ORBIS

Modern Rolling-Element Bearing Analysis Software

USER'S MANUAL

Updated for Version 5.0
Apr-2025

Table of Contents

Table of Figures

Table of Tables

1   Getting Started

1.1   System Requirements

The following <u>minimum</u> system requirements are needed to run ORBIS.

- Windows family operating system
- Display monitor with minimum resolution of 1024 by 768 pixels
- 128 MB of free disk space
- 256 MB RAM
- Available USB port
- Java Runtime Environment 1.8 or greater

1.2   Installation Instructions

Run the installer steps below to complete installation. Administrator rights are needed to complete the installation properly. Note: ORBIS will install to 'All Users' on a given machine.

1. Obtain installation files from developer
2. Double click the ***ORBIS_VXX_Setup.exe*** (where XX will be numeric based on version)
3. Follow installer instructions to complete installation. Note: user has option to define where the application is installed. The default location is C:\\Program Files (x86)\Orbis.
4. For USB dongle type licenses there is no further steps needed. If your license requires an update, copy license file (license.dat) provided by developer to the root installation directory.

1.3   Java Runtime Environment

ORBIS requires the host computer to have a compatible Java Runtime Environment (JRE). The minimum compatibility requirement is Java version 1.8. There are numerous distributions of Java available online, however, only the distributions from Oracle and Amazon Corretto have been tested and verified. Other distributions may or may not work properly. Amazon Corretto, version 8 or greater, is recommended due to its free license terms, certified compatibility with the official Java 1.8, and commitment to long-term support. Amazon Corretto can be downloaded at https://aws.amazon.com/corretto/.

2   General Data Input and Conventions

Proper bearing analysis within ORBIS requires various data inputs and conventions to be followed. This section discusses such matter, and it is highly recommended that the first-time user read through this section carefully.

2.1   Coordinate Systems

ORBIS uses a standard right-handed coordinate system. The user must define the location of the system origin but cannot change its orientation. As shown in the figure below, the x-axis is aligned with the shaft spin axis, with

positive direction pointing rightward on the page, and the positive y-axis is pointing upward on the page. Positive moments/rotations then follow right hand rule along their respective axes.
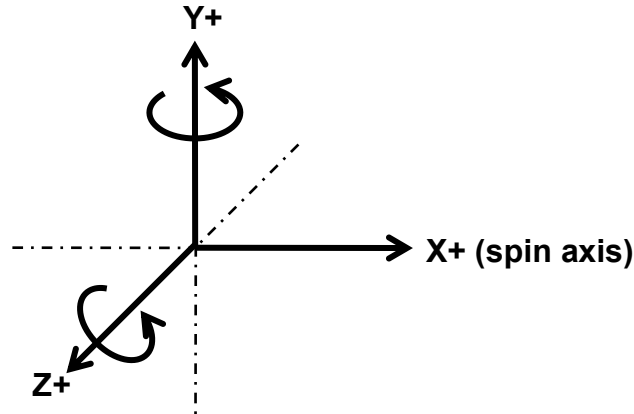
Figure 2-1. Coordinate System

2.2   Numerical Input Formatting

ORBIS accepts different numerical inputs but there are a few that are not allowed. In general, the U.S. convention (i.e. decimal point) is allowed while the European convention (comma) is not. The table below shows examples of valid and invalid numeric input formatting.

Table 2.2-1. Numeric Input Formatting Examples

Number to input	Valid Inputs	Invalid Inputs	Description
One thousand	1000, +1000, 1e3, 1.00e3, 1.0e+3, 1.0E3	1,000, 1.0e 3, 1.0x10^3	• Plus signs are optional for positive values • Scientific notation uses the "e" or "E" to specify powers of 10 • Comma notation is not allowed • Spaces anywhere in input string are not allowed
One thousandth	0.001, .001, 1.0e-3, 1E-3		• Leading and trailing zeros are optional

2.3   Input Field Expressions/Calculations

Most input fields provide a built-in calculator that can interpret basic math functions. To use this functionality the user simply types the desired equation into the input field, hits the [Enter] key, and ORBIS automatically returns the answer back to the same input field. Note: the [Enter] key must be used to tell ORBIS to parse the input field for expressions.

As an example, suppose we are defining a new bearing and want the pitch diameter to be halfway between the bearing I.D. (1.375 dia.) and O.D. (1.875 dia.). To do this with the built-in calculator type (1.375+1.875)/2 in the pitch diameter field and hit the [Enter] key. Orbis will return the result of 1.625 to the field. Note the parentheses is needed here to properly sum the I.D. and O.D. before dividing by 2.

Allowable mathematical operators are shown in the table below.

Halpin
Engineering,
LLC

Phone: (310) 650-8982
Email: info@HalpinEngineeringLLC.com
Web: http://www.HalpinEngineeringLLC.com

Table 2.3-1. Input Field Math Operators

Operator	Mathematical Function
+	Addition
-	Subtraction
*	Multiplication
/	Division
()	Parentheses are used to group values and control order of operations
^	Power or exponent

2.4   Unit Conversions

Most ORBIS input fields contain a built-in unit converter, allowing the user to specify inputs in various units and have them automatically converted to the proper system units. To perform a unit conversion, the user must append the unit abbreviation within square brackets to their input value. Note: this functionality may also be used in conjunction with an input expression.

The following example illustrates a conversion of 100 Newtons to the default ORBIS load units (lbf). Note that a space between the number and the square bracket is optional.

100 [N] [Enter]

Available unit abbreviations are shown in the table below. In general, both uppercase and lowercase options exist for all available units.

Table 2.4-1. Unit Conversion Syntax

Unit Type	Abbreviation	Unit	ORBIS Default Unit
Force	[N], [n]	Newton force	Pound Force
	[GF], [gf]	Gram force	
	[OZF], [ozf]	Ounce force	
Length	[MM], [mm]	Millimeter	Inch
	[CM], [cm]	Centimeter	
	[M], [m]	Meter	
	[FT], [ft]	Foot	
Pressure	[PA], [pa]	Pascal	Pound Per Square Inch
	[MPA], [mpa]	Megapascal	
	[GPA], [gpa]	Gigapascal	
Torque	[NM], [nm]	Newton*meter	Inch Pound Force
	[NCM], [ncm]	Newton*centimeter	
	[NMM], [nmm]	Newton*millimeter	
	[FTLBF],[ftlbf]	Foot*pound-force	
	[INOZF], [inozf]	Inch*ounce-force	
	[GFCM], [gfcm]	Gram-force*millimeter	
	[GFMM], [gfmm]	Gram-force*millimeter	
Angle	[RAD], [rad]	Radian	Degrees
Rotational Velocity	[RPS], [rps]	Radians per second	Rotation Per Minute
	[DEGPS], [degps]	Degrees per second	

3   User Interfaces

This section describes the graphical interfaces within ORBIS.

3.1   Main Graphical User Interface (GUI)

The main graphical interface (see Figure 3-1) is used to configure a system of bearings, apply any external loading or temperatures, and submit the setup for analysis. As depicted in Figure 3-2, the main window is organized to group key input types for easy access. Each of these regions are discussed in subsequent sections.

Halpin Engineering, LLC

Figure 3-1. Main Graphical Interface
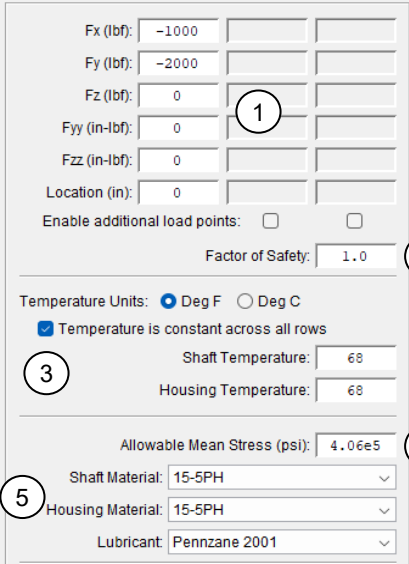
Figure 3-2. Main Window Input Categories

As shown in Figure 3-2, the main window contains the following regions:

- **System Inputs** – located in the upper left region, is where external loading, temperatures, shaft/housing materials and lubricants are defined. Refer to section 3.1.1 for details.
- **Dynamic Analysis** – located in the lower left, is where parameters such as velocities, fatigue life (reliability and life factor), viscous torque factor, rotational member and load fixity are defined. Refer to section 3.1.2 for details.
- **System Display** – located in the upper right, provides an engineering sketch based on the user defined system. Refer to section 3.1.3 for details.
- **Bearing Row Inputs** – located in the lower right, is where parameters for defining the configuration of each bearing row in the system. Refer to section 3.1.4 for details.
- **Dashboard** – located bottom center, provides key details and helpful information for each input field, and contains the 'Analyze' button for job submittal. Refer to section 3.1.5 for details.

3.1.1   System Inputs

The System Inputs region contains inputs for external loading, bulk temperature definitions, and assignment of housing/shaft/lubricant materials. See the following figure for descriptions of each input field.
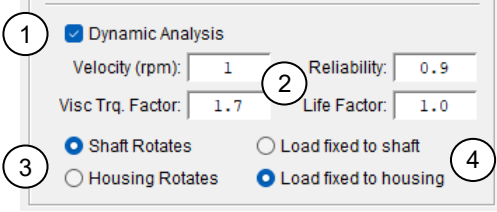
#	Title	Description
1	External Loads	External loading fields have the following features: • Full 5 dof load component specification (Fx, Fy, Fz, Fyy, and Fzz follow system coordinate definition) • Load location is the x-axis coordinate of the load • Up to 3 simultaneous load points can be defined by selecting the check boxes.
2	Factor of Safety	The factor of safety is a linear scale factor that is multiplied by each load component prior to solving the system equilibrium.
3	Temperature Assignment	Radio button toggles between Fahrenheit and Celsius units. Checkbox specifies if temperature should be constant across all rows. If not, temperature values are assigned separately for each bearing row (see section 3.1.4).
4	Allowable Mean Hertzian Stress (psi)	Allowable stress input sets threshold at which any elements with contact stress above the specified allowable will be highlighted in the output file.
5	System Materials	System materials for the shaft, housing, and lubricant are assigned via drop-down selections. Available selections are based on user database definitions (see section 3.2 for discussion on databases).

Figure 3-3. System Inputs

3.1.2   Dynamic Analysis Inputs

ORBIS has two different analysis modes: static and dynamic. Static mode means the system is at rest (i.e. zero speed on the bearings). This mode is useful for applications where dynamic effects are negligible. The static solver is the most robust and quickest due to the reduction in equilibrium equations. Dynamic analysis mode provides full analysis output parameters (such as torque, fatigue life, film parameters, centrifugal and gyroscopic forces, etcetera). See the following figure for descriptions of the inputs and controls in this region.

#	Title	Description
1	Dynamic Analysis Checkbox	Selecting this checkbox activates the dynamic analysis mode and enables the inputs below. Default is un-checked.
2	Dynamic Inputs	Dynamic input fields consist of the following: • **Velocity** - Defines the rotational velocity, in RPM, of the rotational member • **Reliability** – Defines the reliability for fatigue life calculations. Valid inputs are between 0 and 1 exclusive. • **Viscous Torque Factor** – Factor to adjust lubricant torque resistance. Default is 1.7, which is a reasonable initial guess for an oil lubricated bearing (no oil bath or jet conditions). This factor is per the Palmgren torque model (reference section 6). • **Life Factor** – Generic fatigue adjustment factor. This factor should combine all fatigue adjustments except for lubricant and reliability (i.e., material factors, cleanliness, etc).
3	Rotational Member	Radio button selection for rotational member
4	Load Fixity	Radio button selection for relative fixity of external loading

Figure 3-4. Dynamic Analysis Inputs

3.1.3   System Display

The system display area provides a sketch of the defined bearing system. This sketch takes key user input entries, such as bearing locations, load locations, bearing contact angle orientations, and generates an engineering layout sketch of the user's setup. Review of the sketch prior to analysis submittal is advised as this provides a quick and easy means to check that the system of bearings is configured as intended by the user. Furthermore, the system sketch is included in the results window as a figure to provide a complete analysis report. See the following figure for a detailed description of the information provided in the system display panel.

#	Title	Description
1	Bearing Labels	Bearing name is shown above the assigned row. Note: long bearing names will get truncated when the window is small.
2	Spring Icon	Rows with a preload spring will be identified with a spring icon adjacent to the row. ORBIS will place spring in correct load path based on contact angle direction.
3	Bearing Sketch	Bearing sketch depicts angular contact, 4-Point, and cylindrical roller type bearings. For angular contact type bearings, the contact angle direction is also sketched based on convergent/divergent orientations.
4	Entity Labels	Load points identified with 'L' prefix and bearing rows identified with 'R' prefix
5	Dimensioning Region	Ordinate dimensions for all defined entities shown in bottom of sketch region.
6	Coordinate Frame	System coordinate frame
7	Sketch Button	Sketch button will refresh the system display
8	Database Buttons	Bearing button opens the Bearing Database dialog window (see section 3.2.1) Materials button opens the Material Database dialog window (see section 3.2.2) Lubricant button opens the Lubricant Database dialog window (see section 3.2.3)

Figure 3-5. System Display

3.1.4   Bearing Row Inputs

The Bearing Row Input area consists of tabbed window panels for each bearing row in the system. See the following figure for a detailed description of each input.

A common error for new users is improper sign convention on the 'Row Preload' field. Row preload direction, or sign, is related to the orientation of the contact angle. Contact angles are defined using convergent and divergent terminology (see section 3.1.4.2 for details). These terms relate to whether the contact angle line of action converges or diverges toward the spin axis while traversing along the x-axis in the positive direction (rightward along the spin axis). For example, the leftmost bearing in a duplex pair of bearings, configured in a back-to-back (DB) orientation, would have a divergent contact angle, while the rightmost bearing has a convergent contact angle. In this example, the preload sign of the leftmost row would be positive, and the sign of the rightmost row would be negative. Another way to visualize preload force is to consider the inner ring and determine which direction the force acts on the ring.

The term preload generally implies force balance or equilibrium is held at the bearing system, hence, to achieve this the sum of all row preload values should be zero. However, ORBIS does not enforce or check that the sum of all preload values equals zero. In the case where there is a net preload other than zero, when the system equilibrium

solver completes, the shaft will have moved axially until equilibrium is found (assuming the solver can find equilibrium).

#	Title	Description
1	No. Bearing Rows	Drop-down selection allows up to 5 bearing rows on a common shaft. Row tabs will be activated based on number of bearing rows selected here.
2	Row Tabs	Selecting an active tab allows the user to define parameters for that row.
3	Bearing	Button opens the bearing selection window for assignment of the bearing in this row position. See section 3.1.4.1 for the bearing selection window.
4	Contact Angle	Radio buttons specify contact angle orientation. A divergent contact angle extends away from, or diverges, from the spin axis as you traverse in the positive x-axis.
5	Preload Condition	Radio buttons specify condition at which the row preload (see #6 below) is defined. Un-mounted means rings are radially free at the specified preload. Mounted means the specified preload includes boundary constraints such as fitup and clamping effects.
6	Row Preload (lbf)	Specifies the preload force applied to the active bearing row. Preload forces are directional and must include the appropriate sign convention. For standard preloading specify a positive preload for divergent contact angles and a negative preload for convergent contact angles.
7	Preload Type	Radio buttons for either rigid or spring type preload. Rigid preloading activates input fields for inner and outer ring clamping forces. Spring preloading activates inputs for the spring rate and snubber gap. • Clamp forces effect ring diameters per Poisson's theories. • Snubber Gap specifies the gap from the preloaded position to a spring hard stop ('snubber'). The gap is always specified as a positive value. ○ **CAUTION**: Applying a zero gap will lock the bearing at the preload condition
8	Add Spacer	Selection of this checkbox places a spacer between the active row and the next row. The drop-down selection will activate to specify the material for the spacer.
9	Row Inputs	The following inputs are entered here: • **Row Location (in):** Sets the x-axis position of the active row • **I.R. Fitup (in):** Sets the inner ring fitup (diametral). Fitup is defined as the difference in the shaft O.D. to the free bearing I.D. A positive value indicates interference fits. • **O.R. Fitup (in):** Sets the outer ring fitup (diametral). Fitup is defined as the difference in the free bearing O.D. and the housing I.D. A positive value indicates interference fits. • **Shaft I.D. (in):** Specifies the I.D. of the shaft at the bearing location. For a solid shaft, input a zero value. This parameter is used to determine the hoop stiffness of the shaft. • **Housing O.D. (in):** Specifies the O.D. of the housing at the bearing location. This parameter is used to determine the hoop stiffness of the housing. • **Shaft Temp:** Used to specify unique row-wise shaft temperature. Units follow definition from radio button discussed in section 3.1.1. • **Housing Temp:** Used to specify unique row-wise housing temperature. Units follow definition from radio button discussed in section 3.1.1. • **Coeff. of Ball Friction:** Specifies the rolling contact friction coefficient for the bearing row.

Figure 3-6. Bearing Row Inputs

3.1.4.1 Bearing Selection Dialog

The bearing selection dialog is used to assign a specific bearing object from the user database to a row in the system. Refer to the figure below for a full description of the dialog.

#	Title	Description
1	Select Bearing Type	Drop-down menu specifies type of bearing to use. Options are Angular Contact/Radial, 4PT Gothic or Cylindrical Roller.
2	Bearing Database List	This list shows all defined bearings in the user's database for the selected bearing type. List items are selectable with the mouse button.
3	Parameter Summary	Once a bearing is selected (left mouse click) from the Bearing Database List, the parameters of the bearing are shown here.
4	Cancel Button	Pressing 'Cancel' exits the window without selecting anything and the window is closed automatically.
5	Select Button	Pressing the 'Select' button assigns the selected bearing to the row in the main ORBIS window.

Figure 3-7. Bearing Selection Dialog

3.1.4.2 Contact Angle Orientation

Contact angle orientation is assigned within the bearing row input region and terminology for convergent and divergent is further illustrated in the figure below.

Halpin Engineering, LLC

Divergent Contact Angle Orientation

Convergent Contact Angle Orientation

X+

X+
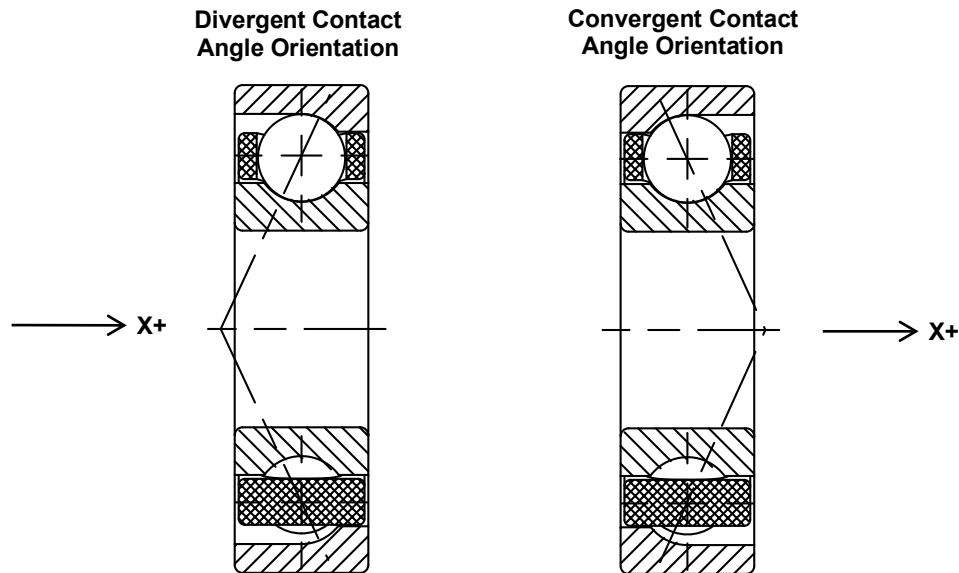
Figure 3-8. Convergent and Divergent Contact Angles

3.1.5 Dashboard

The dashboard area (lower portion of the main window) provides helpful user information and is also where completed setups are submitted for analysis. The following figure illustrates the various components of the dashboard area.

#	Title	Description
1	Input Field Description	When the user places their cursor in one of the input fields within main window (done by clicking the mouse in a field or pressing the tab button to advance to the next input field) a description about that field appears in the Input Field description bar.
2	Exit Button	Select this button to safely exit. User will be prompted to save any changes that have occurred to the active setup.
3	Analyze Button	The analyze button is used to submit the user setup for final analysis. Upon completion of an analysis, the Results window (reference section 3.3) will automatically display.
4	Display Console Checkbox	Selecting this checkbox will display the console window. See section 3.1.5.1 for further description of the console.

Figure 3-9. Dashboard Area

3.1.5.1   Console Window

The console window, which is normally hidden, is used to display solver messages. Information, such as solver progress, is automatically output to this window. Selecting the <clear> button at the bottom of the console window will erase all current contents of the console, otherwise text within the console is persistent and will remain available during the entire session.

Common information displayed to the console window includes:

- Convergence criteria (see section 4.2)

- Norm of the residuals for each solver attempt

- Element equilibrium cannot be achieved

- Row expansions due to thermal loads exceed initial preload

- Number of iterations analyzed and if truncation is found when running a Worst-Case Tolerance analysis

- Various step convergence information when running a Flexible Shaft analysis

- Confirmation of convergence
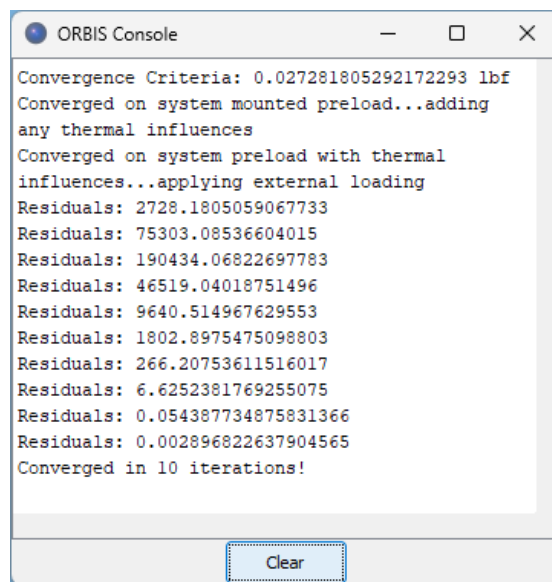
Figure 3-10. Console Window

3.1.6   User Menus

User menus are available within the main graphical interface. See the following figure for a detailed description of the available menu options.
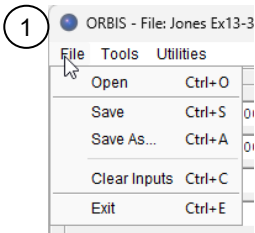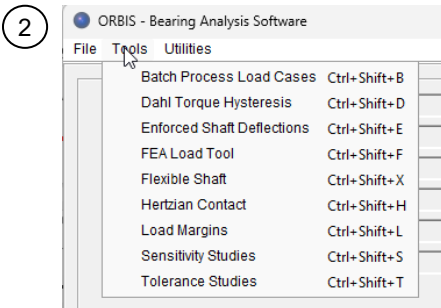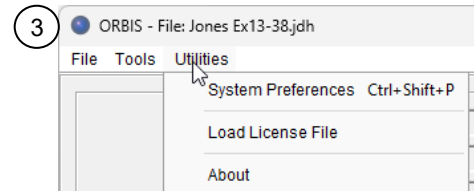
#	Title	Description
1	File Menu	The file menu provides the following options: • **Open:** This option generates a file browser for loading an existing ORBIS analysis file (*.jdh). • **Save:** Used to save all settings in the main window to a file for later use. • **Save As:** Used to save an existing file to a different name. • **Clear Inputs:** This option clears all the inputs in the main window and resets any default values. • **Exit:** Exits the application (user will be prompted to save if there are non-saved changes pending).
2	Tools Menu	This menu provides advanced analysis tools shown below: • **Batch Process Load Cases:** Launches the Batch Process tool. See section 3.1.6.1. • **Dahl Torque Hysteresis:** Launches the Dhal Torque Hysteresis window. See section 3.1.6.2. • **Enforced Shaft Deflections:** Launches the Enforced Shaft Deflections tool. See section 3.1.6.3. • **FEA Load Tool:** Launches the FEA Load Tool window. See separate FEA Load Tool User Manual. • **Flexible Shaft:** Launches the Shaft Flexibility Analysis dialog. See section 3.1.6.4. • **Hertzian Contact:** Launches the Hertzian Contact tool. See section 3.1.6.6. • **Load Margins:** Launches the Load Margins tool. See section 3.1.6.7. • **Sensitivity Studies:** Launches the Sensitivity Studies window. See section 3.1.6.8. • **Tolerance Studies:** Launches the Tolerance Analysis window. See section 3.1.6.9.
3	Utilities Menu	This menu provides the following utilities: • **System Preferences:** Selecting this utility launches the System Preferences window. See section 3.1.6.10. • **Load License File:** Utility to load a new license file. See section 3.1.6.11. • **About:** This menu provides an 'About' window to show key parameters regarding the current software version and lease expiration date.

Figure 3-11. User Menus

3.1.6.1 Batch Process Load Cases

This tool is used to automatically analyze a list of predefined load cases from a comma delimited text file (*.csv). When run, each load case defined in the external file is copied into Load Point #1 and analyzed. Note that any load points specified in Load #2 or #3 will remain in effect throughout all batched cases. The batch process user interface is shown in Figure 3-13 below.

How to use the tool:

1. Start with a fully defined bearing system in the main window. It is recommended to verify the system is properly setup by running one load case manually from the main window and reviewing the results.

2. Setup load cases via an external *.csv file. To setup an external *.csv file refer to the steps defined in Figure 3-12.

3. Open the Batch Process tool: Tools Menu → Batch Process Load Cases

4. Load cases from step 2 are entered into the tool by selecting the [Browse] button at the top of the tool window. An informational dialog will remind that a *.csv file is required, select [OK] to proceed. Next, a file browser dialog will prompt to select the *.csv file. Locate and select the *.csv file and press the [Open] button. Each defined load case in the file will be populated into the 'Batch Load Cases' table on the top of the tool.

5. Specify a factor of safety in the Factor of Safety input field. Each load case will be multiplied by this factor prior to analyzing.

6. Specify run options by selecting the desired checkboxes from the 'Run Options' region. To save a result file for each load case, select the first checkbox and either accept the default filename prefix or type a new prefix. All files will be saved to the same directory where the *.csv file is located. The second run option determines if a warning dialog is presented to the user when any of the load cases cannot be solved. Unchecking this checkbox results in no warnings presented to the user.

7. Select the [Analyze] button to start processing the results. The progress bar at the bottom of the window will display progress. Once all cases have been analyzed, the 'Results Summary' table will be updated with the specified summary parameters from each load case. If a load case could not be solved, the results for each parameter of that load case will be filled with zeros.

8. To see a standard analysis results window for a particular load case, select the desired row from the Results Summary table and then select the [Show Selected Case Results] button.

9. To save the Results Summary table to a text file, select the [Save Result Summary] button.

STEP 1:
Create external file with load cases to be analyzed. File must be formatted as follows (see table and corresponding example with Microsoft Excel®):
- Each row contains a unique load case
- Columns contain load components and load location as shown in table below

	Fx	Fy	Fz	Fyy	Fzz	Location
Case 1→	100	500	0	0	1000	1.5
Case 2 →	200	350	0	0	980	1.5
...	300	275	0	0	880	1.5
Case n →	125	600	0	0	1200	1.5

STEP 2:
Save file as a .csv format (recommend using the MS-DOS CSV version to save the file)

Figure 3-12 Batch Processing Load Cases

Figure 3-13 Batch Process Load Cases Tool

3.1.6.2 Dahl Torque Hysteresis

This tool is used to analyze the torsional stiffness (torque versus angle) of the bearing system during startup or direction reversal. This phenomenon occurs through small finite angles of rotation, often most apparent when direction of rotation is reversed, at speeds sufficiently slow such that viscous drag is negligible. The utility provides quick inspection of the reversing torque slope and steady state torque. Additionally, the tool can quickly generate small angle hysteresis loops for both graphical plot inspection and data export. The analysis procedures of this utility follow those set forth by Todd and Johnson (1986). The Dahl torque hysteresis tool is shown in Figure 3-14.

How to use the tool:

1. Ensure a bearing system is fully defined in the main window. It is recommended to verify the system is setup by running one load case manually from the main window and reviewing the results.

2. Open the Dahl Torque Hysteresis tool by going to the Tools Menu → Dahl Torque Hysteresis

3. Select desired units for both torque and angle from the dropdown selections

4. Optionally edit the rolling contact coefficient of friction for each row defined in the system

5. Select the [Generate Dahl Parameters] button to get the torque stiffness and steady state Coulomb torque values. These results will have the units specified in step 3, where torque stiffness has units of torque/angle and Coulomb has units of torque.

6. To plot torque loops, perform the following steps

   a. Specify the number of solutions points per toque loop.

   b. Enter a list of increasing reversal angles in the Reversal Angles table. A good start would be 0.1° increments up to about 1.0°. Do not enter zero for any of the points. The values entered will represent a zero-peak value and the plot loops will be full peak-to-peak cycles.

   c. Optionally select the [Export Plot Data] checkbox to have solved torque loop data saved to a text file.

   d. Select the [Plot Torque Loops] button to generate the plot.
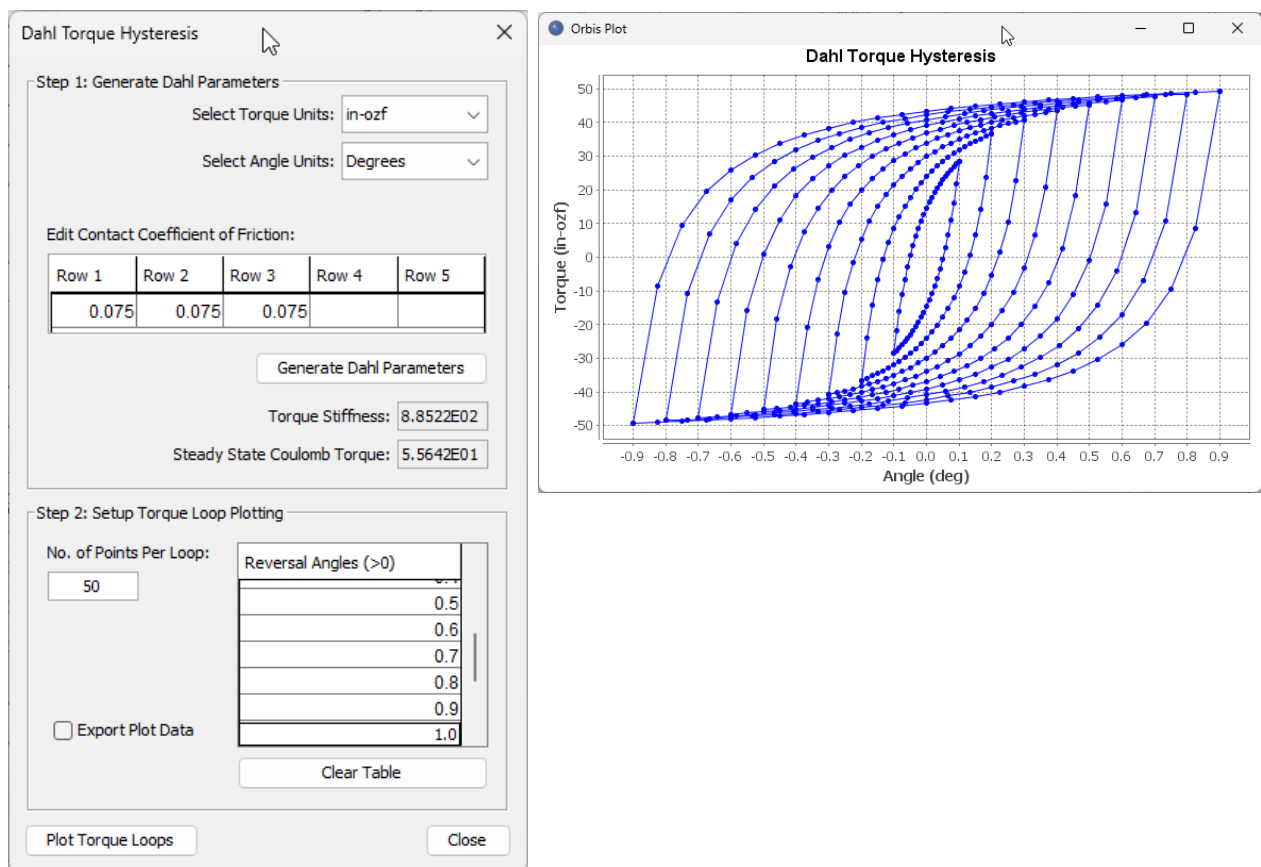
Figure 3-14 Dahl Torque Hysteresis Tool

3.1.6.3 Enforced Shaft Deflections

This utility enforces a prescribed deflection on the shaft and generates a full set of results. Specified deflections are relative to the mounted and preloaded state. Any defined external loading will be ignored. The figure below shows the Enforced Shaft Deflections dialog.

How to use the tool:

1. Ensure a bearing system is fully defined in the main window. It is recommended to verify the system is setup by running one load case manually from the main window and reviewing the results.

2. Open the Enforced Shaft Deflections tool by going to the Tools Menu → Enforced Shaft Deflections

3. Complete the fields to defined shaft deflections at a specific location. All fields must be filled out (use zeros to specify no deflection).

4. Select the [Analyze] button to run the analysis. A standard Results Window will appear with the deflected state results.
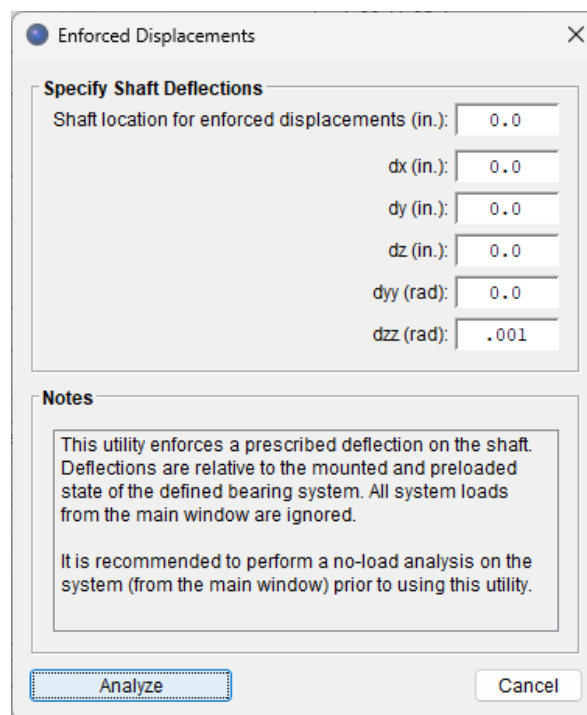
Figure 3-15 Enforced Shaft Defections Tool

3.1.6.4 FEA Load Tool

The FEA Load Tool is a finite element modelling environment that can model larger structure and resolve bearing reaction forces from common structural loading environments, such as static point loads, quasi-static gravitational loads, and random vibrations. The tool interacts with existing bearing setup files in two distinct ways: first, it can pull bearing stiffnesses properties, which are used to define spring elements in the finite element model, and second, recovered bearing loads from the finite element model can be directly analyzed with the bearing setup files.

The tool employs matrix methods to express real continuous structure with a discrete mathematical model; where elastic and inertial properties of basic structural elements are developed into local stiffness and mass matrices. These local matrices then become building blocks, which, when assembled in accordance with the theories of elasticity, represent the static and dynamic properties of the continuous structure. Linear modal dynamics are used to solve random vibration problems, and the displacement method is used to solve static problems.

	Phone:	(310) 650-8982
	Email:	info@HalpinEngineeringLLC.com
	Web:	http://www.HalpinEngineeringLLC.com

Halpin
Engineering,
LLC

A top-level workflow for the FEA Load Tool is as follows:

1. Bearing setups are completed prior to using the FEA Load Tool. These setups represent the bearing configurations in their final assembled state (i.e. mounted and preloaded), which should be the state in which loading will occur. If the mechanism under consideration has multiple bearing axes, a setup file for each axis should be completed. All bearing setups must be saved (i.e. *.jdh files created).
2. The FEA Load Tool is used to construct a larger structural model, which will include spring elements to represent the bearings. Stiffness properties for the bearing springs are imported from the bearing setup files, which run with the standard ORBIS solver to compute the tangential stiffness of the non-linear bearings.
3. The structural model will define load cases, such as a random vibration base motion, that are then solved. Each load case will contain the bearing spring reaction forces.
4. The bearing reaction forces are then transferred back to the bearing setups for detailed bearing analysis (i.e. Hertzian contact stresses).

The above process, which uses the FEA Load Tool and the non-linear bearing solver, is integrated to promote an efficient workflow. Furthermore, the two processes operate in separate threads, thereby allowing the user to switch between them at any time without losing work from either application.

The remainder of this section steps through the FEA Load tool interfaces and wraps up with a step-by-step example problem.

How to use the tool:

1. Ensure a bearing system is fully defined in the main window. It is recommended to verify the system is setup by running one load case manually from the main window and reviewing the results.

Halpin
Engineering,
LLC

Phone: (310) 650-8982
Email: info@HalpinEngineeringLLC.com
Web: http://www.HalpinEngineeringLLC.com
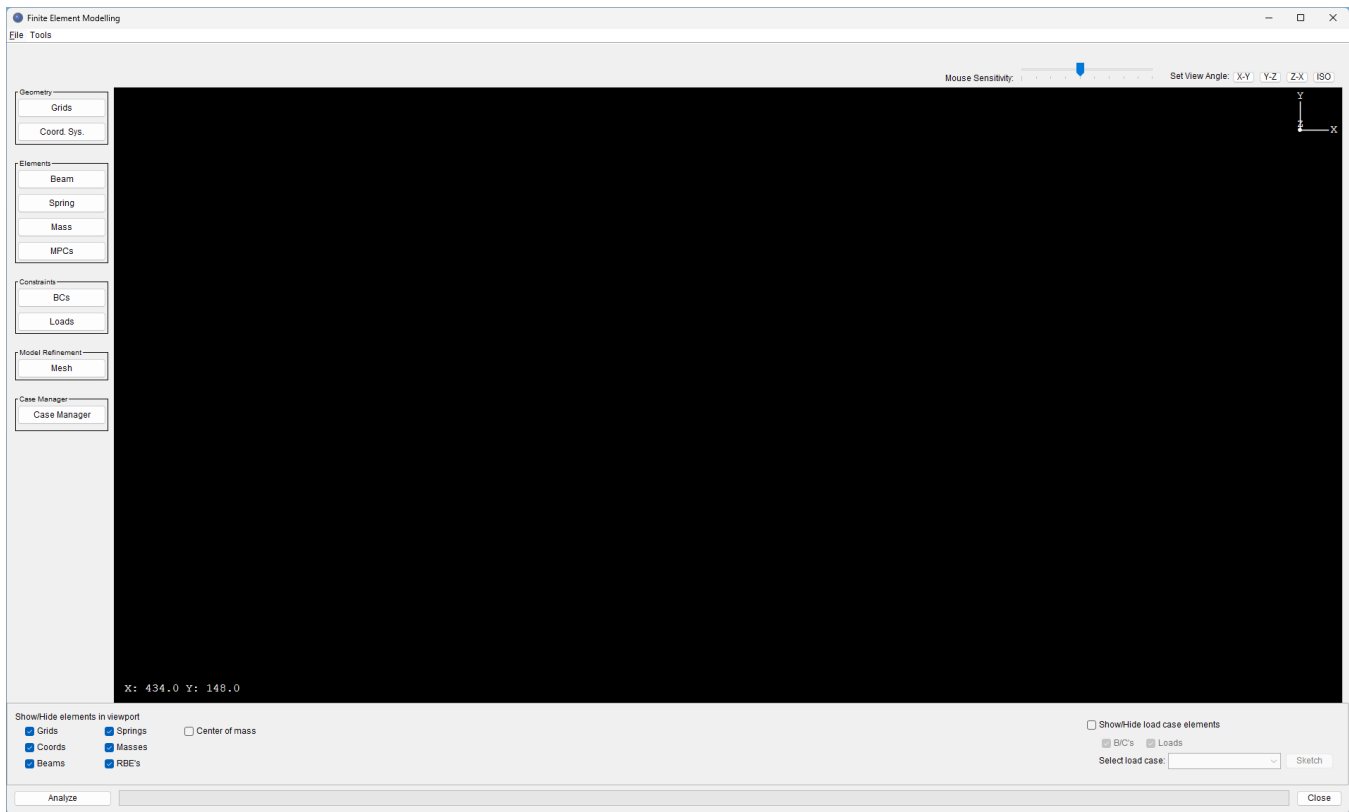
Figure 3-16 FEA Load Tool

3.1.6.5 Flexible Shaft

The Flexible Shaft tool is used to account for elastic compliance of the bearing shaft. ORBIS models the shaft with classical Timoshenko beam element formulations that account for both bending and shear deflections in the shaft. The interface allows shaft definition of up to 25 unique circular beam elements; each of which may be defined with unique section dimensions and/or materials. See figure below for descriptions of the flexible shaft window.

If the initially defined rigid system has constant section dimensions, determined by whether all defined bearing I.D.'s and shaft I.D.'s are constant, ORBIS will prepopulate one shaft element within the Flexible Shaft window that extends through the complete system. The user may override this assumption by editing the table of shaft elements. For cases where the bearing I.D.'s or shaft I.D.'s are not constant throughout the system, no prepopulated elements are provided and the user will need to define shaft elements that extend through all bearing and load locations.

How to use the tool:

1. Ensure a bearing system is fully defined in the main window. It is recommended to verify the system is setup by running one load case manually from the main window and reviewing the results.

2. Open the Flexible Shaft tool by going to the Tools Menu → Flexible Shaft

3. Specify the shaft elements by one of the methods below:

a. *New Analysis Definition*: Each row of the table defines a beam element that may have unique geometry and material. The entire span of the system, which includes all defined bearing rows and load points, must have connected beam elements defined. Start by defining X-Start, X-End, I.D., O.D., and Material fields in the first row of the table. Each subsequent row in the table must have an X-Start entry that is equivalent to the prior element X-End. The materials column will have a dropdown selection of the material database. If a new material is needed, select the [Edit Materials] button to open the material database editor. Once the new material is added to the database and the material database editor is closed, the Flexible Shaft dialog will be returned, and any/all work is preserved.

b. *Re-use Existing Analysis*: Select the [Load Setup] button and browse to the file containing the desired shaft information (this would come from a prior ORBIS Flexible Shaft analysis that saved a setup). The shaft element table should now display all the element settings defined in the file.

4. Sketch the setup by selecting the [Sketch] button and review the sketch.

5. The Solver Convergence region of the dialog allows user inputs for the max step error and max number of solution attempts. It is recommended to start with the defaults initially and adjust as needed after reviewing initial results. Smaller step errors produce a more precise solution but require more steps to solve. At the completion of a flexible shaft analysis, ORBIS displays a dialog stating the convergence metric. This metric is equivalent to the norm of the residual equilibrium load component errors. Use this information to determine if the max step error should be adjusted.

6. The output options region has two checkbox options: [Show Rigid Analysis Results] and [Plot Shaft Deflections]. Selecting the first option produces a separate results window containing the solution when the shaft is rigid. Selecting the plot shaft deflections checkbox produces five separate plots containing deflections of the shaft at each node defined in your system. A node is placed at every X-Start and X-End data point from the shaft element definition. To make smoother plots add more shaft elements to the shaft definition table.

7. Select the [Analyze Flexible Shaft] button to submit the analysis. Results will be automatically displayed.

8. The [Save Setup] button allows the defined shaft elements to be saved to a file for later reuse.
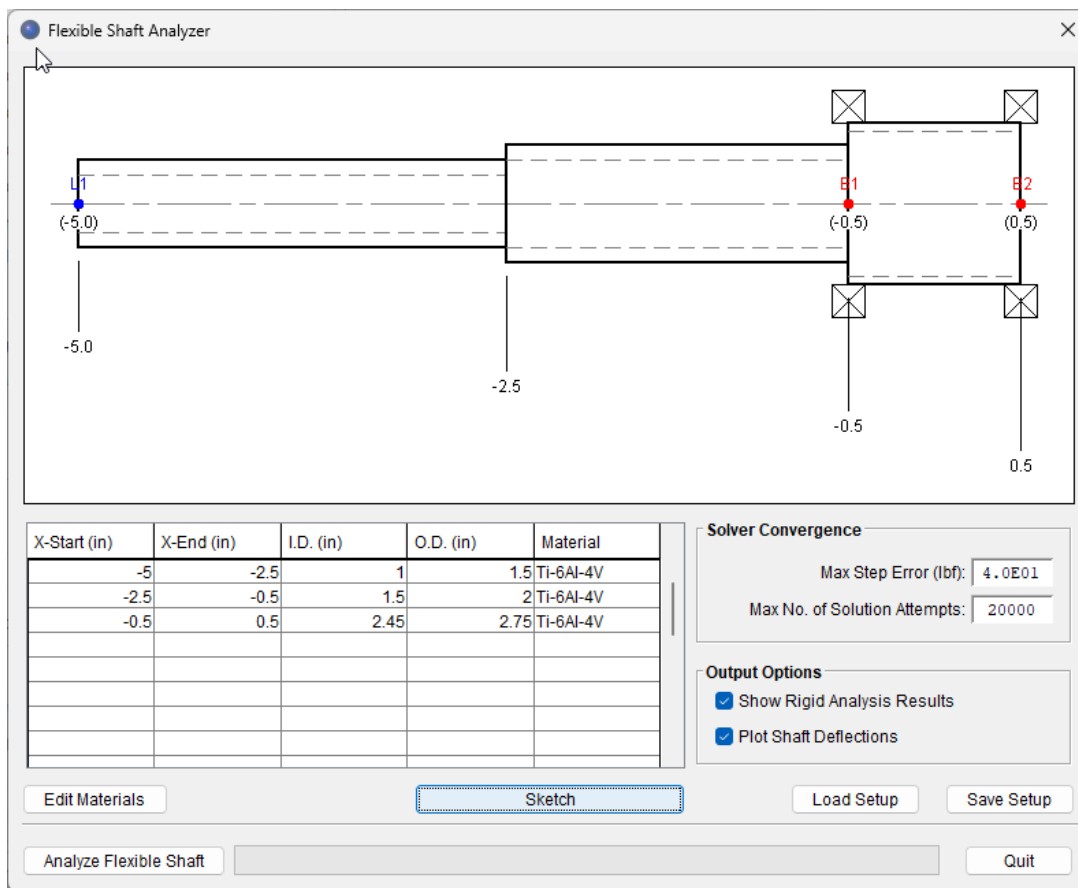
Figure 3-17 Flexible Shaft Tool

3.1.6.6   Hertzian Contact

The Hertzian Contact tool is a general engineering tool to solve classical Hertzian point, and line, contact problems. This tool will solve any classical two body Hertzian contact problem. Materials are assigned from the main material database and results are provided in a formatted text window for printing and exporting to a text file.

How to use the tool:

1. Specify the type of contact problem to be solved by selecting the appropriate Point Contact or Line Contact radio button.

2. Enter the normal load at the contact.

3. Define the geometry at the contact for body 'A' and body 'B' by completing radii of curvature fields for both principal planes in each body definition region. All radii must be entered as a positive value. If the radius is internal or concave for a particular principal plane, select the appropriate checkbox. Likewise, if the radius is infinite, or flat, select the appropriate [Flat] checkbox.

4. Assign materials for body 'A' and 'B' by selecting from the dropdown combo boxes. To define a new material to be used select the [Edit Materials] button to launch the material database editor.

5. Select the [Show Diagram] button to update the figure in the diagram region. If a valid set of inputs have been entered, the diagram will update to show the user defined contact problem. If the entered geometry is invalid, an error dialog will be thrown with information regarding what was found to be invalid. Note the

Halpin
Engineering,
LLC

Phone: (310) 650-8982
Email: info@HalpinEngineeringLLC.com
Web: http://www.HalpinEngineeringLLC.com

diagrams are only illustrative and do not graphically display proportionalities of the geometry entered (e.g. a body with two convex radii will be illustrated with a sphere regardless of values entered for the two principal radii).

6. Select the [Analyze] button to submit the analysis to the solver and see results. Use the scroll bar to see the entire set of results within the text window.

7. To save an analysis, select the [Save Results] button. This saves all text in the results window to a text file.

8. Select the [Print Results] button to bring up a printer dialog and submit the results to a printer.

9. The [Close] button closes the Hertzian Contact tool and returns to the main ORBIS interface.
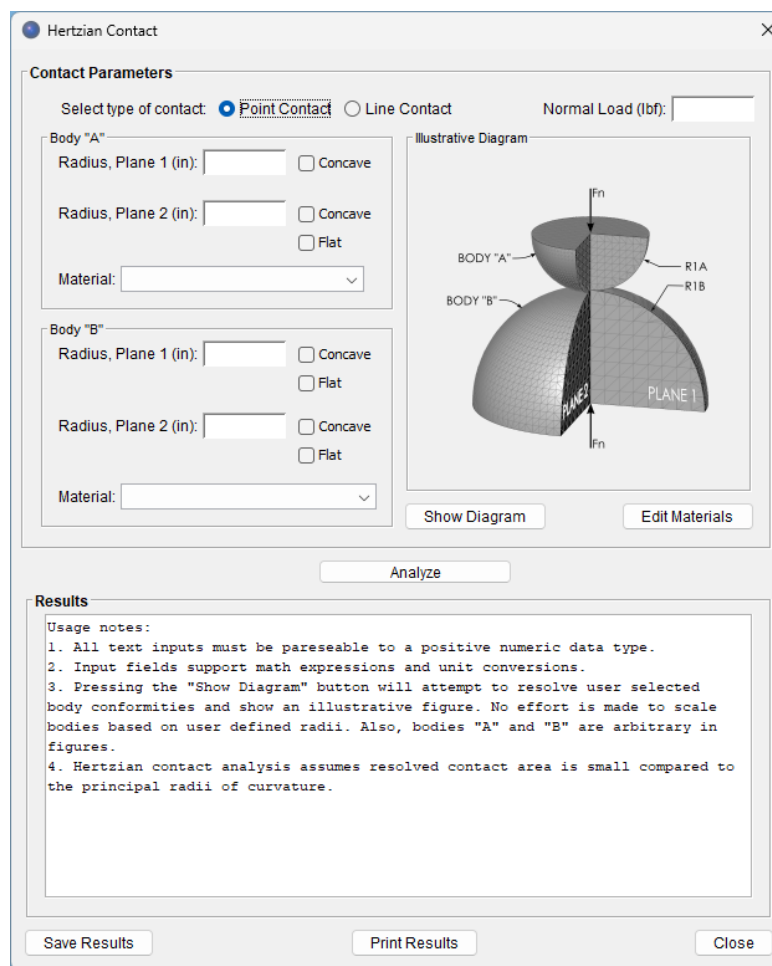
Figure 3-18. Hertzian Contact Tool

3.1.6.7 Load Margins

The Load Margins tool (see Figure 3-19) solves the margin of safety against specified design limit loads. Here, the user specifies the design limit load, the factor of safety, what bearing limit criteria to enforce, and the tool then computes the resulting margin of safety. Note the relationship between load and stress on a bearing is non-linear.

Limit criteria options include an allowable maximum mean stress and/or limits on allowable percent truncation. All limit criteria apply to the entire system of bearings and every contact within the system is assessed against these criteria. The reported margin will be based on whichever limit criteria is achieved first. The factor of safety is directly applied to the design limit load prior to computing the margin of safety.

How to use the tool:

1. Ensure a bearing system is fully defined in the main window. It is recommended to verify the system is setup by running one load case manually from the main window and reviewing the results.

2. Open the Load Margin tool by going to the Tools Menu → Load Margins

3. Update the design limit load components as necessary

4. Update the factor of safety as necessary

5. Specify margin options:

    a. If no truncation is to be allowed, leave the checkbox for [Don't allow truncation] selected. If this checkbox is un-checked, specify a percent truncated length allowed. The percent should be entered in decimal format (i.e., 10% allowable truncation is specified as 0.1).

    b. Specify the allowable Mean Stress.

    c. To see a Results Window with the scaled result (at zero margin), select the [Display Results Window] checkbox.

6. Select the [Analyze] button to generate the margin of safety results. These results will be shown in the text window at the bottom of the tool. Since either stress or truncation can be limiting, the results delineate which limit criteria the margin is based on, shown as either "TRUNCATION" or "MEAN STRESS". When limited by truncation, the max mean stress at the load to cause truncation is provided for reference. In all truncated case results, the displayed mean stresses do not include any adjustments for truncation (i.e. results are based on non-truncated Hertzian Contact assumptions).

Figure 3-19 Load Margin Tool

3.1.6.8 Sensitivity Studies

The Sensitivity Studies tool (see figure below) is used to study the relationship between any independent and dependent parameter. The user can select from different input parameters, specify a range of values for that parameter, and plot them against an output parameter. Multiple output parameters may be selected at a time, thereby producing separate plots for each output parameter. In addition to generating plots, the user can export all raw solution data for post processing. Exported data is saved in a delimited text file that can be easily imported into programs such as Microsoft Excel®.

How to use the tool:

1. Ensure a bearing system is fully defined in the main window and is set to Dynamic mode. Prior to opening the Sensitivity Studies tool, it is recommended to verify the system is setup properly by running from the main window and reviewing results.

2. Open the Sensitivity Studies tool by going to the Tools Menu → Sensitivity Studies

3. Select the input, or independent, parameter to be studied by locating the parameter and selecting its radio button. Note the input parameter section has multiple tabs to group various parameters (see figure below). If the input parameter is from the [Loading] tab, there is an additional dropdown menu to specify which load point the selected parameter will apply to.

4. Complete the limits and step size to be applied to the input parameter. The [Vary From] field is the starting value of the input parameter and the [To] field is the ending value of the input parameter. The [No. Steps] defines how many equal spaced points to divide the starting and ending values by, which also determines actual solution points to be solved. Certain input variables may be applied to specific bearing rows or all rows via the 'Select Row #' dropdown. The 'Ref Value' shows the value from the system setup for the specified input parameter and is a non-editable reference.

5. Select the desired output row from the [Select Output Row] dropdown in the output parameters section of the dialog.

6. Select each output parameter checkbox corresponding to desired dependent parameters to be studied. A separate plot will be generated for each output selected. When selecting Element Output options there are two further options to setup: the [Criteria] and [Raceway]. The criteria option determines if the minimum, mean or maximum values is to be plotted, while the raceway option specifies which raceway is considered. The [All] option for the raceway considers both the inner and outer raceways. For example, to plot maximum stress for the bearing row select [Maximum] criteria and [All] for the raceway options.

7. To save solution data to a file select the [Save raw data to file] checkbox.

8. Select the [Plot Results] button to run the analysis and generate plots of the chosen parameters. The generated plots have interactive functionality. See the next section for a description of these.

Figure 3-20 Sensitivity Studies Tool

3.1.6.8.1 Sensitivity Plot Windows

The plot windows generated from a sensitivity study are interactive. See the following figure for a description of user options within the plot windows.

#	Title	Description
1	Plot Title	The title contains the selected output variable with any applicable element criteria (min, mean, max) and raceway applicability (I.R./O.R. or both)
2	Curve	Data is solution points are indicated with solid squares and connected with a linear curve. Mouse hovering over a datapoint will show solution point values.
3	Plot Options Dialog	A right mouse button click anywhere in the plot window brings up a context menu with options to modify plot properties, copy to clipboard, save plot as an image, print, and zoom/scale axes.
4	Dependent Parameter	All plots have the dependent parameter along the ordinate. Appropriate units and applicable row will be specified.
5	Independent Parameter	All plots have the independent parameter along the abscissa. Appropriate units and row will be specified.
6	Mouse Zoom - Start	Mouse zooming (rectangular) is achieved by positioning the mouse at a desired upper left-hand corner and dragging the mouse (with left button held) to the lower right-hand corner.
7	Mouse Zoom - End	To complete zoom, release the left mouse button with the mouse pointer at the desired lower right corner of the zoom rectangle.

Figure 3-21. Plot Windows

3.1.6.9 Tolerance Studies

The Tolerance Studies tool performs tolerance studies of key bearing parameters within the user setup. The utility iteratively solves all permutations of specified min/max tolerance conditions and returns a result window with the combination causing maximum Hertzian contact stress and/or maximum percent truncation (if truncation exists). Reference the following figure for the Tolerance Studies tool.

If a tolerance analysis has a case that cannot be solved, ORBIS will display an error dialog with a couple user options. Options include abort the tolerance analysis, continue, and continue silently. Both continue options tell the solver to continue processing any remaining tolerance permutations, however, the continue silently option will no longer show an error dialog if additional non-convergence cases are found. Once all tolerance permutations have been processed, and if there were non-convergence cases, ORBIS will display a dialog showing how many cases solved and how many did not solve.

3.1.6.9.1 How to use the Tolerance Studies tool

Follow these steps to setup and run a tolerances study:

1. Ensure a bearing system is fully defined in the main window. It is recommended to verify the system is setup by running one load case manually from the main window and reviewing the results.

2. Open the Tolerance Studies tool by going to the Tools Menu → Tolerance Studies

3. In the 'Tolerances' section there are tabbed panes for specifying preload tolerances and key tolerances for each row in the setup. Preload tolerances are defined as follows:

    a. Specify the min/max value of the system preload as positive values. For systems with two rows, the preload is the same for all rows and no further consideration is needed. However, for systems with more than two rows, there could be multiple different preload values. For these cases, specify the preload tolerance for the most heavily preloaded row.

    b. Specify the row preload distribution factors. These factors must be between positive and negative one [-1,1], inclusive, and should be signed according to the preload force direction along the x-axis. Essentially, the preload factor sign should match the sign of the preload defined in the main window. If a row does not have preload, set the distribution factor to zero. ORBIS will recognize standard two row systems with preloading and populate the preload factors automatically. The user can choose to accept them or override them.

4. Select each row tab in the 'Tolerances' input region and enter the pertinent min/max tolerance values. The tolerance fields in the row tabs will activate/deactivate certain fields depending on the type of bearing in that row. Radial and angular contact type bearings have all field options and either the free contact angle or free radial play can be defined. 4-Point type bearing inputs are similar but do not allow free contact angle inputs (only the arched radial play can be specified for the internal clearance tolerance). Cylindrical rollers do not require free contact angle or raceway curvature tolerances.

5. Specify whether to vary load component sign in the Permutation Options. If [vary load component sign] is selected, all positive/negative permutations of each load component in load point #1 will be included.

6. Select desired 'Result Options.' The [Find max mean stress condition] checkbox will return the combination of tolerances that result in the highest mean stress. This stress condition does not consider truncation. The [Find max % truncation condition] will return the combination of tolerances that produce the highest percent truncation. If both options are checked, and truncation exists, then two results will be displayed once the solver completes all cases.

Figure 3-22 Tolerance Analysis Tool

3.1.6.10 System Preferences

System preferences settings allow control of the database location, importing external database entries, setting solver convergence criteria, and setting up a network license host. These preferences are persistent; meaning they remain in effect each time the user launches and runs ORBIS. Options available are shown in the figure below.

Halpin Engineering, LLC

#	Title	Description
1	Tabbed Pane Selection	Each system preference category is accessed by selecting the appropriate tab.
2	Database Location	This area contains the folder location to the default database files. The browse button brings up a folder selection dialog that allows the user to specify this location.
3	Select Database Type	To perform a database import, or view contents of a database, the user must select which type of database to be displayed. Once the type is specified, the two scroll panes (see #6 & #7) populate with the appropriate entry names.
4	Load Remote Database	Select this button to bring up a folder selection dialog. Within the folder selection dialog, the user will navigate to a desired database. Note: this button is only active once a database type is specified.
5	Remote Database Entries	Displays entries from remote database. Single entry selection is achieved by mouse selection (left click) on desired entry. Multiple entry selection is achieved by two methods: for consecutive entry selection select the first desired entry and shift+click the last consecutive entry, for non-consecutive entry selection select each entry while holding the control key.

6	Import button	Selecting this button imports the selected remote database entries into the default database. The database does not allow duplicate entries and will check each import entry for existing names within the default database. If duplicates are found the user will be asked to confirm overwriting or skip the duplicate import.
7	Default Database Entries	Displays the entries from the user's default database.
8	System Force Balance Error	This entry specifies the allowable system force balance error for solver convergence (Norm of the residuals). Acceptable values are between zero and one exclusive [0<value<1]. The system force balance convergence criteria will be determined by multiplying this value by the RSS of the applied external loading. In the case where the RSS of the external loading is less than one this value will be used directly. For example, if the RSS of the external loading is 1,000 lbf and the system force balance error is 1.0E-5 (default value), then the system will converge to equilibrium within 0.01 lbf.
9	Max Solver Iterations	Specifies the maximum number of numerical solver iterations before aborting.
10	Max Preload Error	This entry specifies the maximum allowable force error for determining the preloaded state. Acceptable values are greater than zero [0<value].
11	Max Preload Iterations	Specifies the maximum number of numerical solver iterations on preload convergence before aborting.
12	Max Internal Clearance Error	Specifies the maximum allowable internal clearance error for convergence, in inches. Acceptable values are greater than zero [0<value]. During mounting and preloading routines, the solver will converge on each bearing row's change in internal clearance within this value.
13	Reset Defaults	Select this button to restore all solver options to their default values.
14	Select License Type	Radio buttons allow license type to be either USB Dongle (default) or Network. When Network is selected the follow items below become active inputs
15	Server Host Name	Text field to specify the name of the server. The server must be running the Orbis Server software and be located on the same Local Area Network (LAN) as the client.
16	Port Number	This is the dedicated listening port opened by the server. This number will be defined during the server initialization.
17	Test Connection Button	Once the Host Name and Port Number are specified, pressing this button will cause ORBIS to attempt to connect with the server. Results are displayed in the text window (see 18 below)
18	Text Window/Console	This window will display server connection information once the Test Connection button is pressed.
19	Clear Console	Pressing this button simply clears any existing text in the Text Window/Console.

Figure 3-23. System Preferences Dialog

3.1.6.11 Load License File

This menu option is used to update the user license for USB dongle-based licenses only. This function should only need to be performed once per USB key. Prior to running, the user should have a valid license file supplied by the developer. Launching this utility will show a dialog per the following figure. To properly update the user license, the license file must be placed in the ORBIS installation directory, and the USB dongle must be inserted into the computer before pressing the [Proceed] button on the dialog. Note: the dialog shows the path where the license file should be located.
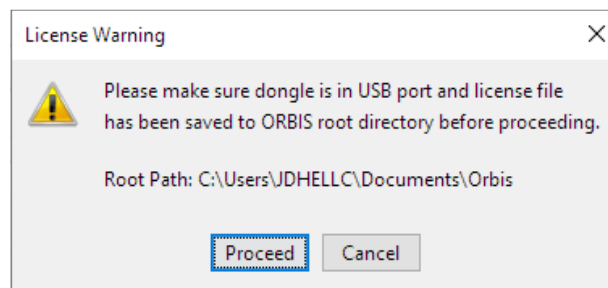
Figure 3-24. Load License File Dialog

3.1.6.12 About

This dialog is purely informational and shows the user's license expiration date, the product version, and the dongle serial number.

3.2   Database Editors

ORBIS requires bearing, material, and lubricant databases to define many input parameters needed to perform a bearing analysis. Once database entries are defined, they can be applied to an analysis via drop-down menus within the pertinent user interface.

Database editors allow the user to view, modify, add, or delete database definitions. The database editors are accessed from the buttons within the System Display area (see section 3.1.3).

Database definitions are stored in five specific files: Bearings.dat, Bearings4PT.dat, CylindricalRoller.dat, Materials.dat and Lubricants_V2.dat. These files are typically located within the ORBIS installation directory. However, database files can be placed anywhere, such as a shared network drive. Storing database files on a network drive is commonly used when there are multiple users, as it allows all users to work with consistent data. See the System Preferences dialog (section 3.1.6.10) for details on how to import entries from external databases or change the default location of the database files.

Saved analysis setup files depend upon database entries being available upon next use. For example, suppose a user creates a material definition titled "440C" and then defines a bearing named "XYZ Bearing" that uses the 440C material for the inner/outer rings and balls. Every time the user wants to run an analysis with XYZ Bearings, ORBIS will search the material database for a valid definition named "440C." If this name is not found, ORBIS will generate an error message. This same situation occurs for all database entries and any user saved analysis setup files.
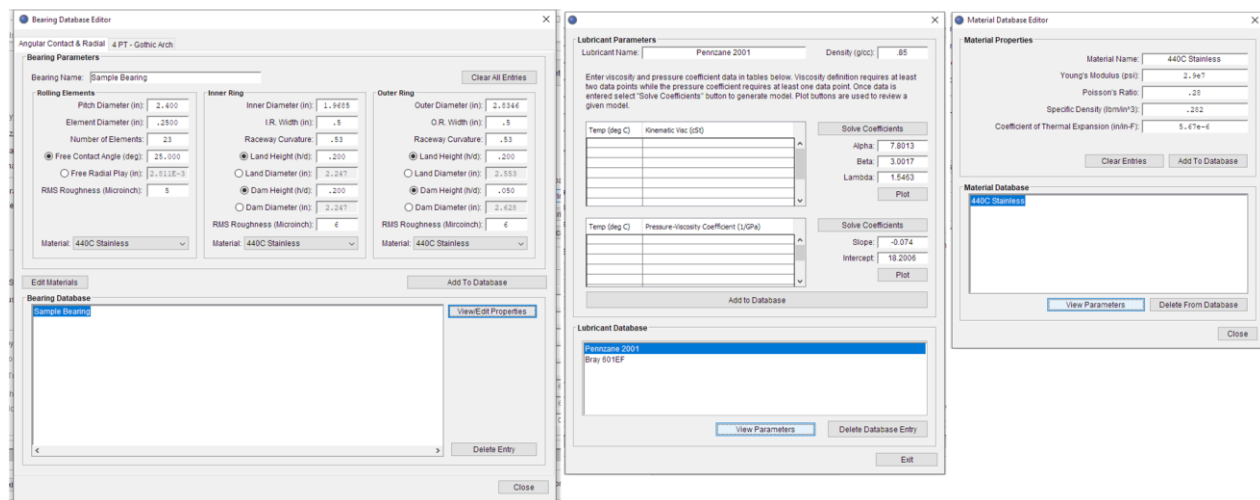
Figure 3-25 Database Editors

3.2.1   Bearing Database Editor

The Bearing Database Editor, shown in the figure below, is used to add, edit, view, or delete bearing database objects. A bearing object consists of required parameters to numerically model a bearing. Tabs on the editor window allow all supported bearing types to be edited. See the following subsections for descriptions of required parameters for each bearing type.

3.2.1.1   Common Bearing Database Editor Usage

The following lists the most common database editor usage scenarios and provides step-by-step instructions.

1. Add a new bearing to the database:

   a. Select the tab corresponding to the type of bearing to be defined.

   b. Specify a unique name for the bearing. The name is important since it is used to refer to the bearing object throughout ORBIS. Although there is little restriction on the number of characters for the name input field, it is recommended to keep it short. ORBIS will use the name in the system sketch figure and in the results report, however, these usages have limits to how many characters can be shown, so long names will get clipped or truncated.

   c. Complete all input fields for the Rolling Elements, Inner Ring and Outer Ring input sections. For inputs with a radio button next to them, such as the Free Contact Angle and Radial Play inputs in the Angular Contact & Radial type bearing tab, only one of the two inputs is needed to define the geometry.

   d. Assign materials to the rolling elements, inner ring and outer ring via the dropdown material selections. If a material is not defined in the material database, select the [Edit Materials] button to open the Material Database Editor. Once the new material is added and the Material Database Editor is closed, the Bearing Database Editor will resume.

   e. Select the [Add To Database] button. The new bearing name will show up in the database region on the bottom of the editor window.

2. Edit an existing bearing in the database:

   a. Select the tab corresponding to the type of bearing to be edited.

   b. In the Database section, locate and select the bearing to be edited (select by left mouse button clicking on the name of the bearing).

   c. Select the [View/Edit Properties] button. This will populate all the parameters for the selected bearing into the Bearing Parameters section of the editor.

   d. Edit the desired parameters.

   e. Select the [Add To Database] button. Unless the name is changed, a warning will be shown stating the bearing name already exists. Select [Yes] to overwrite the existing database entry.

#	Title	Description
1	Bearing Type Tabs	Bearing type options include Angular Contact/Radial or 4PT Gothic Arch. See section 3.2.1.2 for a description of the Angular Contact/Radial type bearing.
2	Clear All Entries Button	Clears all current input field entries. This does NOT clear the database entries.
3	Bearing Parameters	The bearing parameters contains all inputs required to defined the selected bearing type. See following figures for details on each bearing type.
4	Edit Materials Button	Opens the Material Database Editor. Once the material database editor is closed the user will be brought back to the Bearing Database Editor.
5	Add To Database Button	Commits the specified input entries into the database under the bearing name specified. A warning will occur if the specified bearing name already exists within the database.
6	Database Entry	This area shows listing of all defined database entries. Each bearing type has it's own database.
7	View/Edit Properties Button	Once a database entry is selected within the database window, selecting this button will populate the bearing parameters into the input fields.
8	Delete Entry Button	With a database entry selected, this button will permanently delete the entry from the database.
9	Close Button	Closes the Bearing Database Editor.

Figure 3-26. Bearing Database Editor, General Functions

3.2.1.2   Angular Contact and Radial Bearing Parameters

Figure 3-27 illustrates terminology for an angular contact bearing. The radial/deep groove/Conrad bearing shares all these parameters but typically the free contact angle is replaced by the parameter for radial play. ORBIS allows either value as an input.

Additional parameters, not shown in Figure 3-27, required to define angular contact and/or radial bearings include surface roughness and curvatures. Surface roughness must be defined for the balls and inner/outer raceways and is specified in units of micro-inch RMS. Inner and outer raceway curvatures must also be defined. This is a unitless ratio of the cross-raceway radius to the ball diameter. This ratio should be greater than 0.5.
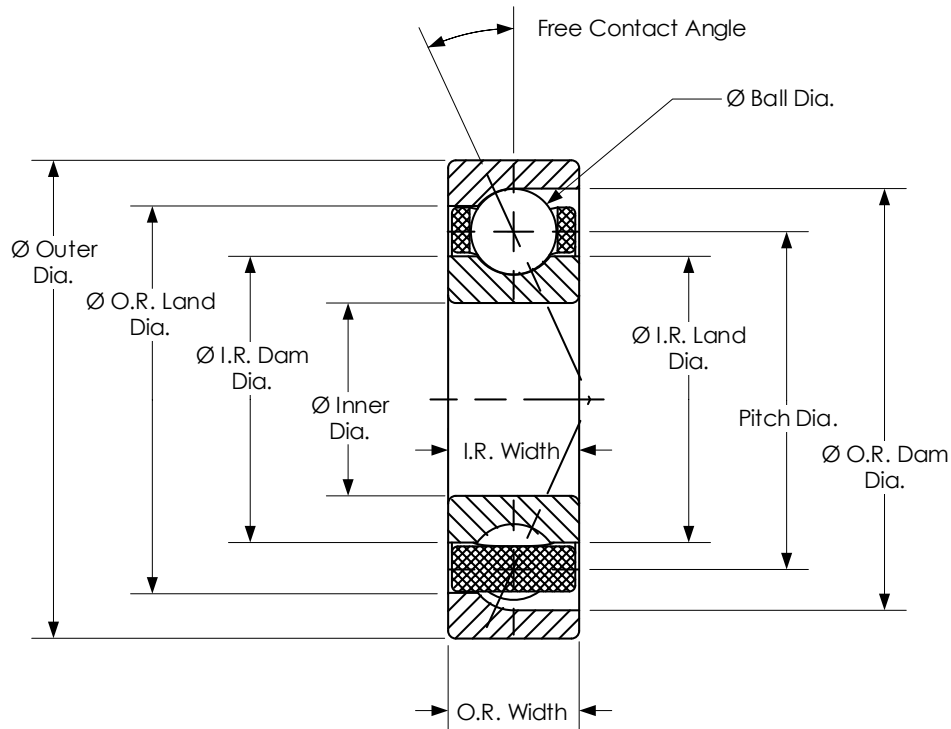
Figure 3-27. Angular Contact (2PT) Terminology

3.2.1.3 Shoulder Height Definitions

Land and dam heights can be defined as h/d ratios or diameters. The figure below illustrates the land and dam heights (h). As shown, the term 'dam' refers to the non-contacting shoulder and the term 'land' refers to the contacting side of the raceway.

Halpin
Engineering,
LLC

Phone: (310) 650-8982
Email: info@HalpinEngineeringLLC.com
Web: http://www.HalpinEngineeringLLC.com
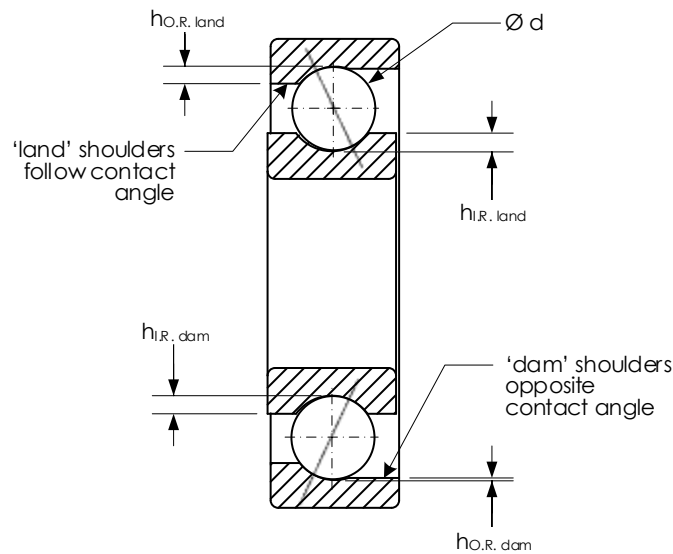
Figure 3-28. Shoulder Height Definitions for h/d Values

3.2.1.4   4PT Gothic Arch Bearings

Definition of a 4PT Gothic Arch bearing in ORBIS requires a few additional parameters compared to a standard deep groove radial bearing. In particular, the arched radial play or free contact angle must be defined along with the shim size or resting angle for both the inner and outer races. See Figure 3-29 and Figure 3-30 for definition of the arched radial play, arching shim size, resting angle and free contact angle.
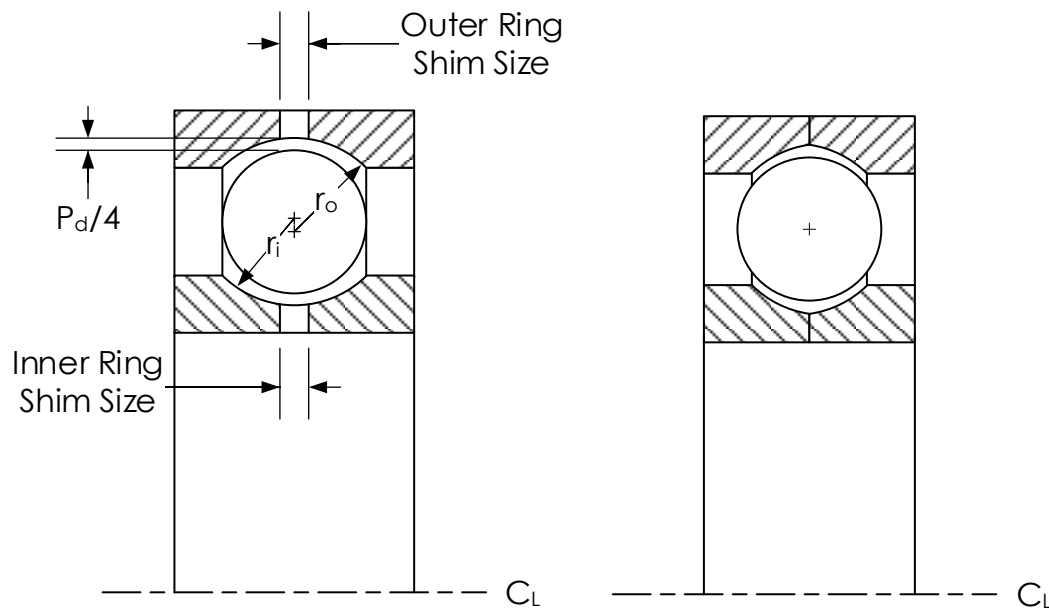
Figure 3-29. Gothic Arch (4PT) Arching/Shim Terminology
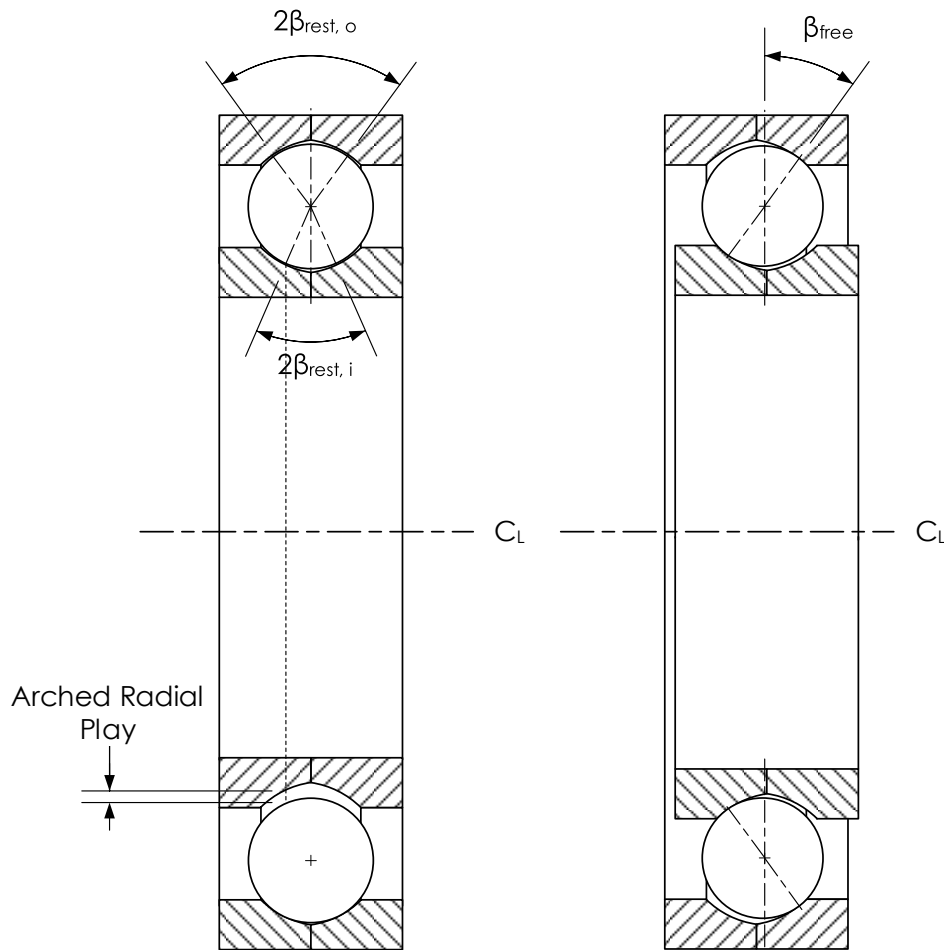
Figure 3-30. 4PT Radial Play, Resting Angle and Free Contact Angle Terminology

3.2.1.5 Cylindrical Roller Bearings

Cylindrical roller bearings require definition of the roller profile. ORBIS allows four different profile types: flat, partial crown, full crown, and logarithmic. The figure below illustrates the flat, partial crown and logarithmic type. A full crown profile has a constant radius over the full length of the roller (i.e., no flat length).

Other inputs required to define a cylindrical roller are similar to all bearing types (e.g., I.D., O.D., width, surface finishes, etcetra) and need no further explanation.
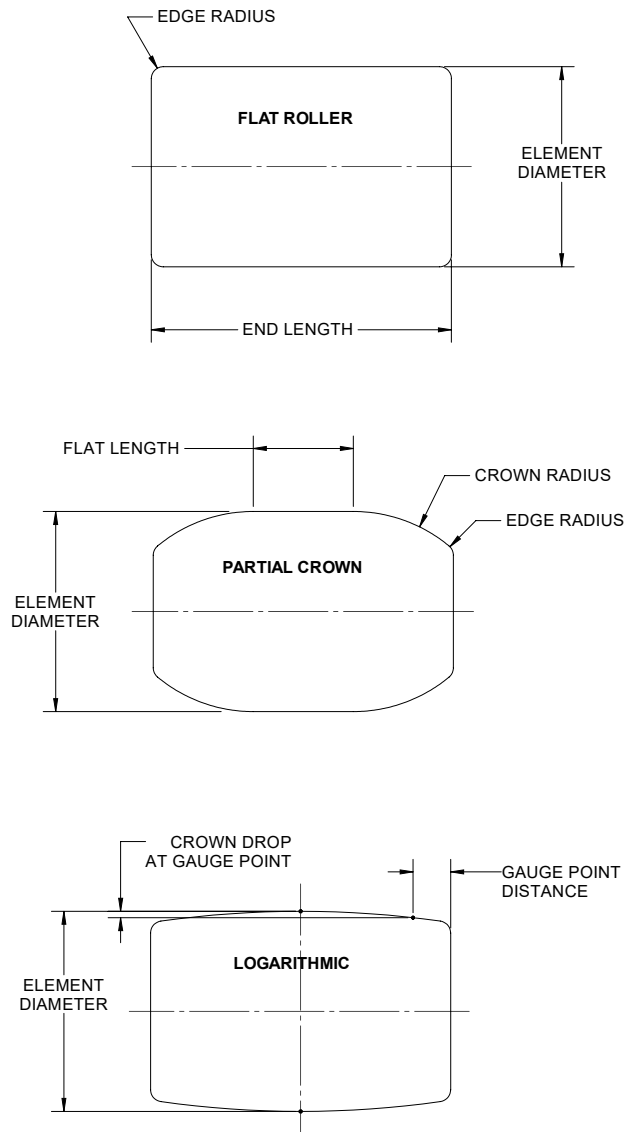
Figure 3-31 Roller Profile Terminology

3.2.2   Material Database Editor

The material database editor is used to define or edit materials. Since material definitions are needed for both bearing definitions and shaft/housing definitions it is recommended that the user initially take the time to define their most widely used materials before proceeding with analysis setups. See the following figure for a detailed description of the material editor.

#	Title	Description
1	Parameter Inputs	Each material requires the following parameters: • **Material Name:** Specify a unique name for the material database entry • **Young's Modulus (psi):** Specify the modulus of elasticity of the material • **Poisson's Ratio:** Specify Poisson's ratio for the material • **Specific Density (lbm/in^2):** Specify the density of the material • **Coefficient of Thermal Expansion (in/in-F):** Specify the coefficient of expansion for the material
2	Clear Entries Button	Clears all input entries (does NOT clear the database entries).
3	Add To Database Button	Adds the new entry to the Material Database.
4	Material Database	Shows all defined entries in the Material Database.
5	View Parameters Button	Displays the parameters of a selected material database entry.
6	Delete From Database Button	Deletes the selected database entry from the database.
7	Close Button	Closes the Bearing Database Editor.

Figure 3-32. Material Database Inputs

3.2.3   Lubricant Database Editor

The lubricant database editor is used to define or edit lubricants. ORBIS models key lubricant properties across temperature. This is done by entering temperature/parameter data point into the editor tables and then ORBIS will derive a model with coefficients that fit the entered data in a least-squares error sense. Note that ORBIS only stores the model coefficients in the database and any raw data entered will not be saved. However, the editor does allow the user to plot the lubricant model across any temperature range of interest. See the following figure for a detailed description of the lubricant editor.

Figure 3-33. Lubricant Database Inputs

#	Title	Description
1	Lubricant Name	Input field to name the lubricant. This name will be shown in the database list once added.
2	Density (g/cc)	Input field to specify the density of the lubricant in grams per cubic centimeter.
3	Kinematic Viscosity Vs. Temperature Table	User can enter multiple temperature and kinematic viscosity data to be modeled. Note that at least two temperature and corresponding kinematic viscosity data points are required. Adding more than two points generally improves the curve fit model.
4	Solve Coefficients Button	After entering temperature/viscosity data, this button must be selected to generate a curve fit of the viscosity data. A plot window will appear showing the entered data and resulting viscosity model.
5	Viscosity Model Coefficients	These fields are populated by ORBIS once selecting the 'Solve Coefficients' button. Coefficients match the general model described in ASTM D341 (Annex) with one exception: Lambda will only be 0.7 if there are only two data points specified. Otherwise, Lambda is refined such that the model matches the data in a least squares sense.
6	Viscosity Plot Button	Allows the user to generate a plot of the viscosity model at any temperature range and step size.
7	Pressure-Viscosity Vs. Temperature Table	Here the user inputs at least one temperature and corresponding pressure-viscosity coefficient. Again, adding more than one data point will improve the model generated.
8	Solve Coefficients Button	After entering P-V coefficient data in table select this button to generate the model coefficients.
9	P-V Model Coefficients	These entries are solved for once the 'Solve Coefficients' button is pressed. P-V model is a simple linear model.
10	P-V Plot Button	Allows the user to generate a plot of the P-V model at any temperature range and step size.
11	Add to Database Button	Once all inputs are defined and satisfactory model coefficients for both kinematic viscosity and P-V coefficient have been generated this button is selected to add the lubricant to the database.
12	Lubricant Database	This window shows all of the user defined database entries. Entries are selectable by left mouse click.
13	View Parameters Button	Upon selection of a lubricant in the database, selecting this button will populate the Lubricant Parameters into the dialog.
14	Delete Database Entry	Selecting this button with a database entry selected will delete the lubricant from the user database.

3.3   Analysis Results Window

The Results Window is shown in the figure below. This window will appear as the top window once an analysis has reached a solution. As described in the figure below, there are several features of this window. However, its main intent is to display a complete set of results that can be reviewed and printed to a final report. Refer to section 4.2.1 for a description of all available output.

All results windows 'float', which enables the user to compare results from multiple runs in side-by-side windows. Furthermore, the 'Detailed Results' section of the window is a text pane with simple text editor properties (user can copy/paste and even type inside). Results are also categorized with check boxes to allow the user to show/hide any combination of result sections they want.

In a general analysis workflow, the setup file is usually saved rather than the analysis results. However, options do exist to save the results to a text file or print them.

#	Title	Description
1	Result Summary	Key result parameters are tabulated for each bearing row. This area provides quick access to available key parameters.
2	Detailed Results	A scrollable/selectable text window containing the complete analysis output. See section 5.0 for a comprehensive listing of available output contained in the detailed results section.
3	Result Highlighting	Each element Hertzian contact stress exceeding the user specified allowable is highlighted. Additionally, all elements with truncation are automatically highlighted.
4	Output Filter Selection	User can select various categories of output to be displayed by selecting the various check boxes.
5	Select All Button	Select this button to have all output categories selected
6	Clear All Button	Select this button to clear all output checkboxes
7	Post Selected Button	Select this button to post all output categories actively selected
8	Adjust Font Size	Font slider adjusts font size, in 1pt increments, for all the text in the Detailed Results pane.
9	Print Button	Brings up a standard print dialog. Only contents showing the detailed results are printed.
10	Save Button	Saves the contents of the results window to a tab delimited text file.

Figure 3-34. Results Window

4 Brief Technical Background

ORBIS uses numerical techniques to solve the nonlinear elastic behavior of the user defined system of one or more bearing rows. The model considers each ball-to-race contact for all bearing rows defined in the system; resulting in complete knowledge of the element load distributions and their raceway attitudes. A solution to the system is achieved when the sum of all bearing row reaction forces is sufficiently close to the external applied forces (system equilibrium). Core analysis methods follow the mathematical theories developed and published by A.B. Jones (1964).

The system model essentially follows the logical process necessary to assemble a rotational system: initial conditions are defined, the bearings are fit into the assembly, preload is applied to the bearings, and external loading is finally applied to the mounted and preloaded bearing system. The parameters describing relative axial ring displacements and internal clearance changes are tracked at each step of the process; ultimately leading to the final state of the bearing system.

ORBIS implements a ring compliance model based on classical thick ring theories to determine the final mounted and preloaded state of the bearing system. The compliance model is also used for assessing thermal strains. Once all mounting, preloading and thermal strains have been considered the traditional fixed ring theories are applied to calculate effects due to external loading.

The compliance model makes a key assumption that, in the local vicinity of the bearing, the housing, shaft and bearing rings can be expressed with a series of nested rings. Thick ring theory assumes the representative rings have uniform constant wall thickness and all deflections remain within the linear elastic region of the material. Since the groove side of a bearing race has a non-constant cross section, an equivalent diameter has been developed to represent bearing rings. The equivalence model is shown in section 5.7 and has been correlated to detailed finite element models of standard bearing ring cross-sectional proportions. The compliance model also handles various sudden changes in the boundary conditions of each nested ring, such as when the fit between a bearing O.D. and housing I.D. transitions from clearance to interference due to preload expansion or thermal expansion.

4.1   Preloading

Axial preloading creates radial forces on the bearing rings that increase the bearing's internal clearance—the rings stretch—and change the inner-to-outer ring stickout (axial offset of ring faces when bearing is in is free contact angle position). To find the preloaded state, with account for ring stretch, each bearing row is iteratively solved until the change in internal clearance equals the resultant ring deflection due to preload pressure on the raceways. This algorithm also accounts for nonlinear behavior of axial load versus deflection and potential nonlinear radial ring stiffness due to fitup gap closure.

The preload value entered within each bearings row's preload input (see section 3.1.4) includes options to specify the preload *condition* and preload *type*. These parameters, as discussed below, allow the user to specify at what state the entered preload value is true.

4.1.1   Preload Condition

Since ORBIS treats bearing rings as flexible objects (note: ring flexibility is only considered during the preload algorithms), knowledge of an initial state is needed to determine the final solution. The preload condition radio buttons (reference section 3.1.4) are used specify whether the preload value applies at the mounted or un-mounted condition.

The term 'mounted' means the bearing has been fit, or mounted, into the shaft/housing system and all boundary influences from interference fitting and ring clamping are included. This means the specified preload will be exactly as input at the mounted state of the bearing. The compliance model will determine what the resulting mounted and preloaded contact angle is and use that as the starting point for any external loading.

Conversely, an un-mounted preload condition means the specified preload value is true when the bearing rings are pressed together axially but are radially unrestrained (free to expand/contract). This is very common for pre-ground matched duplex pairs supplied by most bearing manufacturers. For an un-mounted preload condition, it is not necessarily true that the final mounted preload will equal the preload input by the user. This difference will depend on the influence of mounted boundary conditions, such as interference fitting, and the specified ***Preload Type***.

4.1.2   Preload Type

ORBIS handles two different preload types: rigid and spring (reference section 3.1.4). The preload type determines how the preload reacts to mounting and thermal influences.

When the preload type is specified as 'rigid' there is no further axial motion allowed between the rings and their adjacent boundary (shaft or housing). Additionally, when the preload condition is specified as un-mounted, the preload gap determined from radially unrestrained rings will be enforced during the mounting routine. If the mounting fits are tighter than the ring expansions due to radially unrestrained preloading, then this preload gap enforcement will always result in a mounted preload that is higher than the specified un-mounted value. Perhaps the best way to describe a rigid preload type is when one uses a thick clamp plate, or jam nut, that is torqued down on the bearing with forces much higher than the bearing preload to ensure all pre-ground face gaps have been securely seated.

When the preload type is specified to be 'Spring' then relative motion between the inner or outer rings and their mating shaft or housing is permissible. Once the preload force is overcome, the bearing reaction follows the spring rate curve. Spring type preloading essentially bypasses unmounted preload specifications since typical spring rates prohibit noticeable force increase for small deflections—this assumption will be valid for spring rates much lower than the axial stiffness of the bearing, which is often the case.

4.2   Convergence Criteria

ORBIS uses the IEEE 754 technical standards for all floating-point arithmetic. All calculations use at least 64-bit precision. Key calculations pertaining to matrix inversion and the overall system Jacobian are extended to 128-bit precision to improve accuracy of the solver. The default criteria for convergence are show below. Refer to section 3.1.6.10 for instructions on how to change these settings.

Table 4.2-1. Solver Convergence Criteria

Parameter	Default Error (±)	Comments
System Equilibrium Error	0.001%	The allowable system equilibrium error is defined as a percentage of the applied external loading (Euclidean norm of force components). Error is defined as the difference between the norm of all bearing row reaction forces and the norm of the external applied forces. In the case where there is zero external loading the error defaults to 1.0E-5.
Preload Force Error	1.0E-5 lbf.	Preload force error is defined as the difference between the bearing reaction forces and the applied preload force.
Ring Expansion Error	1.0E-7 in.	The ring expansion error is defined as the difference between ΔP_D input into the Jones model (a 'fixed ring' model) and the resulting ΔP_D due to ring deflections. Ring deflections are determined by using the resulting ball normal forces and associated contact angle to determine an equivalent radial pressure on the bearing ring.

4.2.1   Non-Convergence

Non-convergence occurs when the numerical solver is unable to reach a solution that meets the above convergence criteria. When this occurs, a warning dialog (see figure 4.2-1) is shown with options to 'Cancel', 'Override', or 'Show Best Solution.' These options are described as follows:

Halpin
Engineering,
LLC

Phone: (310) 650-8982

Email: info@HalpinEngineeringLLC.com

Web: http://www.HalpinEngineeringLLC.com

- **Cancel** – This option aborts the solver and returns to the main window.

- **Override** – This option provides the user with a dialog to change the initial guess (see Figure 4-2 below). Once the new guess is entered, ORBIS will re-run the solver based on the new initial guess.

- **Show Best Solution** – Selecting this option shows a result dialog with the best, or closest, solution found. When selecting this option, there will be an information dialog showing the norm of the residuals and percent convergence for the best solution. Percent convergence is the ratio of the root sum square (RSS) of the applied force components divided by the RSS of the reaction forces. Because this ratio consists of terms that are positive definite magnitudes, the percent convergence result is not guaranteed to be meaningful. For instance, if the reaction magnitude is equal to the applied force magnitude, but they do not cancel vectorially, the ratio will be misleading. However, when including the norm of the residuals output, which provides the root sum square of the force errors, there is now sufficient information to judge how close the solution is to force balance. Ultimately, when using the 'Show Best Solution' for non-convergence cases, the user is advised to review the presented solution carefully.
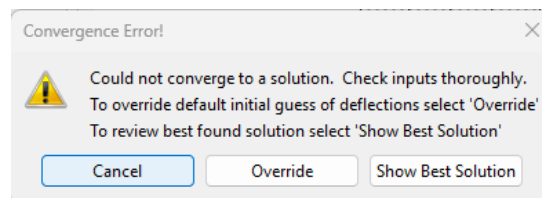
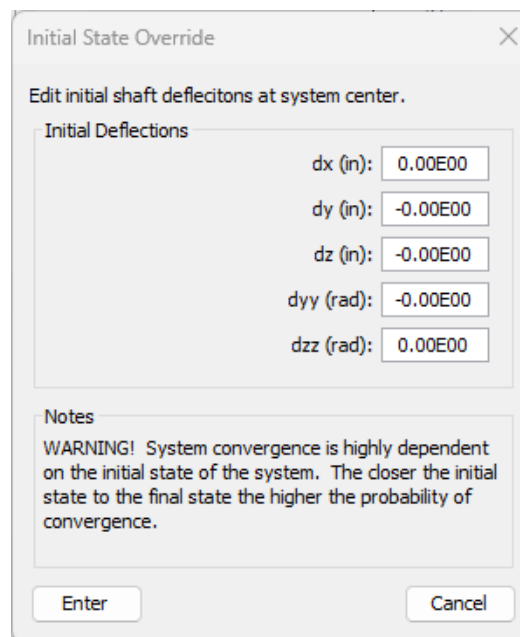

Figure 4-1. Convergence Error Dialog

Figure 4-2. Initial State Override Dialog

4.2.1.1   Common Causes for Non-Convergence

ORBIS uses Newton-Raphson techniques to solve equilibrium of the user-defined system of bearings. Although the Newton-Raphson technique is generally robust, its ability to solve will depend on the non-linearity of the system

and how close the initial guess is to the final solution. Furthermore, the Newton-Raphson technique requires the system stiffness matrix to be inverted, so any loss in stiffness will lead to divide-by-zero exceptions.

Therefore, non-convergence cases are generally caused by systems that either have non-smooth stiffness-to-load characteristics or regions where the system stiffness is zero. Common bearing configurations with these aspects are:

- **Non-Preloaded Bearings –** Bearings that do not have preload, and contain axial/radial free play, have zero stiffness regions. These regions are where none of the rolling elements have contact with the raceways.

- **Preload Springs –** Often bearing preload spring rates are orders of magnitude softer than the bearing axial stiffness, thereby causing a large nonlinearity to the stiffness versus load of the system. This nonlinearity occurs at the point where external forces overcome the preload.

- **Non-Reactive Load Components –** When a bearing type, or system of bearings, has no stiffness in a particular load direction, there can be non-convergence if forces are applied in this direction. For example, if a system only has cylindrical roller bearings, the system has zero stiffness along the thrust, or axial, degree of freedom.

4.2.1.2 Managing Non-Convergence
When faced with a system that does not solve, here are the recommended actions to try and remedy the problem:

- Make sure the system setup is modelled as intended (i.e. there are no setup errors). Double check contact angle orientations are correct (this can be done by inspecting the direction of the contact angles in the sketch). Also check that any applied bearing preloading has the correct sign convention.

- Activate the console window (see section 3.1.5.1) and inspect the trend of the residuals. The first output line in the console provides the convergence criteria, which is the threshold for which the residuals must be below to converge. Based on the trending of the residuals in the console window, the following scenarios should be looked for:

  o Residuals are decreasing but the number of solution attempts were exceeded. In this case allowing more solution attempts might allow convergence. See section 3.1.6.10 for how to change the number of solution attempts in the System Preferences.

  o There are solution attempts where the residual is small but still larger than the convergence criteria. Otherwise, the residuals tend to dither between small and larger values. In this case, using the best solution may be acceptable. To always allow this level of convergence, consider changing the system equilibrium criteria (also done from the System Preferences dialog).

- Specify a better initial guess by selecting the 'Override' button at the non-convergence dialog. Theoretically, if the true system solution exists where the system Jacobian is positive definite, then there will be an initial guess that will lead to this solution. Practically, making a good initial guess is not always easy. However, with some iteration to understand the system behavior, this may be achievable. One recommendation is to study individual force components and/or their magnitudes. If convergence can be achieved with single force components, or at lower or higher magnitudes, and one reviews the resulting deflections of these converged cases, one can develop an estimated region of where the solution point must be.

- If the non-convergent analysis is a Dynamic analysis, try changing it to Static analysis. The Static analysis is a more robust solver and will have a better chance at converging. If the Static analysis converges, record the deflection of the solution, then try using that deflection as the override to the initial guess for the Dynamic analysis.

- Consider adding a small sacrificial load component to the system. This should be done judiciously but can help certain systems maintain contact to avoid zero stiffness.

5 Output Descriptions

The following sections describe the analysis output generated by ORBIS.

5.1 Results Summary

The result summary table, as displayed in the top region of the Results Window, is copied to the detailed results section so that it may be included in the printable report. This table contains maximum mean Hertzian stresses for each row, any percent truncation (maximums are only reported), final preload values, and if the analysis is run in dynamic mode, the total resistive torque for each row.

5.2 Input Parameters

This section provides a list of the user inputs used to generate the results. The section is divided into system inputs, bearing inputs, and row inputs. System inputs include shaft/housing material names, system rotational speeds, and lubricant properties for each row. Bearing inputs contain all geometry used to define each bearing within the user's setup. Row inputs include all row information for each row (i.e. row locations, fitups, clamp loads, spring rates, etcetra).

5.3 Material Properties

The material properties section provides a list of the materials used in the system along with their defined properties. Properties include Young's Modulus, Poisson's Ratio, specific density, and coefficient of thermal expansion.

5.4 External Applied Loads

This section displays the user defined system load components and the application point at which they are applied (location is along the x-axis). Loading information for up to three separate load points will be displayed. The loads shown are the original defined loads and are not scaled by the factor of safety. However, the factor of safety is output directly after the applied loads.

5.5 Ball Crossing Angles

Ball Crossing Angles are displayed for each row and are defined as the angular rotation required by a given ring, inner or outer, to cause a ball to travel to an adjacent ball station. Ball crossing angle results are based on the ambient mounted bearing contact angle.

5.6 Internal Clearances

This section provides bearing internal clearances in the diametral (denoted by 'PD') and axial (denoted by 'PE') directions. Note: diametral clearance is also referred to as 'radial play', which is meant to indicate play in the radial direction (not a radial dimension that is ½ the diametral dimension). Diametral play is defined as the total linear radial distance the inner ring can move relative to the outer ring with negligible applied force (peak-to-peak radial play). Axial play is defined as the total axial displacement the inner ring can move relative to the outer ring (peak-to-peak axial play).

Result descriptions for each of the internal clearance outputs are as shown in the following table.

Table 5.6-1. Internal Clearance Output Descriptions

Result Title	Parameter Description
Diametral, PD (free)	Radial play of the bearing in its unmounted, or free, state.
Axial, PE (free)	Axial play of the bearing in its unmounted, or free, state.
Delta PD Mount	Change to radial play due to interference and clamping effects (ambient temperature)
Delta PD Preload	Change to radial play due to preloading (ambient temperature)
Delta PD Temp (free)	Change to radial play due to thermal expansions of bearing rings and balls. This is a free bearing condition with applied temperatures.
Delta PD Temp Mount	Change to radial play due to interference and clamping effects at temperature. Fitup at bearing ID and OD are adjusted based on relative thermal expansions. Ring clamping forces are always as input by user and do not get adjusted for temperature.
Delta PD Temp Prld	Change to radial play due to effective preload at temperature. This change represents the difference in radial play from the mounted temperature state to the mounted and preloaded state.
Delta PD Film	Change to radial play due to generated film thickness. Only film thickness above the root of the sum of the squares (RSS) of the contacting body surface finishes are considered
Delta PD Speed	Change to radial play due to ring expansion from centrifugal forces.
Diametral, PD (tot)	Final effective radial play, which considers all deltas discussed above.
Axial, PE (tot)	Final effective axial play after all mounting, preloading and temperature effects.

5.7   Bearing Ring Properties

This section provides output for the equivalent raceway diameters and the bearing inner and outer diameter expansions due to mounting, preloading, thermal expansions, and speed (centrifugal). The equivalent raceway diameter is the approximate diameter through the grooved side of the race such that the ring hoop stiffness is equal to that of a prismatic ring analyzed with thick-ring theories. This diameter is used for the mounting/preloading algorithm. The equations for equivalent raceway diameters are defined as function of the bearing geometry as shown below.

$$\mathcal{D}_{IR} = d_{IR_{race}} + 0.68(h_l + h_d) - \frac{2h_l}{d_{IR_{race}}}(h_l - h_d)$$

$$\mathcal{D}_{OR} = d_{OR_{race}} - 0.68(h_l + h_d) + \frac{2h_l}{d_{OR_{race}}}(h_l - h_d)$$
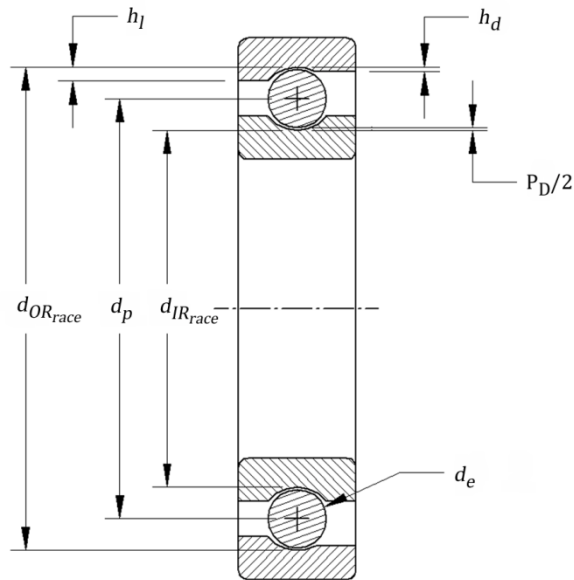
Figure 5-1. Bearing Nomenclature for Equivalent Ring Calculations

The following table provides a detailed description of each of the different bearing ring outputs.

Table 5.7-1. Bearing Ring Output Descriptions

Result Title	Parameter Description
I.R. Equiv Diam (in)	Equivalent inner ring diameter as shown in FIG 23 equations
O.R. Equiv Diam (in)	Equivalent outer ring diameter as shown in FIG 23 equations
I.D. Expand, Free (in)	Expansion of bearing inner diameter due to preloading the bearing in a free, radially unrestrained, condition. Applies at ambient temperature.
I.D. Expand, Mnt (in)	Expansion of bearing inner diameter due to ambient mounted and preloaded state. This includes effects from interference fits, ring clamping and preloading.
I.D. Expand, Temp (in)	Expansion of bearing inner diameter due to mounted and preloaded state at temperature. This includes fitup changes due to temperature, ring clamping, and effective preload at temperature.
I.D. Expand, Speed (in)	Expansion of inner diameter due to centrifugal effects.
O.D. Expand, Free (in)	Expansion of bearing outer diameter due to preloading the bearing in a free, radially unrestrained, condition. Applies at ambient temperature.
O.D. Expand, Mnt (in)	Expansion of bearing outer diameter due to ambient mounted and preloaded state. This includes effects from interference fits, ring clamping and preloading.
O.D. Expand, Temp (in)	Expansion of bearing outer diameter due to mounted and preloaded state at temperature. This includes fitup changes due to temperature, ring clamping, and effective preload at temperature.
O.D. Expand, Speed (in)	Expansion of outer diameter due to centrifugal effects.
I.D. Fitup, Mnt (in)	Effective fitup (diametral interference/clearance) between bearing inner diameter and shaft outer diameter at mounted and preloaded state at ambient temperature. This includes fitup changes due to interference, ring clamping, and preloading.
I.D. Fitup, Temp (in)	Effective fitup (diametral interference/clearance) between bearing inner diameter and shaft outer diameter at mounted and preloaded state at temperature. This includes fitup changes due to interference, ring clamping, and preloading.
I.D. Fitup, Dyn (in)	Final fitup at inner diameter due to all mounting, temperature, and speed effects.
O.D. Fitup, Mnt (in)	Effective fitup (diametral interference/clearance) between bearing outer diameter and housing inner diameter at mounted and preloaded state at ambient temperature. This includes fitup changes due to interference, ring clamping, and preloading.
O.D. Fitup, Temp (in)	Effective fitup (diametral interference/clearance) between bearing outer diameter and housing inner diameter at mounted and preloaded state at temperature. This includes fitup changes due to interference, ring clamping, and preloading.
O.D. Fitup, Dyn (in)	Final fitup of outer diameter due to all mounting, temperature, and speed effects.

5.8   Preload Data

Preload data output section shows the applied preload, which pertains to setup specified preload condition, the mounted preload, the mounted preload at temperature, and dynamic preload. Furthermore, the corresponding axial ring deflections for mounted preload and preload at temperature are provided. The dynamic preload includes all mounting, temperature, and film thickness effects to the preload. Refer to the following table for a description of each output.

Table 5.8-1. Preload Output Descriptions

Result Title	Parameter Description
Applied Preload (lbf)	The applied preload is the value entered into the preload input field. Note that it's applicability depends on the preload condition (i.e. unmounted or mounted).
Mounted Preload (lbf)	Mounted preload is the preload after all mounting conditions have been applied. For example, if the system of bearings has been installed with all clamp loads and fitups, and is at ambient temperature with no external loading, the axial row reactions would be equal to the mounted preload.
Dx, Mounted Prld (in)	Shows the relative motion of the inner ring with respect to the outer ring in the mounted preload state. Zero is defined to be the initial contact of the bearing at its free contact angle with no external influences.
Preload at Temp (lbf)	Preload at temperature is the mounted preload with temperature expansions applied to the system. No external loading applied.
Dx, Prld at Temp (in)	Shows the relative motion of the inner ring with respect to the outer ring in the 'Preload at Temp' state.
Dynamic Preload (in)	Shows final preload of each bearing row due to mounting, temperature, and oil film effects.

5.9   Reaction Forces on Shaft

This section provides the resulting bearing reaction force components, expressed in global coordinates, on the shaft at each bearing row. These forces include all mounting, preloading, speed, and external loading conditions. Forces apply at the center of their respective bearing row locations.

5.10 Inner Ring Displacements

This section provides the total displacement, expressed in global coordinate components, of the inner rings due to mounting, preloading, temperature, and external loading. Initial, or zero position, is defined to be the unmounted free initial contact condition. Note: the axial (dx) displacements due to external loading only can be obtained by taking the difference between the total displacements shown and the axial displacements due to preloading as shown in the Preload Data output section (section 5.8). Alternatively, if system is run with and without external loading, the difference in inner ring displacements represents the deflections due to external loading.

5.11 Stiffness Outputs

ORBIS provides three different types of stiffness outputs: axial stiffness with ring compliance considerations, system Jacobian diagonal terms, and complete 5x5 stiffness derivatives for each bearing row. In all cases, reported stiffness values are local tangent stiffnesses (i.e. dF/dx) and not secant stiffness. Refer to subsequent sections for a description of each type of stiffness result.

5.11.1   Axial Stiffness with Ring Compliance

This output is only available for bearing systems that have internal preloading specified. For such systems, the total axial stiffness of the shaft, which includes effects from shaft/housing/bearing ring hoop stiffnesses along with the Hertzian contact stiffness are provided. The reported stiffness applies at the mounted and preloaded state, with no influences from external loading (i.e. the stiffness prior to any external loading).

5.11.2   System Jacobian

The full system Jacobian diagonal terms are shown here. Results apply in the fully loaded condition. Additionally, the system Jacobian assumes a rigid system during external load application. Results are provided at the system center and each specified load point.

5.11.3   Stiffness at Load Point

When performing a flexible shaft analysis, the output contains additional complete stiffness matrices for each load point specified in the system. This matrix represents the full 5 x 5 stiffness matrix of the system at the specified load point. Stiffness results include effects of shaft compliance and bearing stiffness's in their final loaded equilibrium state.

5.11.4   Row Stiffness Matrix

This section provides complete 5x5 stiffness derivatives for each bearing row. Stiffness results correspond to the quasi-static equilibrium state of the system in its final loaded state.

5.12 Fatigue Life

This section provides individual ring and total system fatigue cycles for various conditions. Note: Fatigue results are only generated for dynamic analysis runs. Outputs include L10 life, adjusted L10 life, and adjusted life with consideration for film thickness. L10 Fatigue life calculations are based on the Lundberg – Palmgren theories as shown in Jones (1964). The adjusted life output is based on the life factor theory adopted in AFBMA (1990) standards. This result includes the user defined life factor input along with a computed factor based on the user reliability input.

The life factor input allows the user to enter a combined factor to account for items such as materials, cleanliness, and misalignments. Bamberger (1971) provides a useful reference for computing various life factors. The adjusted life with film includes an additional lubricant factor, which is also provided for reference. The film parameter is from Bamberger (1971) and follows the AFMBA recommended average curve and uses the minimum film parameter as discussed below. All fatigue calculations use the individual rolling element results and do not require determination of an equivalent radial load on the bearing row.

5.13 Bearing Torque

Bearing torque results are only generated for dynamic analysis runs. Two types of torque are considered: friction torque and viscous torque. Friction torque output represents the torque associated with rolling and spinning within the contact area when the balls start rotating. Precisely, this is the torque due to interfacial slip (aka Heathcoate Slip) at the contact ellipse. ORBIS uses the 'Race Control' theory from Jones (1964) for 2PT bearing types and therefore only allows spin to occur on one raceway. 4PT bearing types use a minimum energy theory that allows spin and slip to occur simultaneously. The computed torque output can be scaled with the coefficient of friction input parameter. Reference Jones (1964) for the friction torque calculations used by ORBIS.

Viscous torque output is based on the Palmgren model, which was republished by Harris (2001). This model accounts for lubricant viscosity and requires use of a viscous torque factor. Hence, the viscous torque output can be scaled by direct modification of the viscous torque factor.

To achieve optimal torque predictions the user will need to tune both the coefficient of friction at the ball contact and the lubricant viscous torque factor. This is most accurately done by use of existing test data of a known configuration. The coefficient of friction should be tuned first based on breakaway or slow speed test data. The

viscous torque factor is then tuned based on test data for two different operational speeds. Note that viscosity is highly sensitive to temperature.

5.14 Ball Excursions

Dynamic runs include output showing the maximum ball excursion for each bearing row. This output represents the maximum circumferential excursion, in units of inches, a ball travels relative to the average ball path. Another way to view this is the maximum amplitude a ball travels within the retainer pocket. This result is achieved by taking the orbital velocities of each ball, determining the average velocity of all balls, integrating the orbital velocities through one full revolution, and finally computing the difference between actual integrated ball positions and the average ball positions. Final reported excursion represents the maximum ball departure from the average.

5.15 Row Outputs (Element-Wise results)

Detailed outputs for each element of each row are provided in tabular form. Output tables are repeated for each bearing row in the user's system. These tables differ depending on whether the analysis is static or dynamic. All outputs correspond to the system equilibrium state after application of all external loading.

Results for a 4PT bearing will have detailed element output for all four possible contacts. The different contacts are distinguished with the following terminology: IR-L would read 'inner ring left' and thus denotes the left side of the inner race, OR-R would be the right side of the outer raceway.

5.15.1 Element Number

The Element Number is simply an indexing scheme to identify each of the rolling-elements uniquely. The first ball is always placed along the –Z axis (the three o'clock position when looking down the X axis at the Y-Z plane).

5.15.2 Normal Ball Load

The Normal Ball Load is the load applied by each ball into each raceway contact. This load is directly normal to the contact ellipse and has an attitude defined by the contact angle.

5.15.3 Contact Angle

The Contact Angle output describes the angle of the normal ball load vector to the plane extending through the Y-Z plane.

5.15.4 Mean Hertz Stress

Mean Hertz Stress output represents the average Hertzian contact stress over the contact area. For ball elements, the contact area is defined by the ellipse dimensions provided in subsequent output. For cylindrical elements, the contact area used for the mean stress calculation varies. If the roller has non-flat profiling, then the length of the contact area is divided into laminar slices and the reported stress is the laminae with the highest stress – where mean stress is the local laminae force divided by the area of the lamina. For flat rollers, the area is defined by the traditional width and length of contact. Additional output for the most heavily loaded roller is provided (see section 5.15.27) which provides laminae dimensions and stresses at each lamina. ORBIS automatically highlights, in red, all values that exceed the user defined allowable mean Hertzian stress from the main user interface. All stress results assume the contact ellipse if fully contained within the raceway.

5.15.5   Truncation Analysis

When truncated elements are found additional output is provided directly after the first table of the affected row. This output consists of peak center stresses and peak edge stresses for all elements exhibiting truncation. The method used follows the publication by Frantz and Leveille (2001). Stresses are reported as peak, not mean, and all edge stresses include a nominal 1.8X edge concentration factor. Since the edge concentration factor is linearly applied to the stress, the user may manually adjust edge stresses based on alternate edges stress factors by multiplying the results by the ratio of desired concentration factor divided by 1.8.

5.15.6   Truncated Length

The Truncated Length represents the percent of the total length of the contact ellipse, along the major axis, that is truncated due to shoulder or dam override. ORBIS will automatically highlight all elements that have any truncation.

5.15.7   Ellipse Semi Major

The Ellipse Semi Major output represents one half of the major dimension of the contact ellipse.

5.15.8   Ellipse Semi Minor

The Ellipse Semi Minor output represents one half of the minor dimension of the contact ellipse.

5.15.9   Contact ½ Width

Roller elements report the contact ½ width instead of the ellipse semi-minor dimension. The contact ½ width is one half of the full contact width.

5.15.10   Max Sub-Surface Shear

The Max Sub-Surface Shear is the peak shear stress developed below the raceway surface due to contact stress.

5.15.11   Max Shear Depth

The Max Shear Depth is the distance along the normal to the contact area, below the raceway surface, at which maximum shear stress is developed.

5.15.12   Upper Edge Location

The Upper Edge Location represents the edge of the contact ellipse that is closest to the land diameter. This value is represented as a ratio of its height from the center of the raceway to the ball diameter.

5.15.13   Lower Edge Location

The Lower Edge Location represents the edge of the contact ellipse that is closest to the dam diameter. This value is represented as a ratio of its height from the center of the raceway to the ball diameter.

5.15.14   Contact Normal Approach

Contact Normal Approach represents the total combined deflection of the contacting bodies (rolling-element and raceway). This deflection is along the normal direction to the contact area.

Halpin Engineering, LLC

5.15.15 Contact Normal Stiffness

Contact Normal Stiffness represents the stiffness of the rolling-element to raceway contact area stiffness in the normal direction.

5.15.16 Spinning Velocity

Spinning Velocity is the angular velocity of the rolling element about the axis of rotation that is normal to the contact on the un-controlling race. Per Jones' (1964) race control theory, spin can only occur on one raceway while pure rolling occurs on the other. Based on the spinning velocity output one can deduce race control (i.e. if there is zero spinning velocity on a given raceway than that raceway is 'in control').

5.15.17 Rolling Velocity

Rolling Velocity is the relative angular velocity of the rolling element about its own axis of rotation parallel to the contact on the controlling race.

5.15.18 Spinning Torque

Spinning Torque is the component of torque generated by interfacial slip within the contact area due to rolling-element spin.

5.15.19 Rolling Torque

Rolling Torque is the component of torque generated by interfacial slip within the contact area due to pure rolling.

5.15.20 Element Roll Velocity

The Element Roll Velocity represents the rotational velocity of the rolling elements as seen relative to the pitch orbit velocity.

5.15.21 Pitch Orbit Velocity

The Pitch Orbit Velocity is the rotational velocity of the bearing pitch diameter about its spin axis. This is essentially the angular velocity of rolling-element cage or retainer.

5.15.22 Minimum Film Height

The Minimum Film Height represents the thinnest point of the lubricant along the center line of the contact ellipse. Two types of minimum film heights are provided: H-D, which stands for Hamrock-Dowson, and C-Z, which stands for Coy-Zaretsky. The H-D calculation is based on the 'Hard-EHL' theory by Hamrock and Dowson (1981) for fully flooded conditions. The C-Z calculations apply a starvation correction factor to the H-D results.

5.15.23 Minimum Lambda Value

Lambda is a dimensionless parameter that is often used to describe the lubricant regime of the bearing. Its value is determined by taking the ratio of the minimum film height to the root sum squares (RSS) of the contacting surface roughness. Mathematically, lambda is defined as follows. The lambda values output in ORBIS are based on the C-Z minimum film height.

$$\lambda = \frac{h_{min}}{\sqrt{R_{raceway}^2 + R_{ball}^2}}$$

5.15.24 Centrifugal Force

The Centrifugal Force output represents the radial body force of the rolling element due to its orbital velocity and mass. This force tends to create differing contact angles between the inner and outer race contacts and is treated in the analysis per Jones (1964).

5.15.25 Gyroscopic Moment

The Gyroscopic Moment output represents the spinning body moment of the rolling element due to its angular velocity and inertia. The influences of this force are handled per Jones (1964).

5.15.26 Ball Spin Axis Angle

The ball spin axis angle defines the instantaneous axis of spin for each ball. This result is only available for 4PT bearings. Further definition of this angle can be found in Jones (1964) and its derivation for 4PT bearings is described in Halpin (2016).

5.15.27 Detailed results for most heavily loaded roller

All roller bearing rows will contain a table showing further details for the stress distribution along the roller axis. These results pertain to the most heavily loaded, or stressed, roller only. ORBIS analyzes roller elements by slicing the roller into multiple laminae along its axis. Since the roller is not allowed to have non-symmetric loading, only one half of the roller is unique. The lamina results in the table start at the center of the roller and progress toward the end of the roller. See the following subsections for descriptions of the contents of this table.

5.15.27.1 Lamina ID
The lamina ID is a simple numerical indexing scheme for each lamina of the roller. The start of the first lamina will be at the center plane of the roller.

5.15.27.2 Lamina Width
This output shows the width of each lamina. ORBIS determines widths based on a fixed number of slices for each type of geometry describing the roller profile. A flat profile is covered with a single lamina. Crowning is broken into two options: full crowning, or logarithmic profiles, will divide the region into twenty-eight laminae, and partial crowning will use nineteen laminae. The edge radius is broken into four laminae.

5.15.27.3 Dist To Lam Center
This output provides the distance between the prior lamina center and the current lamina center. The first lamina result will be one half the lamina width.

5.15.27.4 Crown Drop
Crown drop is the radial drop from the reference diameter to the surface of the roller at the middle of the respective lamina.

5.15.27.5 Mean Hertz Stress IR/OR
The last two columns provide the Hertzian stress at each lamina for the inner and outer raceways. These stresses are titled "Mean Hertz Stress" since they represent local lamina total force divided by the contact area of the lamina.

Halpin
Engineering,
LLC

6   References

Anon.,(1990) *Load Ratings and Fatigue Life for Ball Bearings*, ANSI/AFBMA 9-1990, The Anti-Friction Bearing Manufacturers Associations, Washington, DC.

Bamberger, E. N. (1971) *Life Adjustment Factors for Ball and Roller Bearings – An Engineering Design Guide*, American Society for Mechanical Engineers, New York

Frantz, P. P. and Leveille, A. R. (2001), *An Approach to Predicting the Threshold of Damage to an Angular Contact Bearing During Truncation*, Aerospace Report No. TR-2001(8565)-4, El Segundo, CA.

Halpin, J. D. and Tran, A. N. (2016) *An Analytical Model of Four-Point Contact Rolling Element Ball Bearings*, ASME J. Tribology, 2016, doi:10.1115/1.4033134

Hamrock, B. J. and Dowson, D. (1981) *Ball Bearing Lubrication – The Elastohydrodynamics of Elliptical Contacts*, Wiley, New York, N.Y.

Harris, T. A. (2001), *Rolling Bearing Analysis,* 4th Edition, John Wiley & Sons, New York, NY

Jones, A. B. (1964) *The Mathematical Theory of Rolling-Element Bearings*, in *Mechanical Design and Systems Handbook*, H. A. Rothbart, ed., McGraw-Hill, New York, N.Y. article 13.

Todd, M.J. and Johnson, K. L. (1986) *A Model for Coulomb Torque Hysteresis in Ball Bearings*, in *International Journal Mechanical Science,* Vol 29, No. *5*, pp 339-354, Great Britain.