

### Edit Interactions

The availability of and options available in the drop-down menus in the **Edit Interactions** section of the **Interactions** dialog box depend on the **Network Category** and **Interaction Type** selected on the **Interactions** page.

**NOTE:** For **Interconnects** and **Components**, when you select the same **Network Category** on the **Interactions** dialog box that you selected on the **Interactions** page, the analysis case for which you are defining the interactions is automatically selected on the **Interactions** dialog box as the case from which the data is obtained. In this case, the data is obtained from the output of the case itself and fed back into the same case as input.

Use the following drop-down menus on the **Interactions** dialog box as available to specify the component from which to obtain the data for the **Forcing Function**, **Interconnect**, or **Component** selected on the **Interactions** page.

### Network Category

Use **Network Category** to specify the type of network (**Dynamics** or **Internal Flow**).

**NOTES:** Interactions for both network types can be specified; however, you can only work with one network category at a time. Select one **Network Category** and define the interactions for that network analysis case. Then, select the other **Network Category** and define the interactions for the corresponding network analysis case. The interactions you define for both network analysis cases are saved and used when running the *Integrated Network* simulation case.

The **Network Category** drop-down menu is not

available when **Forcing Function** is selected as the **Interaction Type**. This is because forcing functions are only available in *Dynamics Network* analysis cases, not *Internal Flow Network* analysis cases.

### Network Case Name

The **Network Case Name** drop-down menu is always automatically set to the network case selected in the **System Composition** box on the **Options** page for each network included in the *Integrated Network* simulation case.

### Category

The **Category** drop-down menu is always automatically set to the type of component analysis case for which you are defining the interactions on the **Interactions** page.

### Type

The **Type** drop-down menu is only available when **Forcing Function** is selected as the **Interaction Type**. Select between **1-D Hydrostatic** and **2-D Hydrostatic**.

When **Interconnect** or **Component** is selected as the **Interaction Type**, the **Type** drop-down menu is automatically set to the type of case on the **Interactions** page for which you are defining the interactions.

### Name

If there is only one component analysis case available of the **Type** selected, the **Name** drop-down menu is automatically set. Otherwise, you can select from the available component analysis cases of the selected **Type**.

Once you select the component case from which to obtain the data, select the type of data to obtain from the data grid at the bottom of the **Interactions** dialog box.

Use the check boxes in the **Select** column of the data grid to specify the data to be obtained.



## Flow Network and Component columns

If you specify a rotor as being interactive (see Interactive column), you must use the **Flow Network** and **Component** columns to specify the *0-D Flow Elements Pump* or *0-D Flow Elements Turbine* component from which the rotor speed will be obtained.

**NOTE:** These columns are only available when **Rotor Speed** is the selected **Interaction Type**.

## Components Settings page

The **Component Settings** page applies to the **Interconnect** and **Component** interaction types only. This page allows you to set the time segments within the time range specified in the **Analysis Parameter** box on the **Options** page to be used when the interactive cases are updated in the *Integrated Network* simulation case.

## Network Category

Use **Network Category** to specify the network type for which you want to define the settings for the components.

**NOTE:** Settings for both **Interconnects** in the *Dynamics Network* and for **Components** in the *Internal Flow Network* can be set; however, you can only work with one network category at a time. Select one **Network Category** and define the component settings for that network analysis case. Then, select the other **Network Category** and define the component settings for the corresponding network analysis case. The component settings you define for both networks are saved and used when running the *Integrated Network* simulation case.

## Network Case

Use **Network Case** to select the network case within the selected **Network Category** for which you want to define the component settings.

**NOTE:** Currently, only one **Network Case** from each **Network Category** can be used within an *Integrated Network* simulation case; therefore, there is only one case available in the **Network Case** drop-down menu.

## Update Profiles Library button

Use the **Update Profiles Library** button to setup update profiles containing pre-defined time segments and numbers of updates to be used when the *Integrated Network* simulation case is run.

**NOTE:** The **Update Profiles Library** button is only available when **Non-uniform Spacing** is selected as the **Component Update Scheme** in the **Simulation Control** box on the **Options** page. See **Options page** on page 321.

See **Update Profiles Library dialog box** on page 327 for more information.

## Update Profiles Library dialog box

Access the **Update Profiles Library** using the **Update Profiles Library** button on the **Component Settings** page.

Use the **Insert**, **Copy**, **Move**, **Delete** buttons at the top of the **Update Profiles Library** dialog box to add profiles to the data grid and to manipulate them.

**NOTE:** See **Using a grid to enter data** on page 40 for information on entering data using a data grid.

Use the **Name** column to specify a name for the update profile.

Use the **Define/Apply** button to define the selected profile on the **Define Update Profile** dialog box and apply the update profile to all *Dynamics Network* interconnections and/or all *Flow Network* components, if desired.

Use the **Apply the above profile to all dynamics network interconnections** button and/or the **Apply the above profile to all flow network components** button on the **Define Update Profile** dialog box to apply the profile you defined in the data grid to all interactive interconnects and/or interactive components.

**NOTE:** Applying update profiles to interactive interconnections and/or interactive components on the **Define Update Profile** dialog box affects ALL your interactive interconnections and/or interactive components. If you want to only apply an update profile to certain interactive interconnections and/or interactive components, use the **Define** button on the data grid for the desired interactive interconnect or interactive component (see **Define** button and **Update Profile** box for more information).

## Define button

Use the **Define** button for each interconnect or component listed on the data grid on the **Component Settings** page to define the time segments and initial values to be used when the *Integrated Network* simulation case is run. Clicking the **Define** button displays the **Component Settings** dialog box.

## Component Settings dialog box

Use the **Component Settings** dialog box to define the time segments and initial values to be used when the *Integrated Network* simulation case is run.

### Initial Values

Use the **Initial Values** box on the **Component Settings** dialog box to set the initial values for the component case.

The values to be set include **Loss Factor** and **Heat Input** for interconnects and components and coefficient values for interconnects only. The coefficient values available in the **Initial Values** box depend on whether the interconnect has lateral or axial coefficients:

- Lateral: Kxx, Kxy, Kyx, Kyy, Cxx, Cxy, Cyx, Cyy, Mxx, Mxy, Myx,

Myy

- Axial: Kzz, Czz, Mzz

## Update Profile box

If **Non-uniform Spacing** is selected as the Component Update Scheme in the *Simulation Control* (on page **Error! Bookmark not defined.**) box on the **Options** page, the **Update Profile** box is available.

Use **Update Mode** to select whether to use **Full Transient** or a **Specified Profile** when updating the interactive components.

Select **Full Transient** to update the values at every time step.

If you select **Specified Profile** in the **Update Mode** drop-down menu, the **Define** button and the data grid for entering time segments displays.

There are two methods of specifying time segments to use when updating the selected interconnect or component:

- Specify time segments and numbers of updates in the data grid in the **Update Profile** box.

Use the **Insert**, **Copy**, **Move**, **Delete**, and **Import** buttons above the data grid to add and/or import time segments and to manipulate the time segments.

See **Using fields and grids** in **Part IV – The Analysis case managers** of **Tutorial I – The basics** for information on entering data using a data grid.

If you want to import data into the data grid, see **Importing data from external files** in **Part IV – The Analysis case managers** of **Tutorial 1 – The basics**.

- Link to a pre-defined update profile containing time segments and numbers of updates.

Use the **Define** button to select an update profile you want to use for the selected interconnect or component.

## Linked? and Link Update Profile dialog box

The **Yes** or **No** next to the **Linked?** label indicates whether the selected interconnect or component is currently linked to an update profile. If it is, the profile name displays next to **Linked Profile**.

Use the **Define** button to access the **Link Update Profile** dialog box and link to a pre-defined update profile. See **Update Profiles Library** button for

information about defining Update Profiles.

Use the **View** button to view the details for an update profile listed in the data grid on the **Link Update Profile** dialog box. The settings for the selected update profile display on the **View Update Profile** dialog box.

## Part III – Appendices

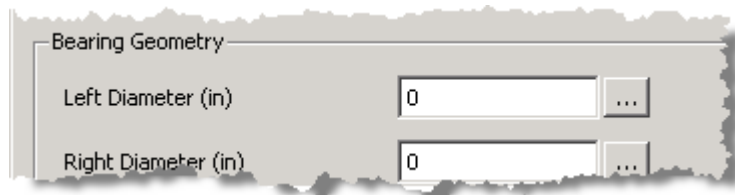
### *Appendix A: Screen elements for entering data*

#### Introduction

There are various screens in RAPPID™ that display and ask you for information. They all work in similar ways. This section describes the different types of elements used for entering data in RAPPID™.

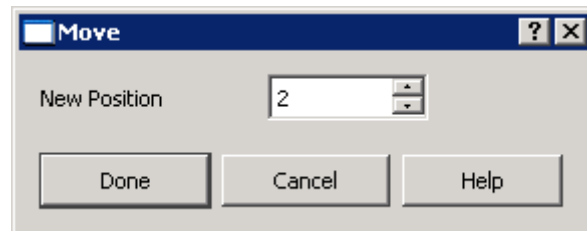
#### Edit boxes

Edit boxes let you type in information like numbers or labels. Just click in an edit box and type in the information.



#### Spin edit boxes

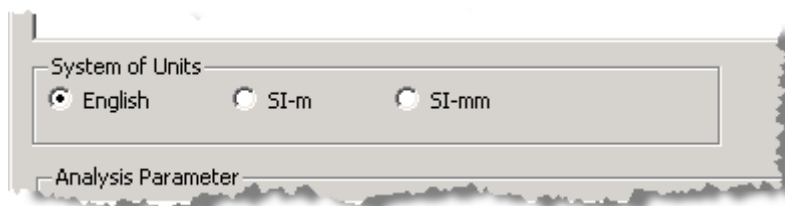
Spin edit boxes are similar to Edit boxes except they have up and down arrow buttons you can use to select the value you want. You can also type in the value like you would in an Edit box.



**NOTE:** You can only enter valid values. Invalid values are not accepted at all.

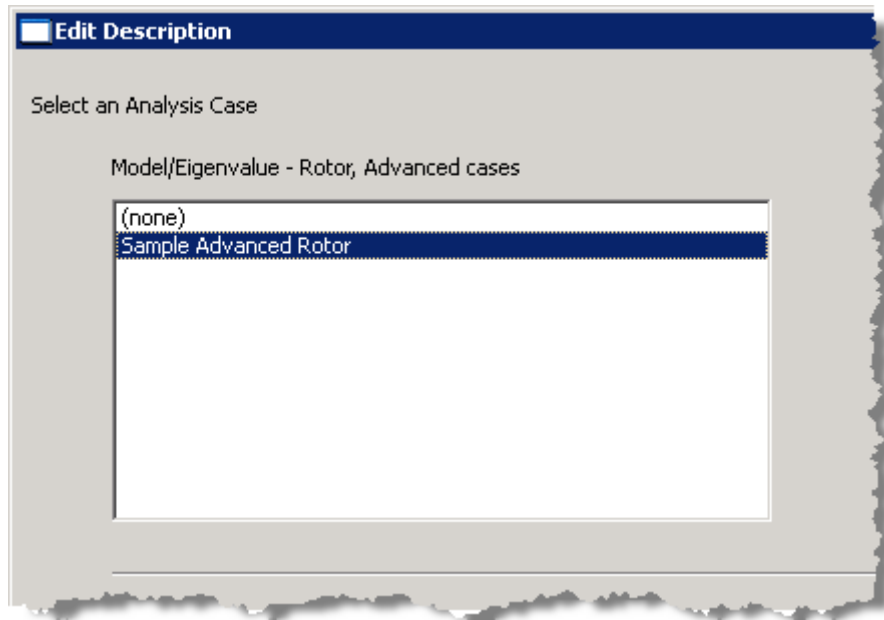
#### Radio buttons

Radio buttons let you select either/or choices. No matter how many radio buttons there are in one box, one and only one can be selected at a time. Click on the button next to the choice you want. Anything that was selected before will automatically be unselected.



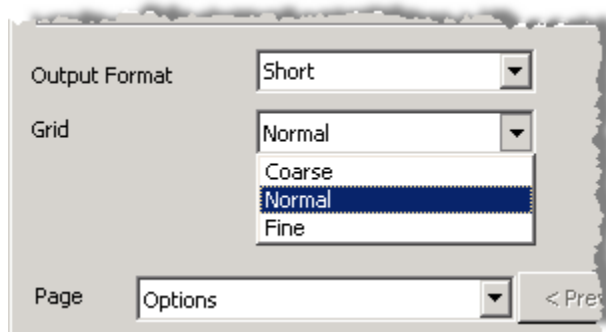
## List boxes

List boxes let you choose one name or item from a list. Click on the name or item you want to select, then click OK, if applicable. The list box may be only part of a screen, in which case you have to make other choices before clicking **OK** or **Done** on the screen.



## Drop-down menus

Drop-down menus contain all the possible choices. Click on the down-arrow to open the menu, then click on the menu item you want.

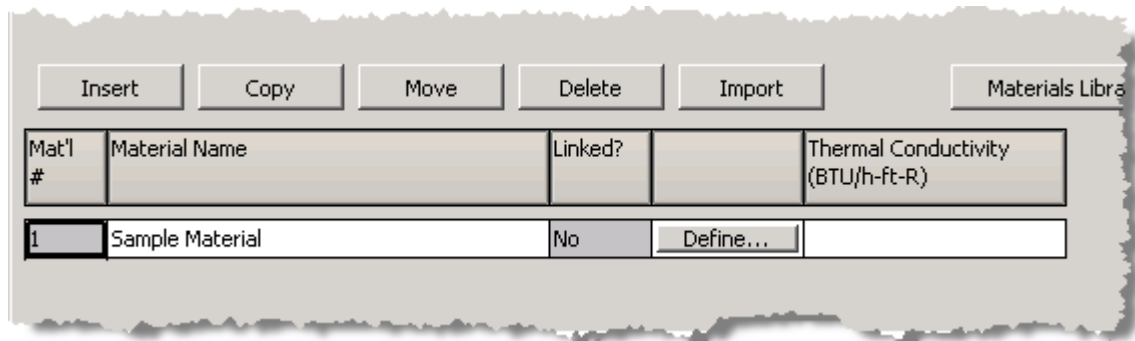


## Data grids

Many cases in RAPPID contain data grids for entering rows of data. These are tables containing cells in rows and columns. All grids generally contain the same capabilities.

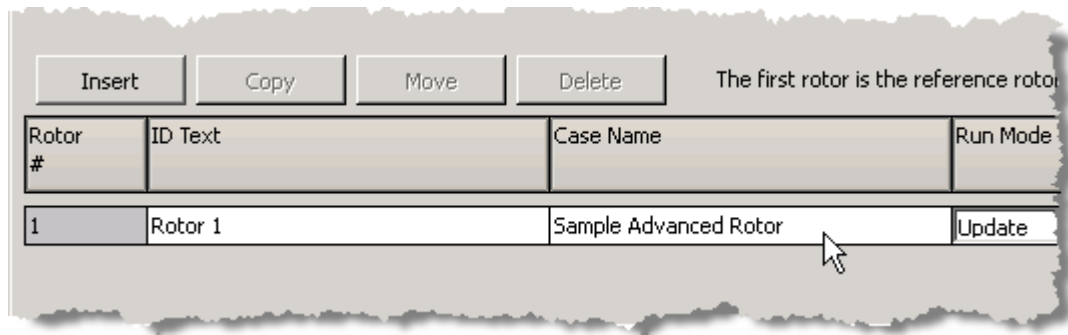
# RAPPID User's Guide

See **Using a grid to enter data** on page 40 for more information.



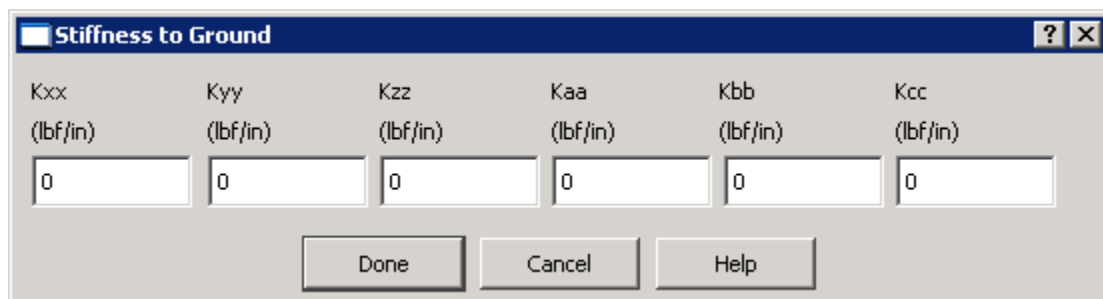
## Button-like cells in data grids

Some data grids have cells that act like buttons. When you click in the cell, a dialog box displays. In the following picture, the **Case Name** column brings up the **Edit Description** dialog box for choosing a *Structural Dynamics Rotor* analysis case.



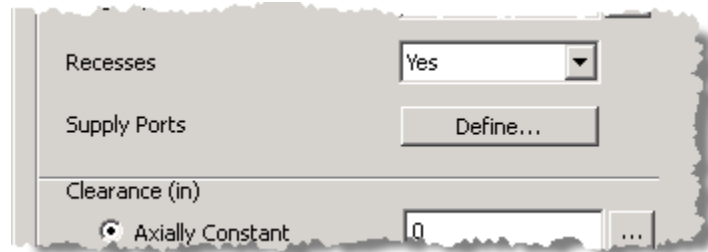
## Cancel and OK/Done buttons

Most screens in RAPPID™ have **Cancel** and **OK/Done** buttons. **OK/Done** accepts and saves your choices or entered information and closes the screen. **Cancel** closes the screen and ignores your choices or entered information.

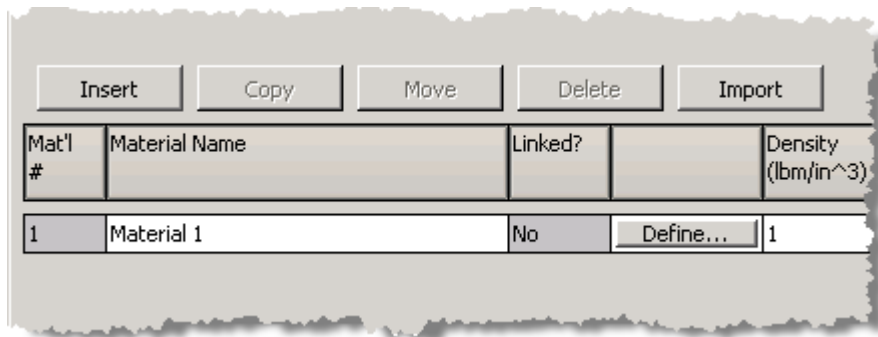


## Other buttons

There are other types of buttons throughout RAPPID™ that open dialog boxes for entering additional data. Examples of other types of buttons include **Define** and **View**, among others.



There may also be buttons in cells in data grids. These buttons operate the same way as buttons not in data grids.



## Appendix B: Troubleshooting

FAQ Questions and Answers on typical problems

## Appendix C: Support information

RSR offers a variety of services to answer your questions about RAPPID™. These services are available in all countries.

- FAX: (916) 660-0222 (24 hours a day)
- Phone: (916) 660-0444 (7 a.m. to 6 p.m. PST)
- Internet: [www.rsr.com](http://www.rsr.com) (24 hours a day)
- E-mail: [rsr@rsr.com](mailto:rsr@rsr.com)

When you call or e-mail RSR please have the following information available:

- Product name and version number
- Brand and model of hardware in your system (computer, printer, etc.)



## **RAPPID User's Guide**

---

- Operating system and version number
- Your network name

If the call concerns a problem, archive the project file and e-mail it to the address listed above.

If the call concerns a problem, send an e-mail (text only) to the address listed above describing the problem, listing the steps to reproduce, and listing any supporting files.

### Glossary

...

### Index

...