ORBIS

Modern Rolling-Element Bearing Analysis Software

FEA LOAD TOOL USER MANUAL

Updated for Version 5.0
Apr-2025

Table of Contents

Table of Figures

Table of Tables

1 Introduction

The FEA Load Tool is a finite element module integrated into the ORBIS software. The module features and functionality are described herein (instead of in the main ORBIS manual). Familiarity with the ORBIS software will benefit the reader since certain features of the ORBIS software are inherited by the FEA Load Tool but not reiterated herein. The following table summarizes these items.

Table 1-1. FEA Load Tool Integration with ORBIS

Item Description (feature or functionality)	FEA Load Tool Integration Remarks
Bearing setup files (*.jdh files)	These files are produced from ORBIS (file → Save) and define a bearing analysis setup. The FEA Load Tool uses these files in two distinct ways: 1.) the file is used to compute bearing stiffness properties in spring elements and, 2.) the file can be used to run recovered loads on spring elements (to analyze the bearings). In general, the FEA Load Tool does not alter saved bearing setup files.
Material Database	The FEA Load Tool will access the ORBIS material database when applying materials to beam elements. The FEA Load Tool can open the material database editor, which allows the user to make changes to the material database.
Input Field Expressions	This functionality, as described in section 2.3 of the ORBIS manual, is available in many input fields within the FEA Load Tool.
Batch files (*.csv files)	The FEA Load Tool can export recovered loads to an ORBIS batch file. The tool has features to create new batch files or append load cases to an existing batch file.

The FEA Load Tool extends the capability of ORBIS by adding traditional finite element modelling and analysis capability. It can model assembly level structure around the bearings, run static and linear dynamic loads on this structure, and resolve the internal bearing reaction forces for subsequent analysis within the main non-linear ORBIS bearing solver. The tool solves arbitrary static point loads, quasi-static gravitational loads, and base motion random vibrations.

A typical workflow for the FEA Load Tool is as follows:

1. Bearing setups are completed within the main ORBIS software prior to using the FEA Load Tool. These setups should represent the bearing systems configured in their final assembled state (i.e. mounted and preloaded), which defines the state in which assembly-level loading occurs. If the mechanism under consideration has multiple bearing axes, a setup file for each unique bearing configuration must be completed. All bearing setups must be saved (i.e. *.jdh files created) to be accessible from the FEA Load Tool.
2. The FEA Load Tool is used to construct the assembly-level structural model containing the bearing systems defined in step 1. This model will include spring elements to represent the bearings, and the stiffness properties for these springs can be imported directly from the bearing setup files.
3. The FEA Load Tool is run to solve the defined structural load cases, such as a base motion random vibration case, and recover bearing reaction forces at each defined bearing spring element.
4. The recovered spring element forces are transferred to the main ORBIS bearing model for non-linear bearing analysis (i.e. Hertzian contact stresses).

The above process, which uses a linear finite element solver and the non-linear bearing solver, is integrated to promote an efficient workflow.

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

1.1 Notation

The following notation is used in this manual. When text has the bold blue font depicted, and the notation characters, then it shall have the meaning identified in the table.

Table 1-2. Notation

Notation Character	Meaning
[abc]	Square brackets identify a graphical interface object, such as a button or checkbox. The text inside the brackets will be the name of the object.
<abc>	Less than and greater than characters, or angle brackets, are used to identify text strings that should either be entered by the user or selected from a list within a graphical interface (such as a list or drop-down selection).

2 User Interfaces

This section describes the user interfaces of the FEA Load Tool.

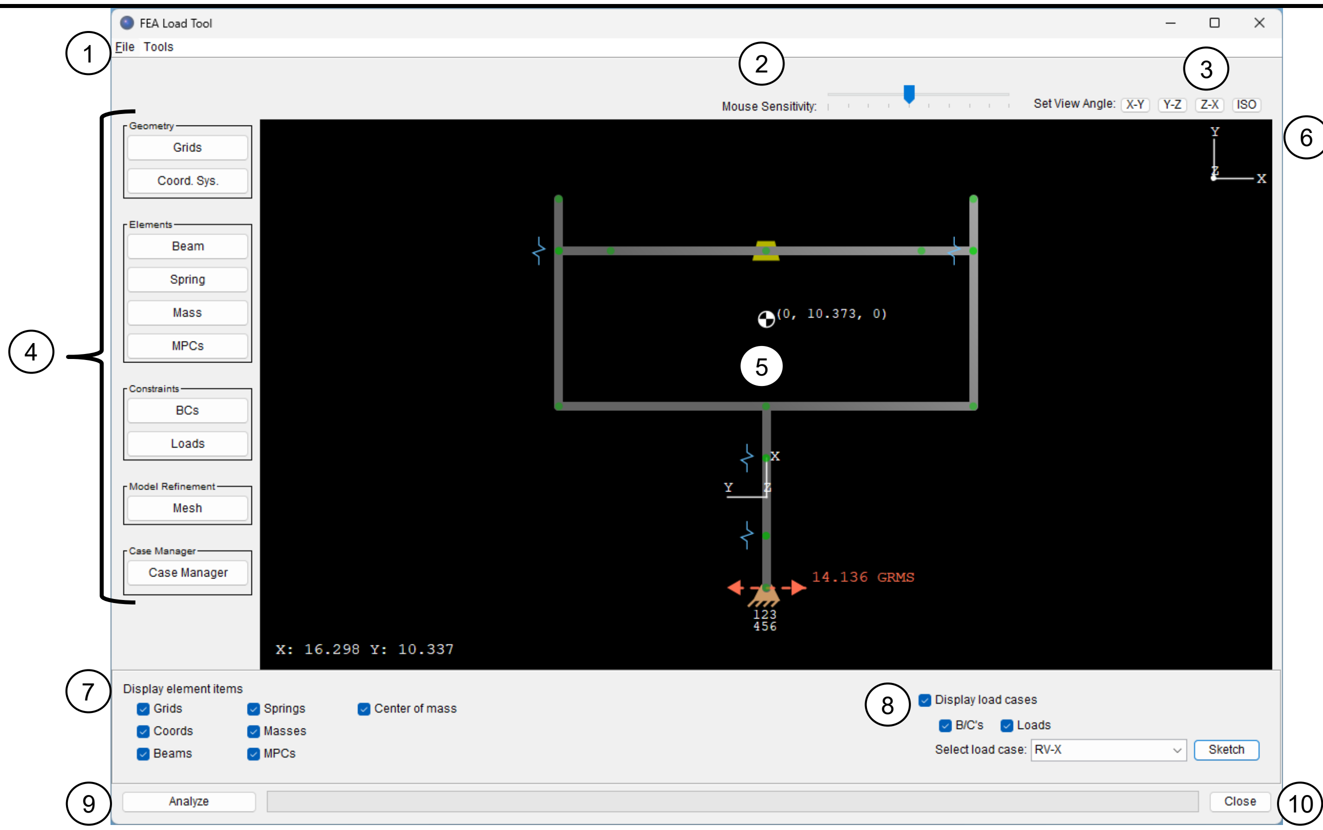
2.1 FEA Load Tool Interface

The FEA Load Tool is opened from the Tools menu within ORBIS. Figure 2-1 shows the FEA Load Tool interface. The tool functions like a stand-alone application, however, the main ORBIS window will always be available while using the tool and can be found as a minimized window in the desktop taskbar.

Model development in the tool is accomplished via a series of model editors. These editors are discussed in section 2.1.2. The display area of Figure 2-1 illustrates an example model, which shows a graphical representation of the developed model. A description of the graphic objects rendered by the tool is provided in section 2.1.3.

Halpin Engineering, LLC

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



#	Title	Description
1	Menus	File Menu – contains options for saving and opening FEA Load Tool model files. Tools Menu – access the Mass Tuner tool (see section 2.1.2.10)
2	Mouse Sensitivity	Slider adjusts mouse sensitivity for model rotation in the display window. See section 2.1.1 for further description on mouse gestures.
3	Set View Angle	These buttons reset the model view in the display window. The Model will automatically scale to fit the display window.
4	Model Editor Buttons	These buttons open the model editors. See section 2.1.2 for a description of each editor.
5	Display Window	This window provides a graphical representation of the model in 3D space. The model can be manipulated with keyboard or mouse gestures (see section 2.1.1). See section 2.1.3 for a description of graphical objects rendered in the display window.
6	Global Coordinate Frame	The global coordinate frame is in the upper right-hand corner of the display window. The frame rotates with the model to show its current orientation.
7	Element Rendering	Checkboxes toggle if selected elements are rendered in the display window. See section 2.1.3 for a description of each item.
8	Load Case Rendering	Once load cases are defined, these controls allow them to be rendered in the display. See section 2.1.3 for a description of load and boundary graphical objects.
9	Analyze Button	Submits the model for analysis. All defined load cases will be analyzed.
10	Close Button	Close the FEA Load Tool safely (with prompt to save work).

Figure 2-1. Main FEA Tool Interface

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

2.1.1 Model Manipulation

Model display windows, such as the main display window and various preview windows, support mouse and keyboard gestures to manipulate the model in 3D space. Model rotations operate about invisible horizontal and vertical axes centered on the active model display window, so a combination of panning and rotating may provide the best way to orient the model to a desired position. The table below lists the model manipulation options supported.

Table 2-1. Keyboard/Mouse Gestures

Model Action	Keyboard and/or Mouse Gesture
Panning	• Arrow keys on keyboard • Hold left mouse button and drag
Zooming	• Cntl+ up/down arrow keys • Mouse wheel
Rotating	• Shift+ arrow keys • Hold right mouse button and drag

2.1.2 Model Editors

The buttons on the left side of the main window open model editors, which are used for model creation and editing. These editors have a similar look-and-feel to the database editors in ORBIS, where parameters are entered or viewed in the upper region of the dialog window and defined objects are listed in the lower section of the dialog. The subsections below describe each editor.

2.1.2.1 Grid Editor

The grid editor is opened by selecting the [Grids] button from the main window. Figure 2-2 shows the Grid Editor dialog. Grid points are the foundation of the finite model and prior to starting a model it is recommended to do some planning to establish these points. Items to consider during model planning are:

1. Model layout in the global coordinate frame.
    - Consider where to put the origin and how to orient the structure along the global X, Y, and Z axes. If possible, align structure such that most of it lies along one or more global axes.
        - Aligning bearing axes with global coordinates, although not required, may simplify the coordinate transformations for the bearing spring element definitions.
2. Key structure
    - Structural interfaces, particularly if they have boundary conditions, must have a grid point.
    - Bearing locations will be modelled with a spring element, which requires two grid points (one for each end of the spring). Bearing spring elements should use coincident grid points.
    - Structural transitions, such as a stepped shaft or bolted joint between two parts, where separate beam elements would better represent the structure, will need a grid point at the transition.
    - Lumped masses will require a grid point to locate the mass.
    - All load points require a grid point to apply the load to.

Once a grid point is defined and subsequently used as a connection point for a beam element, the properties of the beam element become dependent upon the grid point. If the grid point definition is then edited, perhaps moved to a different location, the attached beam element must be updated to reflect its new state. This dependency is checked by the Grid Editor and the user will be prompted to redefine any impacted beam elements. All other element types will move with the edited grid point without the need to be redefined.

How to add a Grid Point:

1. Open the Grid Editor by selecting the [Grid] button from the FEA Tool window.
2. Specify a unique grid ID number (must be an integer).
3. Define the global X, Y, and Z position of the grid point. This input should be numeric with commas to separate each ordinate value. Spaces between the number and comma are acceptable. Optionally, the grid point position can be defined relative to another point as follows:
    a. Enter the X, Y, and Z position of the reference point.
    b. Enter the radius from the reference point to the new point being defined.
    c. Enter the X-Y plane angle and Y-Z plan angle. Thes angles are the projected angles made by a line connecting from the reference point to the new point.
4. Select the [Add Grid Point] button. The new defined grid ID name will appear in the Grid List area.

How to edit an existing Grid Point:

1. Select the grid point to edit by clicking on the name of the grid point in the Grid List area (bottom of the Grid Editor dialog).
2. Select the [View/Edit] button. This will populate the Grid Parameters with the current settings for the selected grid point.
3. Make changes to the grid parameters.
4. Select the [Add Grid Point] button.
5. Select [OK] to confirm replacement of the existing grid point with the new definition.

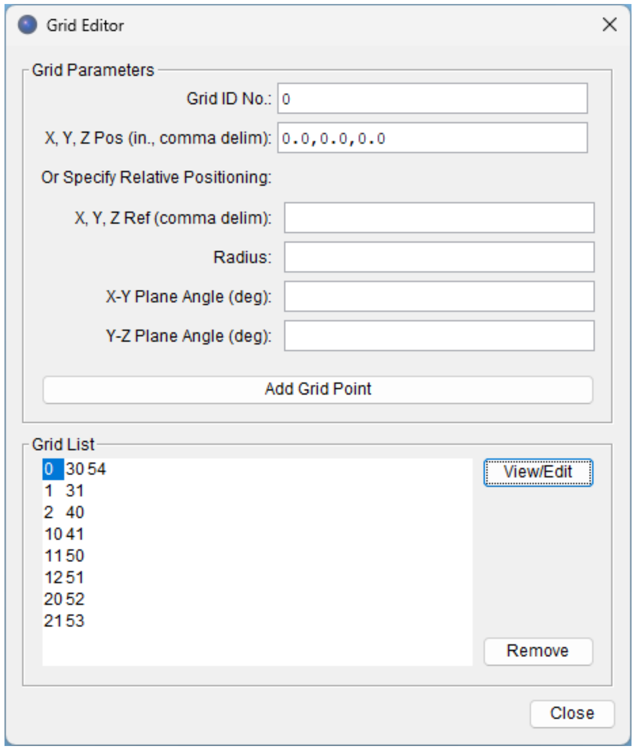


Figure 2-2. Grid Editor

2.1.2.2 Coordinate System Editor

The Coordinate System Editor is opened by selecting the [Coord. Sys.] button from the main window. This editor creates local coordinate reference frames. A global coordinate is provided by default and cannot be edited or removed. Figure 2-3 shows the editor.

How to add a Coordinate System:

1. Open the Coordinate System Editor by selecting the [Coord. Sys.] button from the FEA Tool window.
2. Specify a unique name for the coordinate system. The name can be a string containing all character types (including spaces).
3. Define the global X, Y, and Z position of the coordinate origin as a comma delimited point. Optionally, the origin can be defined by selecting a defined grid point from the drop-down menu.
4. Define the first axis of the new coordinate as follows:
    a. Select the X, Y, or Z radio button to specify which axis is to be defined.
    b. Either specify a point that is on the positive axis or specify the vector defining the orientation of the axis. To specify a point, use the first input field and type in the comma delimited X, Y, Z values. To specify a vector, use the second input filed and type in the vx, vy, vz vector components. The entered vector does not need to be a unit vector.
5. Define the second axis of the coordinate system in the same manner as the first axis. Note that whichever axis is defined by the first axis definition will not be available for defining the second axis.

6. Optionally select the [Preview Coordinate Frame] button to open a model preview window with the proposed coordinate frame rendered in model space. Close the preview window when finished.
7. Select the [Add Coordinate System] button. The new coordinate system name will appear in the Coordinate List area.

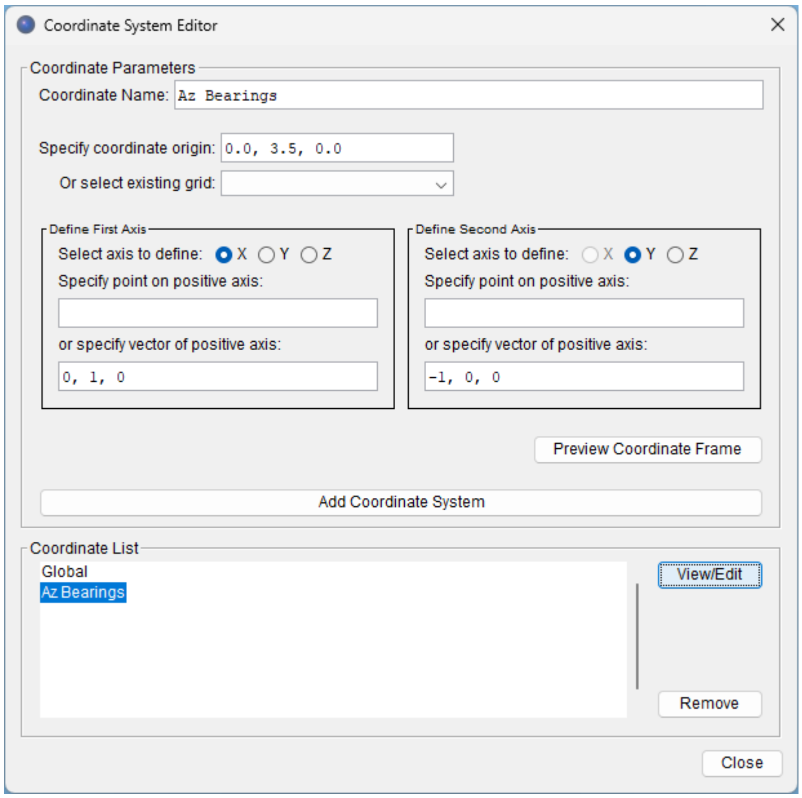


Figure 2-3. Coordinate System Editor

2.1.2.3 Beam Element Editor

The Beam Element Editor is opened by selecting the [Beam] button from the FEA Tool window. This editor creates structural beam elements between defined grid points (reference Figure 2-4).

Beam elements use a Beam Section to describe the cross section of the beam. The Beam Section Editor is shown in Figure 2-5 and is accessed within the Beam Element Editor. Four beam section types can be defined: cylindrical, rectangular, I-beam, and general. Each section type has a figure showing its dimensional nomenclature and reference frame convention (n1/n2 direction vectors). When creating a beam between two grid points in the model, the orientation of the section n1 vector must be specified.

How to add a Beam Element:

1. Open the Beam Element Editor by selecting the [Beam] button from the FEA Tool window.
2. Specify a unique name for the beam element.
3. Select the grid attachment points from the drop-down menus. The two specified grid points will be the end points for the created beam element.

4. Assign the beam section from the drop-down menu. If a new section needs to be created, select the [Edit Sections] button. See below instructions for how to create a new section.
5. Assign the beam material from the drop-down menu. If a new material needs to be defined, select the [Edit Materials] button to open the material editor. Once the new material is added and the editor is closed, focus will return to the Beam Element Editor.
6. Set the orientation of the beam by specifying the direction of the section n1 vector in global coordinates. Refer to the Beam Section Editor for diagrams of the n1 vector for each section type. The specified vector must be orthogonal to the beam axis. If the entered n1 vector is not normal to the beam axis, when adding or previewing the beam element, a warning will occur and a new vector, that is orthogonal to the beam axis, will be recommended.
7. Optionally select the [Preview] button to open a preview window with a rendering of the proposed beam element, along with its defined n1/n2 vector orientations. Previewing beam elements prior to committing them to the model is recommended.
8. Select the [Add Beam Element] button to add the beam element to the model. The name of the new beam element will appear in the Beam Element List area.

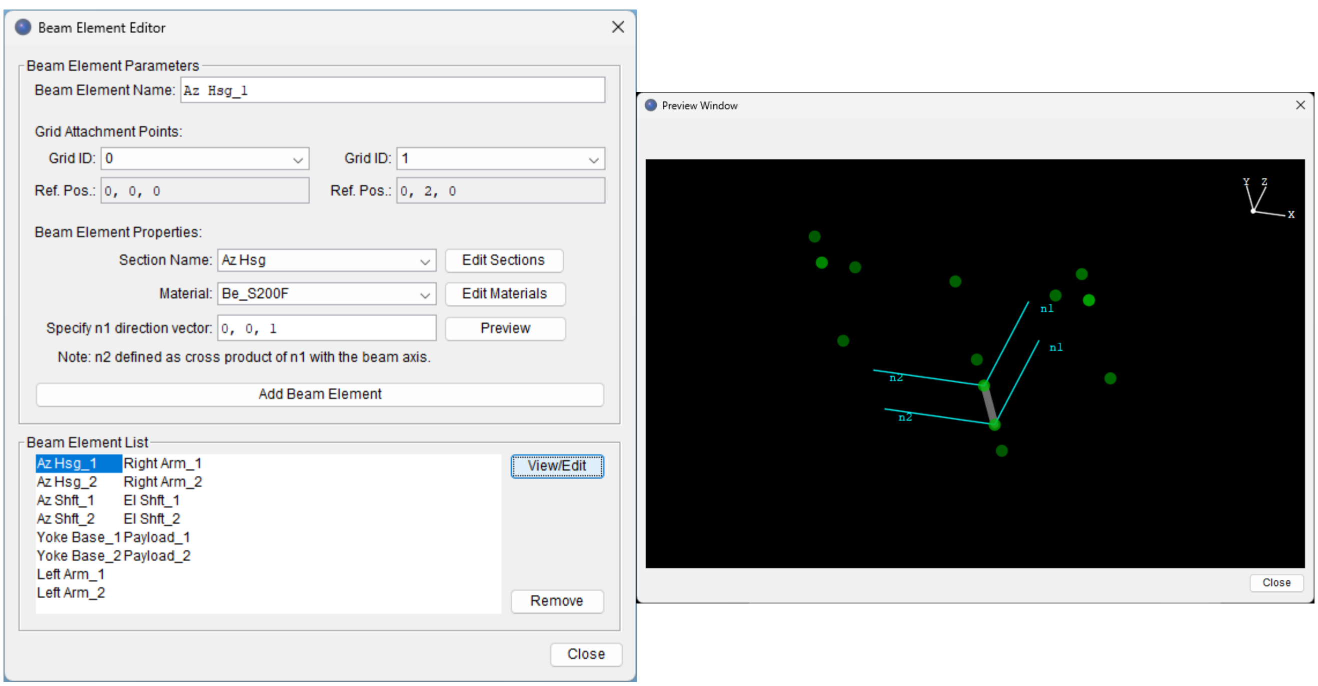


Figure 2-4. Beam Element Editor (with Preview Window)

<u>How to add a Beam Section:</u>

1. Open the Beam Section Editor by selecting the [Edit Sections] button from the Beam Editor.
2. Specify a unique name for the beam section.
3. Select the Cylindrical, Rectangular, I-beam or General tab to specify which type of section to create.

4. Complete the parameters for the selected section type. Refer to the corresponding section figure in the Beam Section Editor for definitions of each input. Also, take note of the n1 vector for use when defining the beam element in 3D model space.
5. Select the [Add Section] button to add the beam section to the model. The name of the new section will appear in the Section List area.
6. Select [Close] to close the Beam Section Editor and return to the Beam Element Editor.

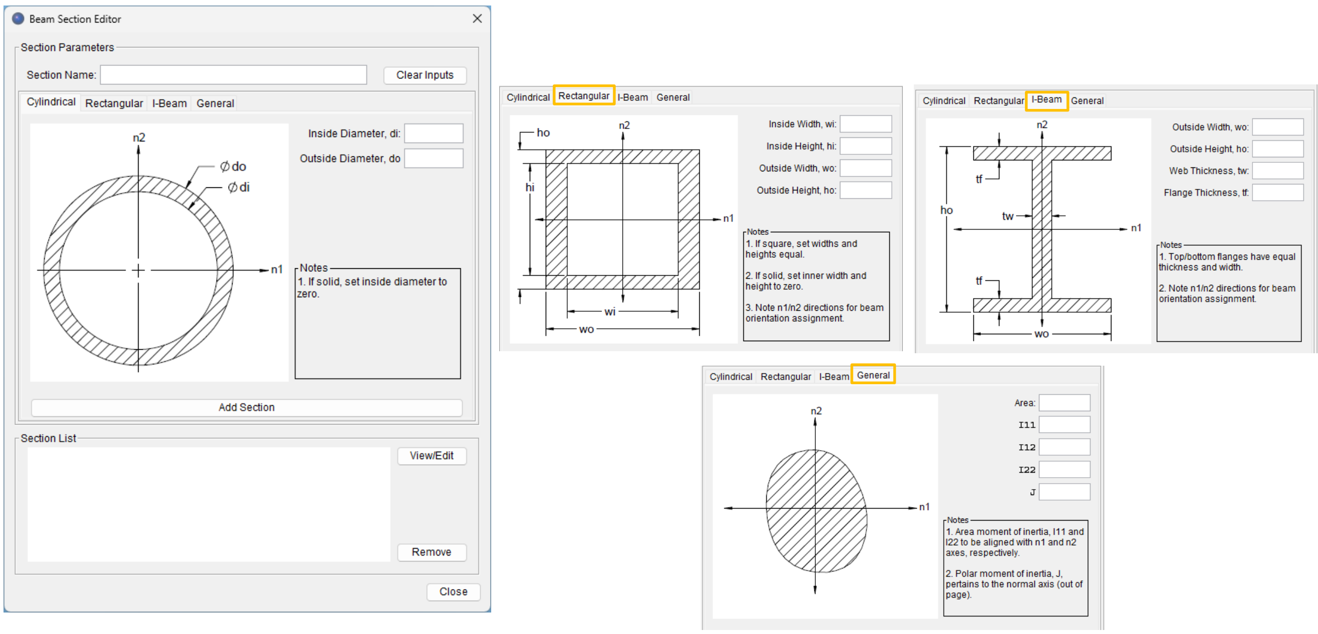


Figure 2-5. Beam Section Editor

2.1.2.4 Spring Element Editor

The Spring Editor is opened by selecting the [Spring] button on the main FEA Tool interface (reference Figure 2-6). Spring elements are linear stiffness elements that connect two grid points and have no mass. Their stiffness is defined by a 6x6 matrix that must be positive semi-definite (i.e. symmetric with diagonals that are positive or zero). Additionally, spring elements use a local coordinate reference frame to define how their stiffness terms relate to global coordinates (via transformations).

Spring force recovery is a key objective of the FEA Load Tool and is always output in the analysis results. Hence, by adding a spring element anywhere in the model, the user can get a force recovery output at the defined spring location.

Bearing stiffnesses can be imported into the spring editor via the Bearing Stiffness Editor (see Figure 2-7). The Bearing Stiffness Editor can only be accessed from within the Spring Editor by selecting the [Load Bearing Stiffness] button. When importing bearing stiffnesses, only the diagonal terms are imported, and K44, which is the torsional stiffness about the bearing rotation axis, is left blank and must be manually completed.

How to add a Spring Element:

1. Open the Spring Editor by selecting the [Spring] button.

Halpin
Engineering,
LLC

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

2. Select the grid attachment points by selecting the first and second grid IDs from the drop-down menus. If the grids for the spring have not been defined yet, select the [Edit Grids] button to open the grid editor and define them.
3. Select the local coordinate system for the spring element by choosing the coordinate system from the drop-down menu. The [Edit Coords] button can be used to define a new coordinate system.
4. Define the spring stiffness terms by either using the ORBIS non-linear bearing solver to compute stiffnesses of an existing bearing setup file or manually entering the stiffness values. See following procedure for how to use ORBIS to compute bearing stiffness values.
5. Select the [Add Spring] button. The new defined spring name will appear in the Spring List area.

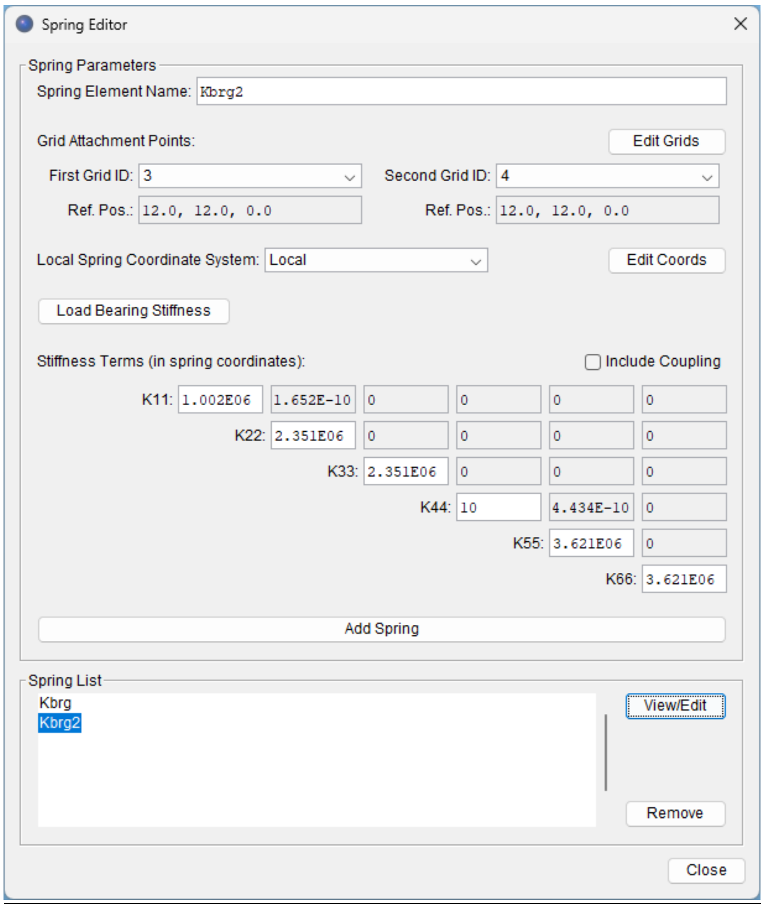


Figure 2-6. Spring Element Editor

<u>How to load bearing stiffness terms into a spring element:</u>

1. Select the [Load Bearing Stiffness] button from the Spring Editor dialog.
2. Select [OK] on the 'Bearing Spring Coordinates' message if the current spring element has specified a local coordinate system with its x-axis oriented along the bearing rotation axis. If this is not the case, select [Cancel] and correct the spring element's coordinate system.
3. The Bearing Stiffness Editor will open (see Figure 2-7 below).

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

4. Select the [Browse] button in the Bearing Stiffness Editor and open an ORBIS setup file containing the bearings to be analyzed for stiffness.
5. Select the rows to be included in the stiffness calculation by selecting the appropriate checkboxes in the first column of the table. Note: the table contains additional information from the setup file for reference.
6. Select configuration options from the following checkboxes:

Checkbox Title	**Description**
Remove external loading	This option clears any external loading defined in the setup file. It does not clear internal preloads.
Run in static mode	This option sets the setup file to run in static mode (no rotational velocity on the bearings).
Use ring compliance for axial stiffness	This option will use the 'axial stiffness with ring compliance' output from ORBIS if that output is available from the setup configuration.
Compute axial stiffness from secant line	This option overrides the axial stiffness method to use a 2-point secant method instead of the default 'tangent stiffness' method. If this option is selected, inputs for Fx1 and Fx2 must be provided. The solver will run each load, capture axial deflections, and compute a stiffness from (Fx2 – Fx1)/(x2-x1).

7. Optionally adjust the stiffness recovery point. This field automatically populates based on the selection of bearing rows and their defined location in the setup file. However, the user can override it by editing its value.
8. Select the [Compute Stiffness] button to run the ORBIS solver with the selected configuration and recover the selected bearing stiffness. The stiffness results will be shown in the Stiffness Results text area.
9. Select the [Assign Stiffness] button to copy the resulting stiffness values back to the Spring Editor. Notice the K44 term in the Spring Editor is left blank. This is the torsional stiffness term for the bearing and must be entered manually.

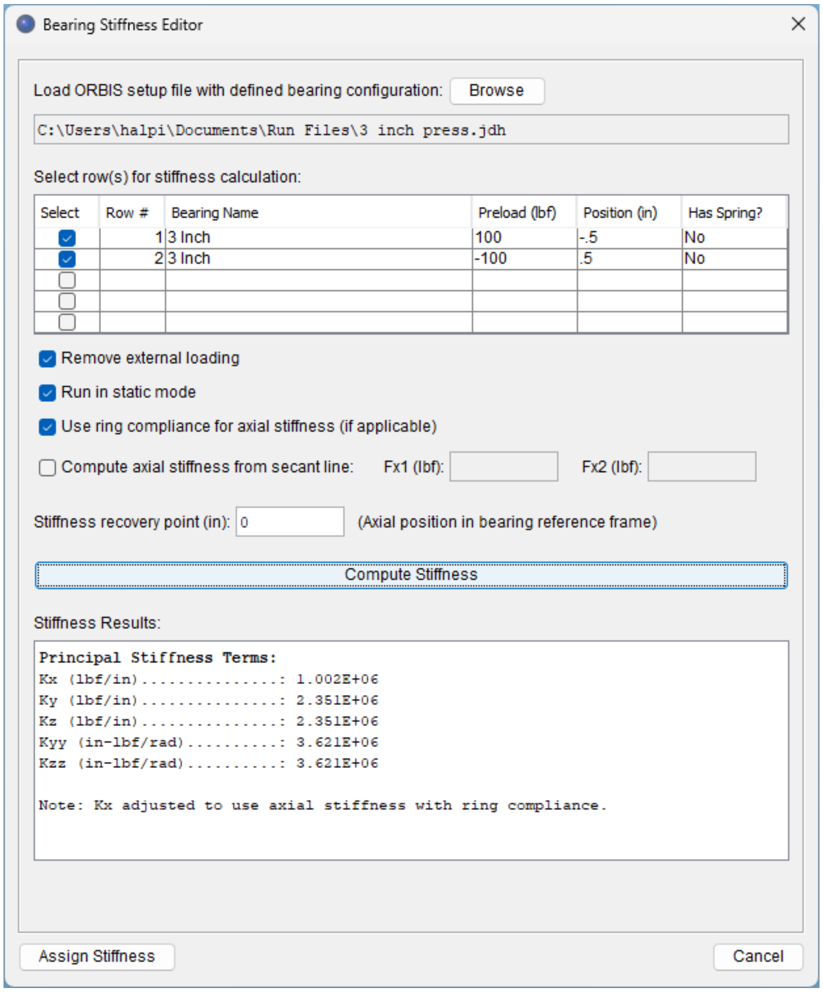


Figure 2-7. Bearing Stiffness Editor

2.1.2.5 MPC Editor

The MPC Editor is opened by selecting the [MPCs] button on the main FEA Tool window. This tool allows a one-to-many constraint between grid points. A common use for MPCs is when defining a random vibration analysis on a model that has more than one ground constraint point. In this case, one of the grids must be set up to have a fixed boundary condition and a MPC is used to connect it to all other grids that require a fixed boundary. The grid with the fixed boundary condition is the independent grid and the remaining grids are dependent grids.

How to add an MPC:

1. Open the MPC Editor by selecting the [MPCs] button
2. Specify a unique name for the MPC.
3. Select the independent grid from the drop-down menu.
4. Select all dependent grids in the dependent grid list area. The dependent grid list supports multi-select functions, such as CTRL+ click or SHFT+ click.
5. Specify the constraint properties by selecting the appropriate radio button (Fixed, Pinned or Other). If the [Other] button is selected, proceed to select the checkboxes for each degree of freedom to be constrained.

6. Select the [Add Constraint] button. The new defined MPC name will appear in the MPC List area.

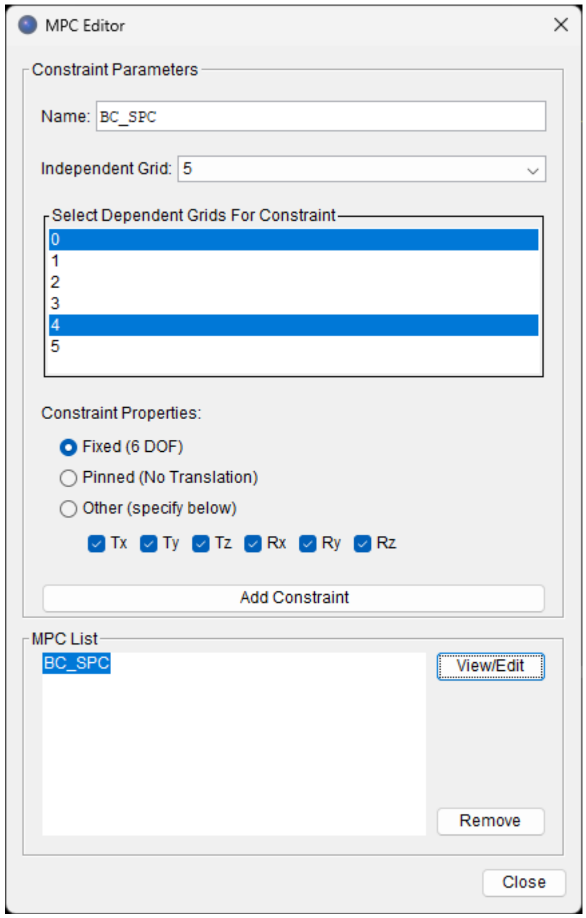


Figure 2-8. MPC Editor

2.1.2.6 Boundary Conditions

Boundary conditions, or constraints, must be defined for all analysis types in the FEA Tool. In general, the defined model must be fully constrained to prohibit free movement of elements within the model.

BCs are not immediately applied to the model when first created. Instead, they become active once added to a load case (see section 2.1.2.9 for the Case Manager). Therefore, multiple different boundary conditions can be defined in a model.

Random vibration analyses require a single grid point with a full 6-DOF boundary constraint. To add more constraint points to a random vibration problem, a multi-point constraint must be added (see section 2.1.2.5 for MPCs).

How to add an BC:

1. Open the BC Editor by selecting the [BCs] button from the main FEA Tool window.

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

2. Specify a unique name for the boundary condition.
3. Select all grids to apply the constraint to. The grid list supports multi-select functions, such as CTRL+ click or SHFT+ click.
4. Specify the constraint properties by selecting the appropriate radio button (Fixed, Pinned or Other). If the [Other] button is selected, proceed to select the checkboxes for each degree of freedom to be constrained.
5. Select the [Add Boundary Condition] button. The new defined BC name will appear in the Boundary Condition List area.

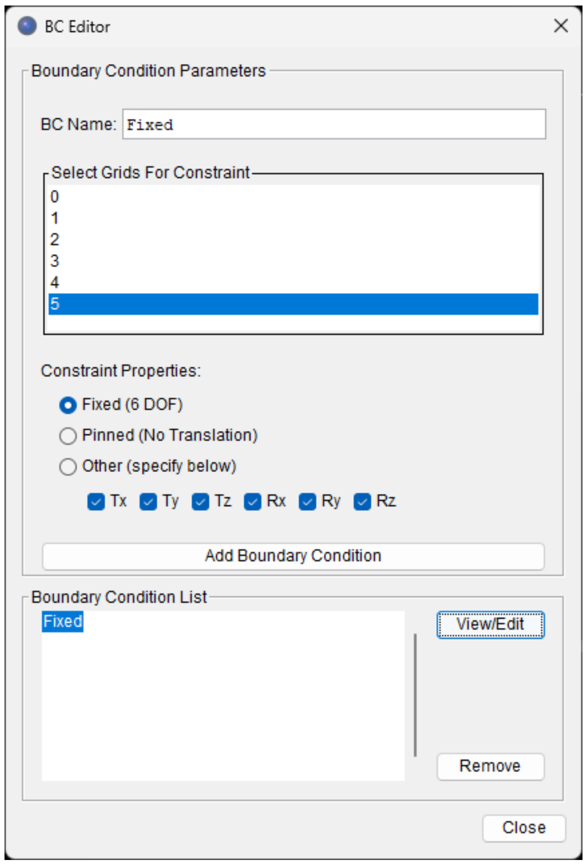


Figure 2-9. BC Editor

2.1.2.7 Load Editor

Like boundary conditions, loads are not immediately applied to the model when defined in the Load Editor. Instead, they become assigned to the model once they are added to a load case (see section 2.1.2.9). Hence, the user can create multiple different loads to be later set up and analyzed independently with the Case Manager.

Three types of loads can be defined: static forces or point loads, mass acceleration loads, and random vibration loads. Each type is discussed in the following sections.

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

2.1.2.7.1 Static Forces

Static forces are full 6 DOF loads that can be applied to any grid point in the model. They are defined with a coordinate reference, which allows the user to define component magnitudes in any reference frame desired. Furthermore, as discussed in section 2.1.2.9, multiple point loads can be applied either simultaneously or individually.

How to add a Static Force:

1. Open the Load Editor by selecting the [Loads] button from the FEA Tool window.
2. Specify a unique name for the load.
3. By default, the Static Force tab is selected (first tab), so select the load point grid ID from the drop-down selection.
4. Select the reference coordinate system for the load from the drop-down selection.
5. Fill in the load component values for Fx, Fy, Fz, Mx, My and Mz. Note that blank entries are not allowed, so enter zeros for components with no load.
6. Select the [Add Force] button. The new defined load name will appear in the Load List area.

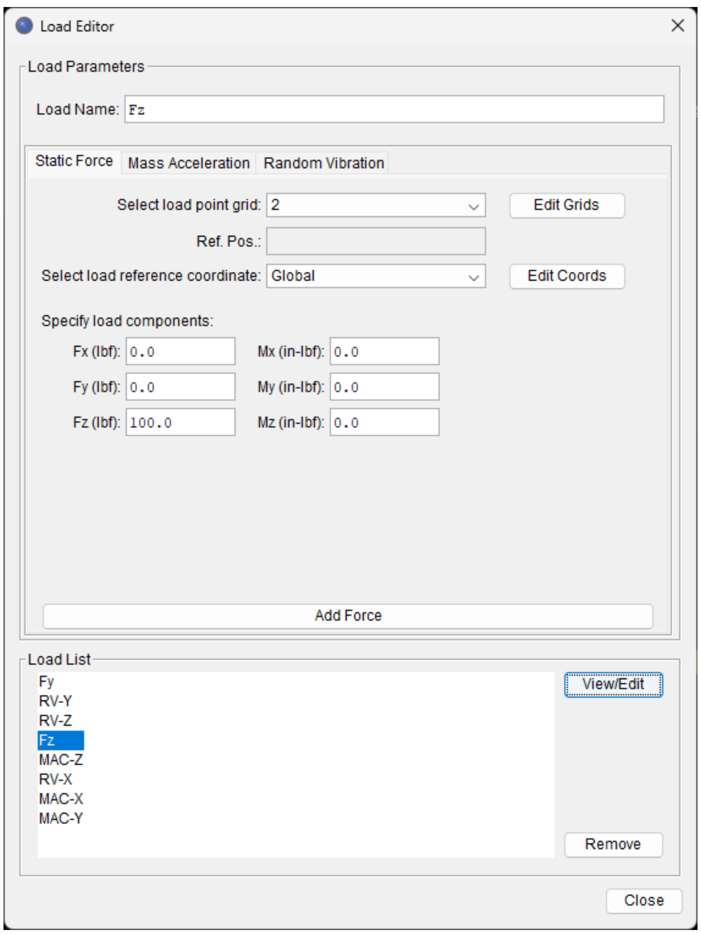


Figure 2-10. Load Editor - Static Force

2.1.2.7.2 Mass Acceleration Loads

Mass acceleration loads are acceleration fields applied to the distributed mass of the model, which is a static analysis. The acceleration level, which is expressed in G's (multiples of gravity), is defined by entering tabular data of a mass-acceleration curve (aka MAC). The solver will compute the mass properties of the model, based on the defined elements, and automatically interpolate the g-load from the defined mass-acceleration curve. Interpolation options include linear and natural cubic. The direction of the acceleration field can be defined by a direction vector, which allows all possible directions to be defined, or by selection the +/- directions of the global coordinate axes.

How to add a Mass Acceleration Load:

1. Open the Load Editor by selecting the [Loads] button from the FEA Tool window.
2. Specify a unique name for the load.
3. Select the Mass Acceleration tab in the Load Editor.
4. Specify the direction of the acceleration field by either selecting the appropriate principal coordinate direction radio button or selecting the [Specify direction vector for acceleration] option and entering the direction vector.
5. Select the mass units: either Kg or Lbs (weight mass).
6. Enter at least 3 mass-acceleration points in the table. The data in the table should have mass entries that envelop the actual model mass.
7. Select the interpolation method (Linear or natural cubic spline).
8. Optionally select the [Plot MAC] button to see a plot of the mass-acceleration curve (see figure below for example). The generated plot will show large, circled poinst for each defined point in the table and a plurality of intermediate points between the table data to illustrate the interpolation.
9. Select the [Add Mass Acceleration] button. The new defined load name will appear in the Load List area.

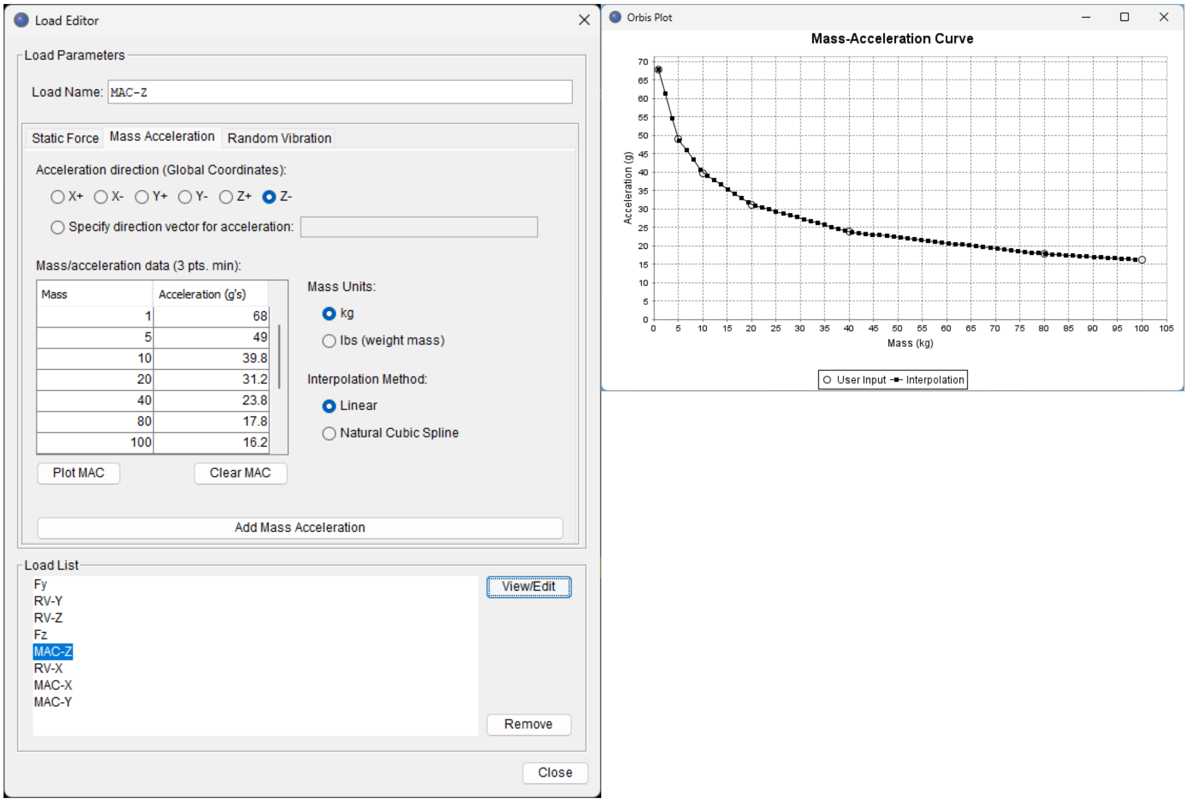


Figure 2-11. Load Editor - Mass Acceleration

2.1.2.7.3 Random Vibration Loads

Random vibration loads are created in the FEA Tool by defining an input power spectral density (PSD) curve and frequency based modal damping curve. The input PSD must represent a stationary (zero mean) ergodic random process and is treated as a base motion acceleration on the model.

How to add a Random Vibration Load:

1. Open the Load Editor by selecting the [Loads] button from the FEA Tool window.
2. Specify a unique name for the load.
3. Select the Random Vibration tab in the Load Editor.
4. Specify the shake axis direction by selecting the appropriate global coordinate direction radio button.
5. Complete the input PSD table by entering frequency, in Hertz, and acceleration density, in g^2/Hz, data points. Note: Select the [Calc. GRMS] button to show the total Grms of the PSD input (area under the curve).
6. Specify the damping ratio as a function of frequency. It is recommended to define damping across the entire frequency range specified in the PSD table. However, if a frequency is outside of the damping definition, the closest defined damping will be used.

Halpin Engineering, LLC

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

7. Optionally adjust the maximum frequency for modal analysis (default is 4000 Hz). This input defines the maximum frequency for modal extraction. To get sufficient modal mass participation, the maximum frequency may need to be adjusted.
8. Optionally adjust the number of solution points per eigenvalue interval (default value is 20).
9. Optionally adjust the bias parameter (default value is 3).
10. Select the [Add RV Load] button. The new defined load name will appear in the Load List area.

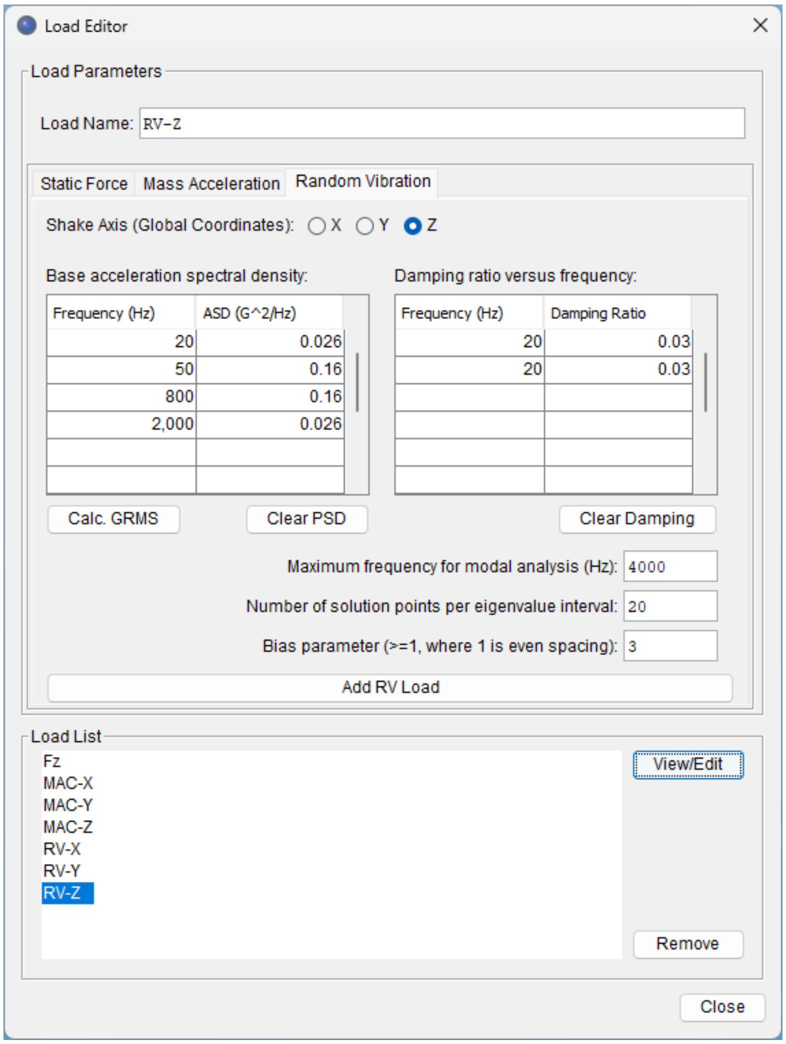


Figure 2-12. Load Editor - Random Vibration

2.1.2.8 Mesh Editor

The Mesh Editor is opened by selecting the [Mesh] button on the main FEA Tool window. This editor is used to refined finite element size for beam elements.

For random vibration load cases, which use linear modal dynamic theories, the element size can influence the accuracy of the solution. As the model is discretized into smaller elements, the mass/stiffness distribution is refined, which creates more degrees of freedom (DOF) for the eigenvalue problem to solve. A larger model DOF means more eigenvalue/eigenvector pairs can be found, which can then influence the modal superposition techniques used for a random response analysis. However, due to floating point precision errors and the non-linear computation time with model size, there will be an upper limit to benefits gained from refining element size.

Conversely, for all static analyses, such as point load and mass acceleration load cases, the element size does not change the accuracy of the solution, so meshing can be omitted entirely.

How to use the Mesh Editor:

1. Open the Mesh Editor by selecting the [Mesh] button from the FEA Tool window.
2. Select one or more beam names from the top list (titled 'Select Elements To Seed/Mesh'). This list window supports multi-select options (CNTL+ click or SHFT+ click).
3. Select the seed method. Options are 'By max size' and 'By element number'. The size option specifies the maximum distance between nodes while the number option specifies how many elements to break the beam into. When applying a mesh to multiple beam elements at once, the specified mesh control will apply to each beam element independently. For instance, if two beams are selected and meshed by element number, then each beam will be divided into the specified number.
4. Specify the desired size or number in the input field. This input has different units depending on the selected seed method. If the seed method is 'max size', then the units on this input are in inches. If the seed method is 'element number', then this input must be an integer and is unitless.
5. Optionally select the [Preview Mesh] button to open a preview window and display the defined mesh seeds. After reviewing the mesh seeds, select the [Close] button to close the preview window.
6. Select the [Mesh Selected Elements] button to update the model with the specified meshing. The element names that are meshed will appear in the bottom list window (titled 'Select Meshed Elements To Preview or Remove'). Furthermore, these elements will be removed from the upper list window. If all elements in the model have been meshed the upper list window will be blank and the lower list window will show all defined beam element names in the model.

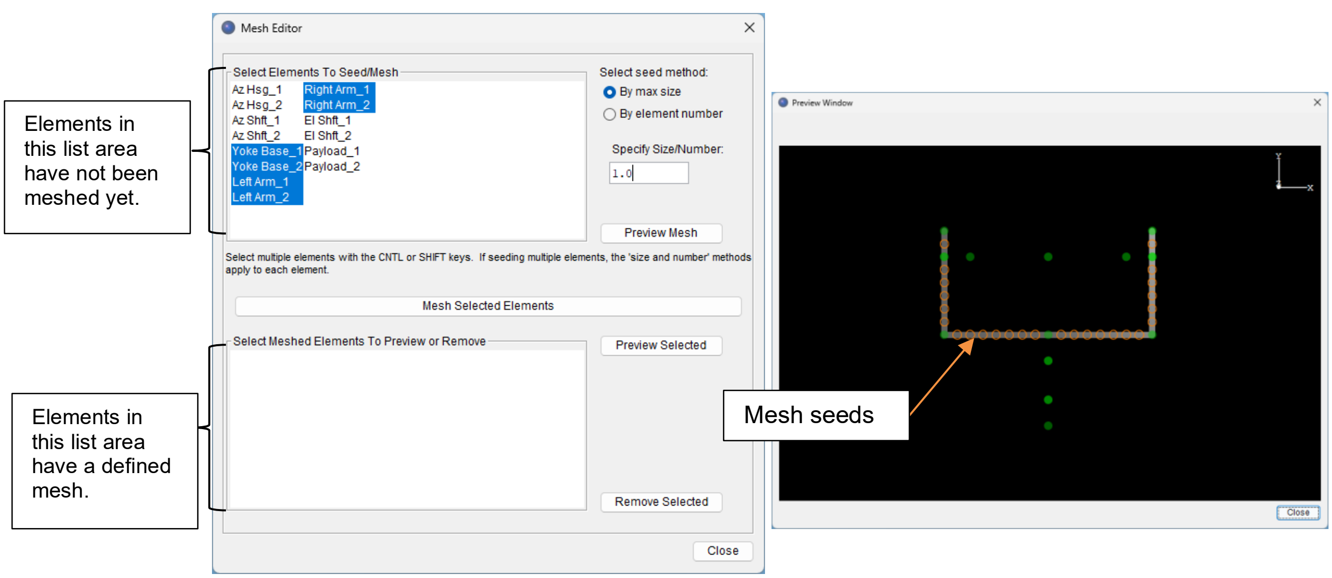


Figure 2-13. Mesh Editor

2.1.2.9 Load Case Manager

The Load Case Manager is opened by selecting the [Case Manager] button on the main FEA Tool window. This editor is used to specify load cases to be analyzed. Each load case must consist of a load and corresponding boundary condition. Multiple load cases can be defined, and all defined cases will be batch processed.

Load cases have restrictions based on the type of load used. When a random vibration load is used, there can only be one boundary condition and random load added to the case. The random vibration boundary condition should be fixed in all degrees of freedom. If the loading is static, such as static forces or mass acceleration loads, then multiple boundary conditions and load points can be added to a single load case.

If only one BC is defined when opening the Load Case Manager, and there are no existing load cases defined, a dialog box will present an option to automatically generate load cases (one case for each defined load). Selecting [Yes] in the dialog will generate the load cases automatically and selecting [No] will not. If load cases are automatically generated, they will be named LC1-XXX, LC2-XXX,…, where the XXX is the name of the defined load (from the Load Editor).

How to add a load case:

1. Open the Load Case Manager by selecting the [Case Manager] button from the FEA Tool window.
2. Specify a load case name for the case to be created.
3. Select a defined load from the load drop-down menu.
4. Select the [Add Load] button to add the load to the case. The load will appear in the ‘Loads’ list area. Note: if the load type is a static force (i.e. a point load), multiple loads can be added to a single load case by repeating steps 3 and 4. This option only applies to static forces.
5. Select a defined boundary condition from the BC drop-down menu.
6. Select the [Add BC] button to add the boundary condition to the case. Multiple BCs can be applied to any static type loading (point loads and mass acceleration loads). A random vibration type load must have only one BC.
7. Select the [Add Load Case] button to create the load case. The new load case name will appear in the Load Case List window area.

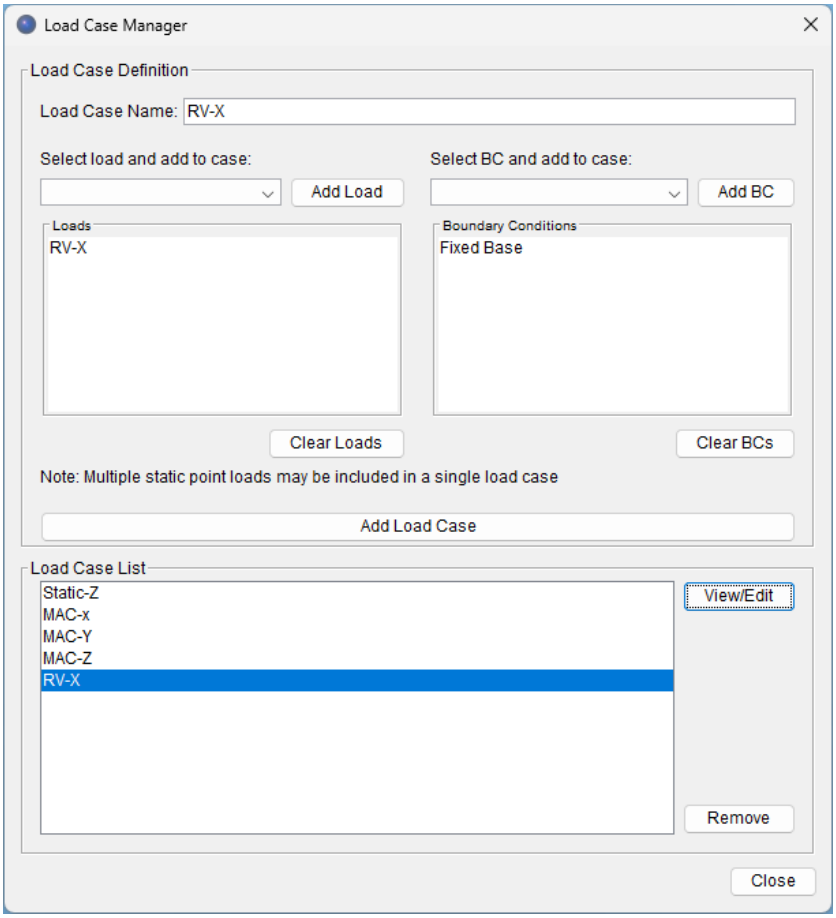


Figure 2-14. Load Case Manager

2.1.2.10 System Mass Tuning

The Mass Tuner tool can be used to adjust mass properties of a model. The tool, as shown in Figure 2-15, provides a list of all beam and mass elements defined in the model and allows the mass of these elements to be adjusted with a scale factor. The tool can also be used to calculate mass properties of the model, including queries on a selection of specific elements.

Mass adjustments, in the context of object data, are implemented differently depending on the element type being adjusted. Mass elements, when scaled with the tool, will update the mass element properties permanently (old values are overwritten and saved with new values). Beam elements, however, keep a scale factor field in their object data that saves the applied scale factor. This difference is provided so that material density, which is defined in the material database, is not altered.

When the Mass Tuner is opened, it pulls the list of beam and mass elements from the defined model and populates a default scale factor for each element. Mass elements will always have a default scale factor of 1. Beam elements, however, use the saved scale factor field from the element object data. If the scale factor for a beam element has never been adjusted, its value will be 1.

How to use the Mass Tuner:

1. After creating a model with at least one beam or mass element, open the Mass Tuner by navigating to [Tools Menu] → [Mass Tuner] from the FEA Tool window.
2. To compute mass properties of the entire model, select the [All elements] radio button and click the [Calculate] button to see the mass properties results for the entire model. To compute mass properties of a subset of elements, first select the elements to be included in the calculation, then click the [Selected elements] radio button and, finally, select the [Calculate] button.
3. To make temporary adjustments to the mass of an element, locate the element in the list and edit the [Scale Factor] cell. After adjusting scale factors, updated mass properties can be calculated by repeating step 2 above. Iterate on scale factors until the desired mass properties are achieved.
4. To commit the element mass adjustments to the model, select the [Apply Scaling] button. A dialog will appear asking to confirm changes to the model, select [OK] to commit the changes or [Cancel] to abort.

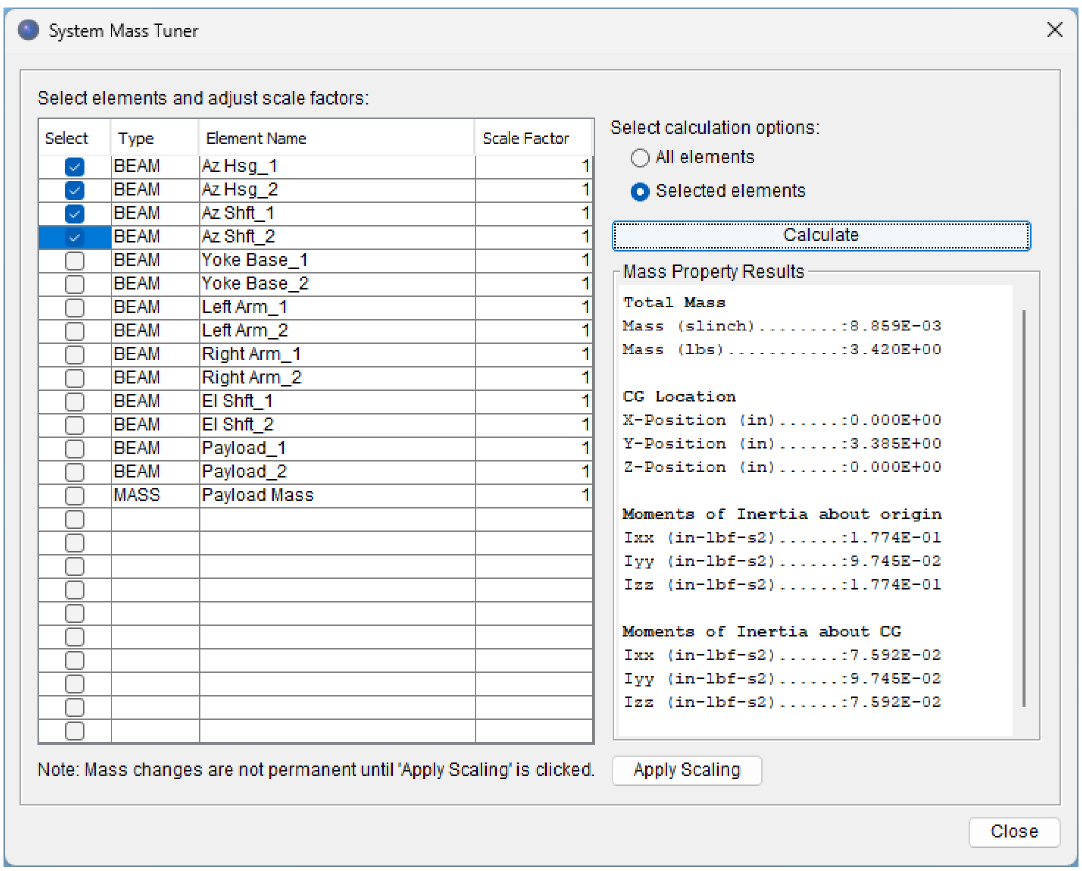
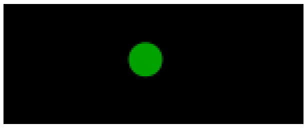
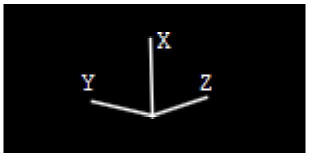
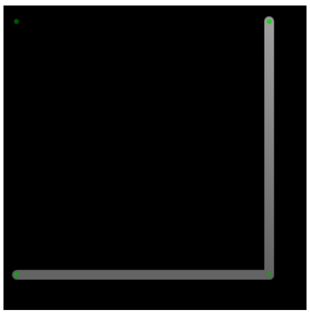
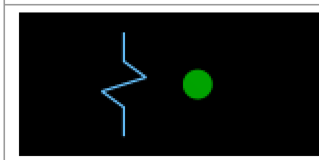
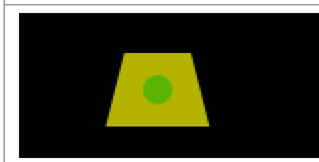
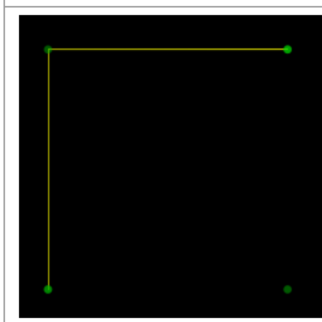
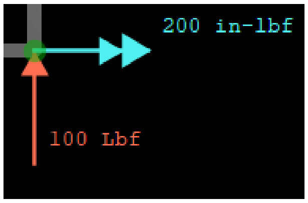
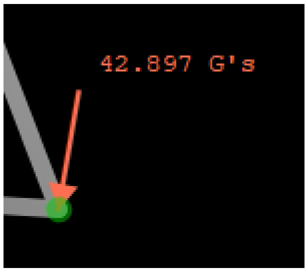
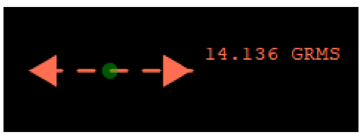
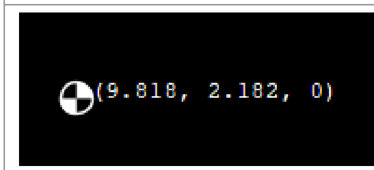


Figure 2-15. System Mass Tuner

2.1.3 Model Graphics Objects

As a model is developed with the model editors, the display area renders the model with certain graphical objects. These rendered objects provide a visual representation of the defined model. The table below provides a description of each graphical object type.

Graphics Object	Description
	Grid Points are identified as green dots and positioned by their defined coordinates.
	Local Coordinate Systems are rendered at their specified center coordinate and orientation.
	Beams are shown with a thick greyscale gradient line connecting the beam’s end point grids.
	Spring Elements are symbolized with a blue ‘resistor’ icon that is placed to the left of their attachment grid points.
	Mass Elements are symbolized with a yellow ‘anvil’ and are located on their attachment grid point.
	MPCs are drawn as thin yellow connecting lines. The lines connect all dependent grid points back to the independent grid point.
	Boundary Constraints are represented with the icon shown. The numbers in this icon represent the fixed degrees of freedom defined in the BC, so they will be unique to the BC definition. The example shown has all 6 DOFs fixed.

	Static shear forces are shown with an orange arrow, and static moments are shown with a light blue double-headed arrow. Shear forces have the tip at the grid point and moments place the tail at the grid point. Load magnitudes are displayed with text next to the arrows.
	Mass acceleration loads are shown as a static shear force arrow. However, they will appear on all grid points in the model and provide the interpolated g-value instead of force values.
	Random vibration loads are shown with a dashed bi-directional arrow and the total Grms value is printed. Random vibration load icons are centered on the grid point containing the fixed BC and oriented along the defined shake direction.
	The center of gravity icon, which includes the (x, y, z) position, is shown here.

2.2 Finite Element Analysis Results

FEA results are generated by selecting the [Analyze] button on the lower left corner of the FEA Load Tool window. If all defined load cases are successfully solved, the FEA Results Window will appear as shown in Figure 2-16. This window provides three tabs: [FEA Result Preview], [Direct Bearing Analysis], and [Batch File Export]. These three functions of the FEA Results Window are discussed in the following subsections.

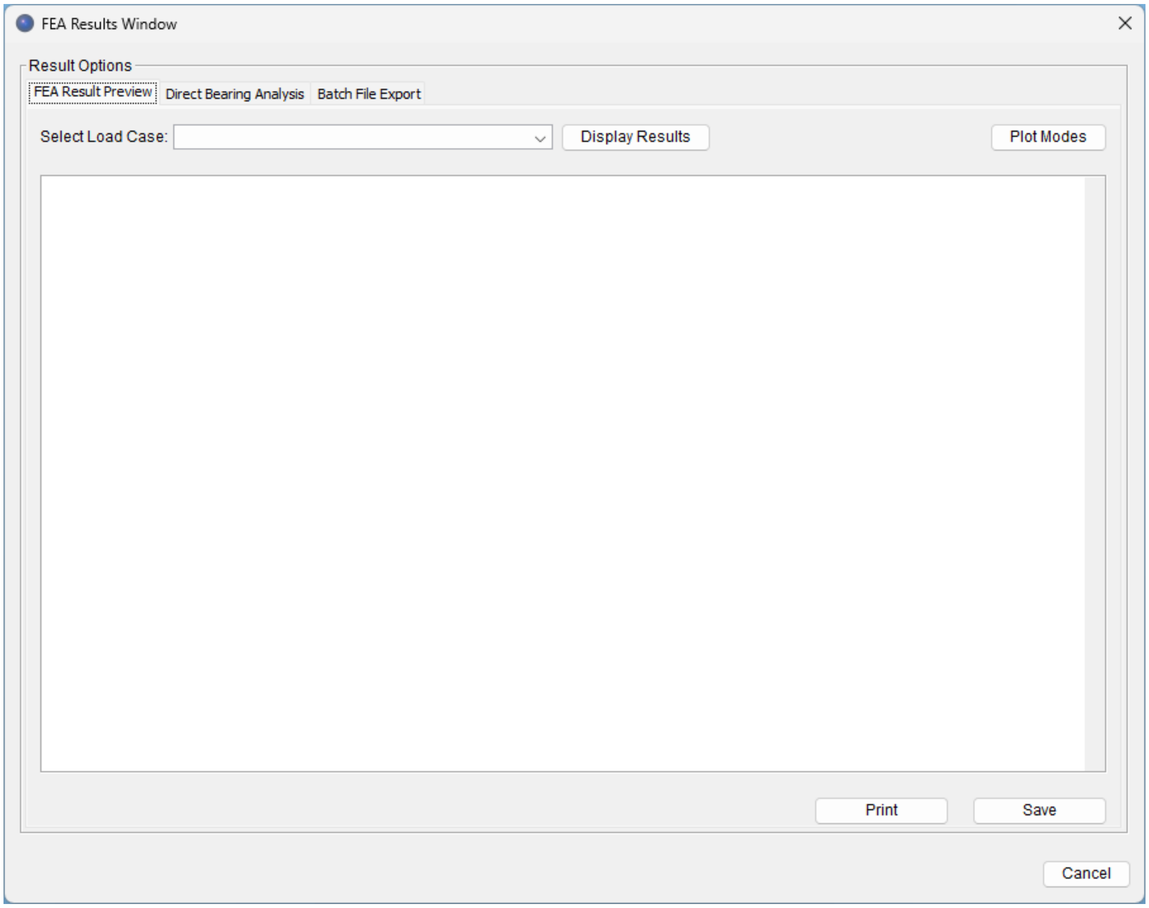


Figure 2-16. FEA Results Window

2.2.1 FEA Result Preview

The first tab of the FEA Results Window, titled FEA Result Preview, will automatically be selected when the window is first shown. This tab is used to see a text-based report of each defined load case. These reports can be printed or saved to a text file. Additionally, for random vibration load cases, the [Plot Modes] button can be selected to open a new window and render mode shape animations.

The text-based report contains different result parameters depending on the type of load case shown. The table below shows the different types of results provided.

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

Table 2-2. Result Output Descriptions

Load Case Type	Analysis Output
Static Force	Mass properties • Total mass, CG location, moments of inertia about both the CG and the origin External Forces • Load name, coordinate system, applied grid ID, and force components Spring Forces • Complete 6 DOF load components for each spring element in the model Grid Point Deflections • Complete 6 DOF grid point deflections for every grid in the model
Mass Acceleration	Mass properties • Total mass, CG location, moments of inertia about both the CG and the origin External Forces • Load name, coordinate system, acceleration (g's), and the acceleration direction vector. Spring Forces • Complete 6 DOF load components for each spring element in the model Grid Point Deflections • Complete 6 DOF grid point deflections for every grid in the model
Random Vibration	Mass properties • Total mass, CG location, moments of inertia about both the CG and the origin External Base Motion Acceleration • Load name, shake axis, total Grms, PSD table, and the damping table Eigenvalues Modal Effective Mass • Mass component for each mode, mass summation across the modes, and the fraction of total mass for the translational directions. RMS Spring Forces • Complete 6 DOF RMS load components for each spring element in the model

How to show text-based load case results:

1. Select the [FEA Result Preview] tab in the FEA Results Window if it is not already selected.
2. Select a load case from the drop-down menu of load cases.
3. Select the [Display Results] button to post the results to the text area of the dialog.
4. Scroll through the results to review them.
5. Optionally, select the [Print] or [Save] button to send the displayed load case results to a printer or save them to a file.

How to view mode shape animations:

1. Select the [FEA Result Preview] tab in the FEA Results Window if it is not already selected.
2. Select a random vibration load case from the drop-down menu of load cases. IMPORTANT: the selected load case must be a random vibration load case to see mode shapes.
3. Select the [Plot Modes] button to open the Modal Preview Window. See Figure 2-17 below.
4. In the Modal Preview Window, select the mode to animate from the drop-down menu.

5. Select the [Plot Mode Shape] button. The preview window should animate the mode shape. In some cases, it may be necessary to rotate the model to get a better perspective on the mode shape.
6. Optionally, select the [Playback Speed] slider to change the animation frequency.
7. Select [Close] when finished reviewing the mode shapes to return to the FEA Results Window.

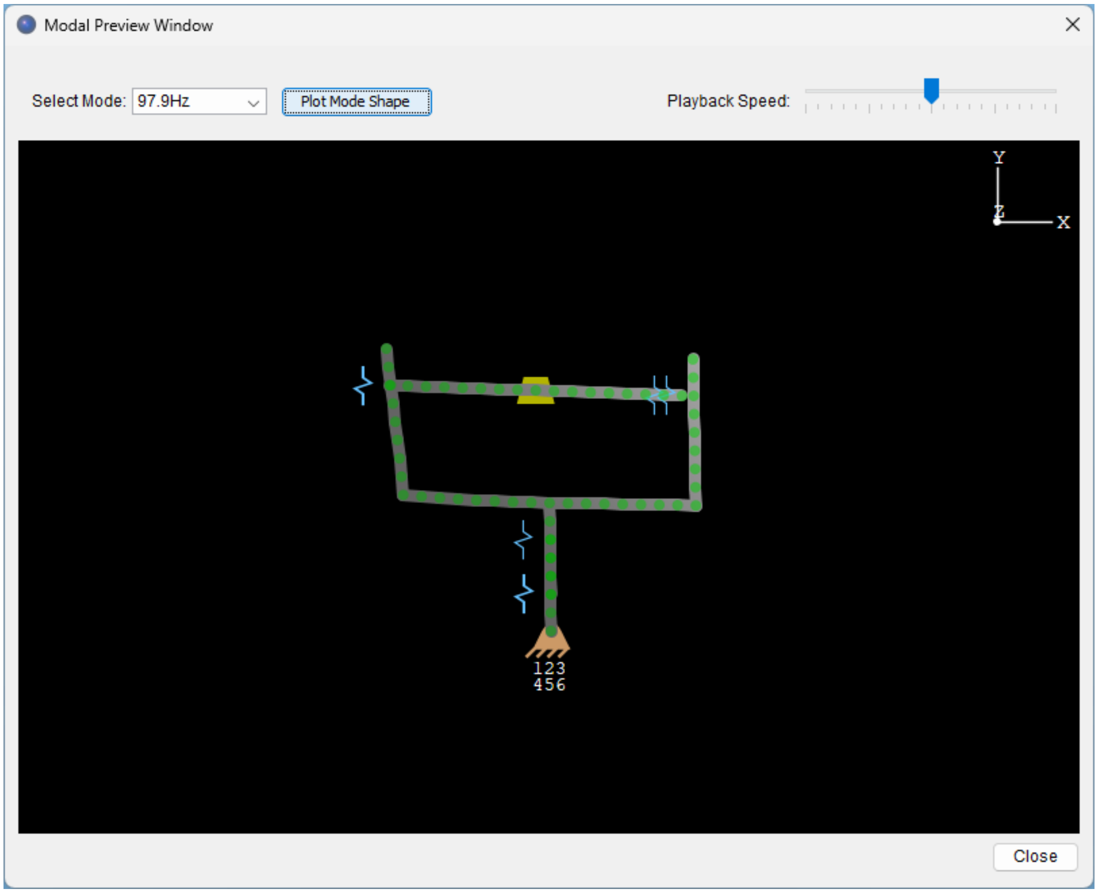


Figure 2-17. Modal Preview Window

2.2.2 Direct Bearing Analysis

The Direct Bearing Analysis tab in the FEA Results Window provides a convenient way to apply spring recovery forces from the finite element model on an existing ORBIS bearing setup. The process automates how it one would do this manually: loads are gathered from the finite element model results and inserted into one of the load points within the ORBIS bearing setup and then the bearing are analyzed. See the following procedure for how to do a direct bearing analysis and Figure 2-18.

How to directly analyze a load case with an ORBIS bearing setup file:

1. Select the [Direct Bearing Analysis] tab in the FEA Results Window.
2. Select the [Browse] button and use the file dialog to select the ORBIS bearing setup file to be used (*.jdh file). This setup file should be configured for a bearing, or system of bearings, that is contained within the mechanism modelled with the FEA Load Tool.

3. Select the load case to analyze from the drop-down menu.
4. Assign an FEA load recovery spring element to an ORBIS setup file load point. ORBIS setup files can have up to 3 different load points on the shaft. In the Direct Bearing Analysis window, the three columns aligned with the table represent the three possible load points for assignment to the ORBIS setup. Most ORBIS setups will only use the first load point. Assuming only one load point, select the spring element in the first column drop-down menu that represents the bearings in the selected ORBIS setup from step 2 above.
5. Select the [Add to Load Pt. 1] button. This will populate the Load Pt. 1 column of the table with the selected spring reaction forces for the selected load case.
6. Specify a Factor of Safety in the input field.
7. Optionally, select the [Save bearing setup file…] checkbox. If this is selected, after running the loads, a Save-As dialog will appear to save the ORBIS setup file with the new load.
8. Select the [Analyze Bearings] button. If the bearing analysis was solved, a standard ORBIS Results Window will be shown with the bearing analysis results.

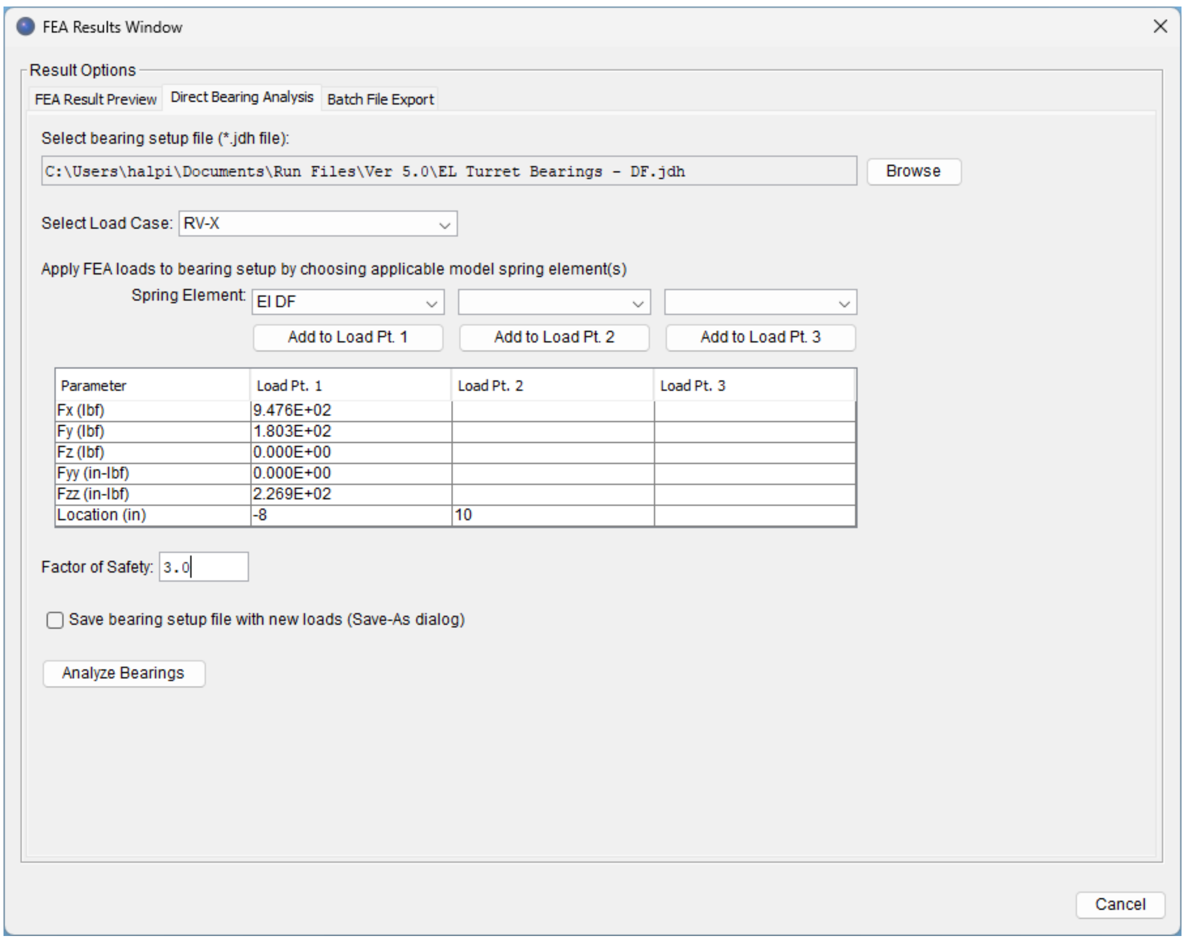


Figure 2-18. FEA Results Window - Direct Bearing Analysis

2.2.3 Batch File Export

The Batch File Export tab in the FEA Results Window is used to write load case results to a formatted ORBIS batch file. The interface is shown in Figure 2-19.

How to export loads to a batch file:

1. Select the [Batch File Export] tab in the FEA Results Window.
2. Select the load case to export from the drop-down menu.
3. Select the spring element from the drop-down menu.
4. Optionally, select the [Generate all +/- sign combinations…] checkbox. Selecting this checkbox will write 32 load cases to the batch file. Each load case will have the same component magnitude, but the sign convention will be altered to cover all sign combinations.
5. Optionally, select the [Scale load components] checkbox and specify the load factor. This option is useful when writing multiple different load cases to a common batch file. For instance, random vibration load cases may carry a 3-sigma scale factor while other load cases may not. Hence, each load case can be individually factored such that the final batch file is ready for direct analysis with no further factoring.
6. Enter the load position, in ORBIS setup file coordinate frame, corresponding to the selected spring element from the finite element model.
7. Select the [Browse] button to either select an existing batch file or specify a filename and location for a new batch file.
8. Select the appropriate radio button to either append this load case to the batch file or overwrite it.
9. Select the [Write Batch File] button to write the file. A confirmation dialog box will confirm the file was written.

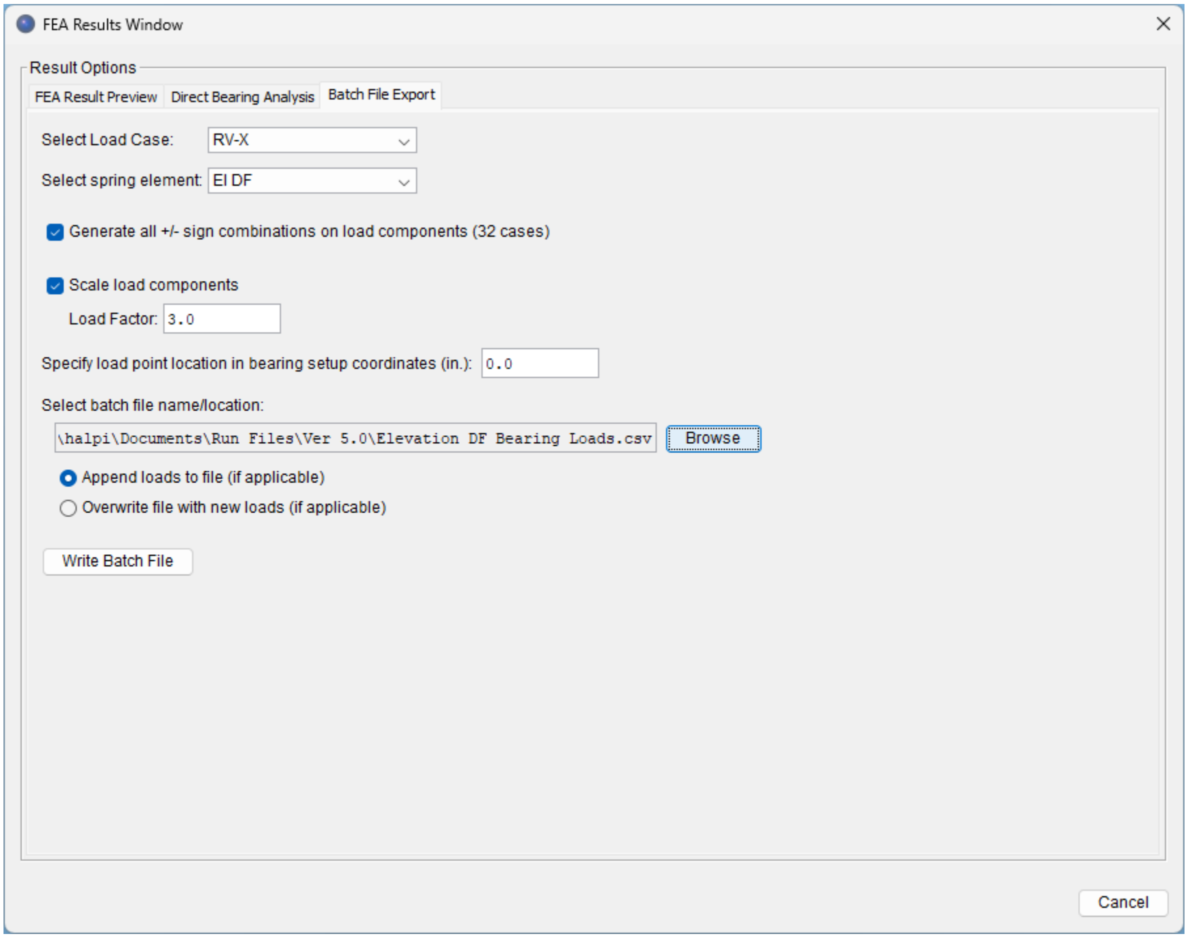


Figure 2-19. FEA Results Window - Batch File Export

3 Example Problem

The biaxial gimbal mechanism illustrated in Figure 3-1 will be modelled with the FEA Load Tool. This mechanism has two identical single axis rotary drive mechanisms configured in a right angle. Each drive mechanism contains a rigidly clamped and preloaded duplex pair of angular contact bearings.

This example is selected to provide a comprehensive usage case for the tool, including modelling a complex structure, running quasi-static and random vibration loading, and analyzing the bearings with recovered bearing loads. Upon completion of this example, the user should be able to use the tool to build and analyze their own models.

For experienced ORBIS users, who are familiar with setting up bearing analyses, this example is estimated to take about 2 hours to complete. The process outline is:

1. Setup the bearing configuration in ORBIS
2. Gather the mechanism structural layout information
3. Create the model
4. Tune the mass properties of the model
5. Define boundary conditions, loads and load cases
6. Run the analysis
7. Analyze the bearings with recovered loads

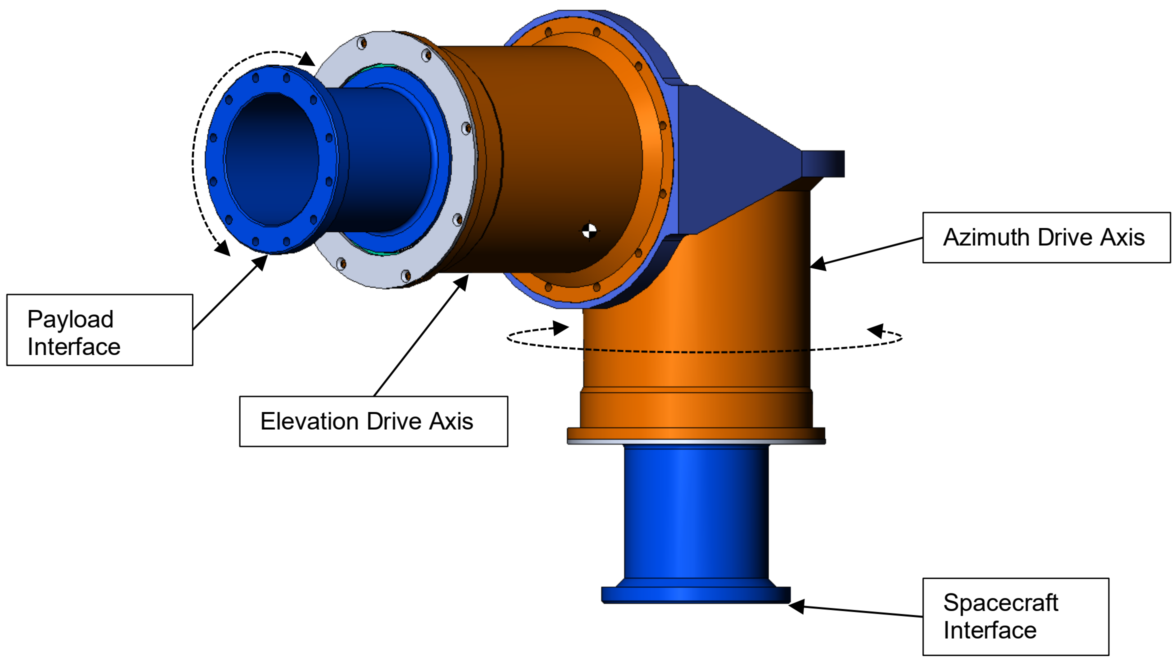


Figure 3-1. Biaxial Gimbal Example

3.1 Bearing Setup

The azimuth and elevation drive mechanisms have identical bearing configurations, so only one bearing configuration needs to be set up in ORBIS. The bearing pair will be configured in the mounted and preloaded state, which defines their stiffness for subsequent mechanism load events. Follow these steps to set up the bearings:

1. Open ORBIS.
2. Create a <440C Stainless> and <Ti-6Al-4V> material in the material database as shown below.

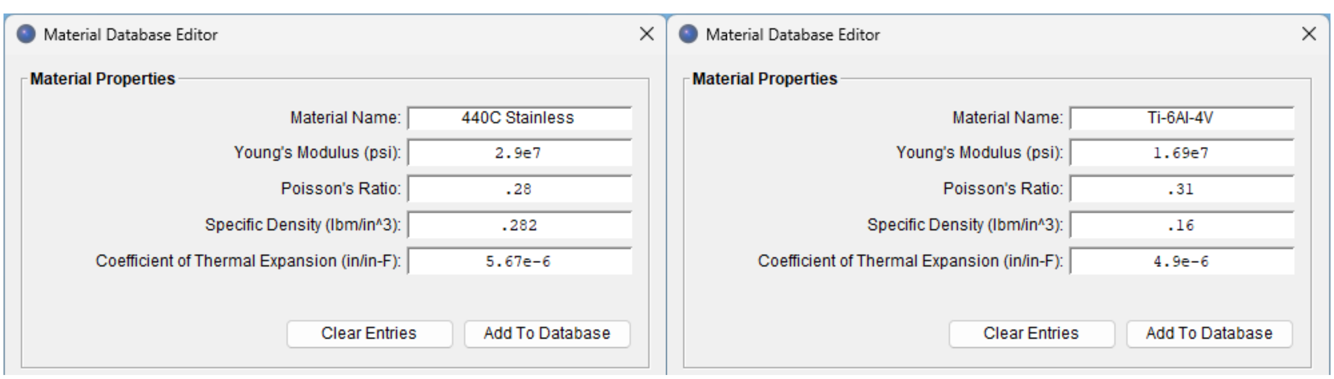


Figure 3-2. Example - Materials

3. Create a bearing called <FEA Ex DB> in the bearing database with the properties shown in Figure 3-3.

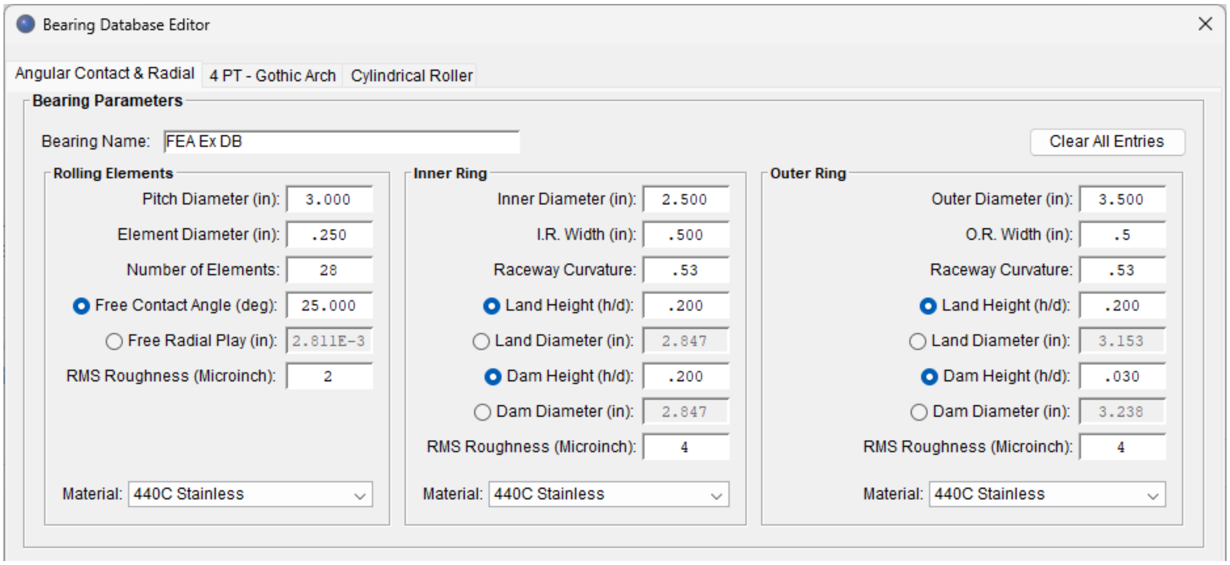


Figure 3-3. Example - Bearing Parameters

4. Configure the duplex pair as shown in Figure 3-4. The DB pair is centered about the origin. Row 2 is located at .250 and has the same fitups and shaft/housing dimensions as row 1. The preload value for row 2 is -120 lbf and its contact angle is ‘Convergent’, which creates the back-to-back configuration shown in the sketch.

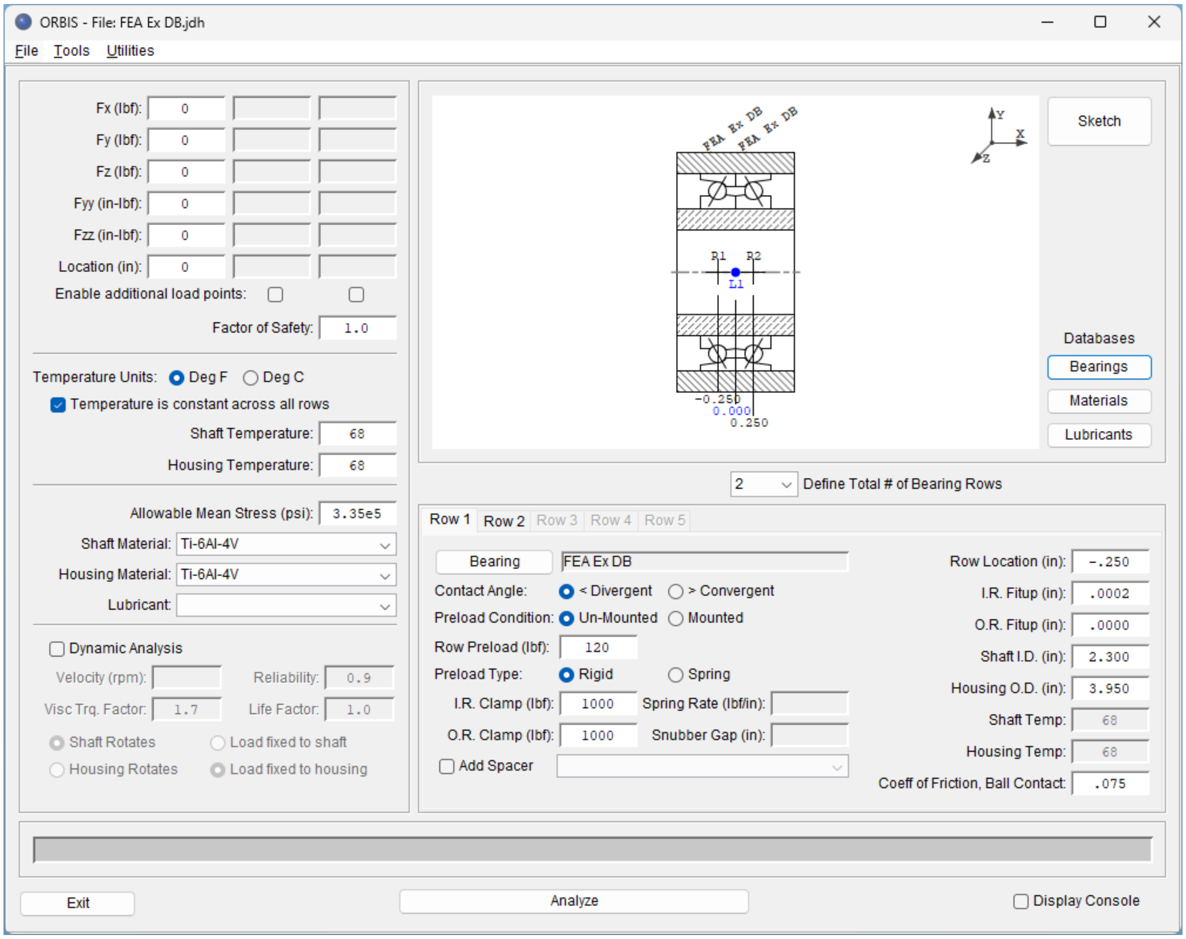


Figure 3-4. Example - DB Configuration

5. Analyze the duplex bearing pair with no external loading (just internal preloading) and verify results match the highlighted items in Figure 3-5. The highlighted stiffness parameters represent the principal stiffness of the duplex pair of bearings at their geometric center, which will be used to model the bearing in the FEA tool later.

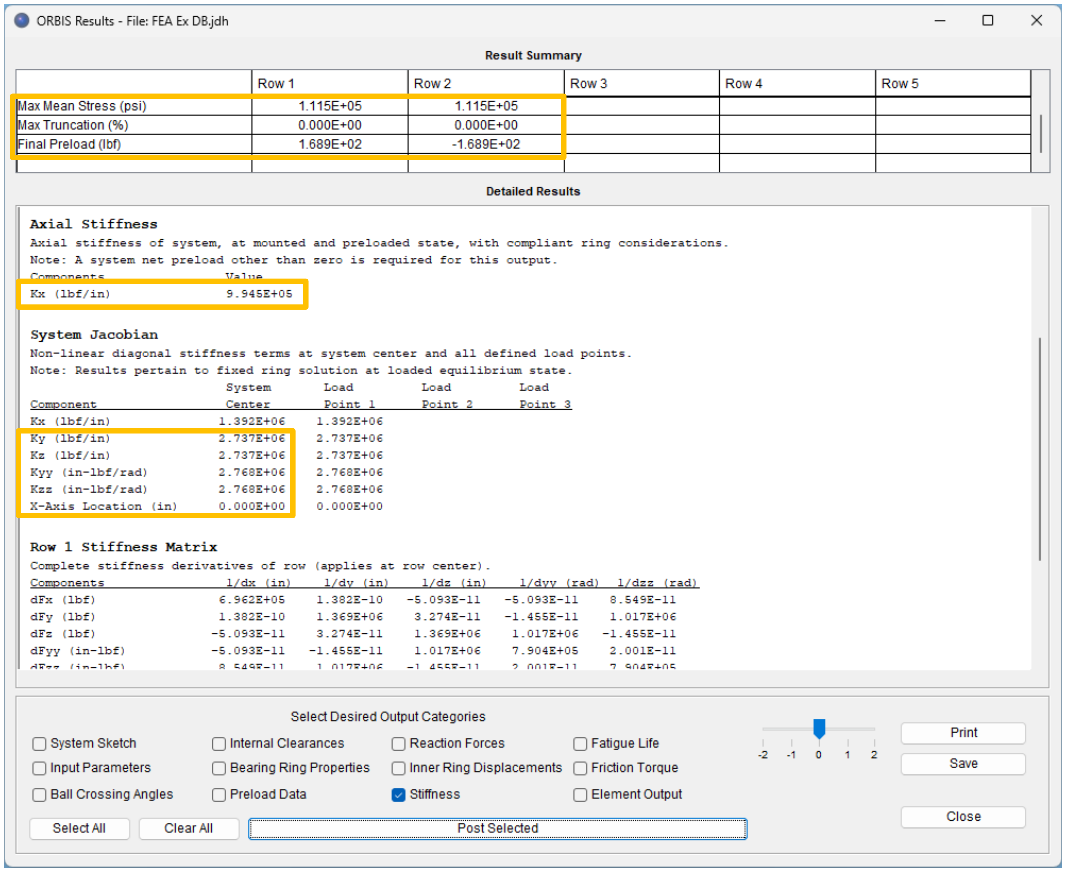


Figure 3-5. Example - DB Results

6. Save the setup file as <FEA Ex DB.jdh>. This file will be used later in this example, so save it somewhere accessible.

3.2 Model Layout and Properties

To help with finite element modelling, the layout shown in Figure 3-6 is provided. The objective of this layout is to orient the structure with a global reference frame and identify key grid point locations for the finite element connections. The method used here is outlined as follows:

- Each bearing pair is modelled with a single spring element at the center of the duplex pairs.
- Each drive mechanism is modelled with three different constant wall thickness cylindrical beam elements: a drive shaft beam, a drive housing beam and a motor shaft beam. Note these parts have various features, such as flanges or local wall thickness changes, that are omitted with this idealization.
- The mass properties of each drive mechanism are modelled as follows:
  - Resolver rotor and stator are modelled with point masses.
  - Total bearing mass is split, half and half, and modelled with point masses at the shaft and housing grids.
  - The motor rotor and stator mass are modelled by scaling the associated beam element densities.
  - Drive mechanism mass, and its center location, will be tuned to CAD predictions.
- The full biaxial mechanism mass and center location will be tuned to CAD predictions.
- The structure connecting the two drive mechanisms (the "Axis Bracket") is modelled with an I-beam. This is a simplification due to element limitations with the tool. Mass errors from this simplification will be mitigated by mass tuning. Stiffness differences, however, will be present but should have a second order

impact (i.e. small influence) on recovered structural modes. Rationale for this assertion is that eigenvalues are proportional to the square root of K/M and the Axis Bracket is suspended between the two drive axis bearing springs, which have much softer stiffness than either the real bracket or the modelled I-beam.

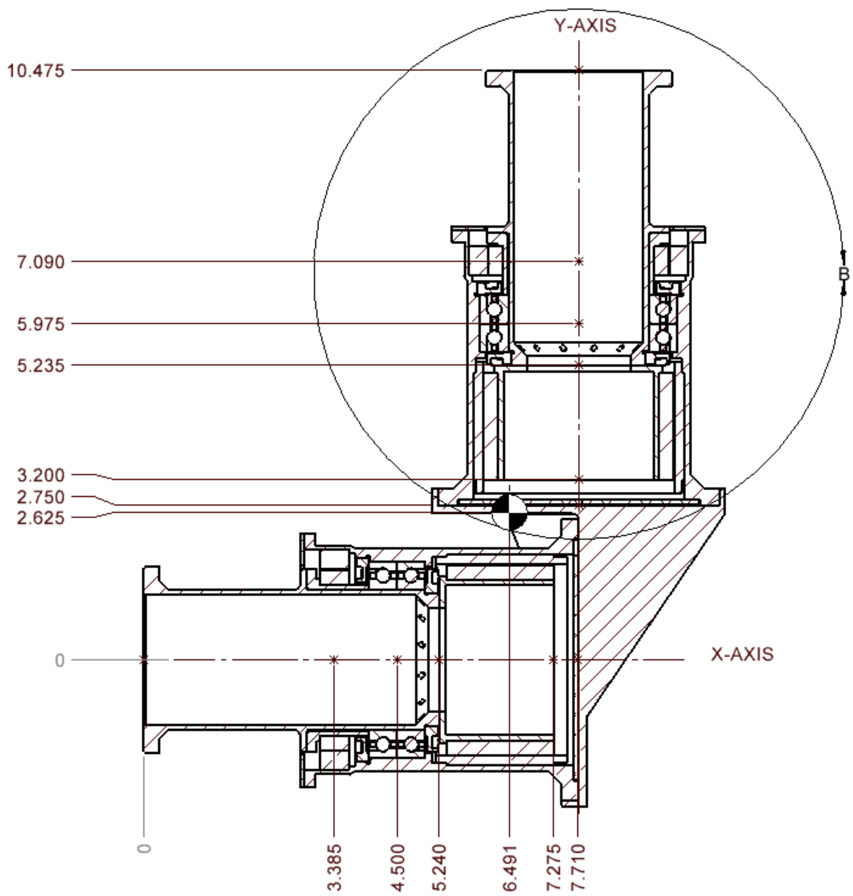


Figure 3-6. Biax Example Layout

Table 3-1 summarizes the structural grid points from the layout. The last grid point, ID 800, is added for use as a multipoint constraint. The location of this point is arbitrarily chosen to be the upper left corner of the model space.

Grid ID numbers can be any unique positive integer. However, it will be helpful to develop a numbering convention that implies how element connectivity is associated with the grids. The convention here assigns blocks of 100's for each region that will have a common beam section. For example, grids 100, 110, 120, and 130 are along the azimuth shaft and will be sequentially connected with the same beam section. Furthermore, the tens digit for coincident bearing grids are matched to help identify them during the spring element connections (e.g. the azimuth bearing connects to shaft grid 120 and housing grid 320).

Table 3-1. Grid Points

GRID ID	POSITION X	Y	Z	MODEL DESCRIPTION
100	0	0	0	Az shaft, SC interface
110	3.385	0	0	Az shaft, resolver mass
120	4.5	0	0	Az, shaft, bearing spring
130	5.24	0	0	Az shaft, motor shaft interface
200	7.275	0	0	Az motor shaft end
300	3.385	0	0	Az housing, resolver mass
320	4.5	0	0	Az housing, bearing spring
330	7.71	0	0	Az housing, axis bracket interface
400	7.71	2.75	0	Axis bracket to El housing interface
510	7.71	5.975	0	El housing, bearing spring
520	7.71	7.09	0	El housing, resolver mass
600	7.71	3.2	0	El motor shaft end
700	7.71	5.235	0	El shaft, motor shaft interface
710	7.71	5.975	0	El shaft, bearing spring
720	7.71	7.09	0	El shaft, resolver mass
730	7.71	10.475	0	El shaft, payload interface
800	0	10.475	0	Ground point multipoint constraint

Now that grid points are defined, the structural beam properties and mass elements are gathered. The following figure shows a detailed view of the drive mechanism, identifying components of interest.

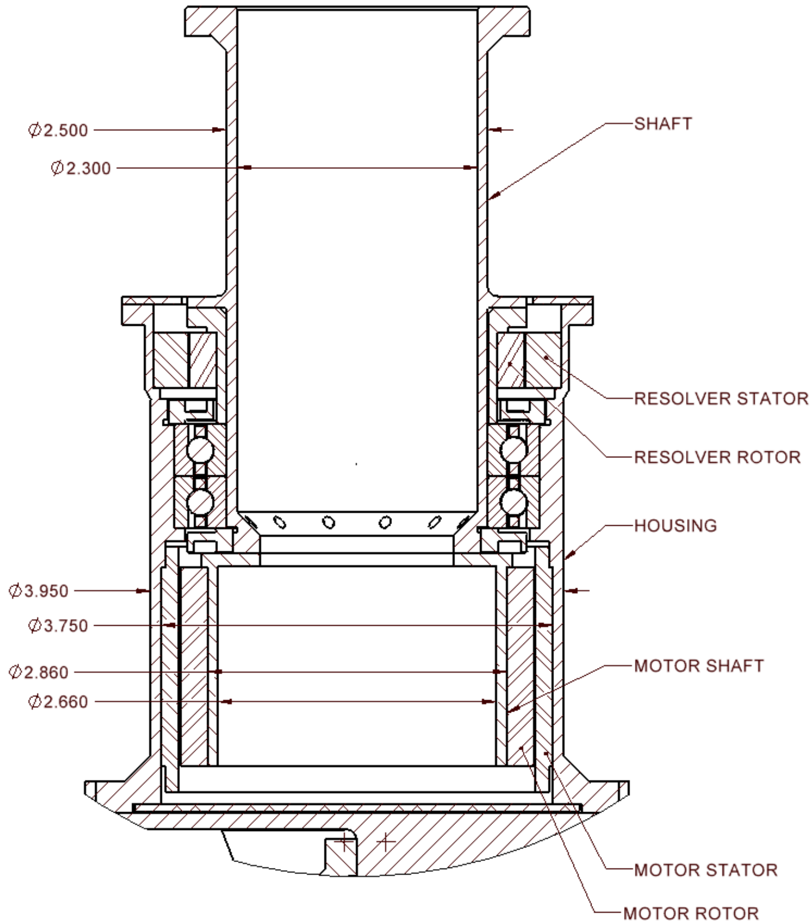


Figure 3-7. Drive Structure

Table 3-2 provides beam section properties and Table 3-3 shows the drive mechanism component masses to be modelled with point mass elements. These two tables will be used during the finite element model creation. Table 3-4 shows assembly mass properties for each drive mechanism and the full biaxial mechanism, which will be used during mass tuning.

Table 3-2. Beam Section Properties

Part	Section Type	Dimensions	Material
Drive Shaft	Cylindrical	I.D. = 2.3 O.D. = 2.5	Titanium
Drive Housing	Cylindrical	I.D. = 3.75 O.D. = 3.95	Titanium
Motor Shaft	Cylindrical	I.D. = 2.66 O.D. = 2.86	Titanium
Axis Bracket	I-beam	Outside Width = 2.6 Outside Height = 5.2 Web Thickness = .46 Flange Thickness = .1	Titanium

Table 3-3. Component Mass Properties

Part	Mass (lbs)
Resolver Rotor	0.6
Resolver Stator	0.6
Bearing Pair	1
Motor Rotor	1.4
Motor Stator	1.2

Table 3-4. Assembly Mass Properties

Assembly	Mass (lbs)	X	Y	Z
Azimuth Drive	7.8	5.005	0	0
Elevation Drive	7.8	7.71	5.47	0
Biax	17.1	6.491	2.625	0

3.3 Finite Element Model Creation

Model creation is accomplished by working through the FEA Load Tool editors in a sequential manner. Buttons on the left side of the FEA Tool, which access the editors, are arranged to be worked through in a top-down fashion. Following this order will ensure any prerequisite items for a given editor are established from a previous step. The following steps work through each editor to create the biaxial gimbal model.

Grid points:

1. From the main ORBIS window, select the Tools Menu → FEA Load Tool.
2. Select the [Grids] button to open the Grid Editor.
3. Create the 17 grid points shown in Table 3-1 by typing their grid ID and comma delimited grid point position. Select the [Add Grid Point] button after each point definition to commit the grid point to the model. Once completed, the Grid Editor should match the figure below.

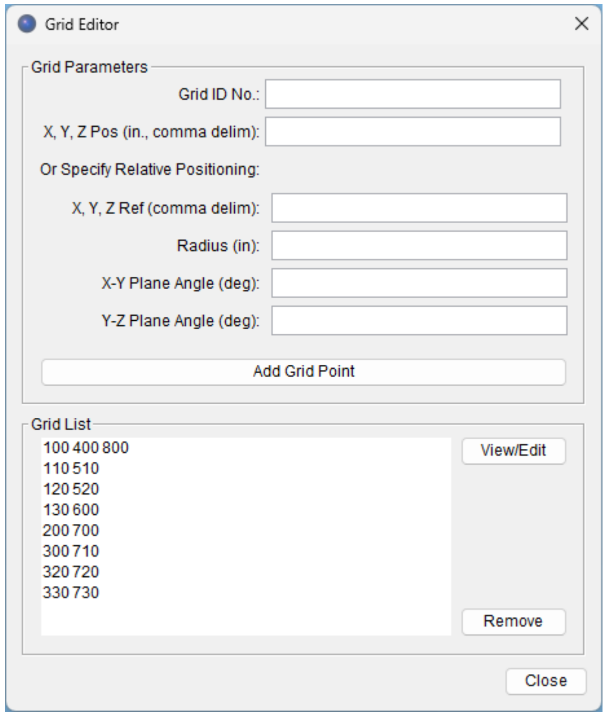


Figure 3-8. Grid Points

4. Close the Grid Editor by selecting the [Close] button. The display window in the FEA tool should now show the defined grid points (depicted with green dots).

Coordinate Systems:

The layout of the biaxial gimbal structure has the azimuth bearing axis aligned with the global x-axis and the elevation bearing axis aligned with the global y-axis. Bearing spring elements require a local coordinate with x-axis pointing along the bearing rotation axis. Therefore, the default global coordinate frame will be used to define the azimuth bearing spring and a new coordinate frame is needed for the elevation axis bearing spring. Follow these steps to create a coordinate frame for the elevation axis bearing:

1. Open the Coordinate System Editor by selecting the [Coords.] button on the FEA Tool window.
2. Set up the coordinate system as shown in the figure below. Note: the origin of the coordinate system, as shown, was defined by selecting grid 710, which is the point on the shaft at the center of the elevation bearings. When selecting a grid point to define the origin, coordinates are automatically populated.
3. Select the [Preview Coordinate Frame] button to see a display window with the defined coordinate frame. Rotate/pan/zoom the model to validate the coordinate frame has its x-axis aligned with the elevation axis grids. When done previewing, select the [Close] button to close the preview window and return to the Coordinate System Editor.
4. Select the [Add Coordinate System] button to commit the new coordinate frame to the model.
5. Select [Close] to close the Coordinate System Editor. The model display window will render the new frame.

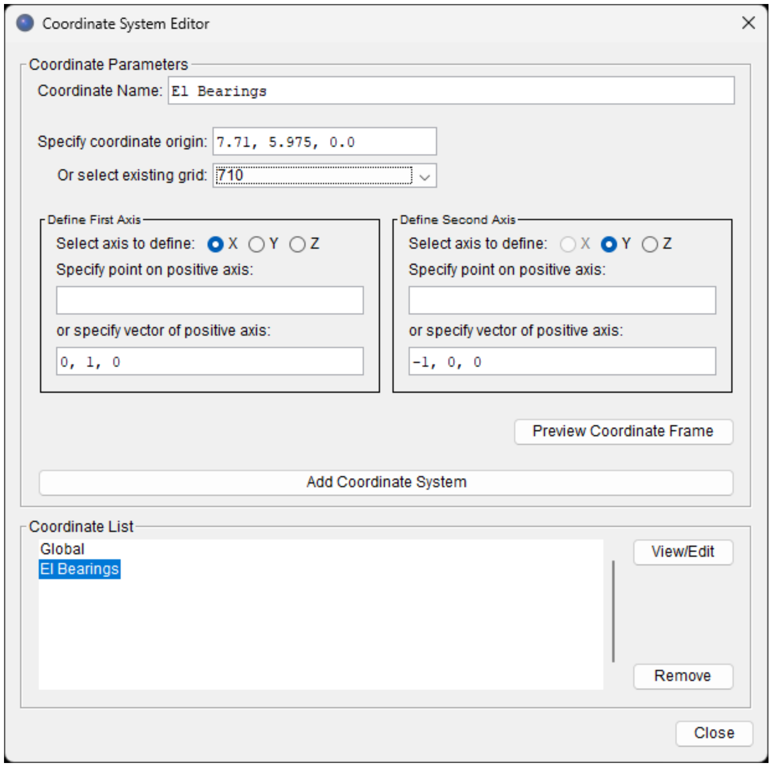


Figure 3-9. Coordinate System Editor

Beam element creation, which is the next step, involves connecting two grid points with a beam section, assigning a material to the beam, and specifying clocking of the section (about the beam axis) in global coordinates. Beam sections, which should not be confused with beam elements, are created with a separate editor that can only be accessed from the beam editor dialog. The approach taken for this example is to create all beam sections first and then create the beam elements.

Beam sections:

1. Open the Beam Element Editor by selecting the [Beam] button from the FEA Tool window.
2. Select the [Edit Sections] button in the Beam Element Editor (located midway down on the right-hand side of the dialog) to open the Beam Section Editor.
3. Create the 4 sections from the properties in Table 3-2. Use the tabs in the Section Editor to select the section type and complete the required parameters for the section. When defining the Axis Bracket as an I-beam section, take note of the n1 vector (see Figure 3-10 below). This section is non-axisymmetric and its clocking orientation must be correctly defined in the beam element editor later.
4. Once all sections have been added, select the [Close] button to close the Section Editor and return to the Beam Editor.

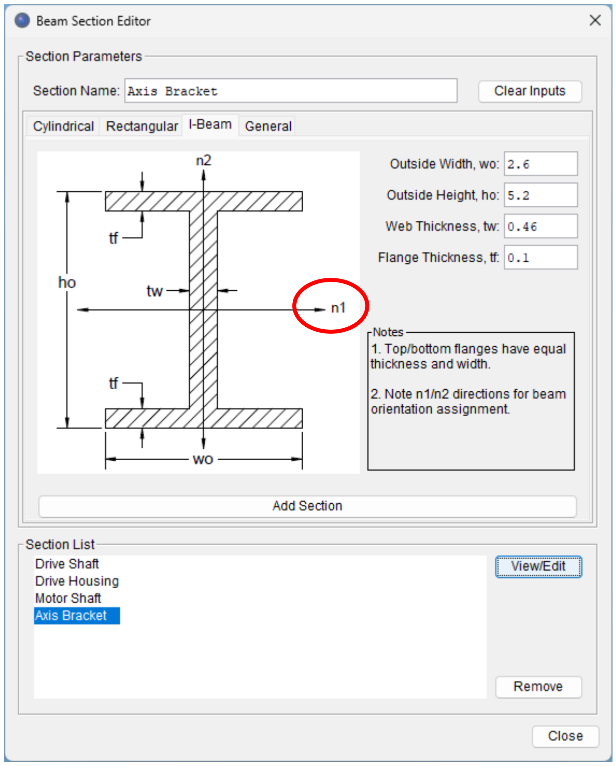


Figure 3-10. Beam Sections

Beam elements:

Create the beam elements shown in Table 3-5, one for each row in the table, typing the name, selecting the attachment grids, assigning the section and material, and typing the n1 vector. It is recommended, especially for the Axis Bracket beam, to use the [Preview] button to render the beam setup prior to adding it to the model. Figure 3-11 shows the Axis Bracket beam rendering (note: the preview model has been rotated in the figure). Note the n1 vector orientation points along the z-axis.

Table 3-5. Example Beam Element Parameters

Name	Grid ID 1	Grid ID 2	Section Name	Material	n1 Vector
Az Shaft_1	100	110	Drive Shaft	Ti-6Al-4V	0, 0, 1
Az Shaft_2	110	120	Drive Shaft	Ti-6Al-4V	0, 0, 1
Az Shaft_3	120	130	Drive Shaft	Ti-6Al-4V	0, 0, 1
Az Motor Shaft	130	200	Motor Shaft	Ti-6Al-4V	0, 0, 1
Az Housing_1	300	320	Drive Housing	Ti-6Al-4V	0, 0, 1
Az Housing_2	320	330	Drive Housing	Ti-6Al-4V	0, 0, 1
Axis Bracket	330	400	Axis Bracket	Ti-6Al-4V	0, 0, 1
El Housing_1	400	510	Drive Housing	Ti-6Al-4V	0, 0, 1
El Housing_2	510	520	Drive Housing	Ti-6Al-4V	0, 0, 1
El Motor Shaft	600	700	Motor Shaft	Ti-6Al-4V	0, 0, 1
El Shaft_1	700	710	Drive Shaft	Ti-6Al-4V	0, 0, 1
El Shaft_2	710	720	Drive Shaft	Ti-6Al-4V	0, 0, 1
El Shaft_3	720	730	Drive Shaft	Ti-6Al-4V	0, 0, 1

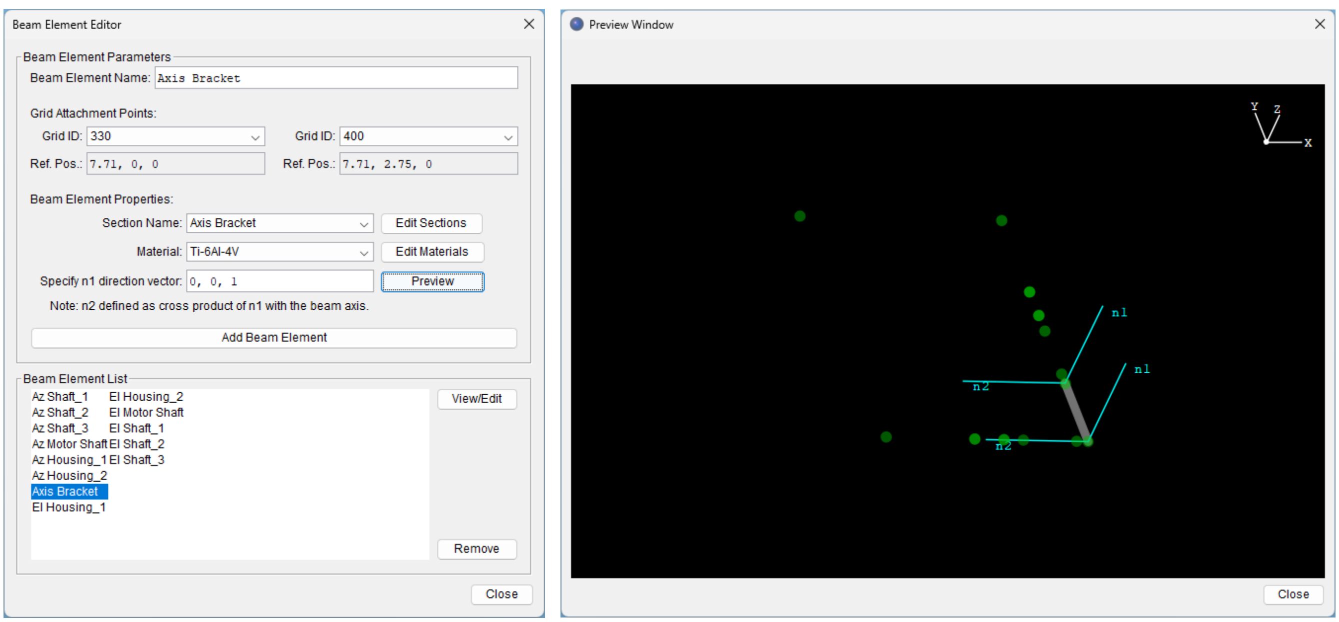


Figure 3-11. Axis Bracket Beam – Preview

Spring elements:

The next step is to add the two spring elements that represent the DB bearing pairs for the azimuth axis drive and the elevation axis drive. These spring elements will connect the respective drive shaft and housing together. Follow steps below to create the spring elements:

1. Select the [Spring] button from the FEA Tool window to open the Spring Editor.
2. Type <Az Brg> for the first spring name.
3. Set the first grid ID to grid <120> and the second grid ID to grid <320> from the drop-down menu selections. Verify these are the coincident grid points inspecting their reference positions.
4. Set the local spring coordinate system to <Global>, which has its x-axis aligned with the azimuth bearing rotation axis.

5. Select the [Load Bearing Stiffness] button to open the Bearing Stiffness Editor (shown in Figure 3-12 below)
6. Select [OK] in the warning dialog regarding the coordinate reference requirements for a bearing spring (x-axis of coordinate must be aligned with the rotation axis).
7. In the Bearing Stiffness Editor, select the [Browse] button and use the file browser to select the <FEA Ex DB.jdh> ORBIS file created at the beginning of this example (section 3.1).
8. Select rows 1 and 2 for stiffness calculations (click the row check boxes) and accept the default options in the Bearing Stiffness Editor.
9. Select [Compute Stiffness] to analyze the bearings for stiffness. See Figure 3-12 below for the complete setup. Review the stiffness results.

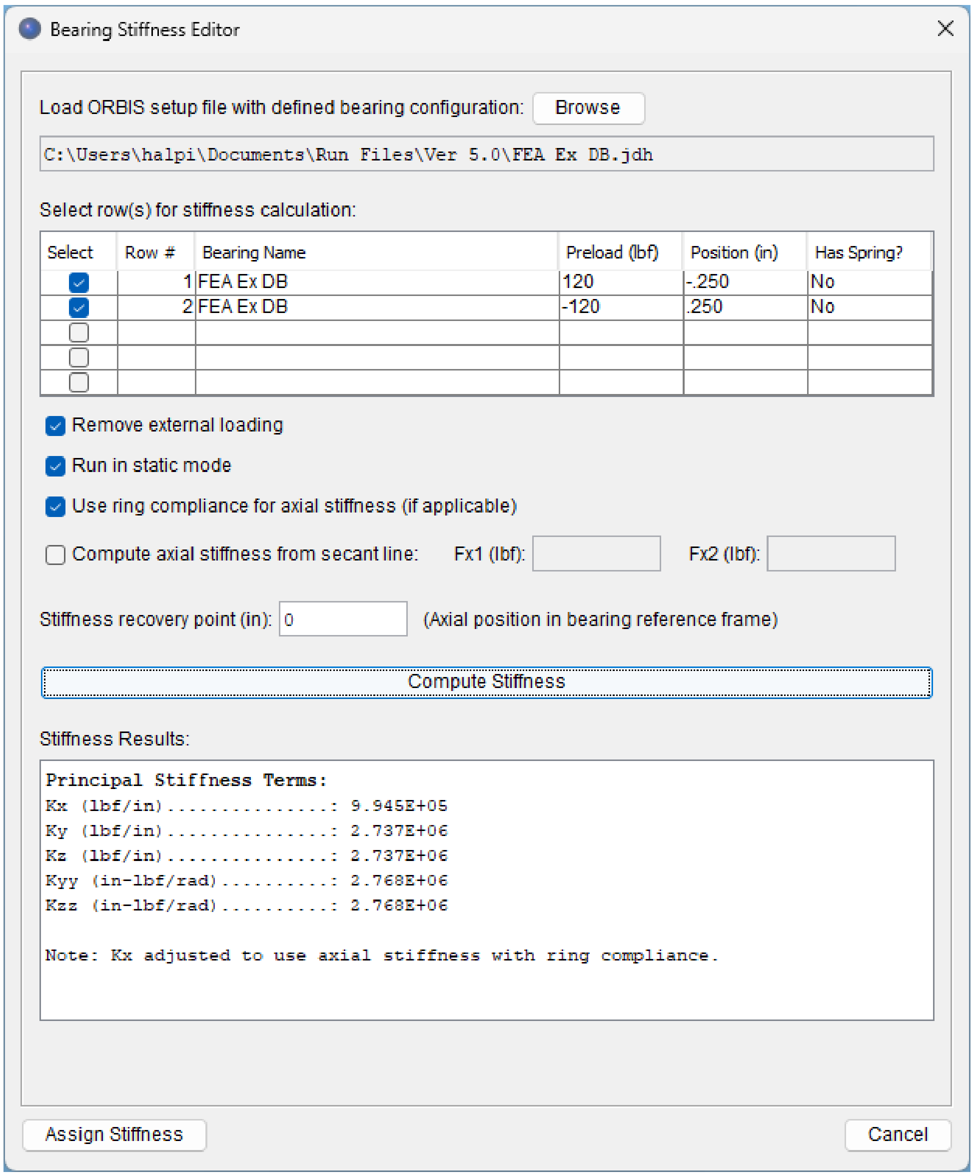


Figure 3-12. Bearing Stiffness Editor

10. Select [Assign Stiffness] to copy the computed bearing stiffness terms back into the Spring Editor. The Bearing Stiffness Editor will close and the Spring Editor will become active.

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

11. The diagonal spring stiffness terms will now have the bearing stiffness values. However, K44 is left blank. This is the torsional stiffness of the bearing, which is not computed from ORBIS and needs to be manually entered. To maintain a positive definite stiffness matrix, which is guaranteed to have real eigenvalues, enter a value of <10> for this term.
12. Review the Az Brg spring setup with the figure below. If everything is correct, select [Add Spring] to commit the spring to the model.

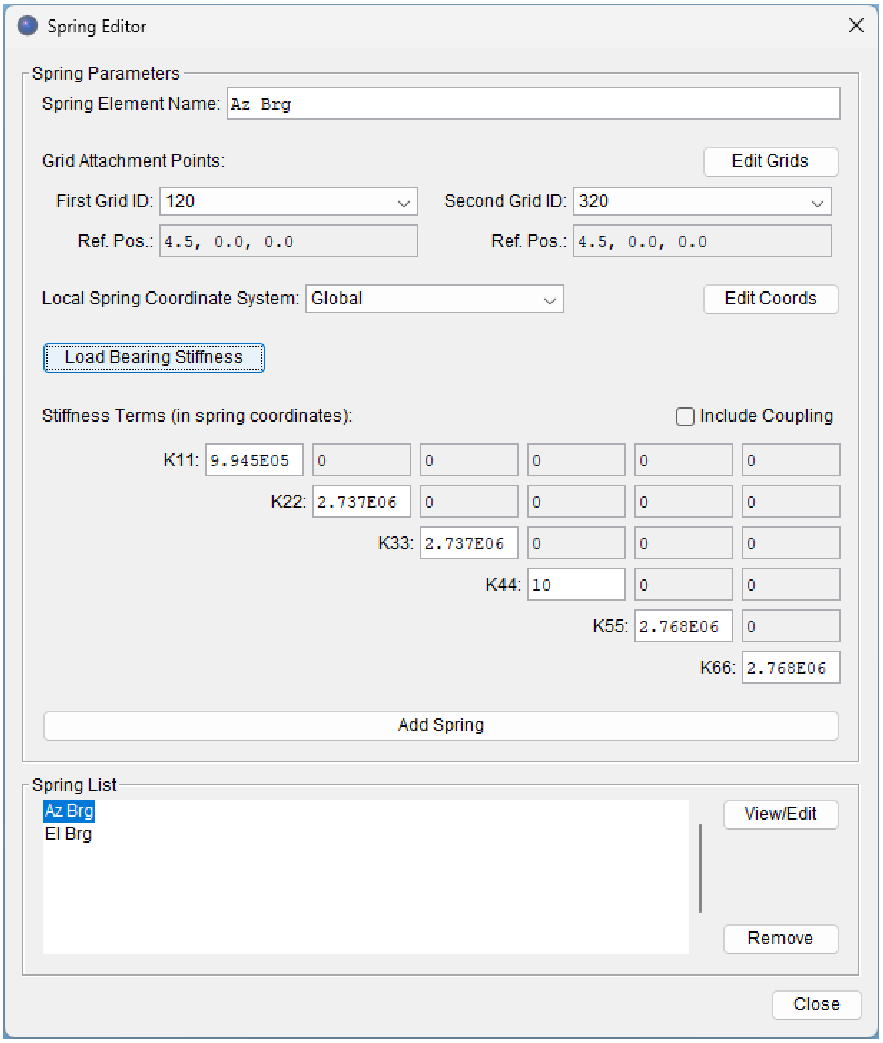


Figure 3-13. Az Bearing Spring

13. Next, define the elevation bearing spring by setting the spring name to <El Brg>.
14. Set the first grid to <510> and the second grid to <710>.
15. Select the <El Bearings> for the local spring coordinate system.
16. Select the [Load Bearing Stiffness] button to open the Bearing Stiffness Editor and select [OK] on the warning dialog.
17. The Bearing Stiffness Editor will remember the last bearing setup. In this case, that is the correct stiffness, so it can be used immediately by selecting the [Assign Stiffness] button.
18. Enter a value of <10> for K44 in the Spring Editor and select [Add Spring].
19. Close the Spring Editor.

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

Mass elements:
Lumped mass elements shown in Table 3-3 are assigned next. The resolver components and the bearings must be added to both the azimuth drive and the elevation drive, creating a total of 8 mass elements. These elements are summarized in the table below. The following procedure goes through creation of the first mass element and should be repeated for each mass element in the table.

Table 3-6. Mass Elements

Name	Mass (lbs)	Grid ID
Az Resolver Rotor	0.6	110
Az Resolver Stator	0.6	300
El Resolver Rotor	0.6	720
El Resolver Stator	0.6	520
Az Bearing IR	0.5	120
Az Bearing OR	0.5	320
El Bearing IR	0.5	710
El Bearing OR	0.5	510

1. Select the [Mass] button from the FEA Tool window to open the Mass Editor.
2. Type <Az Resolver Rotor> for the mass element name.
3. Select grid <110> for the attachment point.
4. Specify the isotropic translational mass by using the built-in equation calculator. Enter <0.6/386.1> in the input field and then hit the <enter/return> key to convert the mass to slug-inch (result should be 1.55E-03).
5. Set the Ixx, Iyy, and Izz rotational inertias to zero.
6. Select the [Add Mass/Inertia] button to commit the mass element.
7. **Tip**: After adding a mass element, the mass parameter input fields are cleared for the next element definition. If the next mass element has identical parameters to a previously defined mass, such as the four resolver masses to be defined here, the following procedure can save some typing:
    a. Select the <Az Resolver Mass> in the Mass List at the bottom of the dialog.
    b. Select the [View/Edit] button to populate the parameters for this mass.
    c. Edit the name and grid ID to match the next mass element to be created.
    d. Select [Add Mass/Inertia] to add the new mass.

Multipoint constraints:
The biaxial gimbal will be analyzed for random vibration loading with a fixed-fixed boundary condition. Specifically, the spacecraft and payload interfaces, as shown in Figure 3-1, will be constrained in all 6 degrees of freedom (DOF) during random vibration loading. A random vibration analysis must have only one fixed grid point boundary condition. To constrain a model at more than one grid point, while having only one fixed grid point BC, a multipoint constraint (MPC) is required. Follow these steps to create the MPC:

1. Select the [MPCs] button on the FEA Tool window to open the MPC Editor.
2. Set the name to <Fixed Interface>.
3. Set the independent grid to <800>.
4. Set the dependent grids to <100> and <730>, which are the two interface grid points to be fixed. To select multiple grids in the editor grid list, select the first grid and then hold the ctrl key while selecting additional grids.
5. Accept the default constraint properties of fixed (6 DOF).
6. Select the [Add Constraint] button to commit the MPC to the model.

At this point the planned elements to represent the biaxial gimbal model are complete. It is recommended to save the model by selecting [Save] from the [File] menu at the top of the FEA Load Tool window. The next steps are to tune the mass properties, specify boundary conditions, loads, and load cases.

3.4 Mass Tuning

The Mass Tuner tool, which is accessible from the Tools menu, allows adjustments to the model mass properties. To avoid iteration, this tool should be used after all beam and mass elements have been created. The following steps adjust the model mass properties, with the goal of tuning until they closely match the assembly mass properties provided in Table 3-4. The approach below will first tune the azimuth drive mechanism elements, then apply those adjustments to the elevation drive elements (both drive axes are identical and will have the same mass properties). Finally, the Axis Bracket mass properties will be adjusted until the full biaxial gimbal properties match the assembly mass properties shown in Table 3-4. Only the translational mass, and their center coordinates, will be tuned (rotational inertias will not be tuned).

1. Open the Mass Tuner by going to the Tools Menu → Mass Tuner.
2. The motor rotor and stator mass, as shown in Table 3-3, was not modelled with mass elements and, therefore, needs to be modelled here. The approach is to add the motor component masses by spreading the additional mass on the associated beam elements. Mathematically, this is achieved by multiplying the beam material density by a scale factor. Follow the steps below to determine and set the scale factors.
    a. Select the <Az Motor Shaft> beam element from the table in the Mass Tuner dialog.
    b. Select the [Selected elements] radio button under the calculation options.
    c. Select the [Calculate] button to get the current mass of the Az Motor Shaft element (should be ~.28 lbs). From Table 3-3, the motor rotor mass is 1.4 lbs., so the total mass of the Az Motor Shaft element should be 1.68 lbs. (1.4 + .28). Therefore, the beam element mass needs to be scaled by 1.68/.28 = 6.
    d. In the element table, set the [Scale Factor] cell for the Az Motor Shaft element to <6>.
    e. Unselect the <Az Motor Shaft> and select the <Az Housing_2>. Beam element Az Housing_2 spans from the center of the bearings to the Axis Bracket, which covers the region where the motor stator resides.
    f. Calculate the mass of this beam by selecting the [Calculate] button (should be ~.62 lbs). From Table 3-3, the motor stator mass is 1.2 lbs. Thus, the scale factor for this beam is computed to be 1.82/.62 = 2.94.
    g. Set the [Scale Factor] for the <Az Housing_2> beam element to <2.94>.
3. With the motor mass added to the azimuth drive, the next step is to query the total mass of the azimuth axis drive subassembly and compare it with the mass properties shown in Table 3-4. Select all azimuth beam elements and mass elements in the table (6 beam elements and 4 mass elements).
4. Select [Calculate] to get mass properties of the selected elements (should be 6.568 lbs. located at x = 4.997"). Per Table 3-4, the azimuth drive should weigh 7.8 lbs. with a CG at x = 5.005. So, the current mass is short by about 1.23 lbs. but is already centered well.
5. Final tuning of the azimuth beam elements, to get mass properties to match Table 3-4, entails iteratively adjusting scale factors and re-calculating mass properties. Once the azimuth drive assembly is tuned, the same scale factors can be applied to the elevation drive assembly elements (azimuth and elevation drive assemblies are identical). Finally, the entire biaxial assembly is tuned by adjusting the Axis Bracket scale factor. For consistency moving forward with this example, enter the scale factors shown in the figure below. Note the naming convention chosen for the azimuth beam elements versus the elevation beam elements are not consistent with respect to the common drive assembly parts. For instance, the shaft region from the spacecraft interface to the center of the azimuth drive resolver is named Az Shaft_1, however, that same structural region on the elevation drive is named El Shaft_3.

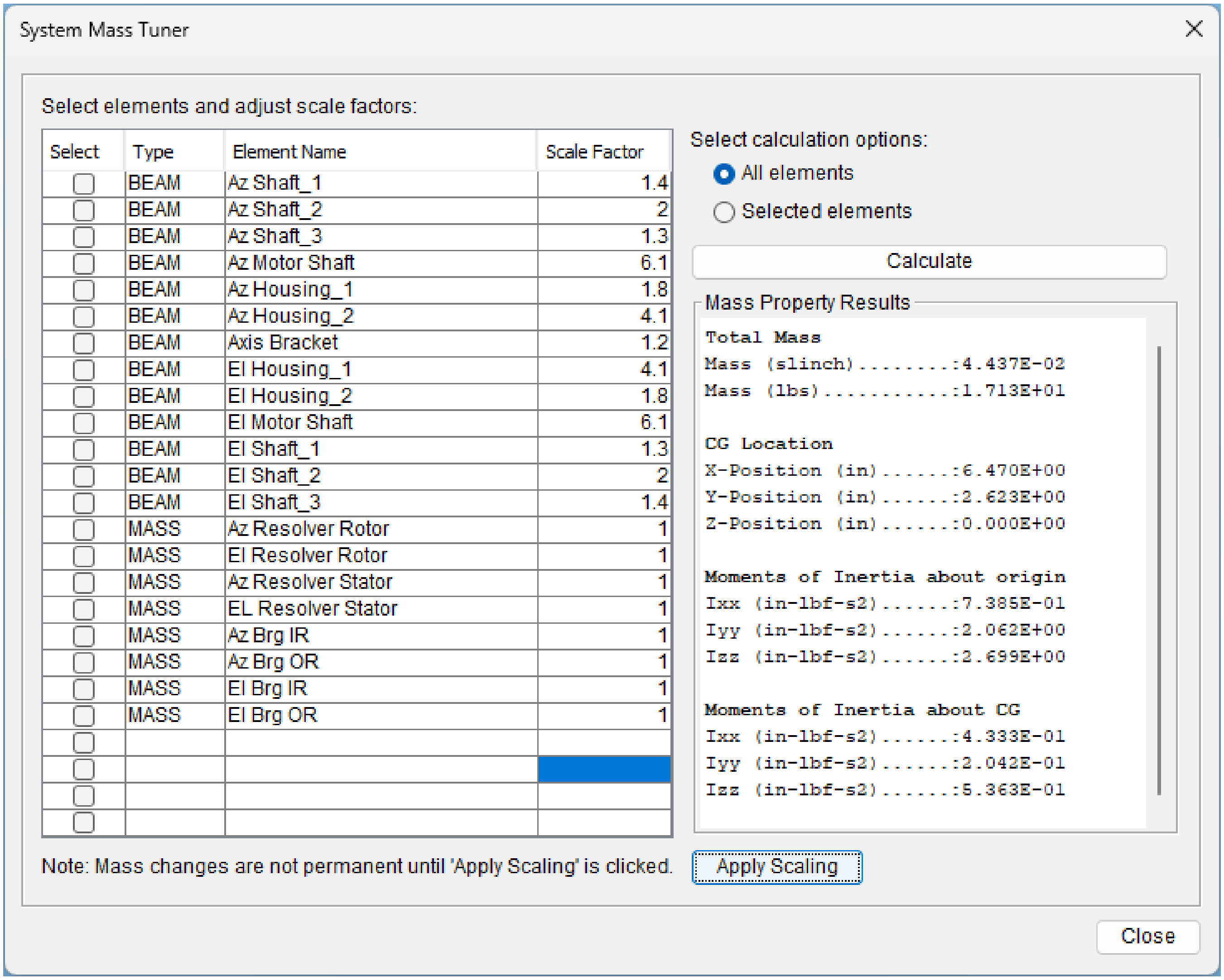


Figure 3-14. Mass Tuner Scale Factors

6. Once scale factors have been updated per the above figure, select the [Apply Scaling] button. Note: model elements are not updated until the scaling is formally applied and then further confirmed as shown in the remaining steps.
7. Select [OK] to the warning regarding intent to commit changes to the model.
8. Select [OK] to the message box stating elements were updated.
9. Select [Close] to close the Mass Tuner.

3.5 BCs, Loads and Load Cases

The biaxial gimbal assembly will be fixed at both the spacecraft and payload interfaces for all loading. However, there is an MPC connection that transforms these two locations back to a common point (grid 800). Hence, only one grid point needs a boundary condition defined. The following steps establish that boundary condition:

1. Select the [BCs] button from the FEA Tool window to open the BC Editor.
2. Specify the BC name to be <Fixed Base>.
3. Select grid <800> for the constraint.

4. Accept the default constraint property of fixed (6 DOF).
5. Select the [Add Boundary Condition] button to commit the constraint to the model. The setup should look like the figure below. When finished reviewing the setup, close the BC Editor.

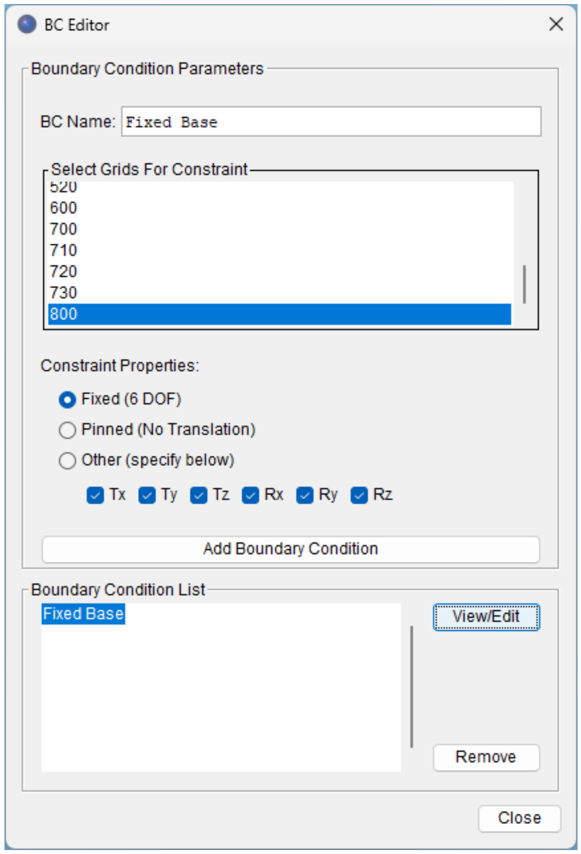
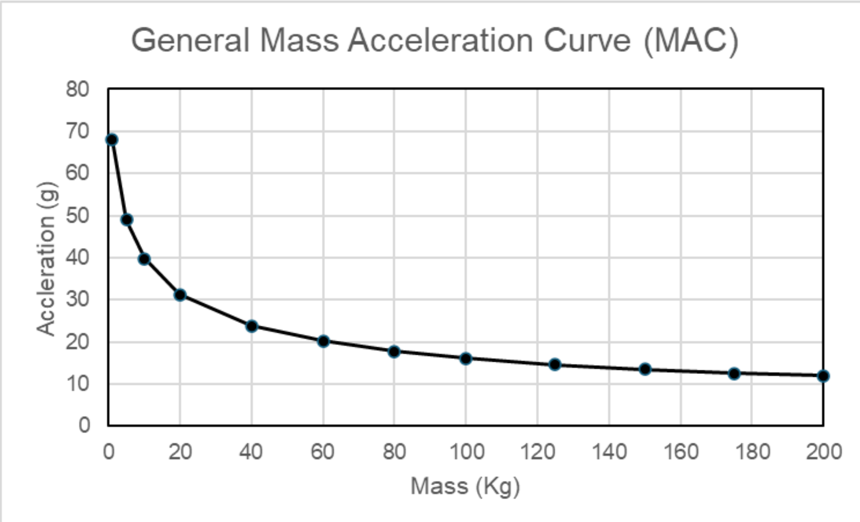


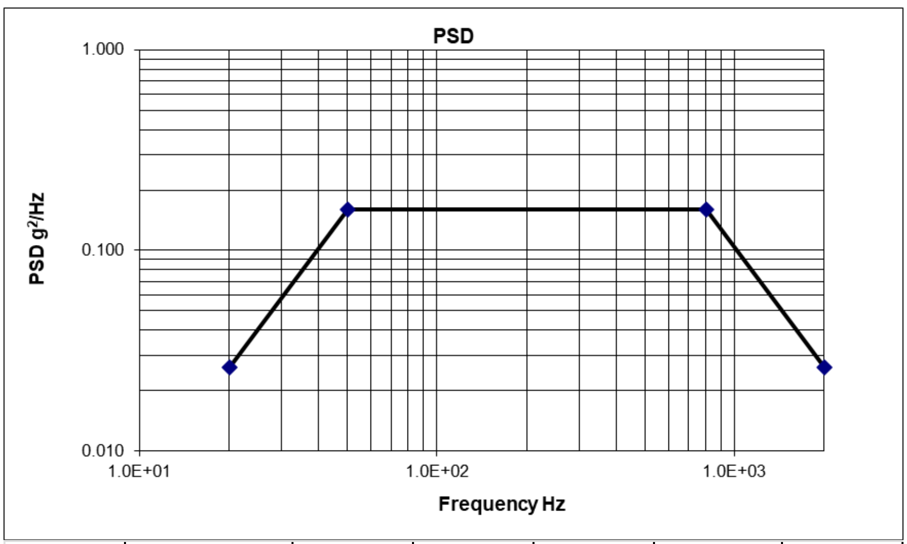
Figure 3-15. Boundary Condition

Loads:
The mass acceleration curve (MAC) shown in Figure 3-16 and the random vibration profile shown in Figure 3-17 will be set up and analyzed. Each of these environments shall be applied along the global x, y, and z axes, creating six load cases to run.



Mass (Kg)	Acceleration (G's)
1	68
5	49
10	39.8
20	31.2
40	23.8
60	20.2
80	17.8
100	16.2
125	14.7
150	13.5
175	12.6
200	12

Figure 3-16. Mass Acceleration Curve



FREQ(Hz)	ASD(G^2/Hz)	dB	OCT	dB/OCT	AREA	Grms
20.00	0.0260	*	*	*	*	*
50.00	0.1600	7.89	1.32	5.97	2.51	1.58
800.00	0.1600	0.00	4.00	0.00	122.51	11.07
2000.00	0.0260	-7.89	1.32	-5.97	199.82	14.14

Figure 3-17. Random Vibration Profile

Procedure for creating MAC loads:

1. Select the [Loads] button from the FEA Tool window to open the Load Editor.
2. Select the [Mass Acceleration] tab in the Load Editor.
3. Specify <MAC-X> for the load name.
4. Select the [X+] radio button for the acceleration direction.

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

5. Define the mass acceleration curve as follows:
    a. Select the [Kg] radio button for the mass units.
    b. Accept the default linear interpolation method.
    c. Enter the MAC data points shown in Figure 3-16 in the Load Editor table. The complete set of MAC data, which extends through 200 Kg, does not need to be entered into the table. However, at least 3 data points must be entered, and those points should envelop the biaxial gimbal mass.
    d. Optionally select the [Plot MAC] button to see a graphical plot of the entered MAC data along with intermediate interpolated points.
6. Select [Add Mass Acceleration] button to commit the new load to the model.
7. After a MAC load has been created and added the dialog will only clear the name field. This is intended to aid creation of subsequent MAC loads when only the name and direction will change. Proceed to add the <MAC-Y> and <MAC-Z> loads now by entering the new name and selecting the acceleration radio button for the [Y+] and [Z+] directions, respectively.

Procedure for creating random vibration loads:

1. From the Load Editor, select the [Random Vibration] tab.
2. Set the new load name to <RV-X>
3. Select the shake axis by selecting the [X] radio button.
4. Enter the base acceleration spectral density table from Figure 3-17.
5. Select the [Calc. GRMS] button to get the computed GRMS from the entered data. Verify it is 14.14 grms and select [OK] to close the message dialog.
6. Specify a 3% critical damping ratio across the 2000 Hz frequency range by entering the following two data points in the damping table: <20, .03>, <2000, .03>.
7. Accept the default maximum frequency of 4000 Hz, the number of solution points per eigenvalue interval (default is 20), and the bias parameter of 3.
8. Select [Add RV Load] to add the load to the model.
9. Create the <RV-Y> and <RV-Z> loads by specifying the new name and corresponding shake axis.

Meshing the model:

The next step, which is not technically related to boundary conditions or loads, is to refine the discrete math model by meshing the beam elements. The procedure is as follows:

1. Select the [Mesh] button from the FEA Tool window to open the Mesh Editor.
2. Select all elements in the 'Elements To Seed/Mesh' list (upper list in the Mesh Editor). To select all items in the list, either click in the list area and type <CNTRL+A> or click on the first list item and hold the <SHIFT> key while clicking on the last list item.
3. Set the seed method to [By max size]
4. Type <.5> in the size/number input field. This sets the maximum span between solution points, or nodes, to .5 inch. Most beam elements will not be evenly divisible by the max size input, so the span will be <.5 inch.
5. Select [Preview Mesh] and review the model. It should look like the figure below.
6. When finished reviewing the model, close the preview window and select the [Mesh Selected Elements] button to commit the mesh settings.
7. Close the Mesh Editor.

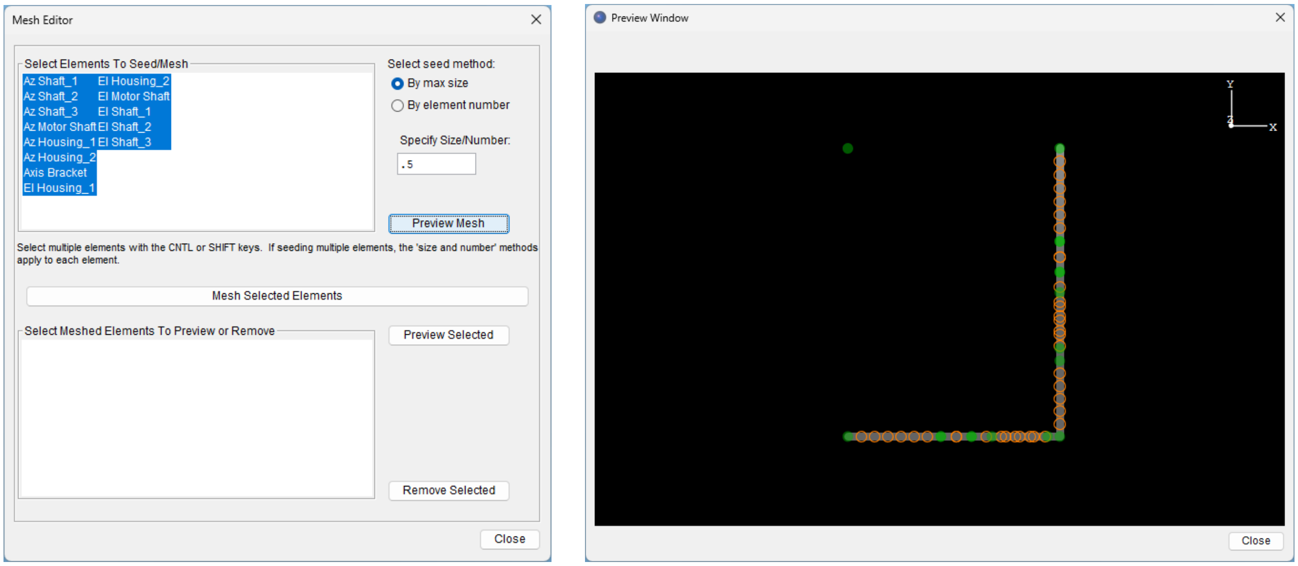


Figure 3-18. Model Meshing

Load Case Manager:

The Load Case Manager is used to specify the list of cases to be solved. A run case requires at least one load and one associated boundary condition. Random vibration loads can only have a single boundary condition and shake axis per case, whereas static type loading can have multiple loads and/or BCs per case. Follow these steps to create the analysis run cases:

1. Select the [Case Manager] button from the FEA Tool window to open the Load Case Manager.
2. Since this model only defined one BC, a dialog box is shown asking if each defined load should be used to automatically create a load case (where each case will use the single defined BC). This is needed here so select the [Yes] button on the dialog.
3. The Load Case Manager will then be shown per Figure 3-19, where the six load cases are defined. You can select a load case and click the [View/Edit] button to see how it is defined within the dialog. After reviewing the load cases select the [Close] button to exit from the load case manager.

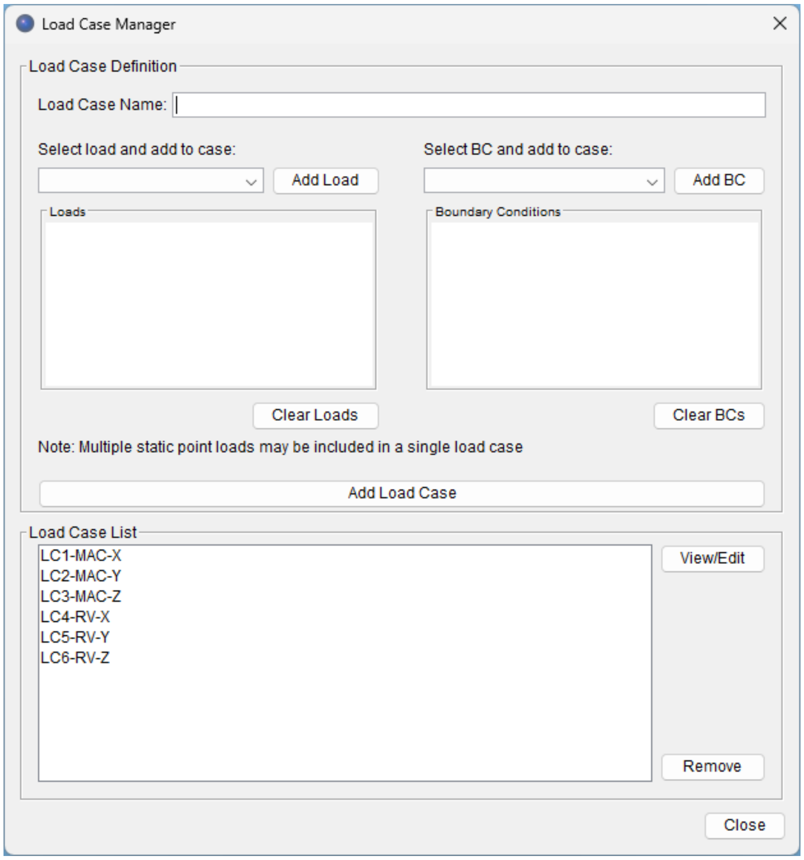


Figure 3-19. Load Case Manager

Save your model (File Menu → Save). At this point, the model is created, meshed, and all load cases are defined. Before running the analysis, it is prudent to validate the model is set up as intended. Follow these steps to validate the model:

1. Select the [Center of mass] checkbox to render the mass center coordinates in the display window. Compare the system mass center coordinates with the values in Table 3-4. Each coordinate should be close the values in the table.

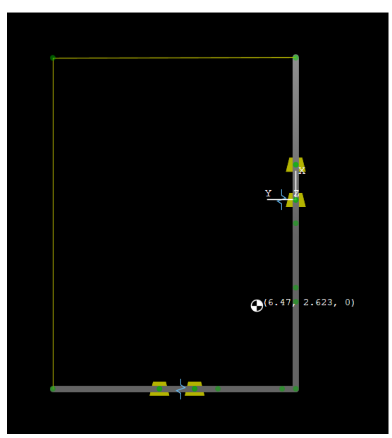


Figure 3-20. Center of Mass Validation

Halpin Engineering, LLC

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

2. Select the [Display load cases] checkbox.
3. Leave the [B/C's] and [Loads] checkboxes selected and select the first load case <LC1-MAC-X> from the drop-down menu.
4. Select the [Sketch] button to render the loads and boundary conditions. The model will update as shown below. Notice there is a load arrow on each grid point that is oriented along the x-axis. Furthermore, the load value of 43.904 G's is displayed. This is the interpolated acceleration from the MAC table at the computed model mass.

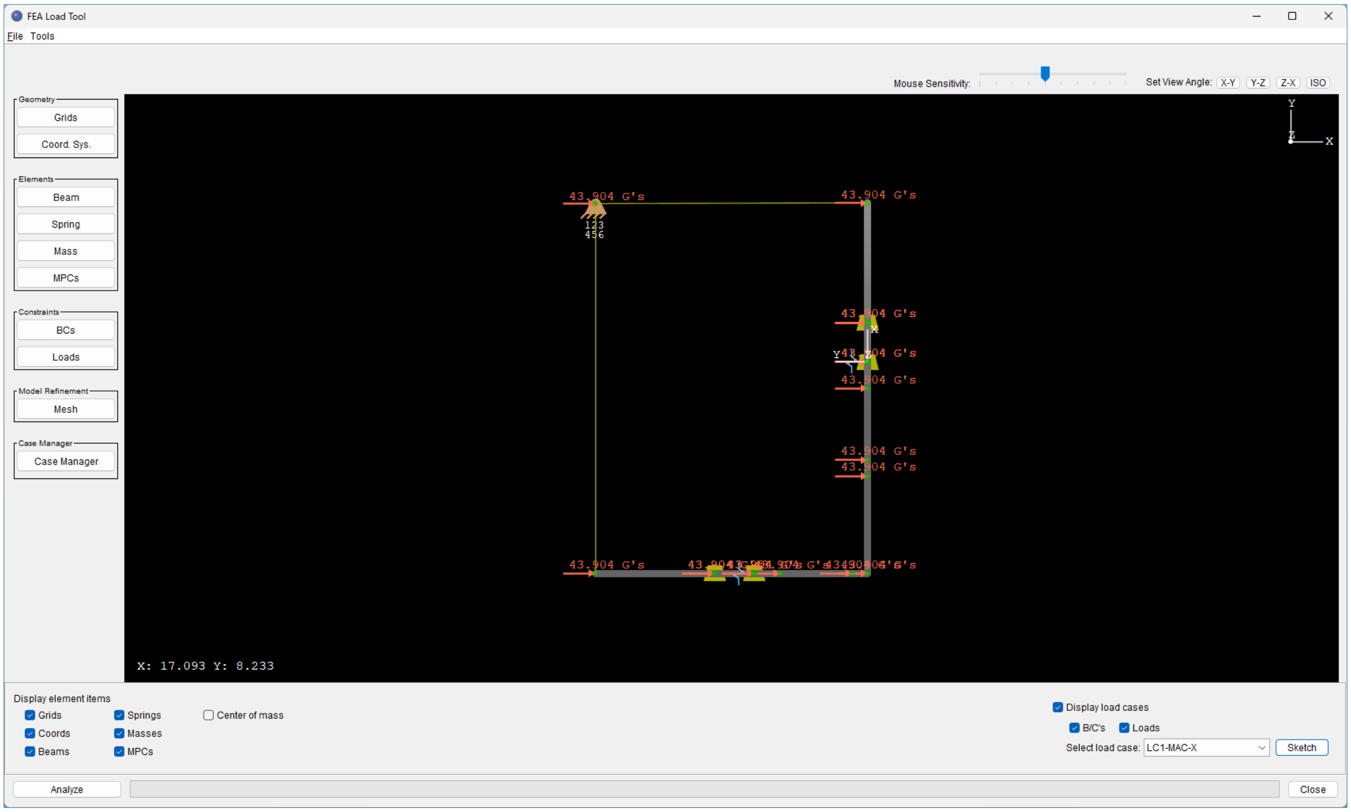


Figure 3-21. Load Case Validation

5. Review the <LC2-MAC-Y> and <LC3-MAC-Z> load cases by changing the load case selection from the drop-down menu and re-sketching the model. These cases should have the same G-level but be oriented along the y+ and z+ axes, respectively.
6. Next, review the random vibration load cases. These cases will show the total GRMS value at the fixed boundary grid point with a bi-directional arrow aligned along the specified shake direction.

3.6 Running the Analysis

To run the defined load cases, select the [Analyze] button. This example will solve six load cases, which will take some processing time to solve (should be less than a minute but computation time will vary depending on computer speed). The progress bar at the bottom of the window will advance as each load case is solved.

Reviewing results:

Once the analysis is completed, an FEA Results window will be shown. To see a formatted text report of each load case, use the [FEA Result Preview] tab (default active tab when the window first appears). In this tab, select a load case from the drop-down menu and select the [Display Results] button to post the load case results in the text pane. Scroll through the report to review the output generated. Outputs posted to the results vary depending on the type of load case selected, so review at least one of the MAC load cases and one random vibration case. For all load case types, bearing spring forces will be output. Table 3-7 shows the recovered bearing forces for all 6 load cases (note: forces are in local spring coordinates).

To see a rendered animation of the mode shapes, first select one of the random vibration load cases from the drop-down menu and then select the [Plot Modes] button to bring up the Modal Preview Window. Within the Modal Preview Window, select an eigenvalue from the drop-down menu and select the [Plot Mode Shape] button to render the eigenvector (mode shape). Note that some modes may have motion in-and-out of the default view, which will appear as a static model. In such cases, it will help to rotate the model around to get the best perspective. When done reviewing modes, close the Modal Preview Window.

Table 3-7. Bearing Reaction Forces

		Mass Accel Cases			Random Vibe Cases		
	Component	**MAC-X**	**MAC-Y**	**MAC-Z**	**RV-X**	**RV-Y**	**RV-Z**
Az Bearing	Fx (lbf)	-343	-3.5	0	327.5	88.9	0
	Fy (lbf)	29.7	-34.2	0	48.4	24.8	0
	Fz (lbf)	0	0	-234.1	0	0	182.5
	Mx (in-lbf)	0	0	0	0	0	0
	My (in-lbf)	0	0	-323.4	0	0	567
	Mz (lbf)	84.7	-292.1	0	246.9	456.9	0
El Bearing	Fx (lbf)	-29.7	-386	0	97.3	479.1	0
	Fy (lbf)	77.2	-3.5	0	80.5	16.3	0
	Fz (lbf)	0	0	186.1	0	0	132.9
	Mx (in-lbf)	0	0	0	0	0	0
	My (in-lbf)	0	0	-102.7	0	0	377.1
	Mz (lbf)	-368	5.5	0	592.2	113.5	0

3.7 Analyze bearings

To analyze the bearings with the recovered bearing loads from the various load cases analyzed, follow these steps:

1. In the FEA Results Window, select the [Direct Bearing Analysis] tab.
2. Select the [Browse] button and select the bearing setup file that was built and saved at the beginning of this example (should be named FEA Ex DB.jdh).
3. Select the <LC1-MAC-X> load case from the drop-down menu.
4. Since the bearings are modelled with a single spring element, there is only one load point at the center of the bearings to use for the bearing analysis. Select the <Az Brg> spring element from the first drop-down menu (see figure below). This will analyze the azimuth bearing location loads.
5. Select the [Add to Load Pt. 1] button to copy the spring forces into the table for load point 1.
6. Set the factor of safety to <1.4>.

7. Select the [Analyze Bearings] button to run the bearing analysis. A standard ORBIS results window should appear with the detailed results. Check the summary table and verify the max mean stress on row 2 is 1.662E+05 psi.
8. The bearing results window can be left open and new cases can be set up and run to compare bearing results from multiple cases. To do this, minimize or move the bearing results window and click on the FEA Results Window to make it active. Change the load case to <LC2-MAC-Y>, then ensure the <Az Brg> spring is still selected and select the [Add to Load Pt. 1] button. Finally, select the [Analyze Bearings] button to analyze the bearings with the new case. There will now be two bearing result windows to review and compare. The titles of the bearing result windows will show the load case name, such as QS-X and QS-Y, making them easy to identify which load case they represent.
9. All load cases and bearing locations can be run by repeating the steps above to configure the load case and bearing spring element.

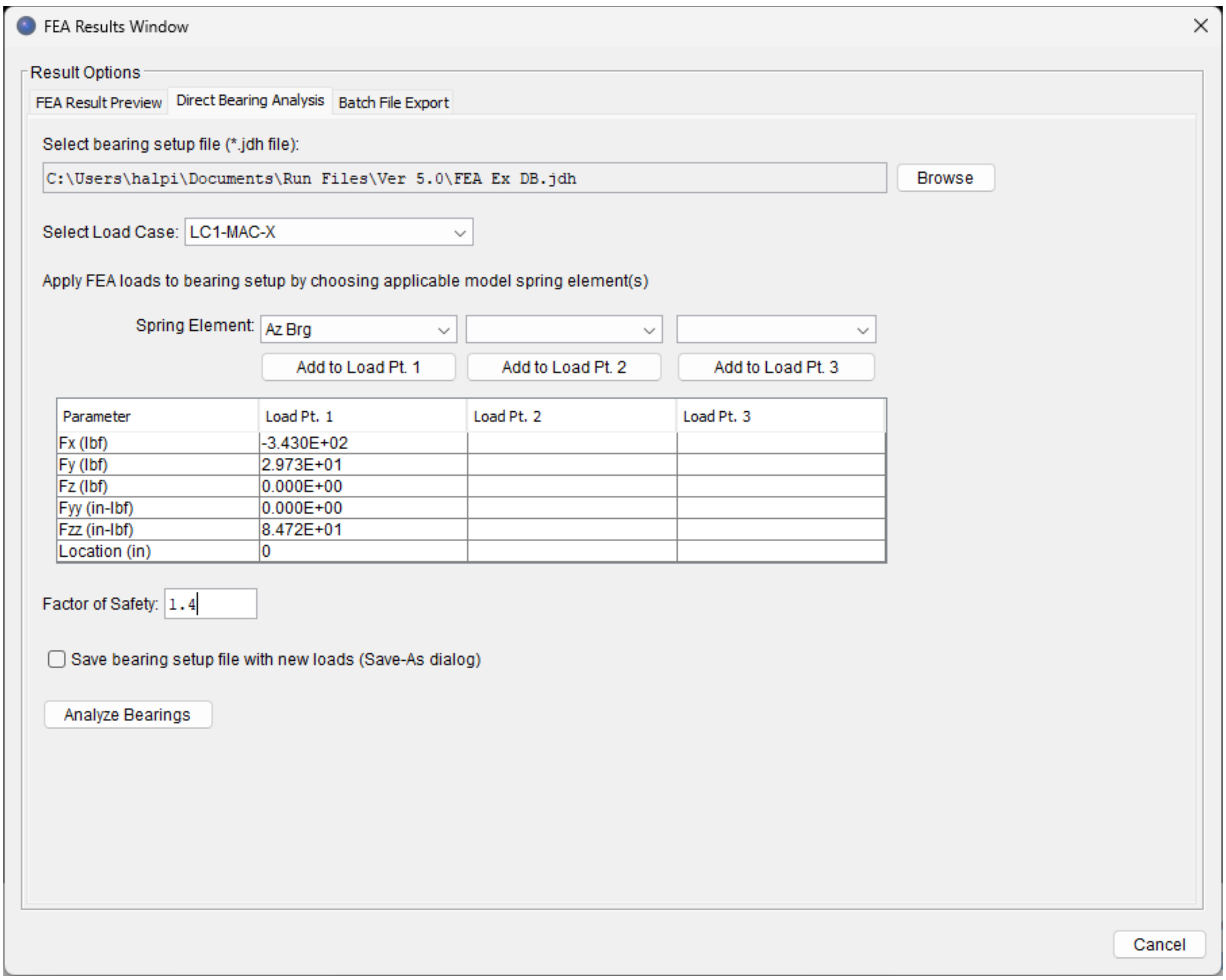


Figure 3-22. Direct Bearing Analysis Setup

Halpin
Engineering,
LLC

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

3.8 Export Loads

The last step is to export all the loads to an ORBIS batch file for later analysis or general record keeping. Follow these steps to create the batch file:

1. From the FEA Results Window, select the <Batch File Export> tab.
2. Select the first load case <LC1-MAC-X> from the drop-down menu.
3. Select the <Az Brg> spring element from the drop-down menu.
4. Select the <Scale load components> checkbox and type <1.4> for the load factor.
5. Type <0> for the load point location.
6. Select the <Browse> button and navigate to the folder where you have saved the bearing setup file. Give the name <Az Bearing Loads> for the file name and select the <Save> button.
7. Accept the <Append loads to file> radio button.
8. Select the <Write Batch File> button and select the <OK> button the message window confirming the file was saved.
9. Now add the <LC2-MAC-Y> and <LC3-MAC-Z> load cases to the same batch file by changing the load case selection and clicking the <Write Batch File> button for each of the two cases. The batch file will now have the three mass acceleration load cases.
10. The random vibration loads are RMS, so it is desired to generate all possible sign conventions on the loads and include a 3-sigma factor. To add these cases to the same batch file:
    a. Select the <LC4-RV-X> load case.
    b. Select the <Generate all +/- sign combinations…> checkbox.
    c. Type <3*1.1> in the Load Factor input filed and then hit the <Enter> key to compute the equation. This factor is the 3-sigma probability factor times a 1.1 safety factor.
    d. Select the <Write Batch File> button.
    e. Repeat the above steps for the <LC5-RV-Y> and <LC6-RV-Z> load cases. When finished the batch file will have 3+3*32 = 99 load cases.

The above process can be repeated for the <El Brg> loads, if desired. The created batch file can then be run within the ORBIS main program (not in the FEA Load Tool).

4 References

Meirovitch, L. (2001). *Fundamentals of Vibrations.* New York: McGraw-Hill.
Przemieniecki, J. S. (1968). *Theory of Matrix Structural Analysis.* New York: McGraw-Hill.
Wijker, J. (2009). Random Vibrations in Spacecraft Structures Design. *165*(Solid Mechanics and its Applications).