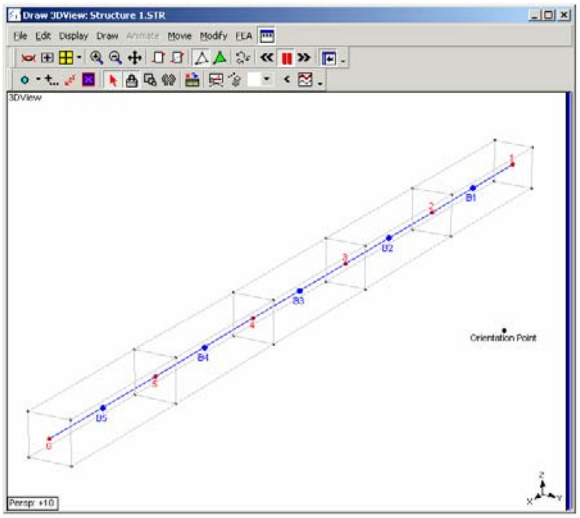
Vibrant Technology, Inc.

ME'scopeVES Application Note #9

Calculating the Modes of a Beam

INTRODUCTION

In this note, **ME'scopeVES** is used to build a finite element model (FEM) of a beam and calculate its free-free modes. Then, one end of the beam is fixed and its cantilever beam modes are calculated. Finally, these answers are compared with modal frequencies of a continuous beam derived from classical *closed form* equations.



Beam model composed of five Bar elements.

The figure above shows the beam model. The beam is 20 inches long and has a 1 inch square cross-section. It will be modeled using 5 bar finite elements called **FE Bars** in **ME'scopeVES**.

Note: A **Bar** element is simply a beam of *fixed cross-section*. Each **Bar** element is described by two end-point locations, material properties, and cross-section properties. A **Bar** attaches to other model elements at its two end-points. At each end-point the Bar imposes translational, rotational and inertia constraints on other elements attached to its end points.

In the above figure, additional points and lines are used to depict each Bar element cross-section. However, only *the six numbered centerline points are required to model the dynamic properties* using five **FE Bar** elements.

Steps described in this Application Note can be duplicated using VT-550 Visual SDM or any package that includes VES-500 Structural Modifications.

OUR CONSTRUCTION SEQUENCE

Since the dynamic equations for a **Bar** element *involve only its two end-points*, we will start by defining the location of the element end **Points** on the *centerline* of the beam. Then we will define each **FE Bar** element between a pair of points by specifying its *cross section* and *material* properties. This will complete the dynamic model needed to calculate the beam modes.

We will calculate the modes of the beam under two different sets of boundary conditions. First we will compute and animate its *free-free* modes, where the beam "floats in space" without connection to earth or any other substructure. Then we will constrain one end of the beam from translating or rotating and calculate its *cantilever* (or clamped-free) modes.

We will build the beam first as a "stick figure" model using only the **FE Bar** end points connected by lines. Then, cross sections and surfaces will be added to make the model more realistic.

All geometric models will be built using the **Drawing Assistant**. This tool defines Substructures using points, lines and surfaces.

BUILDING THE BEAM CENTERLINE

To begin a new Project:

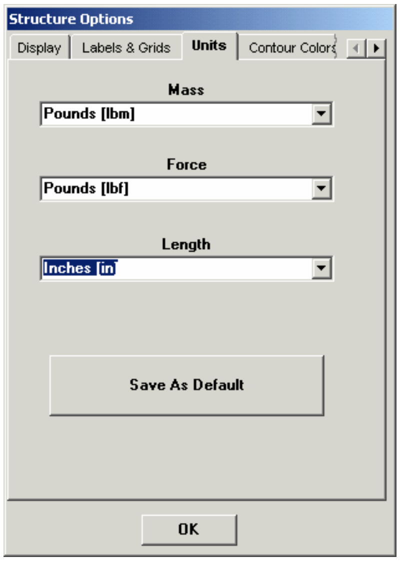
Open **ME'scopeVES**.

Execute: **File | Project | New**.

Execute: **File | New | Structure** to open a new (empty) Structure window.

To set up the units for the beam model,

Execute: **File | Options** in the Structure window. A **Structure Options** dialog will open.



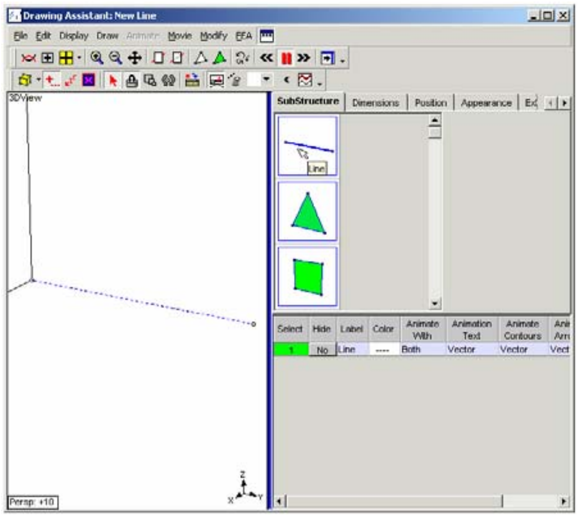
Mass: Pounds [lbm]
Force: Pounds [lbf]
Length: Inches [in]

- On the **Units** tab, choose the following units and then click on **OK**.

  Mass = Pounds (**lb$_m$**)
  Force = Pounds-force (**lb$_f$**)
  Length = Inches (**in**)

Execute: **Draw | Drawing Assistant**.

The **Drawing Assistant** tabs will be displayed.



Select	Hide	Label	Color	Animate With	Animation Text	Animate Contours	Ani Arr
1	No	Line	----	Both	Vector	Vector	Vect

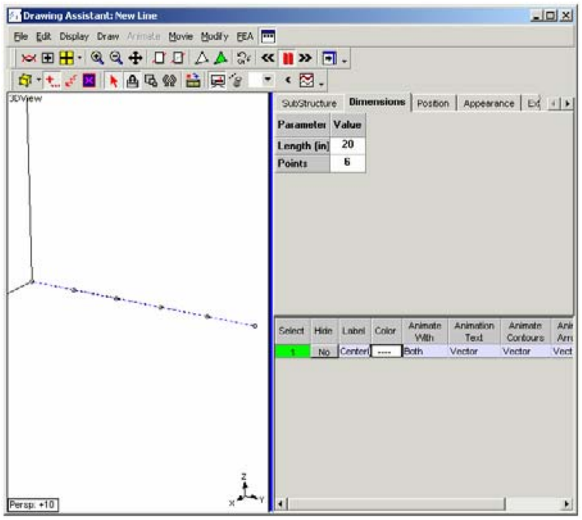
Choosing a Line object in the Drawing Assistant

- On the **Substructures** tab, double click on the **Line** object.

A **Line** Substructure will be added to the Structure window and the remaining **Drawing Assistant** tabs will become active.

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

  Length (in) = 20
  Points = 6



Parameter	Value
Length (in)	20
Points	6

Select	Hide	Label	Color	Animate With	Animation Text	Animate Contours	Ani Arr
1	No	Centerl	----	Both	Vector	Vector	Vect

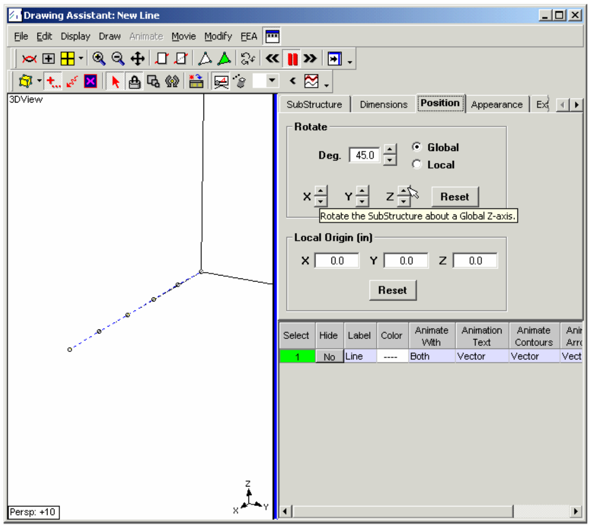
Dimensioning the Line object.

The Line will be redrawn with six points spanning 20 inches along the **Y** axis. However, we want the axis of the beam to be along the **X** axis.

On the **Position** tab:

- Verify that **45 Degrees** and **Global** are selected under **Rotate**.

- Press the **Z** ⇈ button *twice*, to rotate the line by 90° about the Global **Z** axis.

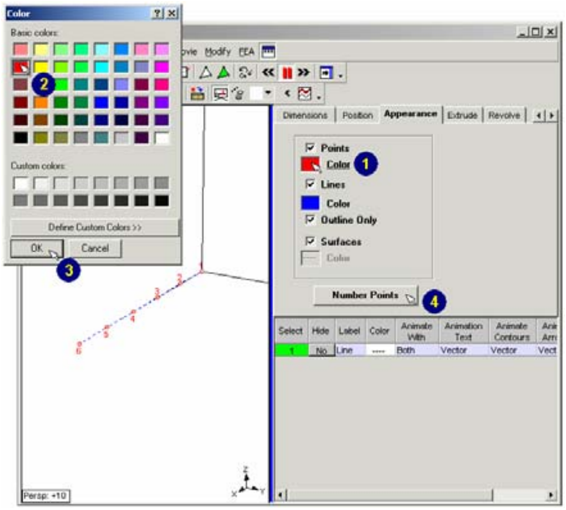


Deg. 45.0
● Global
○ Local
Rotate the SubStructure about a Global Z-axis.
X 0.0 Y 0.0 Z 0.0

Select	Hide	Label	Color	Animate With	Animation Text	Animate Contours	Ani Arr
1	No	Line	----	Both	Vector	Vector	Vect

Centerline along the Global X axis.

Next, the six points will be labeled with **Point Numbers** so that calculated mode Shape components (M#s) can be attached to them. We also want to color these points in red, so that they will stand out clearly.



Setting the Appearance of the centerline.

On the **Appearance** tab:

1. Press the **Number Points** button; the six points are numbered, as shown above.
2. Double-click on the Points **Color**.
3. Click on **red** in the **Colors** dialog that opens.
4. Click **OK** to change the Points **Color** to red.

This completes the definition of the beam centerline.

ADDING CROSS-SECTION ORIENTATION POINT

Before we attach the **FE Bar** elements between the beam centerline **Points**, we must add one more point to the model. This point is called an *orientation point* and it is needed define the "up axis" of the *cross section* of each **FE Bar** element.

Each **FE Bar** element requires an *orientation point* that *lies in the same plane as, but not in line with its end-points*.

Since our beam is straight, a single *orientation point* will serve all five **FE Bar** elements. We will locate this point at **X=10**, **Y=5** and **Z=0** inches.

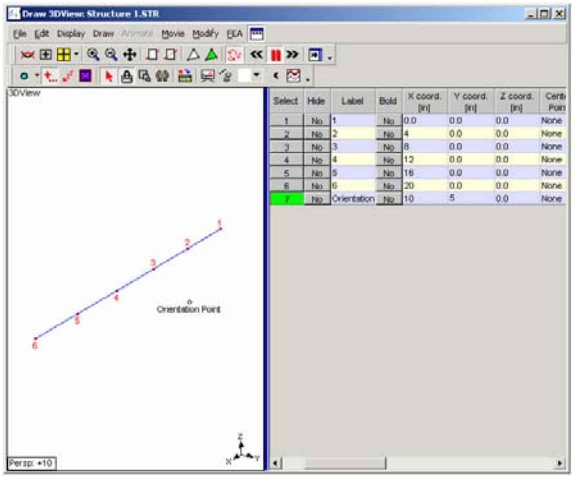
To add the *orientation point*:

Execute: **Edit | Object | Points**.

The **Point** properties spreadsheet will be shown.

Execute: **Edit | Add Object**.

- Click in the drawing area to the right of the centerline as shown below. This will add a seventh **Point** to the model and also to a row to the spreadsheet.
- Execute: **Edit | Add Object** to terminate the **Point Add** operation.



Select	Hide	Label	Bold	X coord. [in]	Y coord. [in]	Z coord. [in]	Cent Poin
1	No	1	No	0.0	0.0	0.0	None
2	No	2	No	4	0.0	0.0	None
3	No	3	No	8	0.0	0.0	None
4	No	4	No	12	0.0	0.0	None
5	No	5	No	16	0.0	0.0	None
6	No	6	No	20	0.0	0.0	None
7	No	Orientation	No	10	5	0.0	None

Adding an Orientation Point.

In the **Points** spreadsheet:

- Edit the coordinate entries for the new (row 7) Point to:

  X coord (in) = 10
  Y coord (in) = 5
  X coord (in) = 0

- Type **Orientation Point** in the **Label** field of the new point.

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

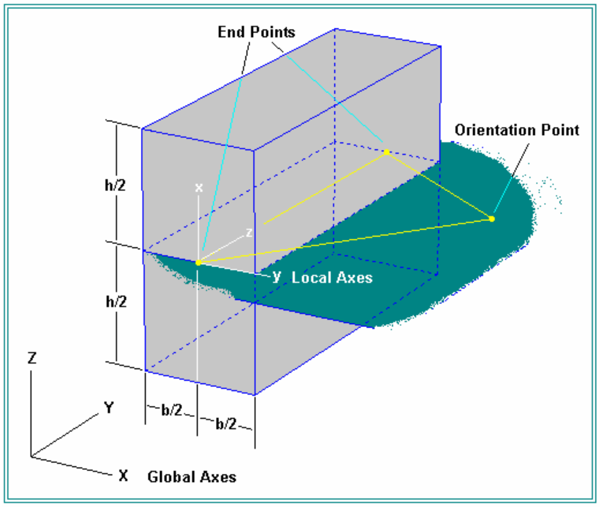
- Click on the **Hide** field for the *orientation point*, changing its entry to **Yes**.

This completes the definition of the *orientation point*.

REQUIRED BAR ELEMENT PROPERTIES

To define each **FE Bar** element, its *cross section properties* and *material properties* must be specified.

Bar Cross Section Properties



Cross-section of a rectangular Bar element.

The **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 drawing above. The cross section properties of a rectangular cross section are:

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

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

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

$$I_{xy} = \int xydA = \int_{\frac{-h}{2}}^{h/2} x \left(\int_{\frac{-b}{2}}^{b/2} ydy \right) dx = 0 \tag{4}$$

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

All of the **FE Bar** elements have the same width (**b = 1** inch) and height (**h = 1** inch) dimensions. Therefore:

Area = (1) x (1) = **1 in^2**
I_{xx} = (1/12) x (1)^3 x (1) = **0.0833 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**

Bar Material Properties

Three material properties are required to describe an **FE Bar**. These are the *Modulus of Elasticity* (Young's Modulus), *Poisson's Ratio* and its *density*. We will assume that the beam is made from 6061-T6 aluminum, which has the following physical properties:

Modulus of Elasticity = 9.9x10^6 lbf/in^2
Poissons Ratio = 0.33
Density = 0.098 lbm/in^3

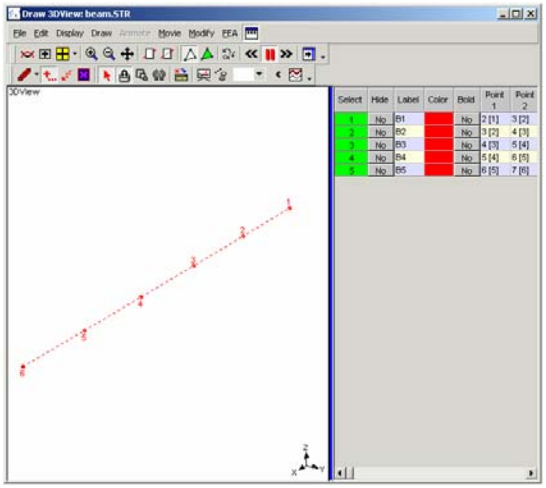
ADDING THE BAR ELEMENTS

We will add five **FE Bar** elements between the six center-line **Points**, and then add the required properties to the **FE Bar spreadsheet**.

Execute: **Edit | Object | FE Bars**.

Execute: **Edit | Add Object** to eneable the addition of elements.

- Click near Point **1**, and then near Point **2** to add the first **FE Bar** element between these two Points.
- Click near point **2**, and then near Point **3** to add the second **FE Bar** element. Keep clicking down the center-line to add all 5 elements, as shown in the following figure.

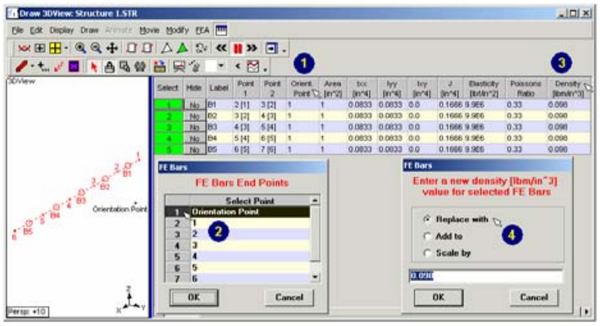


Select	Hide	Label	Color	Bold	Point 1	Point 2
1	No	B1	🟥	No	2 [1]	3 [2]
2	No	B2	🟥	No	3 [2]	4 [3]
3	No	B3	🟥	No	4 [3]	5 [4]
4	No	B4	🟥	No	5 [4]	6 [5]
5	No	B5	🟥	No	6 [5]	7 [6]

Adding FE Bar elements to model.

After completing all five entries:

- Click on the depressed **Add Object** button to terminate the **Add** operation.



Select	Hide	Label	Point 1	Point 2	Orient Point	Area [in^2]	Ixx [in^4]	Iyy [in^4]	Ixy [in^4]	J [in^4]	Elasticity [lbf/in^2]	Poissons Ratio	Density [lbm/in^3]
1	No	B1	2 [1]	3 [2]	1	1	0.0833	0.0833	0.0	0.1666	9.9E6	0.33	0.098
2	No	B2	3 [2]	4 [3]	1	1	0.0833	0.0833	0.0	0.1666	9.9E6	0.33	0.098
3	No	B3	4 [3]	5 [4]	1	1	0.0833	0.0833	0.0	0.1666	9.9E6	0.33	0.098
4	No	B4	5 [4]	6 [5]	1	1	0.0833	0.0833	0.0	0.1666	9.9E6	0.33	0.098
5	No	B5	6 [5]	7 [6]	1	1	0.0833	0.0833	0.0	0.1666	9.9E6	0.33	0.098

FE Bars End Points

	Select Point
1	Orientation Point
2	1
3	2
4	3
5	4
6	5
7	6

Enter a new density [lbm/in^3] value for selected FE Bars

● Replace with
○ Add to
○ Scale by

0.098

Making FE Bar Spreadsheet entries.

Entering Bar Element Properties

- Drag the vertical blue bar to the left to expose the **FE Bar** elements spreadsheet.

In the **FE Bars spreadsheet**, take the following actions to give all five bars the same properties:

1. Double-click on the **Orientation** column heading. A dialog box will open.
2. Select the **orientation point** (row 7) in the list, and click on **OK**.
3. Double-click on the **Area** column heading. A dialog box will open.
4. Select **Replace with**, type the numerical entry (**1 in^2**) and press **OK**.

- Repeat steps (3) and (4) for the **I_{xx}**, **I_{yy}**, **I_{xy}**, **J**, **Elasticity**, **Poissons Ratio** and **Density** columns. Use the following numerical values:

I_{xx} = 0.0833 in^4
I_{yy} = 0.0833 in^4
I_{xy} = 0.0 in^4
J = 0.1666 in^4
Modulus of Elasticity = 9.9x10^6 lbf/in^2
Poissons Ratio = 0.33
Density = 0.098 lbm/in^3

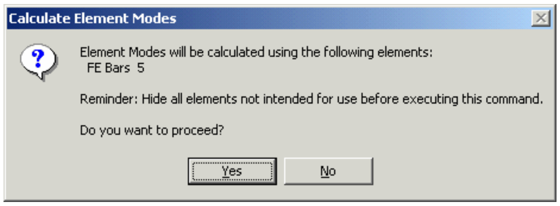
The **FE Bar** elements are now fully defined for the beam model.

CALCULATING THE FREE-FREE MODES

Now that the beam elements have been added to the centerline model, and their properties entering into their spreadsheet, the free-free modes of the beam can be calculated.

Execute: **Modify | Calculate Element Modes**.

A **Calculate Element Modes** dialog will open.



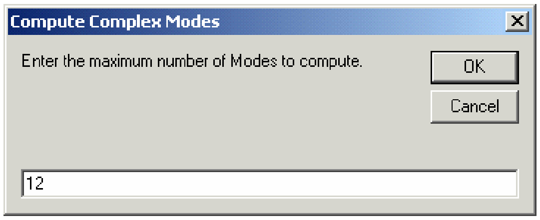
Element Modes will be calculated using the following elements:
FE Bars 5

Reminder: Hide all elements not intended for use before executing this command.

Do you want to proceed?

- Verify that 5 FE Bars will be used in the calculation and press **OK**.

A **Compute Complex Modes** dialog will open.

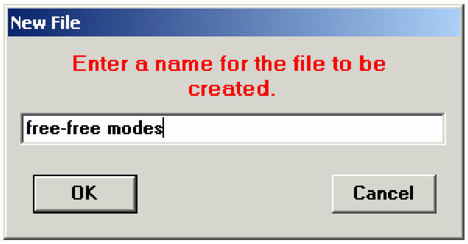


Enter the maximum number of Modes to compute.

12

- Enter **12** as the maximum number of modes to calculate and press **OK.**

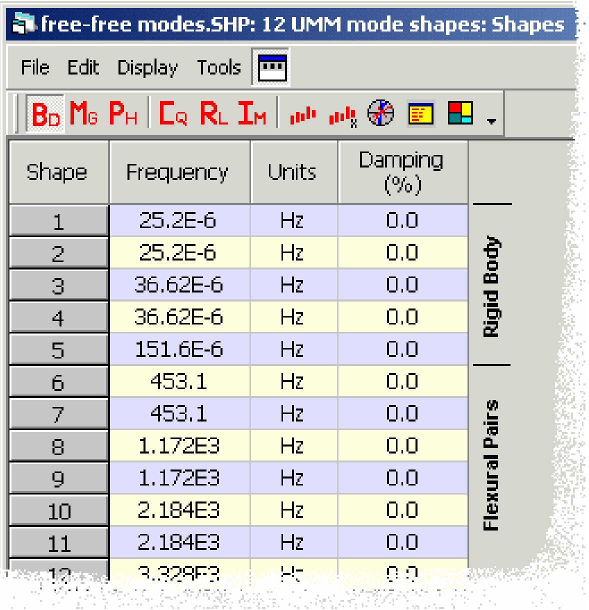
After the modes have been calculated, a **New File** dialog will open.



Enter a name for the file to be created.

free-free modes

- Enter **free-free modes**" as the Shape Table name, and click on **OK**. The Shape Table will open listing the modal frequencies.



Shape	Frequency	Units	Damping (%)	
1	25.2E-6	Hz	0.0	Rigid Body
2	25.2E-6	Hz	0.0	
3	36.62E-6	Hz	0.0	
4	36.62E-6	Hz	0.0	
5	151.6E-6	Hz	0.0	
6	453.1	Hz	0.0	Flexural Pairs
7	453.1	Hz	0.0	
8	1.172E3	Hz	0.0	
9	1.172E3	Hz	0.0	
10	2.184E3	Hz	0.0	
11	2.184E3	Hz	0.0	
12	3.329E3	Hz	0.0	

Frequencies of the free-free modes.

Notice that the first 5 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 that all of the modes have **0 %**, damping since the *FE Bar* elements *contain no damping*.

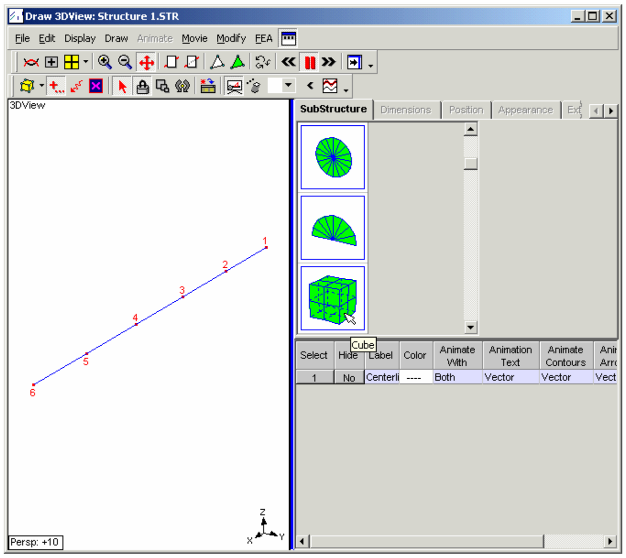
The shapes in the **free-free modes .SHP** file are ready for animated display. Save them for subsequent use.

- Execute: **File | Save**

- Close the **free-free modes** window.

BUILDING A 3D BEAM MODEL

To improve the realism of the Structure, we will add another Substructure to the model to turn the beam into a 3D model.



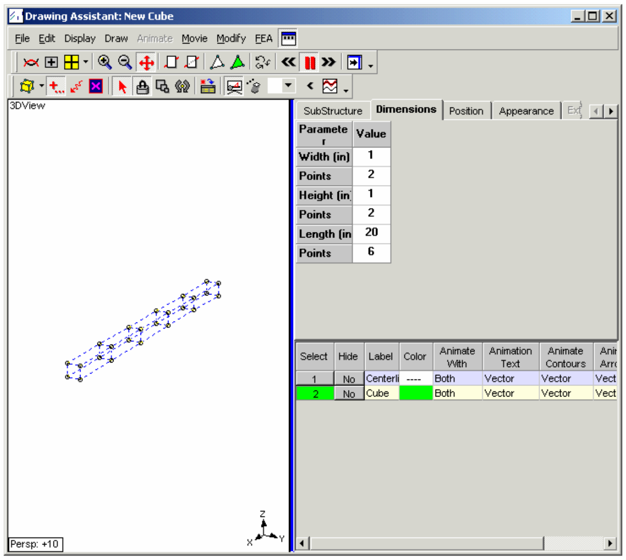
Select	Hide	Label	Color	Animate With	Animation Text	Animate Contours	Anir Arr
1	No	Centerli	----	Both	Vector	Vector	Vect

Cube

Selecting a Cube substructure in the Drawing Assistant.

- Execute: **Draw | Drawing Assistant**.

- In the list on the **Substructure** tab, double-click on the **Cube** substructure.



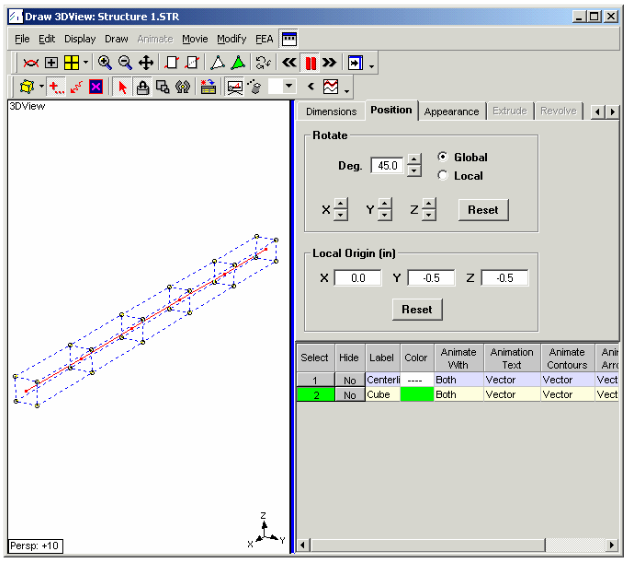
Parameter	Value
Width (in)	1
Points	2
Height (in	1
Points	2
Length (in	20
Points	6

Select	Hide	Label	Color	Animate With	Animation Text	Animate Contours	Anir Arr
1	No	Centerli	----	Both	Vector	Vector	Vect
2	No	Cube	🟩	Both	Vector	Vector	Vect

Dimensioning the Cube substructure.

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

Width (in) = 1
Points = 2
Height (in) = 1
Points = 2
Length = 20
Points = 6



Deg. 45.0 ● Global ○ Local

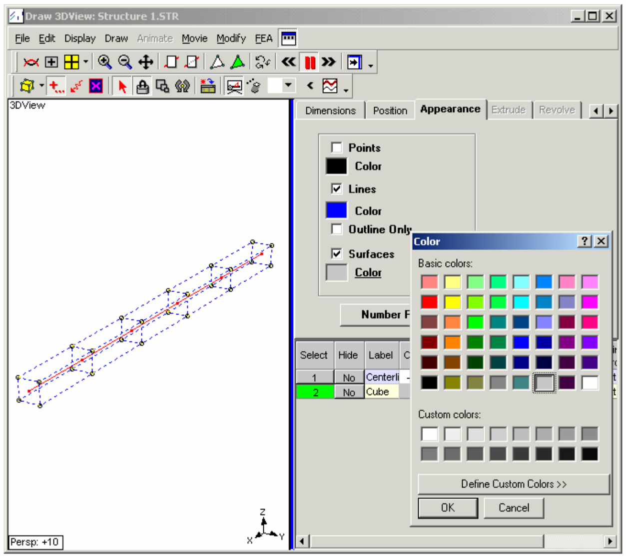
Local Origin (in): X 0.0 Y -0.5 Z -0.5

Select	Hide	Label	Color	Animate With	Animation Text	Animate Contours	Ani Arr
1	No	Centerli	----	Both	Vector	Vector	Vect
2	No	Cube	🟩	Both	Vector	Vector	Vect

Positioning the Local Origin of the Cube substructure.

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

X = 0.0
Y = -0.5
Z = -0.5



☐ Points
Color
☑ Lines
Color
☐ Outline Only
☑ Surfaces
Color

Select	Hide	Label
1	No	Centerli
2	No	Cube

Defining the Appearance of the Cube substructure.

On the **Appearance** tab:

- Double-click on Surface **Color**.
- In the Colors dialog that opens, select **gray** and press **OK**.

Execute: **Draw | Drawing Assistant** to close the Drawing Assistant.

Execute: **File | Save As**

Save the Structure file as **free-free beam.STR**.

COPYING THE STRUCTURE

Next, we will make a copy of the **free-free beam** structure and then modify the copy to make a cantilever beam model.

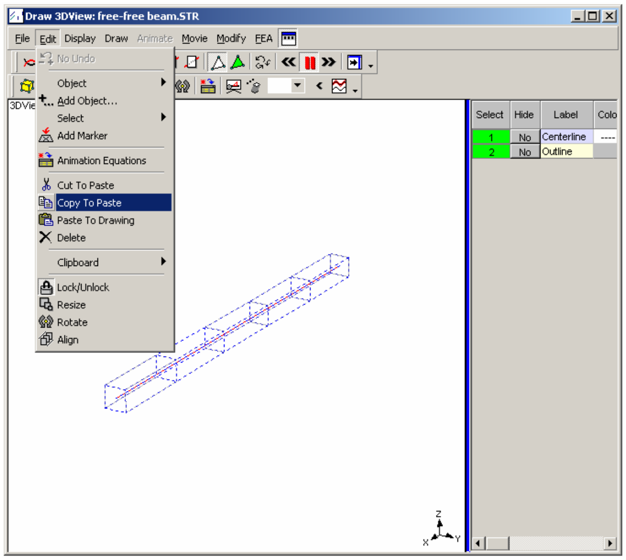
In the Structure window:

Execute: **Edit | Object | Substructure**.

Execute: **Display | Spreadsheet** to view the Substructure spreadsheet.

- **Select** both Substructures in the spreadsheet.

Execute: **Edit | Copy to Paste**.



Select	Hide	Label	Colo
1	No	Centerline	----
2	No	Outline	

Copying the free-free beam Substructures.

Close the **free-free beam.STR** window.

Execute: **File | New | Structure**.

A new (empty) Structure window will open.

In the new Structure window:

- Execute: **Edit | Paste to Drawing**.

A copy of the free-free beam structure will appear in the window. To make a cantilever beam model, we will add a vertical *ground plane* to the fixed end of the beam:

Execute: **Draw | Drawing Assistant**.

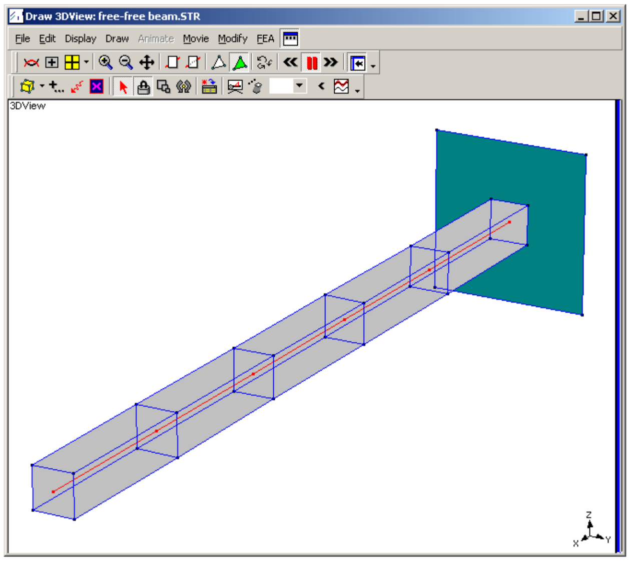
- On the **Substructure** tab, double-click on the **Plate** Substructure in the list of Substructures.
- On the **Dimensions** tab enter:

  Width (in) = **4**
  Points = **2**
  Height (in) = **4**
  Points = **2**

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

  X = **0**
  Y = **-2**
  Z = **-2**

On the Appearance tab, click on Surfaces **Color** and select **blue-green**.



Cantilever beam structure.

Execute: **File | Save As**.

Save the new Structure as **cantilever beam.STR**.

CALCULATING THE CANTILEVER MODES

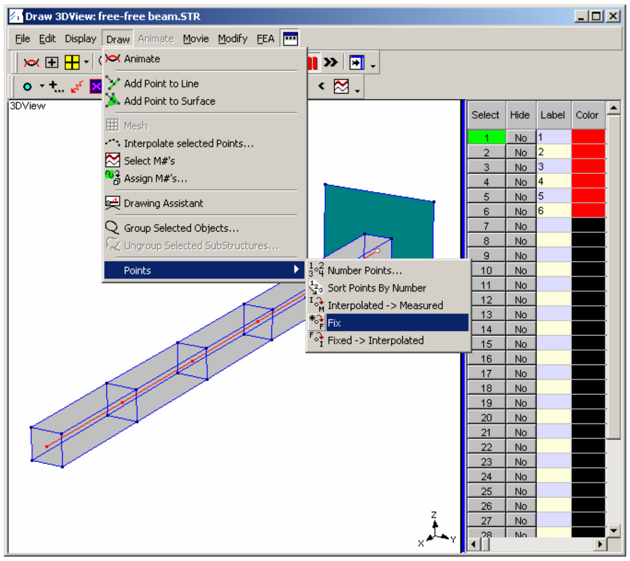
We will now calculate a set of modes for the cantilever beam. To change the boundary conditions at the fixed end, we will fix **Point 1** so that it cannot translate or rotate. This is equivalent to *clamping* the beam to a rigid foundation at **Point 1**.

To fix **Point 1** so that it does not move:

Execute: **Edit | Object | Points**.

Execute: **Display | Spreadsheet** to make the Points Spreadsheet visible.

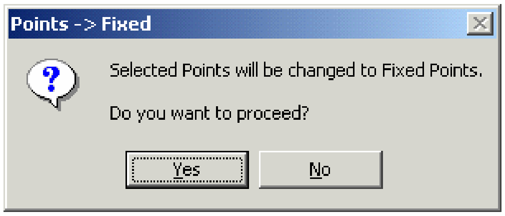
- Select the Point with the **Label 1** by pressing its **Select** button. The selected **Point 1** will be circled.



Select	Hide	Label	Color
1	No	1	🟥
2	No	2	🟥
3	No	3	🟥
4	No	4	🟥
5	No	5	🟥
6	No	6	🟥
7	No		⬛
8	No		⬛
9	No		⬛
10	No		⬛
11	No		⬛
12	No		⬛
13	No		⬛
14	No		⬛
15	No		⬛
16	No		⬛
17	No		⬛
18	No		⬛
19	No		⬛
20	No		⬛
21	No		⬛
22	No		⬛
23	No		⬛
24	No		⬛
25	No		⬛
26	No		⬛
27	No		⬛
28	No		⬛

Fixing Point #1 as the cantilever root.

Execute: **Draw | Points | Fixed** as shown above. A **Points** ⇒ **Fixed** dialog will open.



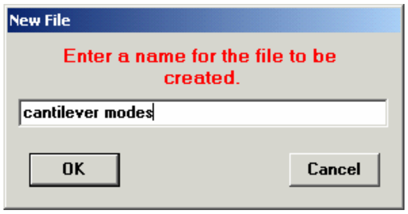
Selected Points will be changed to Fixed Points.

Do you want to proceed?

- Press **Yes** to proceed.

Point 1 is now fixed in all directions.

Repeat the actions described in the **CALCULATING THE FREE-FREE MODES** section to calculate the cantilever modes. Name the resulting new Shape file **cantilever modes**.

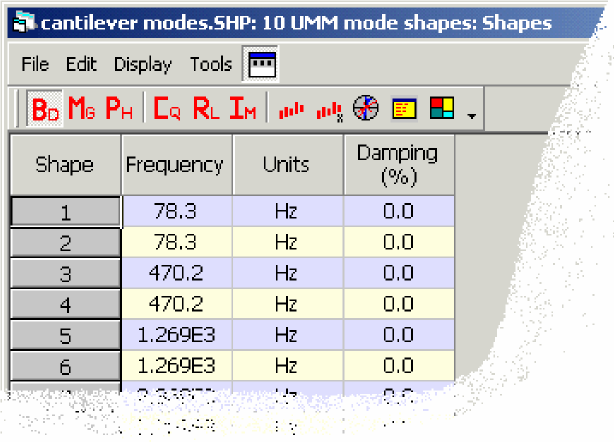


Enter a name for the file to be created.

cantilever modes

The **cantilever modes.SHP** window will open.

Execute: **File | Save** in the Shape Table window to save the file to disk.



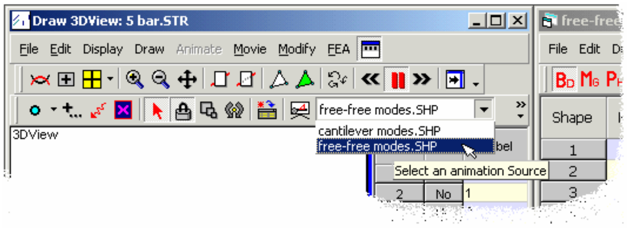
cantilever modes.SHP: 10 UMM mode shapes: Shapes

Shape	Frequency	Units	Damping (%)
1	78.3	Hz	0.0
2	78.3	Hz	0.0
3	470.2	Hz	0.0
4	470.2	Hz	0.0
5	1.269E3	Hz	0.0
6	1.269E3	Hz	0.0

Frequencies of the cantilever modes.

Notice that there are no *rigid body modes* for the cantilever beam. Fixing one end to ground changed the free-free beam into a *constrained structure*. The lowest-frequency cantilever mode (**78 Hz**) is a *flexural* mode.

ANIMATING THE MODE SHAPES



free-free modes.SHP

cantilever modes.SHP

free-free modes.SHP

Select an animation Source

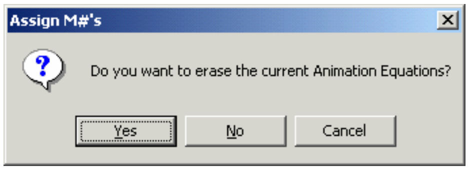
Selecting the Animation Source.

To display the modes of the cantilever beam in animation:

Execute: **Window | Arrange | For Animation**.

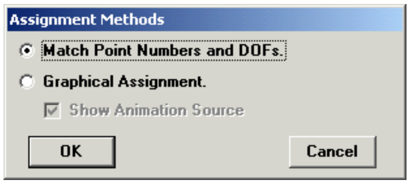
- Select **cantilever modes.SHP** in the **Animation Source** list on the Structure window toolbar.

Execute: **Draw | Assign M#'s** in the Structure window. An **Assign M#'s** dialog will open.



Do you want to erase the current Animation Equations?

- Press **Yes** to continue. An **Assignment Methods** dialog will open.



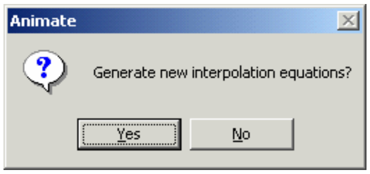
● Match Point Numbers and DOFs.
○ Graphical Assignment.
☑ Show Animation Source

- Verify that **Match Point Numbers and DOFs** is selected and press **OK**. An **Assign M#'s** confirmation dialog will open.

Number of M#'s assigned: 18

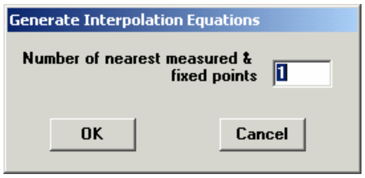
Note: *Always* execute the **Assign M#'s** command (either in the Structure or Shape Table window) before animating a new Shape file.

Execute: **Draw | Animate**. An **Animate** dialog will open.



Generate new interpolation equations?

- Press the **Yes** button to replace the existing *interpolation equations* with new ones. A **Generate Interpolation Equations** dialog will open.

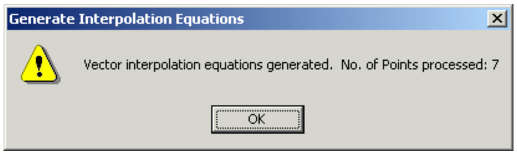


Number of nearest measured & fixed points: 1

- Enter a **1** into the dialog box and press **OK**

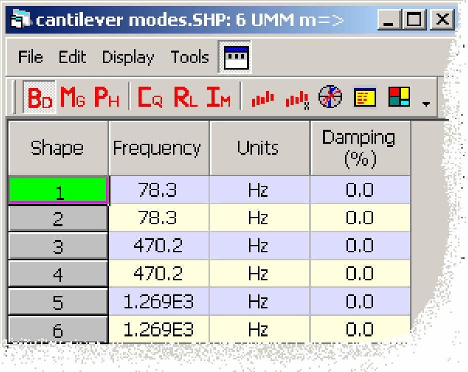
Entering **1** in the above dialog box insures that all Points in each cross section will be interpolated with the motion of the Point at the center.

A **Generate Interpolation Equations** confirmation dialog will open.



Vector interpolation equations generated. No. of Points processed: 7

- Press **OK** to display one of the modes in animation.



Shape	Frequency	Units	Damping (%)
1	78.3	Hz	0.0
2	78.3	Hz	0.0
3	470.2	Hz	0.0
4	470.2	Hz	0.0
5	1.269E3	Hz	0.0
6	1.269E3	Hz	0.0

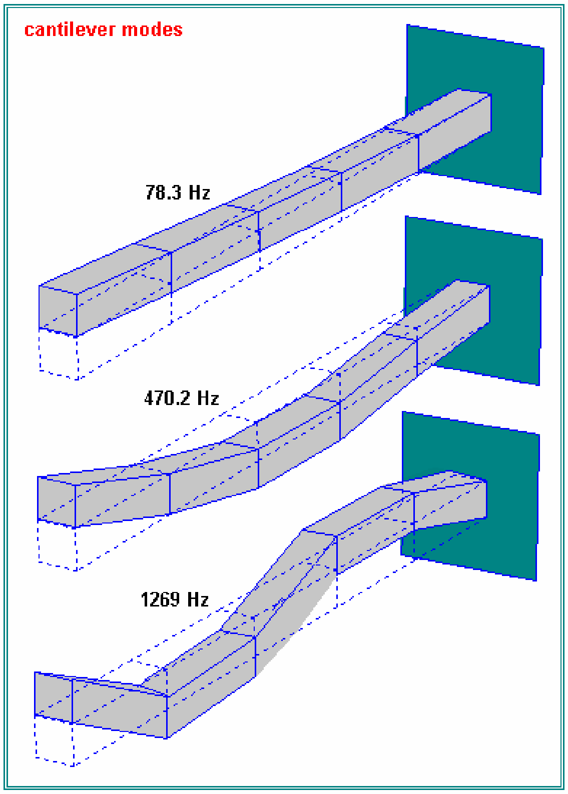
- Select a mode shape to be displayed by pressing its **Shape** button in the Shape Table window.

To show the model with hidden lines, surfaces filled and the undeformed structure outline:

Execute: **Display | Surfaces | Filled**.

Execute: **Display | Hidden Lines | Invisible**.

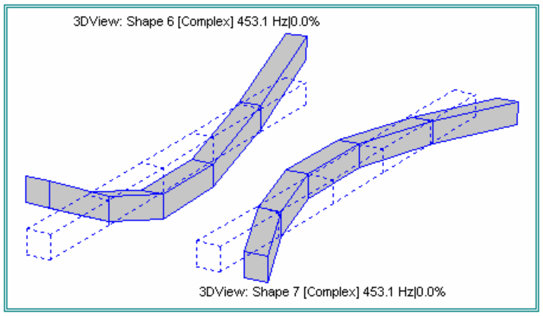
Execute: **Animate | Deformations | Undeformed**



First three cantilever mode shapes.

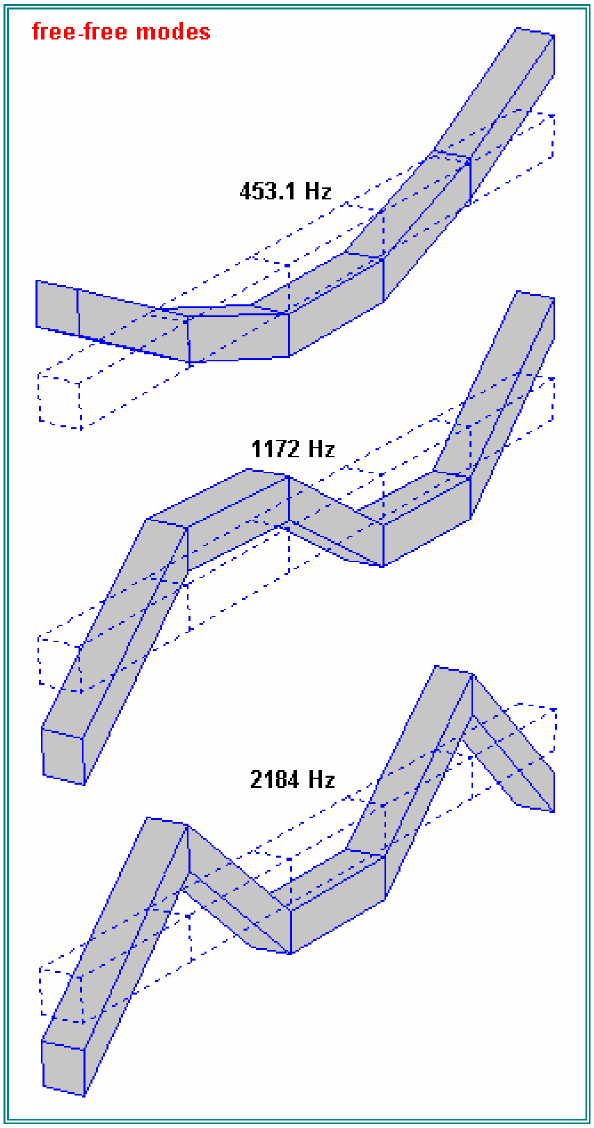
To display the modes of the free-free beam:

- Close the **cantilever beam.STR** and **cantilever modes.SHP** windows.
- Open **free-free beam.STR** and **free-free modes.SHP**.
- Repeat the steps above used to display the cantilever modes.



Free-free repeated root mode shapes at 453 Hz.

Review all of the modes for both beam configurations. You will observe that both beams have multiple pairs of *repeated roots*. For any mode with bending in the **X-Z** plane, there is an identically shaped mode with bending in the **X-Y** plane *at the same frequency*.



First three free-free mode shapes.

COMPARING THE FEM MODAL FREQUENCIES WITH ANALTICAL RESULTS

The reference textbook *Formulas for Natural Frequency and Mode Shape*, Robert D. Blevins, 1979, page 108, contains formulas for the modal frequencies of a continuous beam. 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_i, in Hz.

$\mathbf{L}$ = length of the beam (20 inches)

$\boldsymbol{\lambda_i}$ = 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**)

$\mathbf{E}$ = modulus of elasticity (9.9xE6)

$\mathbf{I}$ = cross sectional inertia (0.0833)

$\mathbf{m}$ = mass per unit length = (density)(area) = (0.098) / (386.4 lbf-sec^2/in/lbm)

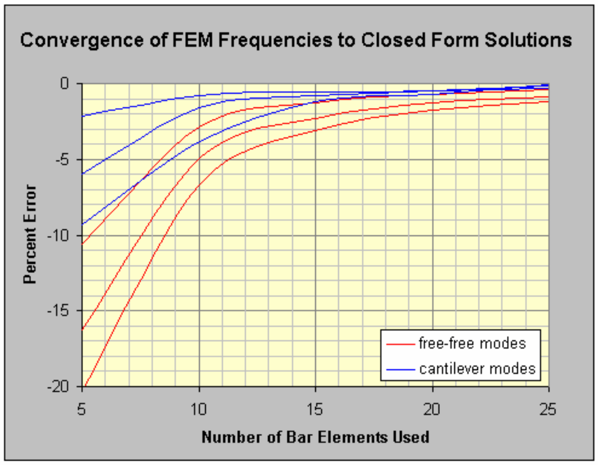
Equation (6) was used to calculate the frequencies of the first three modes of both free-free and cantilever beams. These are compared with the **ME'scopeVES** results below.

Mode	ME'scopeVES	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

ME'scopeVES versus analytical frequencies (Hz)

CONCLUSIONS

Notice that the **ME'scopeVES** modal frequencies are lower, in all cases, than the analytical modal frequencies. This is because the analytical formula is for a *continuous* beam, whereas the **ME'ScopeVES** beams were *approximated* with only **5 FE Bar** elements. The following figure shows how rapidly the first three finite element modal frequencies converge toward the analytic answers when more **FE Bar** elements are used.



FEM modal frequency errors versus number of elements.