

***iSTRDYN<sup>©</sup> - integrated Stress,  
Thermal, and Rotor Dynamics***

**Jeffcott Rotor Analysis Example**

*iSTRDYN* Modeling, Solutions, and Result Processing



July 2007

This presentation shows an analysis sequence using *iSTRDYN* to calculate the natural frequencies and forced response of a classic Jeffcott rotor-bearing system. Linear orthotropic supports are used. A full set of plots are included to highlight the available displays in *iSTRDYN*.

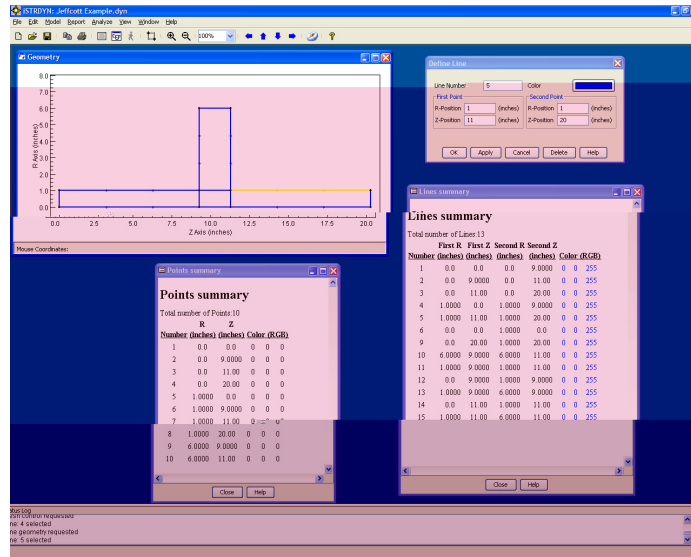
## Description of Jeffcott Analysis

- **Creation of geometry (points and lines)**
- **Auto generation of 2D finite element mesh**
- **Specification of support properties**
- **Linear natural frequency calculation**
- **Linear unbalance response**
- **Result processing**

- 2 -

This example analysis starts with a geometry definition (points and lines) of the cross section, then creates a mesh for use in the analysis. Definition of linear support properties is illustrated, followed by the calculation of natural frequencies and unbalance response. This example can be directly compared to traditional rotor dynamics programs.

# Geometry Definition

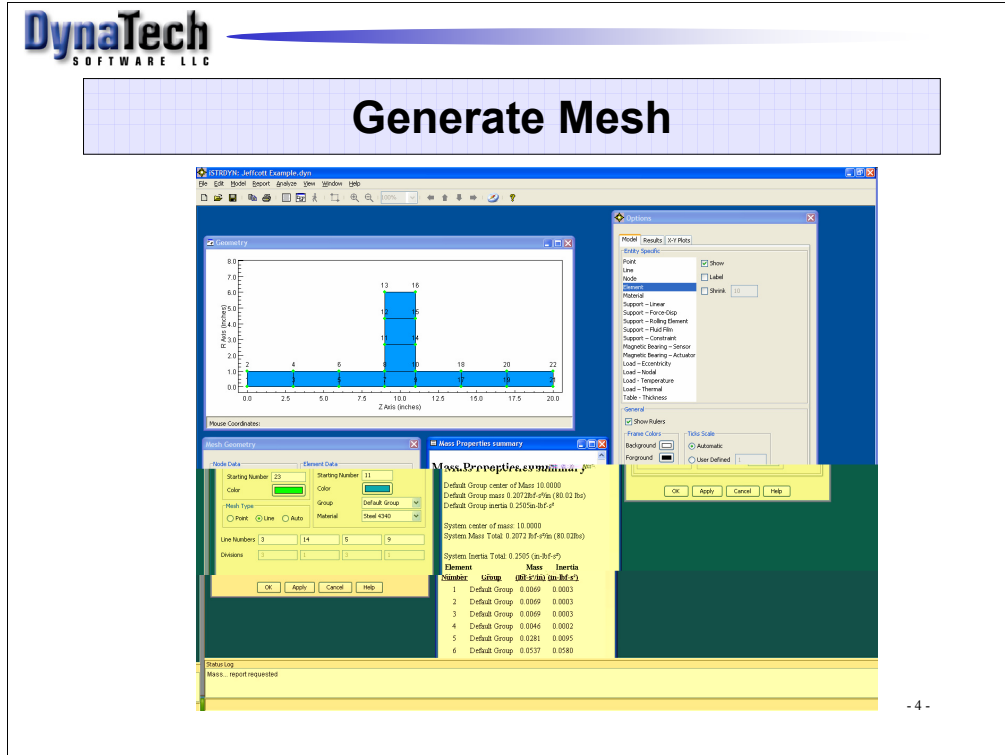


- 3 -

This screen shot shows the point and line geometrical description manually defined, a dialog box used line definition, with a line selected (as shown by the yellow color). While it is not necessary to create a model starting from this point and line geometry, it is easier as will be shown in the next. All of the lines created were specified by the end coordinates, no points were explicitly defined. Note that mesh control has been applied to the horizontal shaft lines and the vertical disk lines. This feature is used to control the number of divisions along a line for node generation. Examination the geometry shows there are 4 separate areas bounded by the lines.

In addition to the geometry window and line definition box, two other text boxes are displayed, one listing all the points in the model, and the other the lines.

## Generate Mesh

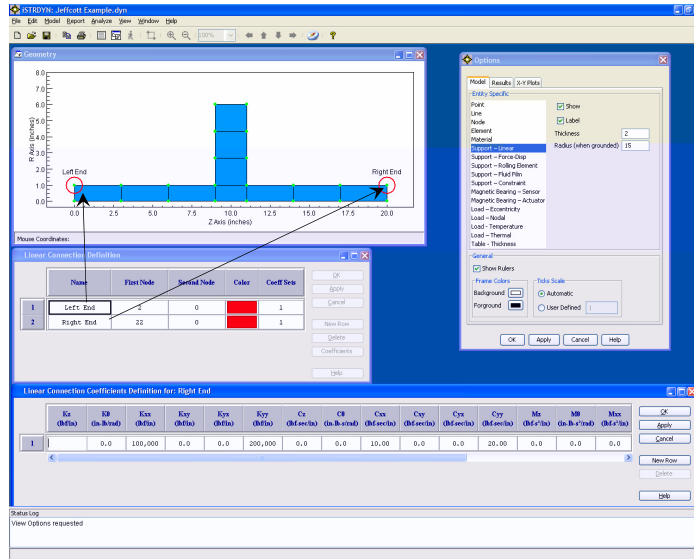


This screen shot shows how the geometry defined in the previous slide has been discretized into 2D axisymmetric elements. The dialog box in the lower left hand corner is the meshing command, which in this case was used to define elements based on 4 enclosed lines. Node and element colors, group name (which has been defaulted, more about this in a subsequent slide), and a material selection are shown. iSTRDYN has 4 default materials available, and a complete data base can be imported using XML files.

The mesh used in this example contains 22 nodes and 10 elements. The exact numbering of these nodes and elements is not important, as the program does a profile sort prior to performing any calculations. One of the more powerful dialogs in the program is shown on the right – the View Options box. This controls the display of all model definition entities, such as nodes and elements, presentation of results, and the format of X-Y plots. In this figure, the Element option is highlighted, and the options indicate the elements will be displayed, but the numbers will not be shown. The “shrink” parameter is used to decrease the element area to check for unmeshed sections.

Finally, a mass summary is displayed. Both individual groups and the total system properties are listed, followed by details on each of the elements.

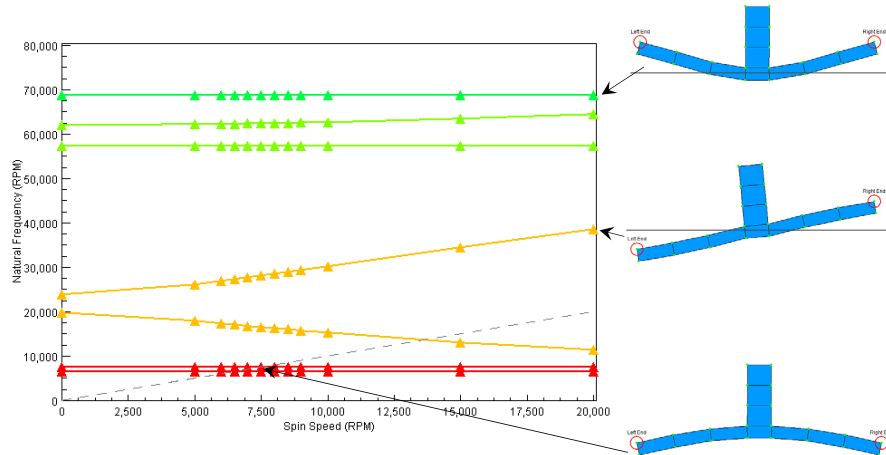
# Linear Supports



- 5 -

The application of linear supports is displayed in this screen shot. The mesh is shown in the top window, the connection definition table is the middle window, and the bottom window is the coefficients. When any connection is defined, iSTRDYN will put a symbol in the geometry window if the entity label is selected, as shown in the Options dialog. The color of the symbol, which in this case is a circle (since the connection is to ground), is entirely the user's choice. The coefficients for the connection are defined by selecting the row number on the left hand side of the definition table, which brings up a long table of 20 columns for the stiffness, damping, and mass terms, along with two columns for a speed value and speed units. Note that in this example, the stiffness and damping are not equal in the two transverse directions, which is an orthotropic system.

## Linear Natural Frequencies



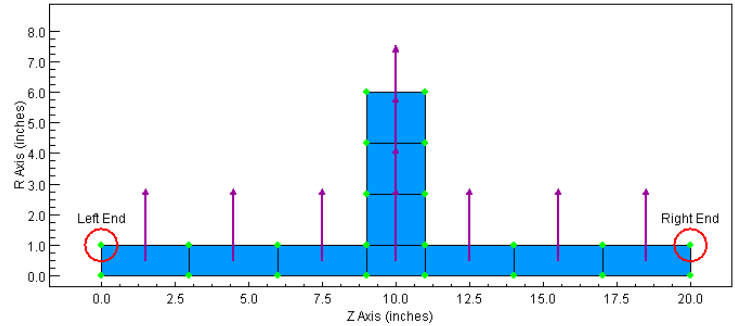
- 6 -

This whirl map and associated modes display the natural frequency characteristics of the linear model. The forward and backward modes are split due to the orthotropic supports used in this example. The first mode is a symmetric translation, and because the disk is not rotating, there is very little gyroscopic stiffening. The second mode is a pitching or out-of-phase displacement, and because the disk is displaced rotationally there is quite a bit of gyroscopic stiffening. The third mode is a classic bending natural frequency.

In the operating range of this example, a single critical speed is predicted, at approximately 7,500 rpm. Note the speed steps around this critical speed, which are narrower than the larger steps that define the analysis range. The ability to “zoom” into a narrow speed range is provided by the Output Set capability, which allows multiple runs to be defined, executed, and combined as in this whirl map.

Although the axisymmetric harmonic elements are much more computationally intensive than equivalent beams, the run time is not excessive, due in part to significant optimizations employed in the calculations. The total execution time on a normal desktop computer for all of these analysis speeds was 31 seconds.

## Unbalance Eccentricity

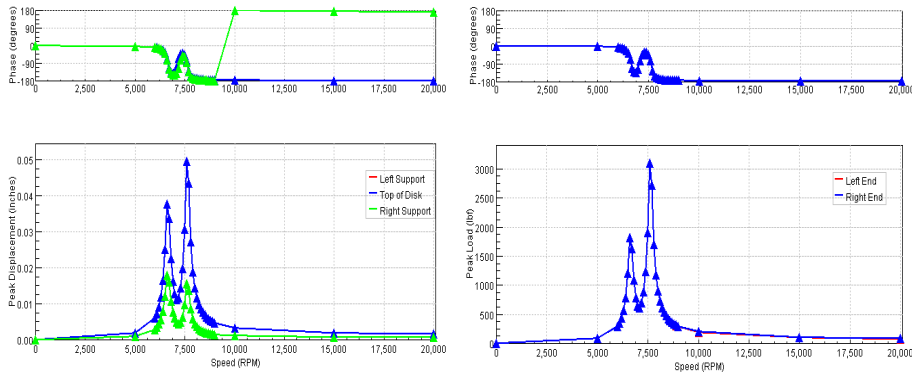


| Label | Color | Start Z (inches) | Start Eccentricity (inches) | Start Angle (degrees) | End Z (inches) | End Eccentricity (inches) | End Angle (degrees) |
|-------|-------|------------------|-----------------------------|-----------------------|----------------|---------------------------|---------------------|
| 1     | Blue  | 0.00             | 0.00                        | 0.00                  | 10.00          | 0.00                      | 0.00                |
| 2     | Blue  | 10.00            | 6.00                        | 0.00                  | 11.00          | 6.00                      | 0.00                |

- 7 -

The unbalance distribution is specified using another dialog table, as shown above. The load is defined in terms of a mass center offset and angular location (in a tangential direction), from one axial position to another. Any element in the axial span will have this distribution applied. Multiple distributions can be applied to a model by changing the axial position values. However, in this case the distribution is purely cylindrical, to excite the first mode.

## Linear Unbalance Response



- 8 -

This shows the deflection of the rotor at three locations – the two bearing locations and the top of the disk, and the loads on each support. The motion is very sharp because of the low amount of damping assumed for the supports. There are two peaks, since any stiffness asymmetry will excite both forward and backward modes. The response at each bearing is identical, as would be expected, and thus it appears that the left support response is not shown. Note that peak response was obtained by running a refined speed increment after the coarse overall analysis was calculated and examined. These two runs were stored in different Output Sets and the combined response plotted. This forced response runs very quickly – the entire combined set of calculations required 16 sections of run time.

## Simple, Easy Modeling and Review

- **Building a model is simple and quick**
  - Geometry (points and lines) can define areas
  - Meshing tools automate process
  - Direct definition of nodes/elements also provided
- **Property definition uses common interfaces**
  - Tables and dialogs provide easy specification of data
  - View options all controlled by user
- **Rapid result generation and review**
  - Output sets allow for detailed examination of response
  - Response from analysis conveyed in graphical displays
  - Run time not excessive

- 9 -

Although this example is intended to contrast a basic rotor dynamics analysis using *iSTRDYN*, the ease-of-use and ability to extract key results is certainly evident. A significant amount of design time went into the interface and organization of the various options, to enable even inexperienced users to build models and obtain results without difficulty. The modeling and review features, coupled with the relatively rapid run times, allow meaningful results to be obtained very quickly.