# MEscope Application Note 09

## Calculating the Modes of a Beam

**NOTE:** The steps in this Application Note can be duplicated using any Package that includes the **VES-8000 Finite Element Analysis (FEA)** option.

**Click here** to download the MEscope Demo Project file for this App Note.

### INTRODUCTION

In this application note, a Finite Element Analysis (FEA) model of a beam is constructed, and its FEA modes are calculated. Then, one end of the beam is ***fixed (or grounded)*** and its ***cantilever beam modes*** are calculated. Finally, these results are compared with the modal frequencies of a ***continuous beam*** derived from classical ***closed form*** equations.

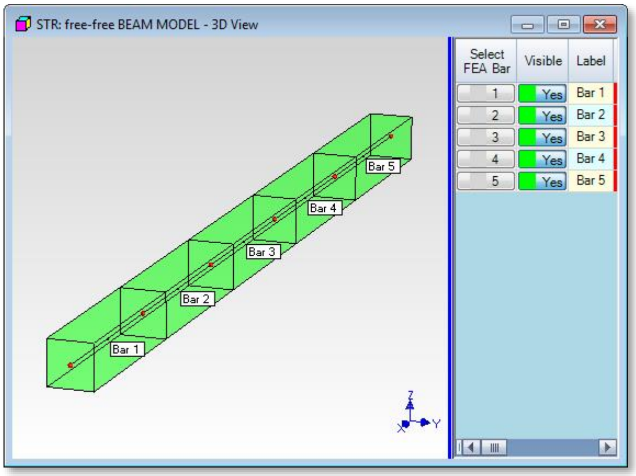



Figure 1. Beam Model with FEA Bars Attached.

Figure 1 shows the beam model. The beam is 20 inches long and has a 1-inch square cross-section. It will be modeled using five FEA bar elements called **FEA Bar Objects** in **MEscope**.

**NOTE:** An **FEA Bar** is a ***beam*** element with a ***fixed cross-section***.

Each **FEA Bar** is defined by its ***two end-points***, ***material properties***, and ***cross-sectional area*** properties. At each end-point, an **FEA Bar** imposes ***translational, rotational & inertia constraints*** on other elements attached to the same Points.

In Figure 1, additional Points, Lines and Surfaces are added to create a 3D beam model. However, only the ***six numbered centerline*** Points are required to model the dynamic properties of the beam using **FEA Bar**s.

### CREATING THE FEA MODEL

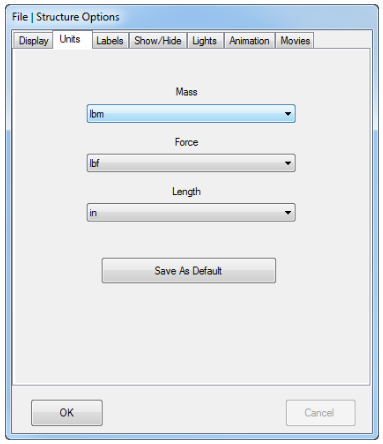
Since the dynamic equations for a **FEA Bar** require ***only its two end-points***, we will start by defining end-points along a ***centerline*** of the beam. Then each **FEA Bar** will be added between a pair of end-points, and each bar defined by specifying its ***cross sectional area*** and ***material*** properties. This will complete the FEA model needed to calculate the beam modes.

#### Creating the Beam Centerline

You can start either by downloading and opening the AppNote09.VTprj Demo Project file for this App Note, or by Creating a new Project file. To create a new Project file,

- Execute **Project | New Project** in the MEscope window, and give the new Project a name
- Execute **File | New | Structure** to open a new (empty) Structure window

- Enter **"Free-Free Beam"** in the dialog box that opens, and ***click*** on **OK**
- ***Right click*** in the graphics area of **STR: Free-Free Beam**, and execute Structure Options.

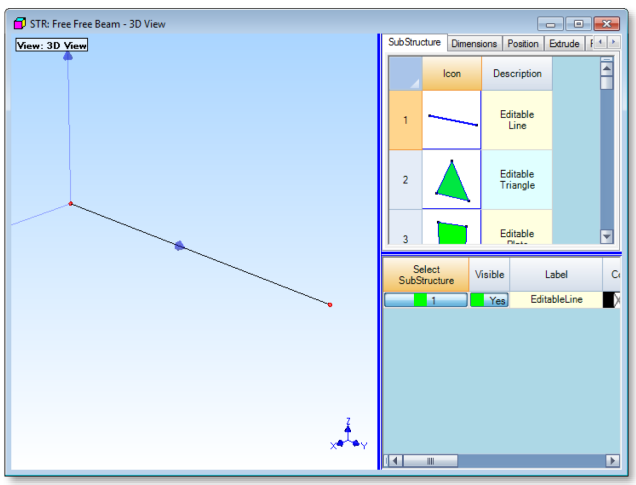



- On the **Units** tab, choose the appropriate **Mass**, **Force**, and **Length** units, and ***click*** on OK

To begin drawing the structure model,

- ***Right click*** in the graphics area of **STR: Free-Free Beam**, and execute **Draw | Drawing Assistant**.

The **Drawing Assistant tabs** will be displayed on the right.



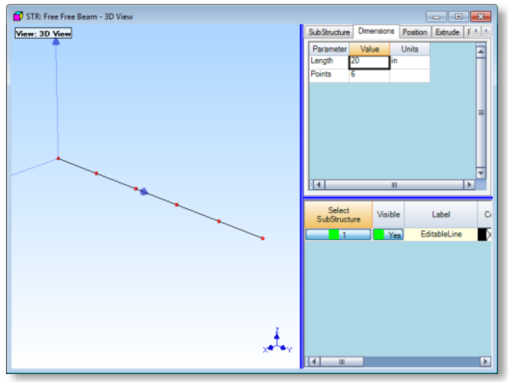

- On the **Substructure** tab, ***double click*** on the **Editable Line** substructure.

A **Line** substructure will be added to the Structure window and additional **Drawing Assistant** tabs will be displayed.

- On the **Dimensions** tab, make the following entries:

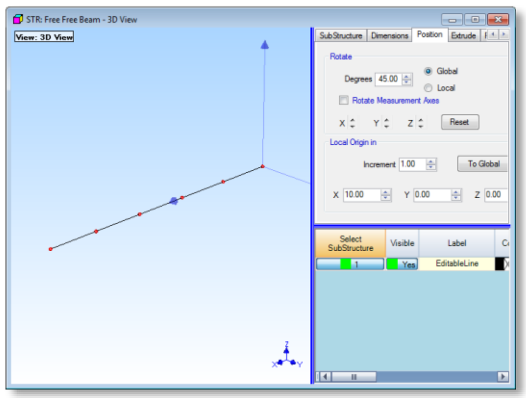
  Length (in) = 20

  Points = 6



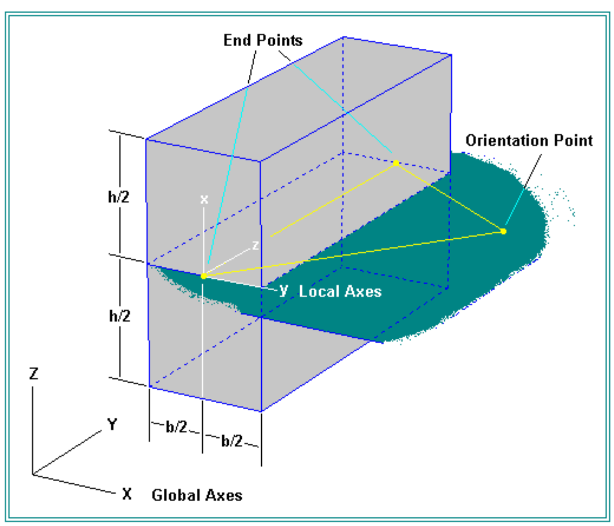
The **Line** substructure will be redrawn with **six points** and a **length of 20 units** along the **Y**-axis. Next, the centerline of the beam will be changed to lie along the X-axis instead of the **Y-axis**.

- On the **Position** tab, verify that **45 Degrees** and **Global** are selected in the **Rotate** section
- Press the **down Z** button ***twice***, to rotate the Line by ***90 degrees*** about the **Global Z-axis**
- On the **Substructure** spreadsheet enter **"Centerline"** in the Label cell



### FEA Bar Properties

To define each **FEA Bar**, its ***cross sectional area properties*** and ***material properties*** must be specified. Cross sectional area properties are specified in the **FEA Properties** dialog box, and material properties are specified in the **FEA Materials** dialog box




Rectangular Cross-Section of FEA Bars

An FEA **Bar** cross-section is described by its ***area*** and ***four area moments***. The area moments (**$I_{xx}$**, **$I_{yy}$**, **$I_{xy}$** and **J**) are computed with respect to the ***local cross section axes*** shown in the figure above. The cross section properties of a rectangular cross section are:

$$Area = \int dA = b \int_{-h/2}^{h/2} dx = h \int_{-b/2}^{b/2} dy = bh \qquad (1)$$

$$I_{xx} = \int y^2 dA = h \int_{-b/2}^{b/2} y^2 dy = \frac{b^3 h}{12} \qquad (2)$$

$$I_{yy} = \int x^2 dA = b \int_{-h/2}^{h/2} x^2 dx = \frac{bh^3}{12} \qquad (3)$$

$$I_{xy} = \int xy dA = \int_{\frac{-h}{2}}^{h/2} x \left( \int_{\frac{-b}{2}}^{b/2} y dy \right) dx = 0 \qquad (4)$$

$$J = \int \left( x^2 + y^2 \right) dA = I_{zz} = I_{xx} + I_{yy} \qquad (5)$$

All of the **FEA Bars** will have the same dimensions (width **b = 1** inch and height **h = 1** inches). Therefore:

**Area** = (1) x (1) = **1 $in^2$**

**$I_{xx}$** = (1/12) x $(1)^3$ x (1) = **0.08333 $in^4$**

**$I_{yy}$** = (1/2) x (1) x $(1)^3$ = **0.0833 $in^4$**

$I_{xy}$ = 0.0 $in^4$

**J** = 0.0833 + 0.0833 = **0.1666 $in^4$**

## FEA Bar Material Properties

Each FEA Bar requires ***three material properties***. Assuming that the beam is made from **6061-T6 aluminum**, it will have the following material properties:

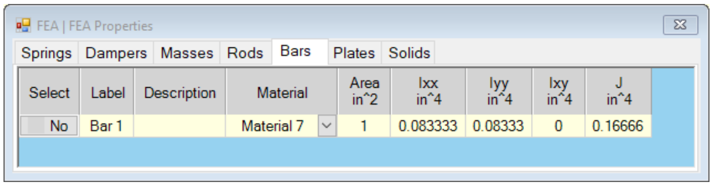
**Modulus of Elasticity (Young's Modulus) = 9.9x$10^6$ lbf/$in^2$**
**Poissons Ratio = 0.33**
**Density = 0.098 lbm/$in^3$**

- Execute **FEA | FEA Materials** in the **STR: Free-Free Beam** window
- Execute **Edit | Add** in the **FEA Materials** window to add a new material to the spreadsheet
- Enter the three properties above into the spreadsheet, (for **Material 7**) as shown below

FEA | FEA Materials

| Select | Label | Description | Elasticity lbf/in^2 | Poisson's Ratio | Density lbm/in^3 |
|---|---|---|---|---|---|
| No | Carbon Steel | | 3E+07 | 0.3 | 0.283 |
| No | Stainless Steel | | 2.8E+07 | 0.3 | 0.289 |
| No | Gray Cast Iron | | 1.4E+07 | 0.26 | 0.264 |
| No | Ductile Cast Iron | | 2.45E+07 | 0.3 | 0.256 |
| No | Aluminum | | 1E+07 | 0.33 | 0.101 |
| No | Copper | | 1.67E+07 | 0.33 | 0.321 |
| No | Concrete | | 3.7E+06 | 0.2 | 0.087 |
| No | Material 7 | | 9.9E+06 | 0.33 | 0.098 |

- Execute **FEA | FEA Properties** in the **STR: Free-Free Beam** window.
- ***Click*** on the **Bars** tab.
- Execute **Edit | Add** to add a new property to the tab
- Enter the properties into the spreadsheet, as shown below




## Adding An Orientation Point

Before attaching the five **FEA Bar** elements between the **six centerline Points**, one more point must be added to the model to complete the bar cross section definition. This point is called the cross section ***orientation point***. It is needed to define the ***"up axis"*** of the cross section of each **FEA Bar**.

**NOTE:** Each **FEA Bar** requires an ***orientation point*** that lies in the ***same plane*** as, but ***not in line*** with the FEA Bar end-points.

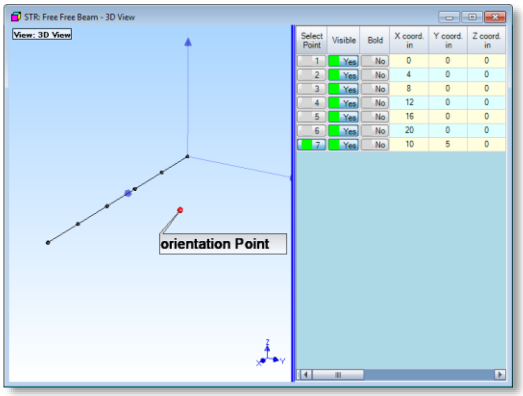
Since the FEA Bars will all lie on the same line, a ***single orientation point*** can be used by all five **FEA Bar**s. The orientation point will be located at coordinates **X=10**, **Y=5** & **Z=0** units

To add the orientation point:

- ***Right click*** in the graphics area of **STR: Free-Free Beam**, and execute **Edit | Current Objects| Points**.
- ***Right click*** in the graphics area again**,** and execute **Edit | Add Objects**
- ***Click*** in the drawing area to the right of the centerline as shown in the figure below

This will add a **Point** to the model and add a row to the Points spreadsheet

- ***Right click*** in the graphics area again**,** and execute **Edit | Add Object *again*** to ***terminate*** the Add Objects operation




- In the Points spreadsheet, edit the coordinates for the new Point (**row 7**) to:

  X coord = 10

  Y coord = 5

  X coord = 0

- Enter "**Orientation Point"** in the **Label** field of the new Point.

To ***hide*** this Point so it will no longer be displayed

- ***Click*** on the **visible** cell for the ***orientation point***, changing its entry to **No**.

To add this Point to the centerline Substructure,

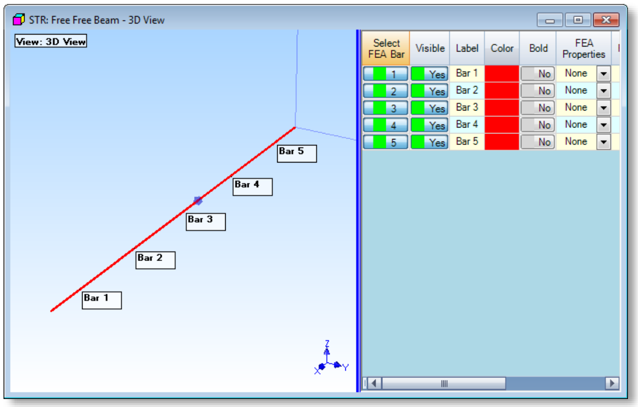
- Make sure that **Point 7** is ***selected***
- Execute **Draw | Substructures | Add Objects to Substructure** in the **STR: Free-Free Beam** window
- Select the **Editable Line** in the dialog box that opens, and ***click*** on **Add To**

This completes the definition of the ***orientation point***.

## Adding the FEA Bars

Next, the 5 **FEA Bar**s will be added to the model between the ***six centerline*** **Points**.

- ***Right click*** in the graphics area of **STR: Free-Free Beam**, and execute **Edit | Current Objects | FEA Bars**.
- ***Right click*** in the graphics area**,** and execute **Edit | Add Objects**.
- ***Click near*** the Point at the end of the **Line**, and then near the adjacent Point on the **Line** to add the first **FEA Bar** between these two Points.
- Keep ***clicking*** down the centerline to add all five FEA Bars, as shown below
- ***Right click*** in the graphics area**,** and execute **Edit | Add Objects *again*** to ***terminate*** the Add Objects operation.




## Properties & Orientation Point

The final steps needed to complete the FEA model are to assign the **FEA Properties** and the **Orientation Point** to each of the FEA Bars.

- ***Drag*** the **vertical blue bar** to the left to expose the **FEA Bar**s spreadsheet
- ***Double click*** on the **FEA Properties** column heading, and select **Bar 1** from the list in the dialog box that opens
- ***Double click*** on the **Orient Point** column heading, and enter **"7"** into the dialog box that opens.

To add the FEA Bars to the **Centerline** Substructure,

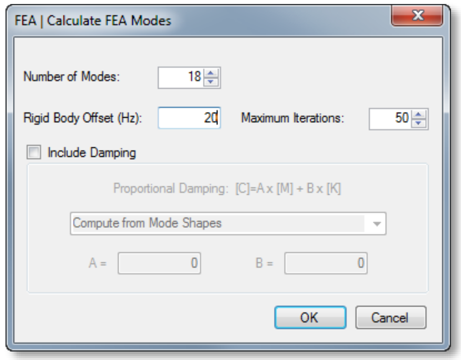
- ***Select*** all of the **FEA Bars**
- Execute **Draw | Substructures | Add Objects to Substructure** in the **STR: Free-Free Beam** window
- Select the **Centerline** Substructure in the dialog box that opens, and ***click*** on **Add To**.

## CALCULATING THE FREE-FREE FEA MODES

Now that the FEA beam has been completely defined, its free-free modes can be calculated.

**NOTE:** All modes depend on the ***boundary conditions*** on a structure. Free-free modes reflect a condition where there are ***no constraining boundary conditions*** at both ends of the beam.

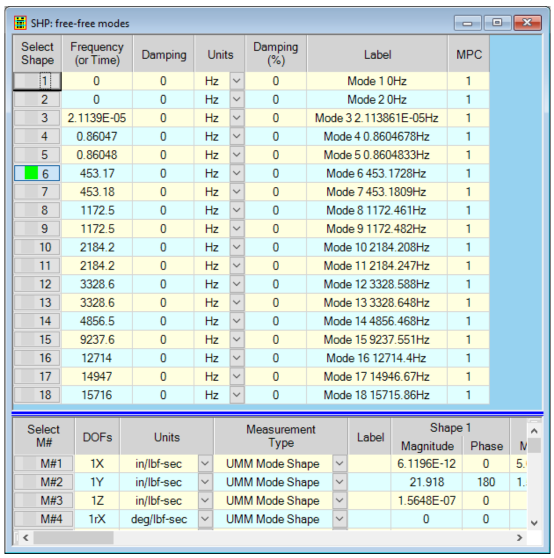
- Execute **FEA | Calculate FEA Modes** in the **STR: Free-Free Beam** window.
- ***Click*** on **Yes** in the dialog box that opens, enter parameters into the next dialog box, as shown below, and ***click*** on **OK.**




After the FEA mode shapes have been calculated,

- ***Click*** on the **New File** button in the dialog that opens, and enter **"free-free modes"** into the next dialog box.

A new Shape Table window will open listing the free-free modes of the beam, as shown below



SHP: free-free modes

| Select Shape | Frequency (or Time) | Damping | Units | Damping (%) | Label | MPC |
|---|---|---|---|---|---|---|
| 1 | 0 | 0 | Hz | 0 | Mode 1 0Hz | 1 |
| 2 | 0 | 0 | Hz | 0 | Mode 2 0Hz | 1 |
| 3 | 2.1139E-05 | 0 | Hz | 0 | Mode 3 2.113861E-05Hz | 1 |
| 4 | 0.86047 | 0 | Hz | 0 | Mode 4 0.8604678Hz | 1 |
| 5 | 0.86048 | 0 | Hz | 0 | Mode 5 0.8604833Hz | 1 |
| 6 | 453.17 | 0 | Hz | 0 | Mode 6 453.1728Hz | 1 |
| 7 | 453.18 | 0 | Hz | 0 | Mode 7 453.1809Hz | 1 |
| 8 | 1172.5 | 0 | Hz | 0 | Mode 8 1172.461Hz | 1 |
| 9 | 1172.5 | 0 | Hz | 0 | Mode 9 1172.482Hz | 1 |
| 10 | 2184.2 | 0 | Hz | 0 | Mode 10 2184.208Hz | 1 |
| 11 | 2184.2 | 0 | Hz | 0 | Mode 11 2184.247Hz | 1 |
| 12 | 3328.6 | 0 | Hz | 0 | Mode 12 3328.588Hz | 1 |
| 13 | 3328.6 | 0 | Hz | 0 | Mode 13 3328.648Hz | 1 |
| 14 | 4856.5 | 0 | Hz | 0 | Mode 14 4856.468Hz | 1 |
| 15 | 9237.6 | 0 | Hz | 0 | Mode 15 9237.551Hz | 1 |
| 16 | 12714 | 0 | Hz | 0 | Mode 16 12714.4Hz | 1 |
| 17 | 14947 | 0 | Hz | 0 | Mode 17 14946.67Hz | 1 |
| 18 | 15716 | 0 | Hz | 0 | Mode 18 15715.86Hz | 1 |

| Select M# | DOFs | Units | Measurement Type | Label | Shape 1 Magnitude | Shape 1 Phase |
|---|---|---|---|---|---|---|
| M#1 | 1X | in/lbf-sec | UMM Mode Shape | | 6.1196E-12 | 0 |
| M#2 | 1Y | in/lbf-sec | UMM Mode Shape | | 21.918 | 180 |
| M#3 | 1Z | in/lbf-sec | UMM Mode Shape | | 1.5648E-07 | 0 |
| M#4 | 1rX | deg/lbf-sec | UMM Mode Shape | | 0 | 0 |

Notice that the first five modes have ***essentially zero*** frequencies. These are the **rigid body modes** of the free-free beam. The first flexible body mode is at **453 Hz**. Notice also that ***all*** of the modes have ***0% damping*** because the FEA model ***contains no damping***.

## ANIMATING THE FREE-FREE MODE SHAPES

- ***Right click*** in the spreadsheet area of **SHP: Free-Free Modes**, and execute **Animate | Animate Shapes**.

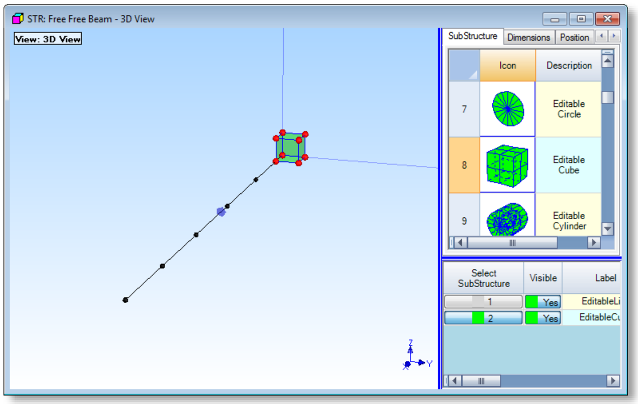
As you click on each Shape button in **SHP: Free-Free Modes** to animate its shape, notice the following,

- Shapes ***1 through 5*** have rigid body mode shapes.
- Shape pairs ***6 & 7, 8 & 9, 10 & 11, 12 & 13*** are all ***pairs of repeated roots.*** Each pair has the ***same frequency*** but a ***different*** mode shape.
- Shapes ***14 through 18*** are ***longitudinal*** mode shapes

## CREATING A 3D BEAM MODEL

To improve the realism of the structure model, another Substructure will be added to the beam model to turn it into a 3D model.

- ***Right click*** in the graphics area of **STR: Free-Free Beam**, and execute **Draw | Drawing Assistant**
- In the list on the **Substructure** tab, ***double-click*** on the **Editable Cube** substructure to add it to the structure model, as shown below




- On the **Dimensions** tab, make the following entries:

  Width = 1

  Points = 2

  Height = 1

  Points = 2

  Length = 20

  Points = 6

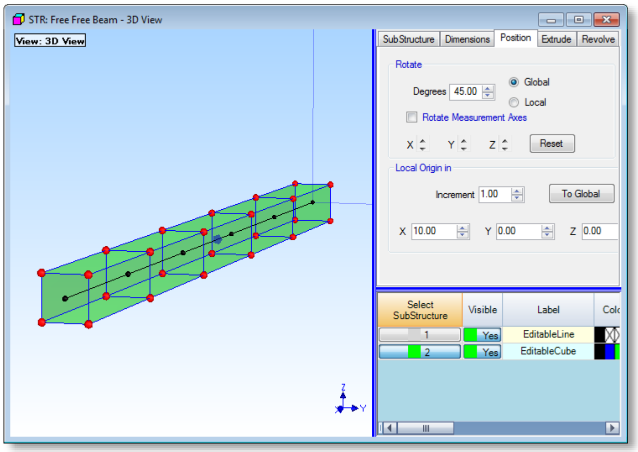
- On the **Position** tab, make the following **Local Origin** entries:

  X = 0

  Y = 0

  Z = 0

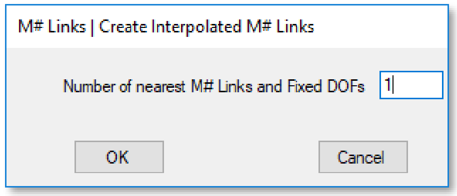
The completed Cube substructure should now ***surround*** the centerline, as shown below



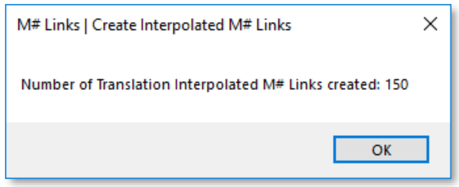

## CREATING INTERPOLATED M# LINKS

To deform the 3D model using the mode shape data at the five Points along the centerline, **Interpolated M# Links** must be created for the Points at each cross section of the 3D model that are closest to a centerline Point.

- ***Right click*** in the graphics area of **STR: Free-Free Beam**, and execute **M# Links | Create Interpolated M# Links**
- ***Click*** through the dialog boxes that open, enter **"1"** into the next dialog box, as shown below, and ***click*** on **OK**

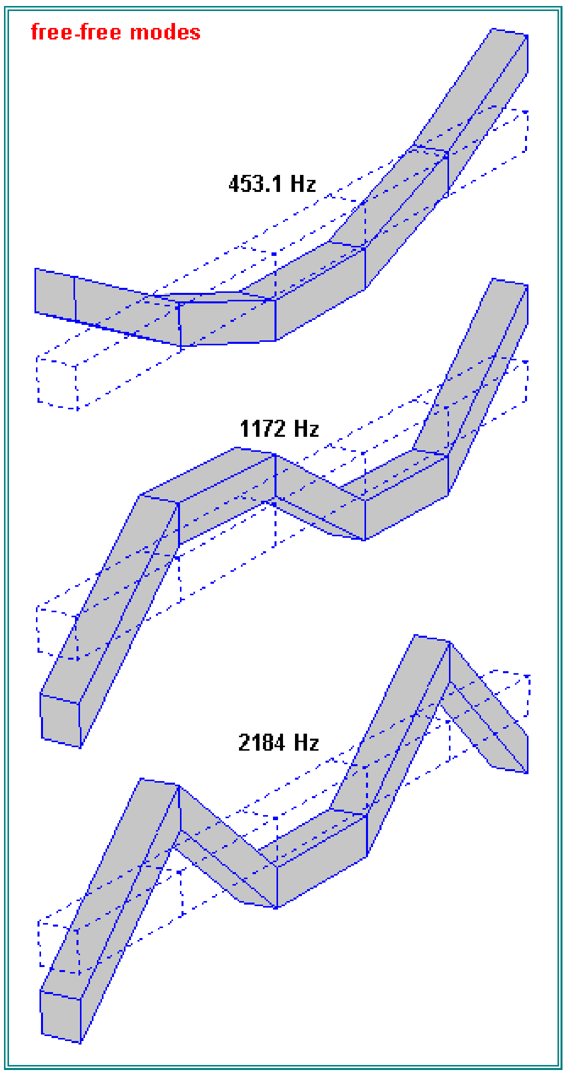



The following will then open stating that 150 M# Links have been created for all of the cross section Points of the 3D model




- ***Right click*** in the graphics area of **STR: Free-Free Beam**, and execute **Draw | Animate Shapes**

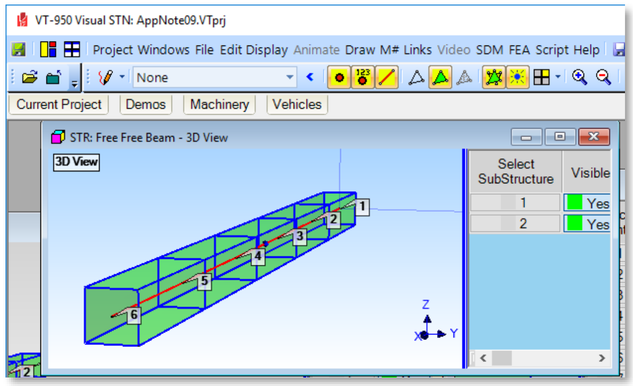
The mode shapes from **SHP: free-free modes** should now be displayed in animation on the 3D beam model. Several of the vertical bending modes are shown below



## CREATING A CANTEILEVER BEAM

To create a cantilever beam structure, the **free-free beam** structure will be copied into a new Structure window and the boundary condition at one end of the beam will be changed from free-free to fixed.

- ***Select*** NONE in the Animation Source list box for the **STR: Free-Free Beam** window, as shown below
- Execute **File | Clone Structure and Source** in the **STR: Free-Free Beam** window
- ***E***nter **"Cantilever Beam"** into the next dialog box, and ***click*** on **OK**




A new Structure window will open with a copy of the beam model in it. To make a cantilever beam model more realistic, a vertical ***ground plane*** will be added to one end of the beam, and its Points will be fixed

- ***Right click*** in the graphics area of the **STR: Cantilever Beam** window and execute **Draw | Drawing Assistant**
- In the list on the **Substructure** tab, ***double-click*** on the **Editable Plate** substructure to add it to the structure model
- On the **Dimensions** tab enter:

  Width = 4

  Points = 2

  Height = 4

  Points = 2

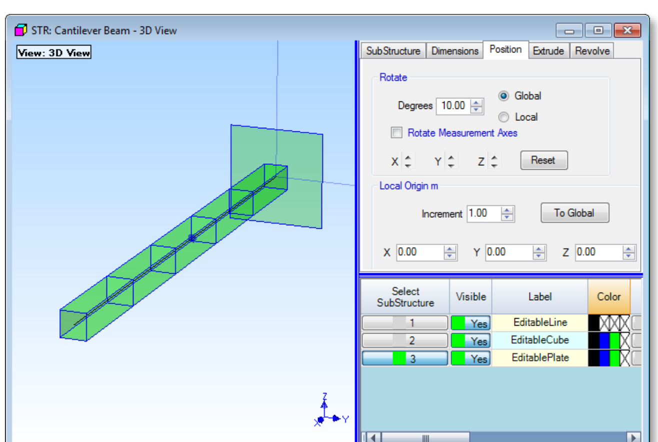
- On the **Position** tab enter the Local Origin (in):

  X = 0

  Y = 0

  Z = 0

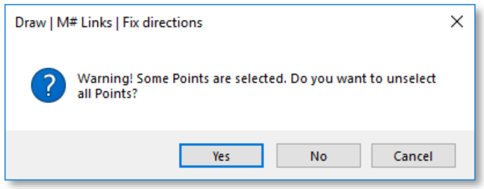
- In the Substructures spreadsheet, enter **"Ground Plane"** into the **Label** cell



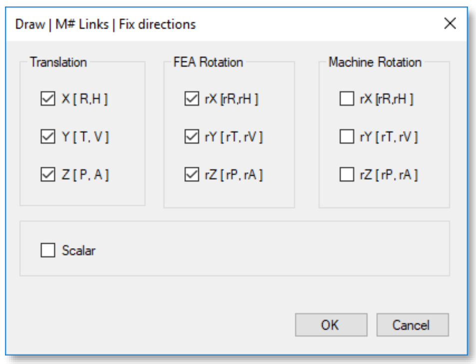
## Fixed One End of the Beam

To model a cantilever beam, one end of the free-free beam must be fixed so that it cannot ***translate*** or ***rotate***. This is equivalent to ***clamping*** the beam to a rigid wall or grounded foundation.

- ***Right click*** in the graphics area of the **STR: Cantilever Beam** window and execute **Edit | Current Objects | Points**
- ***Hold down*** the **Ctrl** key and ***click near*** the Point on the centerline ***closest*** to the **Ground Plane** to ***select*** it
- ***Right click*** in the graphics area, and execute **M# Links | Fix directions**
- ***Click*** on **No** in the following dialog box that opens




- Make the selections in the following dialog box, and ***click*** on **OK**

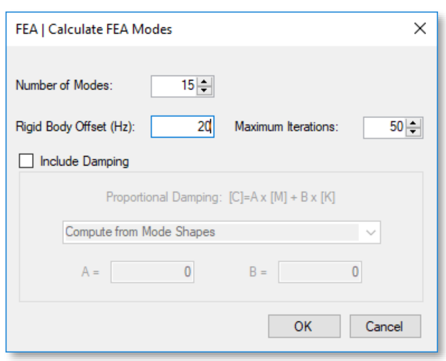



The final dialog box will report that ***6* DOFs** of the ***selected*** end-Point on the **Centerline** have been fixed.

## CALCULATING THE CANTILEVER FEA MODES

The FEA model for the cantilever beam is now ready for calculating the cantilever modes.

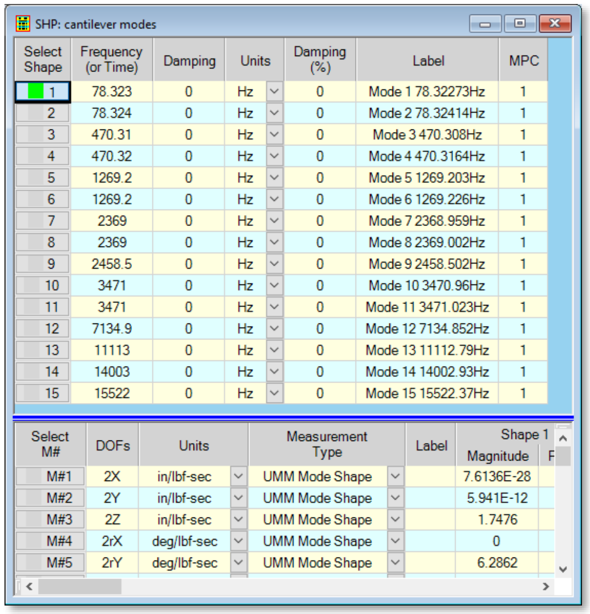
- U***n-select*** all Points in the **STR: Cantilever Beam** window
- Execute **FEA | Calculate FEA Modes** in the **STR: Cantilever Beam** window
- ***Click*** on **Yes** in the dialog box that opens, enter parameters into the next dialog box as shown below, and ***click*** on **OK.**




After the cantilever modes have been calculated,

- ***Click*** on the **New File** button in the dialog that opens, and enter **"cantilever modes"** into the next dialog box.

The **SHP: cantilever modes** window will open listing the cantilever FEA modes of the beam.



SHP: cantilever modes

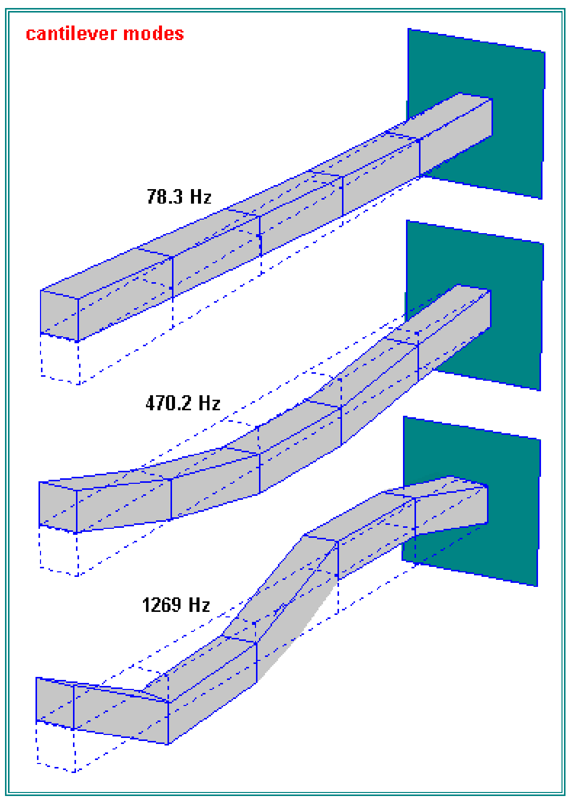
| Select Shape | Frequency (or Time) | Damping | Units | Damping (%) | Label | MPC |
|---|---|---|---|---|---|---|
| 1 | 78.323 | 0 | Hz | 0 | Mode 1 78.32273Hz | 1 |
| 2 | 78.324 | 0 | Hz | 0 | Mode 2 78.32414Hz | 1 |
| 3 | 470.31 | 0 | Hz | 0 | Mode 3 470.308Hz | 1 |
| 4 | 470.32 | 0 | Hz | 0 | Mode 4 470.3164Hz | 1 |
| 5 | 1269.2 | 0 | Hz | 0 | Mode 5 1269.203Hz | 1 |
| 6 | 1269.2 | 0 | Hz | 0 | Mode 6 1269.226Hz | 1 |
| 7 | 2369 | 0 | Hz | 0 | Mode 7 2368.959Hz | 1 |
| 8 | 2369 | 0 | Hz | 0 | Mode 8 2369.002Hz | 1 |
| 9 | 2458.5 | 0 | Hz | 0 | Mode 9 2458.502Hz | 1 |
| 10 | 3471 | 0 | Hz | 0 | Mode 10 3470.96Hz | 1 |
| 11 | 3471 | 0 | Hz | 0 | Mode 11 3471.023Hz | 1 |
| 12 | 7134.9 | 0 | Hz | 0 | Mode 12 7134.852Hz | 1 |
| 13 | 11113 | 0 | Hz | 0 | Mode 13 11112.79Hz | 1 |
| 14 | 14003 | 0 | Hz | 0 | Mode 14 14002.93Hz | 1 |
| 15 | 15522 | 0 | Hz | 0 | Mode 15 15522.37Hz | 1 |

| Select M# | DOFs | Units | Measurement Type | Label | Shape 1 Magnitude |
|---|---|---|---|---|---|
| M#1 | 2X | in/lbf-sec | UMM Mode Shape | | 7.6136E-28 |
| M#2 | 2Y | in/lbf-sec | UMM Mode Shape | | 5.941E-12 |
| M#3 | 2Z | in/lbf-sec | UMM Mode Shape | | 1.7476 |
| M#4 | 2rX | deg/lbf-sec | UMM Mode Shape | | 0 |
| M#5 | 2rY | deg/lbf-sec | UMM Mode Shape | | 6.2862 |

## ANIMATING THE CANTLEVER MODE SHAPES

To display the modes of the cantilever beam in animation

- ***Right click*** in the spreadsheet area of **SHP: Cantilever Modes**, and execute **Animate | Animate Shapes**



As you click on each Shape button in **SHP: Cantilever Modes** to animate its shape, notice the following,

- Shapes pairs ***1 & 2, 3 & 4, 5 & 6, 7 & 8, 10 & 11*** are all pairs of ***repeated roots***. Each pair has the ***same frequency*** but a ***different*** mode shape
- Shapes ***9*** is the first ***longitudinal*** mode shape
- Shapes ***12 through 15*** are all ***longitudinal*** mode shapes

## ANALYTICAL COMPARED TO FEA FREQUENCIES

The reference textbook ***Formulas for Natural Frequency and Mode Shape***, Robert D. Blevins, 1979, page 108, contains formulas for the modal frequencies of continuous beams. The modal frequencies of both free-free and cantilever beams can be determined with the following formula.

$$f_i = \left( \frac{\lambda_i^2}{2\pi L^2} \right) \left( \frac{EI}{m} \right)^{1/2} \qquad (6)$$

where:

$\mathbf{f_i}$ = modal frequency of mode (j), in Hz.

**L** = length of the beam

$\boldsymbol{\lambda_i}$ = 4.730, 7.853, 10.996 i = 1, 2, 3 (**free-free beam**)

$\boldsymbol{\lambda_i}$ = 1.875, 4.694, 7.855 i = 1, 2, 3 (**cantilever beam**)

**E** = modulus of elasticity (9.9xE6)

**I** = cross sectional inertia (0.0833)

**m** = mass per unit length = (density)(area)

= (0.098) / (386.4 lbf-sec$^2$/in/lbm)

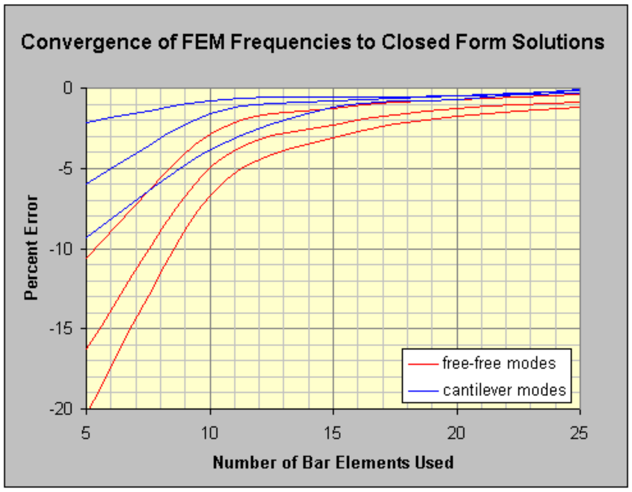
Equation (6) was used to calculate the frequencies of the first three modes of both free-free and cantilever beams. The table below contains a comparison of the analytical frequencies with the FEA frequencies calculated with MEscope in this App Note.

| Mode | MEscope FEA | Analytical |
|---|---|---|
| 1st free-free | 453 | 507 |
| 2nd free-free | 1172 | 1399 |
| 3rd free-free | 2184 | 2742 |
| 1st cantilever | 78 | 80 |
| 2nd cantilever | 470 | 500 |
| 3rd cantilever | 1269 | 1400 |

## CONCLUSIONS

Notice in the Table above that the **FEA** modal frequencies are lower, than the analytical frequencies in all cases. This is because the analytical formula is for a ***continuous*** beam, whereas the continuous beams were modeled in MEscope using only **five FEA Bar** elements.

The figure below shows ***how rapidly*** the first three FEA modal frequencies will converge toward the analytic answers when more **FEA Bar** elements are used.




***FEA Modal Frequency Errors versus Number of Bar Elements.***