# VIBRANT MEscope Application Note 09 Calculating the Mode Shapes of a Beam

The steps in this Application Note can be carried out using any Package that includes the **VES-8000 FEA** option. They can also be carried out using the **AppNote09** project file. These steps might also require MEscope software with a *more recent release date*.

#### **APP NOTE 09 PROJECT FILE**

• To retrieve the Project for this App Note, <u>click here</u> to download AppNote09.zip

This Project file contains numbered Hotkeys & Scripts for carrying out the steps of this App Note.

• Hold down the Ctrl key and click on a Hotkey to display its Script window

#### **INTRODUCTION**

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



Beam Model With Five FEA Bars Attached on Its Centerline.

The figure above shows the beam model. The beam is **20 inches long** and has a **1-inch square cross-section**. It is modeled using **five FEA Bars** called **FEA Objects** in MEscope.

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

Each FEA Bar is defined by its two end-points, its cross-sectional area, and its material properties.

An FEA Bar imposes translational, rotational & inertia constraints on other FEA Objects attached to its end Points.

Other Points, Lines, & Surfaces are added to the model in the figure above to create a 3D beam model.

Only the six centerline Points are required to model the dynamic properties of the beam using five FEA Bars.

## **STEP 1 - CREATING THE BEAM CENTERLINE**

- Press Hotkey 1 Centerline
- To re-draw the Centerline, *click* on Yes in the dialog box that opens

Since each **FEA Bar only requires two end-points**, we start by defining **six end-points along a centerline** of the beam between which to attach **five FEA Bars**.



Structure Window Showing the Beam Centerline.

The centerline is created using the **Drawing Assistant**.

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

A Line substructure is added to the Structure window and additional Drawing Assistant tabs are displayed.

- On the **Dimensions** tab, make the following entries:
  - Length (in) **→** 20
  - Length Points → 6

🗑 '31% Free-Free Beam - 30 View						
3D View	Substructure Dimensions Position Extrude Revolve					
[Pesp. 40]	Parameter Value	Units				
	Length 20	in				
	Length Points 6	in				
	) Cut at such as a					
	Substructures	Ohiut		Ulda Canta		
	Substructure Visible	Label Properties	Transparent T	exture Data 1		
	1 Yes	EditableLine	No	No Translati		
z x↓→v	✓ All Selected ✓ SubStructures Editable	Visible   Has Label	Display Label 🗌 M	Aag & Phs Tra		

Centerline Substructure with Length 20 inches and 6 Points.

The Line substructure is redrawn with six points and a length of 20 units along the Y-axis. Next, the centerline of the beam is rotated so that it lies along the X-axis instead of the Y-axis.

- On the **Position** tab, *press* the down Z button *until* the Centerline is aligned with the X-axis
- On the Substructure spreadsheet edit the Label to Centerline



Centerline Rotated to Align With the Global X-Axis.

# **STEP 2 - ADDING FEA BARS TO THE CENTERLINE**

- Press Hotkey 2 Add FEA Bars to the Centerline
- To Add FEA Bars to the Centerline, *click* on Yes in the dialog box that opens

Cross sectional area properties are specified in the **FEA Properties** dialog box, and material properties are specified in the **FEA Materials** dialog box



• *Click* on the Bars tab to show the FEA Bar Properties

STR: Free-Free Beam Showing FEA Bars, Their Properties & Material Properities.

#### FEA PROPERTIES

The **FEA Properties & Materials** dialog boxes are displayed *on the right* in the figure above. Assuming that the beam is made from **6061-T6 aluminum**, the property labeled **Bar 1** is defined with the **Area & Initials** listed above. **Bar 1** also has material properties defined in the **Materials** dialog box as **Material 7**.



Rectangular Cross-Section of FEA Bars

An FEA Bar cross-section is described by its area & four area moments. The area moments (I<sub>xx</sub>, I<sub>yy</sub>, I<sub>xy</sub>, J) are computed with respect to the local cross-section axes shown in the figure above. The properties of a rectangular cross-section are,

$$\begin{split} Area &= \int dA = b \int_{-h/2}^{h/2} dx = h \int_{-b/2}^{b/2} dy = bh \\ I_{xx} &= \int y^2 \, dA = h \int_{-b/2}^{b/2} y^2 \, dy = \frac{b^3 h}{12} \\ I_{yy} &= \int x^2 \, dA = b \int_{-h/2}^{h/2} x^2 \, dx = \frac{bh^3}{12} \\ I_{xy} &= \int xy \, dA = \int_{-\frac{h}{2}}^{h/2} x \left( \int_{-\frac{b}{2}}^{b/2} y \, dy \right) dx = 0 \\ J &= \int (x^2 + y^2) \, dA = I_{zz} = I_{xx} + I_{yy} \end{split}$$

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

Area = (1) x (1) = 1 in2 Ixx = (1/12) x (1)3 x (1) = 0.08333 in4 Iyy = (1/2) x (1) x (1)3 = 0.0833 in4 Ixy = 0.0 in4 J = 0.0833 + 0.0833 = 0.1666 in4

#### **MATERIAL PROPERTIES**

Each FEA Bar requires three material properties,

Modulus of Elasticity (Young's Modulus) = 9.9x10<sup>6</sup> lbf/in<sup>2</sup> Poissons Ratio = 0.33 Density = 0.098 lbm/in<sup>3</sup>

#### ADDING THE FEA BARS

If you chose to add the **FEA** bars, the Centerline Points are displayed in the **STR: Free-Free Beam** window, as shown below.



STR: Free-Free Beam Window Showing Six Centerline Points.

To add the five FEA Bars between each pair of the six centerline Points,

- Execute FEA | FEA Bars
- Right click in the graphics area and execute Edit | Add Objects to activate the Add Objects operation
- *Click near* the Point at the end of the Line, and then *click near* the adjacent Point on the Line to add the first FEA Bar between those two Points
- Keep clicking down the centerline to add five FEA Bars between pairs of Points as shown below
- Right click in the graphics area and execute Edit | Add Objects again to terminate the Add Objects operation
- Right click in the graphics area and execute Display | FEA Objects | Show Labels



STR: Free-Free Beam Showing Five FEA Bars Added Between Points on the Centerline.

## CREATING AN FEA BARS SUBSTRUCTURE

With the five FEA Bars still selected,

- Right click in the graphics area and execute Draw | Substructures | Add Objects
- *Click* on New Substructure and enter "FEA Bars" in the dialog box that opens

#### ADDING A CROSS SECTION ORIENTATION POINT

One more point must be added to the model to complete the bar cross-sectional definitions. This point is called the **cross-sectional Orientation Point**.

An Orientation Point is needed to define the up axis of the cross-section of each FEA Bar.

Each FEA Bar requires an Orientation Point that lies in the same plane as, but not in line with the FEA Bar.

Since the FEA Bars all lie on the same centerline, a single Orientation Point can be used by all five FEA Bars.

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 | Points
- Right click in the graphics area again, and execute Edit | Add Objects to activate the Add Objects operation
- *Click* in the drawing area to the right of the centerline as shown in the figure below
- Right click in the graphics area, and execute Edit | Add Object again to terminate the Add Objects operation

A new Point has been added to the model and its properties are listed on the last row to the Points spreadsheet.

- In the Points spreadsheet, edit the coordinates for the new Point (row 7) to X Coord → 10, Y Coord → 5, Z Coord → 0
- Enter "Orientation Point" in the Label field of the new Point

To create its own Substructure,

- Right click in the graphics area and execute Draw | Substructures | Add Objects
- Click on New Substructure and enter "Orientation Point" in the dialog box that opens

This completes the addition of the Orientation Point.



Substructures Spreadsheet Showing FEA Bars & Orientation Point.

# FEA BAR PROPERTIES & ORIENTATION POINT

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

- Execute FEA | FEA Bars
- *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" (the **Orientation Point** row in the **Points** spread-sheet) into the dialog box that opens



FEA Bars Spreadsheet Showing FEA Properties & Orientation Point for the FEA Bars.

#### **STEP 3 - CALCULATING THE FREE-FREE MODE SHAPES**

#### Press Hotkey 3 Calculate Free-Free Mode Shapes

Sweep animation of the Free-Free mode shapes will begin on the left, and a new Shape Table window will open on the right listing the Free-Free mode shapes of the beam.

All mode shapes depend on the **boundary conditions** of their structure.

Free-Free mode shapes reflect a condition where there are no constraining boundary conditions at both ends of the beam.



Sweep Animation of the Mode Shapes of the Free-Free Beam.

The first five mode shapes have essentially zero frequencies.

These are the **rigid body mode shapes** of the Free-Free beam.

All the mode shapes have 0% damping because the FEA model contains no damping.

- *Click* on a Select Shape button in SHP: Free-Free Mode Shapes to display its mode shape
- Shapes 1 through 5 have rigid body mode shapes
- Shape pairs 6 & 7, 8 & 9, 10 & 11, 12 & 13 are pairs of repeated roots

Each repeated root pair has the same frequency but a different mode shape.

• Shapes 14 through 18 are longitudinal mode shapes

#### **STEP 4 - 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.

- Press Hotkey 4 Create a 3D Beam Model
- In the dialog box that opens, *click* on Yes to re-draw a 3D Beam, or *click* on No to display the Free-Free mode shapes on a 3D Beam

#### **RE-DRAWING THE 3D BEAM SUBSTRUCTURE**

If you clicked on Yes, the Drawing Assistant tabs are displayed in the STR: Free-Free Beam window as shown below



STR: Free-Free Beam Showing Drawing Assistant Tabs.

- In the list on the Substructure tab, *double-click* on the Editable Cube substructure to add it to the structure model
- On the **Dimensions** tab, make the following entries:
  - Width **→** 1
  - Width Points  $\Rightarrow 2$ Height  $\Rightarrow 1$
  - Height Points  $\rightarrow 2$
  - Length → 20
  - Length Points → 6
- On the **Position** tab, make the following **Local Origin** entries:
  - X → 10
  - Y **→** 0
  - Z → 0
- Edit the Label to "3D Beam" in the Substructures spreadsheet

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



Cube Added to the Centerline of STR: Free-Free Beam.

# DISPLAYING FREE-FREE MODE SHAPES ON THE 3D BEAM

- Press Hotkey 4 Create a 3D Beam Model again
- In the dialog box that opens, *click* on No to display the Free-Free mode shapes



453 Hz Flexible Free-Free Mode Shape of the Beam.



1172 Hz Flexible Free-Free Mode Shape of the Beam.



2184 Hz Flexible Free-Free Mode Shape of the Beam.

# **STEP 5 - CALCULATING THE CANTILEVER MODE SHAPES**

# Press Hotkey 5 Calculate Cantilever Mode Shapes

After the mode shapes of the Cantilever beam are calculated, a new **STR: Cantilever Beam** window will open *on the left* displaying the mode shapes in **SHP: Cantilever Mode Shapes** *on the right* in sweep animation, as shown below.

The only difference between the Free-Free mode shapes and the Cantilever mode shapes is that **Point #1 Centerline has** been fixed to model the cantilever boundary condition.



Sweep Animation of the Mode Shapes in SHP: Cantilelver Mode Shapes.

# **STEP 6 - CREATING A 3D CANTILEVER BEAM MODEL**

- Press Hotkey 6 Create a 3D Cantilever Beam Model
- In the dialog box that opens, *click* on Yes to re-draw Wall, or *click* on No to display the Cantilever mode shapes

If you *clicked* on Yes, the Drawing Assistant tabs are displayed in the STR: Cantilever Beam window as shown below



STR: Cantilever Beam Showing Drawing Assistant Tabs.

#### **DRAWING THE WALL**

- In the list on the **Substructure** tab, *double-click* on the **Editable Plate** substructure
- On the **Dimensions** tab enter

Width **→** 4

Width Points **>** 2

Height 🗲 4

Hight Points → 2

• On the **Position** tab enter the Local Origin (in)

X → 0

Y **→** 0

Z → 0

• In the Substructures spreadsheet, enter "Wall" into the Label cell



STE: Cantilever Beam Showing 3D Beam & Ground Plane.

# DISPLAYING THE CANTILEVER BEAM MODE SHAPES

- Press Hotkey 4 Create a 3D Beam Model again
- In the dialog box that opens, *click* on No to display the Cantilever mode shapes

Sweep animation of the cantilever mode shapes will begin, and a Shape Table will open *on the right* listing the cantilever modes of the beam as shown below.



78 Hz Cantilever Mode Shape of the Beam.



470 Hz Cantilever Mode Shape of the Beam.



1269 Hz Cantilever Mode Shape of the Beam.

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 pairs of repeated roots
- Each repeated root pair has the same frequency but a different mode shape
- Shape **9** is the **first axial mode shape**
- Shapes 12 through 15 are all axial mode shapes

#### FEA COMPARED TO ANALYTICAL MODAL 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 & Cantilever beams** are determined with the following formula.

$$\mathbf{f}_{i} = \left(\frac{\lambda_{i}^{2}}{2\pi L^{2}}\right) \left(\frac{\mathbf{EI}}{\mathbf{m}}\right)^{1/2}$$

**f**<sub>i</sub> = modal frequency of mode (i), in Hz

L = length of the beam

 $\lambda_i = 4.730, 7.853, 10.996 \quad i = 1, 2, 3 \ (Free-Free \ beam)$ 

 $\lambda_i = 1.875, 4.694, 7.855$  i = 1, 2, 3 (Cantilever beam)

**E** = modulus of elasticity (9.9 x E06)

I = cross sectional inertia (0.0833)

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

= (0.098) / (386.4 lbf-sec<sup>2</sup>/in/lbm)

The above formula was used to calculate the frequencies of the **first three mode shape pairs** of both the Free-Free & Cantilever beams. The table below compares the **FEA** modal frequencies calculated from **five FEA Beam elements** versus the analytical frequencies calculated with the above formula.

Mode Shape	MEscope FEA	Analytical
1 <sup>st</sup> Free-Free	453	507
2 <sup>nd</sup> Free-Free	1172	1399
3 <sup>rd</sup> Free-Free	2184	2742
1 <sup>st</sup> cantilever	78	80
2 <sup>nd</sup> cantilever	470	500
3 <sup>rd</sup> cantilever	1269	1400

FEA Mode Shape Frequencies versus Analytical Frequencies

#### CONCLUSIONS

In the Table above, each **MEscope FEA** mode shape frequency is lower than its corresponding **Analytical** modal frequency.

The analytical formula is for a continuous beam, whereas the Free-Free & Cantilever beams were modeled in MEscope using only five FEA Bar elements.

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



FEA Modal Frequency Errors versus Number of Bar Elements.

#### **STEP 7 - REVIEW**

To review the steps of this App Note,

• Press Hotkey 7 Review Steps