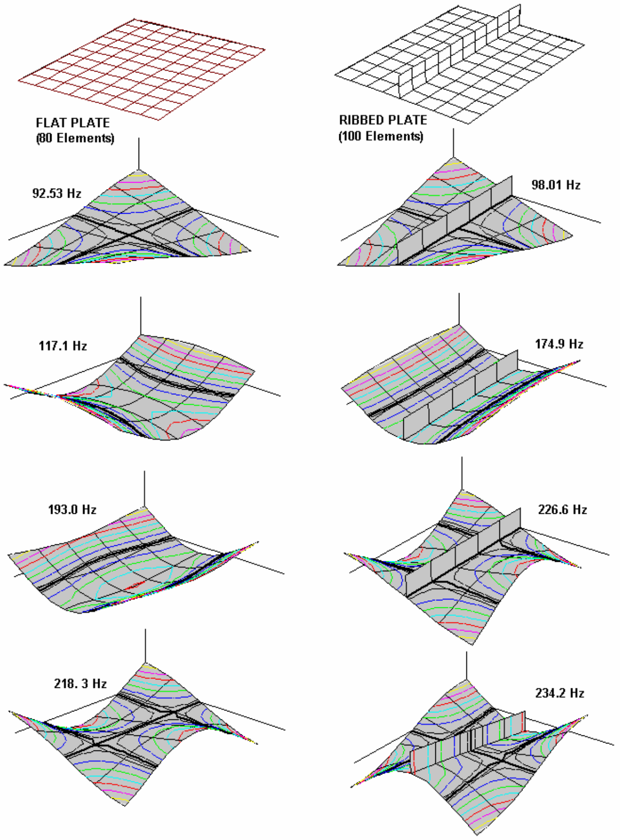
Vibrant Technology, Inc.

ME’scopeVES Application Note #3

Modeling a Rib Stiffener With Structural Modifications

INTRODUCTION

Following instructions in this note, you will use the *Visual SDM* option of **ME’scopeVES** to model the addition of a rib stiffener to a flat aluminum plate. This study will use the mode shapes of the flat plate (alone) produced by finite element analysis (FEA) as a starting point. Both *bar* (beam) and *quadrilateral plate* elements will be used in the modification as alternate ways to model the rib addition. Then you will compare your Structural Dynamics Modification (SDM) results against those of a finite element model (FEM) of the ribbed plate using the Modal Assurance Criteria (MAC).



Comparison of flat and ribbed plate NASTRAN models.

As shown above, the modeled test article is a 3/8-inch thick 20 by 25 inch rectangular plate constructed of 6061-T6511 aluminum. The stiffening rib (25 by 3 by 3/8 inch) is of the same material and runs down the center of the plate’s long dimension. Note that the rib substantially affects those modes involving *bending* along this axis.

NASTRAN® PLATE AND RIB MODELS

The flat plate was modeled in *NASTRAN for Windows*® using quadrilateral plate elements (Quads). Each Quad was defined between nodes 2.5 inches apart. This formed a grid of 99 nodes bounding 80 Quad elements. The elements were given the following material properties:

Modulus of Elasticity = **10^7** lb/in^2
Poisson’s Ratio = **0.33**
Density = **0.101** lb/in^3

The rib was modeled as 2 rows of Quad elements, each 1.5 inches high by 2.5 inches wide. This formed a grid of 33 nodes bounding 20 elements. The bottom 11 rib nodes are also those of the plate’s centerline.

Analytical Data

Two sets of analytical mode shapes were generated using *NASTRAN for Windows*; one set for the plate without the rib, and one set for the plate with the rib attached to its centerline. The modal data and the structure model were imported into **ME’scopeVES** using the FEMAP® neutral format.

These NASTRAN modes were edited to make them more like typical experimental results. The six *rigid body* (zero frequency) modes were eliminated. All *rotational* degrees-of-freedom (**DOFs**) were deleted. Finally, all **X** and **Y** direction **DOFs** were eliminated, leaving only *flexural* modes described solely by vertical (**Z** direction) motion.

These edited NASTRAN modes are contained in Project file **Plate & Rib.PRJ** within the **ME’scopeVES Examples** folder:

> The modes of the *unmodified* structure (plate alone) are contained in the file named **Aluminum Plate.SHP**.
>
> The modes of the *modified* structure (plate with rib) are contained in the file named **Aluminum Plate with rib.SHP**.

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

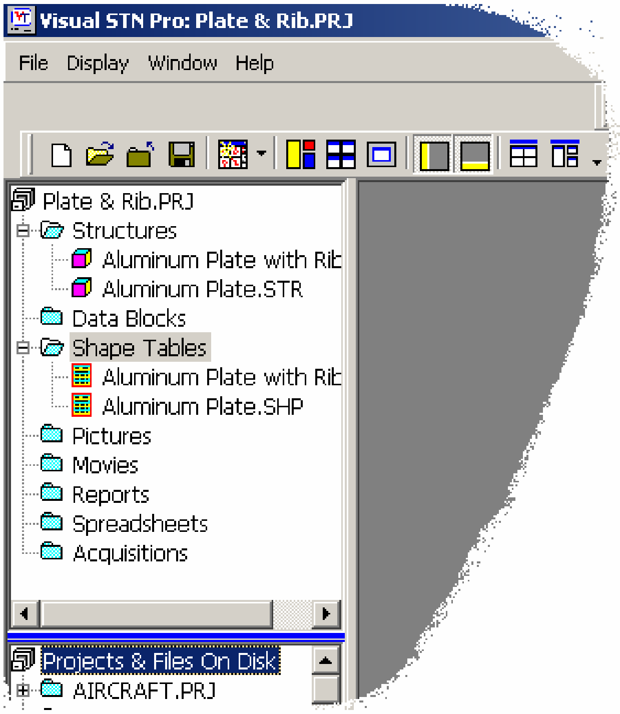
MODES OF THE UNMODIFIED STRUCTURE

To model the rib stiffener, the modes of the *unmodified* structure (plate without the rib) are needed. They are contained in the **Plate & Rib** Project. To open the **Plate & Rib** Project:

Execute: **File | Project | Open**

- Select: **Plate & Rib.PRJ** from the **ME'scopeVES Examples** folder.
- Click **OK**.

The **Project File** will open, providing a **Project Panel** as shown below.

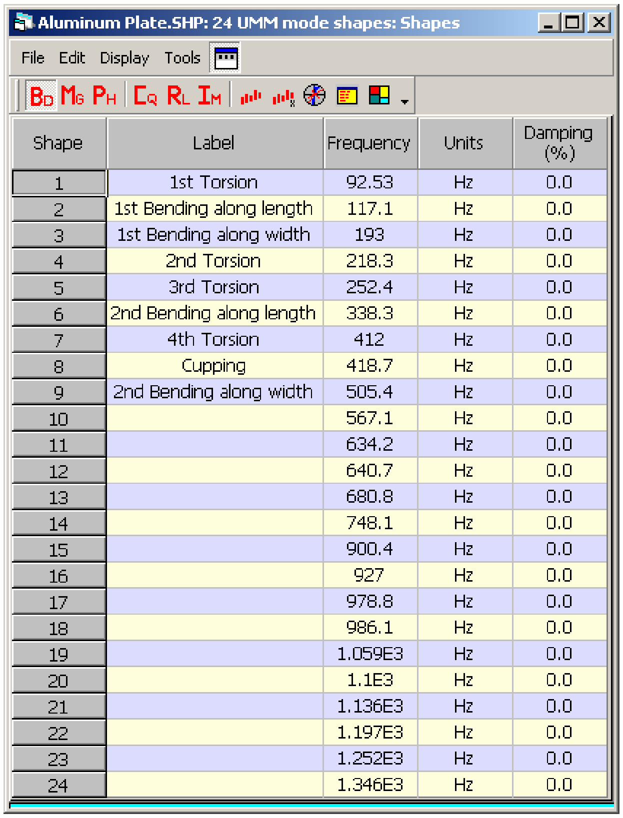


Project Panel.

Note from the upper pane of the **Project Panel** that the project contains two **Structures** and two **Shape Tables**. To open the shape table of the *unmodified* plate:

- Double-click on **Aluminum Plate.SHP** in the **Project Panel**.

The shape table will open, disclosing a list of the natural frequencies and damping factors as shown above right. Note that all of the **Damping (%)** values are **0.0**; this is typical of FEA results *which rarely include model damping*.



Shape	Label	Frequency	Units	Damping (%)
1	1st Torsion	92.53	Hz	0.0
2	1st Bending along length	117.1	Hz	0.0
3	1st Bending along width	193	Hz	0.0
4	2nd Torsion	218.3	Hz	0.0
5	3rd Torsion	252.4	Hz	0.0
6	2nd Bending along length	338.3	Hz	0.0
7	4th Torsion	412	Hz	0.0
8	Cupping	418.7	Hz	0.0
9	2nd Bending along width	505.4	Hz	0.0
10		567.1	Hz	0.0
11		634.2	Hz	0.0
12		640.7	Hz	0.0
13		680.8	Hz	0.0
14		748.1	Hz	0.0
15		900.4	Hz	0.0
16		927	Hz	0.0
17		978.8	Hz	0.0
18		986.1	Hz	0.0
19		1.059E3	Hz	0.0
20		1.1E3	Hz	0.0
21		1.136E3	Hz	0.0
22		1.197E3	Hz	0.0
23		1.252E3	Hz	0.0
24		1.346E3	Hz	0.0

Edited NASTRAN modes of the Plate Without Rib.

To view these mode shapes in animation:

- Double-click on **Aluminum Plate.STR** under **Structures** in the *navigation pane*.

The **Aluminum Plate.STR** window will open. This contains a graphic view of the plate structure and a spreadsheet (which may be hidden).

Select: **Edit | Object | FE Quads** from the object list on the **Aluminum Plate.STR** window toolbar.

If **Quad** elements representing the rib are visible in the model, hide them as follows:

Execute: **Display | Spreadsheet** to view the **FE Quad** spreadsheet.

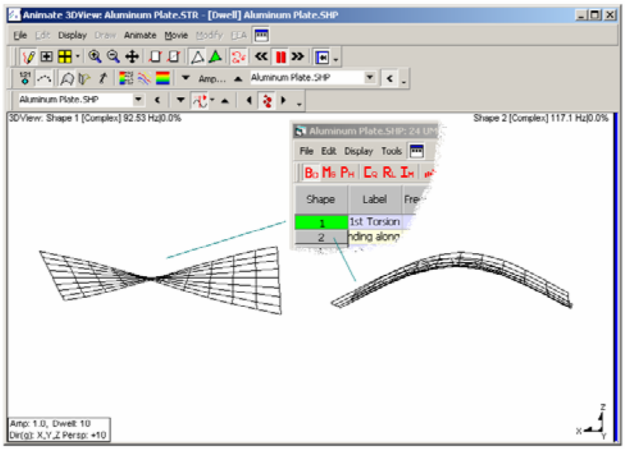
- Double-click on the **Hide** column in the **FE Quads** spreadsheet. A dialog box will open.
- Select **Yes** in the dialog box and click **OK**.

Execute: **Display | Spreadsheet** to hide the spreadsheet.

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

Execute: **Animate the Structure**.

The first mode, a **92.53 Hz** twisting or torsion mode, will be displayed. This is shown at left below and at the upper-left in the lead-in figure.



First two modes of the unmodified plate.

To animate other modes:

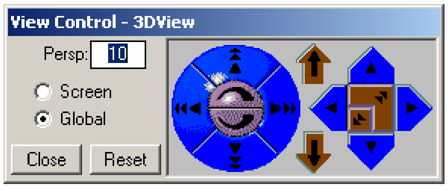
- Click on the desired **Shape** in the **Aluminum Plate.SHP** window (see inset, above).

Step through all of the modes and note the general character of their deformation shapes. Those modes that have a *node line* along the lengthwise center axis of the plate (such as **Shape 1**) will not be affected by a stiffening rib placed over it. Modes that exhibit bending along the length of the plate (such as **Shape 2**, right above), will be changed significantly by the stiffening rib.

To fully appreciate the shape of a given mode, you may want to modify the viewpoint, magnification or perspective.

Execute: **Display | View Control**.

The **View Control – 3DView** dialog will open.



Persp: 10

○ Screen

● Global

View Control – 3DView panel.

Controls in the *round* group cause rotations of the structure about an axis (either a **Global X**, **Y** or **Z** axis or the axies of the **Screen**). Those buttons in the *square* group control translations and magnification. The central *arrows* control perspective correction.

It is also informative to “walk around the structure” as it vibrates. The following controls facilitate this:

Execute: **Display | Spin 3D View | Rotate CW** causes the structure to spin clock-wise about the **Z** axis.

Execute: **Display | Spin 3D View | Rotate CCW** causes the structure to spin counter clock-wise.

Execute: **Display | Spin 3D View | Rotate Stop** ceases the spin.

MODELING THE RIB WITH BAR ELEMENTS

A **Bar** element is simply a beam of *fixed cross-section*. It attaches to the model at its two end-points. At each end-point the Bar imposes stiffness and inertial constraints to all six degrees-of-freedom of the unmodified structure, if they are defined. (In this case, only **Z**-direction translations are defined by the mode shapes.) Each **Bar** is described by its end-point locations, material properties, and cross-section properties.

Checking the Units

Before performing any structural modifications, it is important to make sure that the **Structure** window units match the units of the **Shape Table**. To check this:

Execute: **File | Options** in the **Aluminum Plate.STR** window and select the **Units** tab.

- Verify:
  Mass Units = Lbs
  Force Units = Lbf
  Length Units = In

- Click **OK**.

Adding the Bar Elements to the Model

To model the addition of the rib using beam elements, **FE Bar** elements will be added between all of the points down the centerline of the plate.

In the **Aluminum Plate.STR** window:

Execute: Draw the Structure.

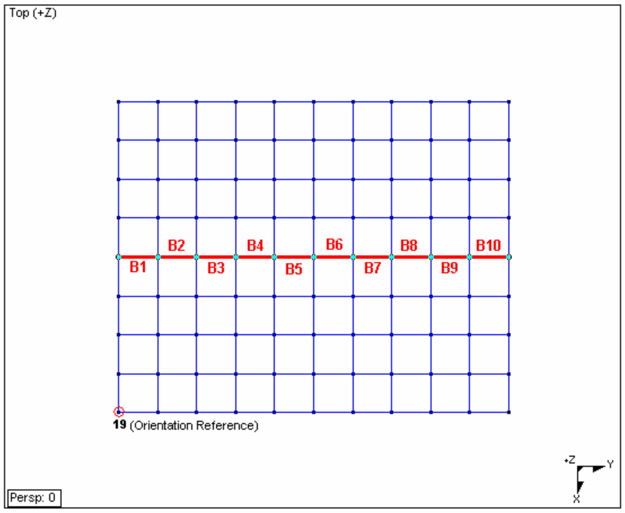
Execute: Edit | Object | FE Bars.

NOTE: If your model already has **FE Bar** elements attached to its centerline, skip the next step.

Select: **Edit | Add Object**.

Beginning at one end of the centerline, click on each **Point** pair down the centerline, to add **Bar** elements between them as shown below.

Deselect: Edit | Add Object.



Top view of plate showing 10 added Bar elements.

Bar Material Properties

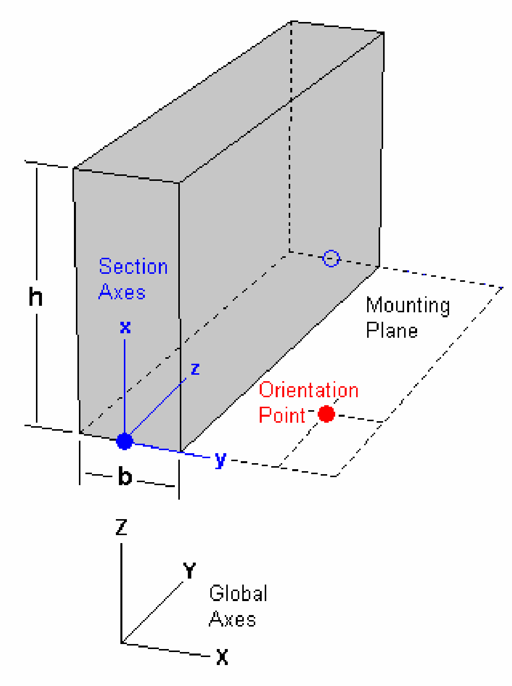
Now, the material properties of the Bar elements must be entered into their respective spreadsheet cells.

Execute: **Display | Spreadsheet**.

- Double click on the **Select** column header to select all of the elements.
- Double click on each respective column header named below, and enter the following physical properties into the opened dialog box.

Elasticity = 10^7 lb/in^2
Poissons Ratio = 0.33
Density = 0.101 lb/in^3

Bar Cross-Section Properties



Cross-section of a rectangular Bar element.

The **Bar** cross-section is described by its **Area** and four *area moments* calculated with respect to the attachment point at the bottom center of the section. The area moments (**I_{xx}**, **I_{yy}**, **I_{xy}** and **J**) are computed with respect to the local *Section Axes* show. For a *rectangular* cross-section, these are:

$$Area = \int dA = b\int_0^h dx = h\int_{-b/2}^{b/2} dy = bh \qquad (1)$$

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

$$I_{yy} = \int x^2 dA = b\int_0^h x^2 dx = \frac{bh^3}{3} \qquad (3)$$

$$I_{xy} = \int xy dA = \int_0^h x\left(\int_{\frac{-b}{2}}^{b/2} y dy\right) dx = 0 \qquad (4)$$

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

All of our **Bar** elements have the same **b** (**3/8** inch) and **h** (**3** inches) dimensions. Therefore:

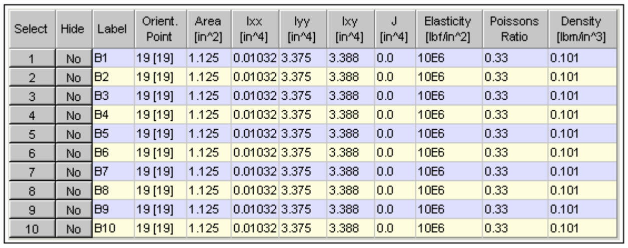
Area = (3/8) x (3) = **1.125 in^2**

I$_{xx}$ = (1/12) x (3/8)^3 x (3) = **0.01318 in^4**

I$_{yy}$ = (1/3) x (3/8) x (3)^3 = **3.375 in^4**

I$_{xy}$ = 0.0 in^4

J = 0.01318 + 3.375 = **3.388 in^4**



Select	Hide	Label	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	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
2	No	B2	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
3	No	B3	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
4	No	B4	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
5	No	B5	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
6	No	B6	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
7	No	B7	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
8	No	B8	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
9	No	B9	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101
10	No	B10	19 [19]	1.125	0.01032	3.375	3.388	0.0	10E6	0.33	0.101

Bar properties entries.

To enter the calculated cross-section properties into the **Aluminum Plate.STR** spreadsheet, select all of the **Bar** elements and:

- Double click on the **Area** column header, enter **1.125** into the opened dialog box, and click on **OK**.
- Double click on the **I$_{xx}$** column header, enter **0.01318** into the opened dialog box, and click on **OK**.
- Double click on the **I$_{yy}$** column header, enter **3.375** into the opened dialog box, and click on **OK**.
- Double click on the **I$_{xy}$** column header, enter **0.0** into the opened dialog box, and click on **OK**.
- Double click on the **J** column header, enter **3.388** into the opened dialog box, and click on **OK**.

Bar Orientation

Note from the figure on directly preceding page 4 that the cross-section properties are computed with respect to a set of local *section axes*, not the *global axes* that describe Point locations. To complete any cross-section specification, we must describe the orientation of these section axes relative to the global axes.

This is accomplished by identifying a single previously defined **Point** termed the **Orientation Point**.

The **Orientation Point** is *any* **Point** in the *mounting plane* of the **Bar** that is *not* in line with both attachment points. The local *section* **y** axis lies in the plane defined by the **Orientation Point** and the two bar-end attachment points.

In our example case, any of 88 points could serve as the **Orientation Point**. Only 44 of these would orient the bars on *top* of the plate as desired. We will orient the **y**-axis of *all* 10 **Bar** elements using corner Point **19**. This point is circled in the Bar-position figure on page 4.

- Make sure that all of the elements are selected.
- Double click on the **Orientation** column heading
- Select **Point 19** from the drop down list, and click **OK**.

MAKING THE MODIFICATION

Now, we are ready to compute the effect of the **Bar** element stiffener on the modes of the plate structure.

Execute: **Display | Elements**.

NOTE: If any **FE elements** other than the **10 Bar** elements have been added to the structure model, use the **Hide** column in their respective spreadsheets to hide them so they won't be used during the modification.

Execute: **Display | Spreadsheet**.

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

A dialog box will open.

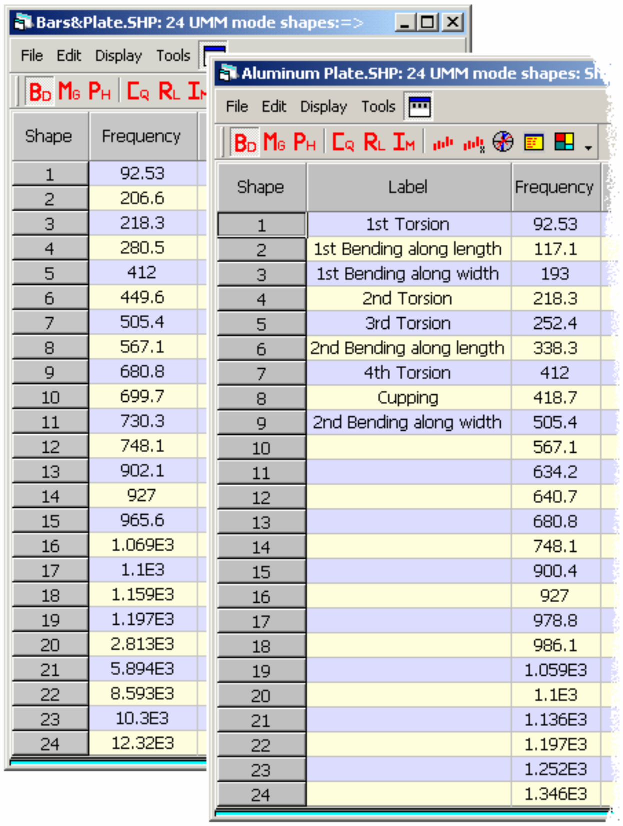
- Verify that new modes will be calculated using only **10 FE Bars**, and click **Yes**.
- Select the **Aluminum Plate.SHP** file with the mode shapes of the unmodified structure, and click on **OK**.

Structural Modifications uses the modes of the unmodified structure plus the **FE Bar** elements to create a set of equations of motion for the modified structure. These equations are then solved for the new modes of the structure.

After the modification has been completed, a dialog box will open asking you to enter the name of the **Shape Table** file for the new mode shapes.

- Enter the name **Bars&Plate** and click on **OK**.

The new **Bars&Plate.SHP** window will open, listing the frequencies of the new modes. Compare these with the original modes of **Aluminum Plate.SHP** as shown below.



Shape	Frequency
1	92.53
2	206.6
3	218.3
4	280.5
5	412
6	449.6
7	505.4
8	567.1
9	680.8
10	699.7
11	730.3
12	748.1
13	902.1
14	927
15	965.6
16	1.069E3
17	1.1E3
18	1.159E3
19	1.197E3
20	2.813E3
21	5.894E3
22	8.593E3
23	10.3E3
24	12.32E3

Shape	Label	Frequency
1	1st Torsion	92.53
2	1st Bending along length	117.1
3	1st Bending along width	193
4	2nd Torsion	218.3
5	3rd Torsion	252.4
6	2nd Bending along length	338.3
7	4th Torsion	412
8	Cupping	418.7
9	2nd Bending along width	505.4
10		567.1
11		634.2
12		640.7
13		680.8
14		748.1
15		900.4
16		927
17		978.8
18		986.1
19		1.059E3
20		1.1E3
21		1.136E3
22		1.197E3
23		1.252E3
24		1.346E3

New modes with Bar stiffener compared to plate alone.

Notice that some natural frequencies in the bar-stiffened list match exactly match frequencies in the original (unmodified) list. The first four *torsion* modes and the 2^{nd} *bending along width* shapes are among these. These are modes that were not changed by the lengthwise stiffener.

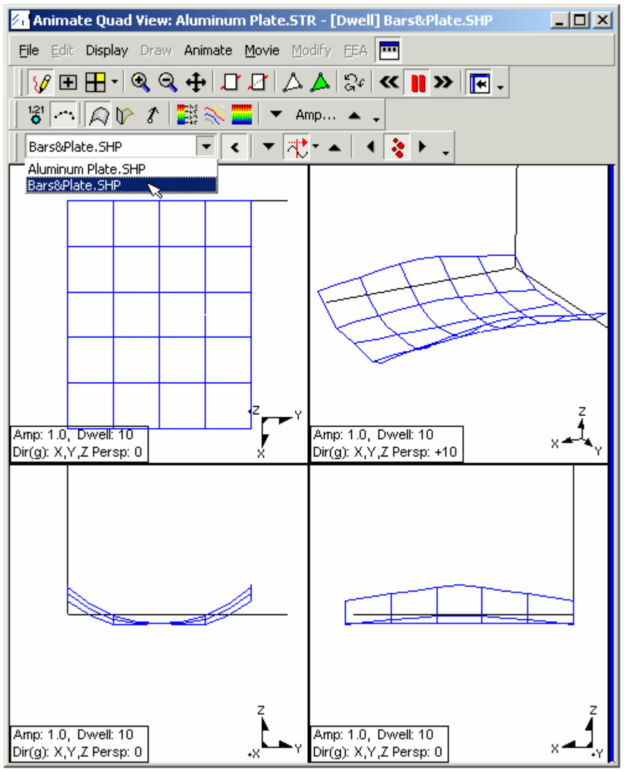
In contrast, other modes (such as *first bending along length*) were replaced by new natural frequencies. These are modes with shapes quite different from those of the unmodified plate and *the added rib caused the changes*.

Let's look at some of these new mode shapes.

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

Execute: **Animate the Structure**.

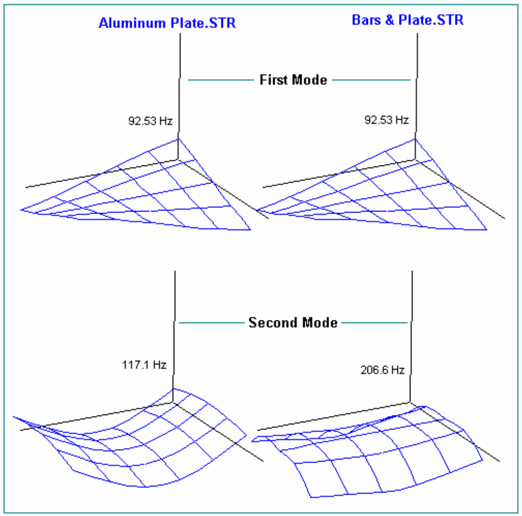
- Click on the **Animation Data Source** and select **Bars&Plate.SHP** as shown at right, above.



Selecting Bars&Plate modes for animation.

Step through these new mode **Shapes** and notice the reduced motion along the center-line of the plate in all modes.

- Double-click in the display area to obtain a four-view display as shown above.
- Double-click in any one of the four displays to make it the single animated display.

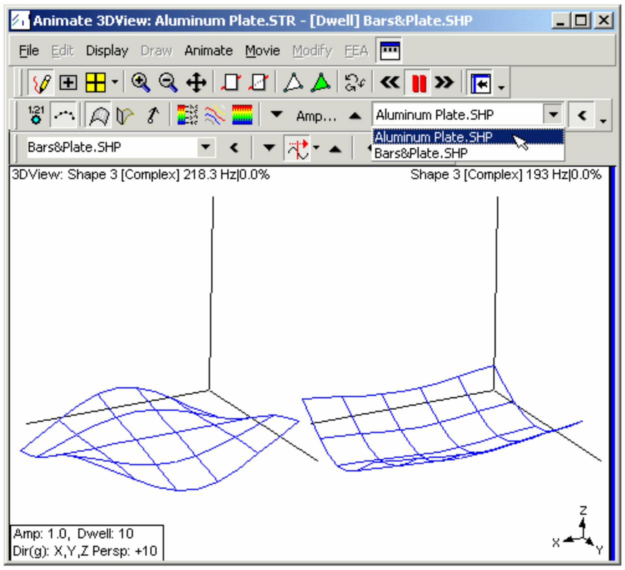


Shape comparisons of 1^{st} and 2^{nd} modes.

COMPARING MODE SHAPES GRAPHICALLY

Now lets compare the animated mode shapes before and after the modification.

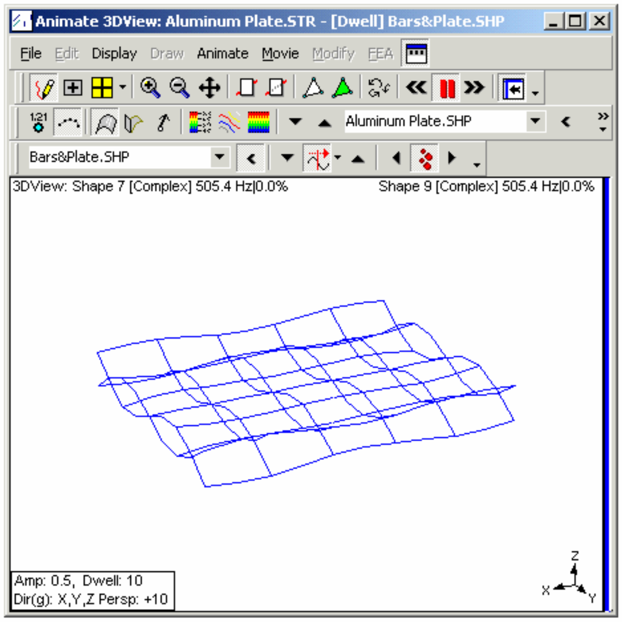
Execute: **Animate | Comparison | Side by Side**.



Side-by-side comparison of before and after modes.

You can make side-by-side comparisons, or overlay two modal animations.

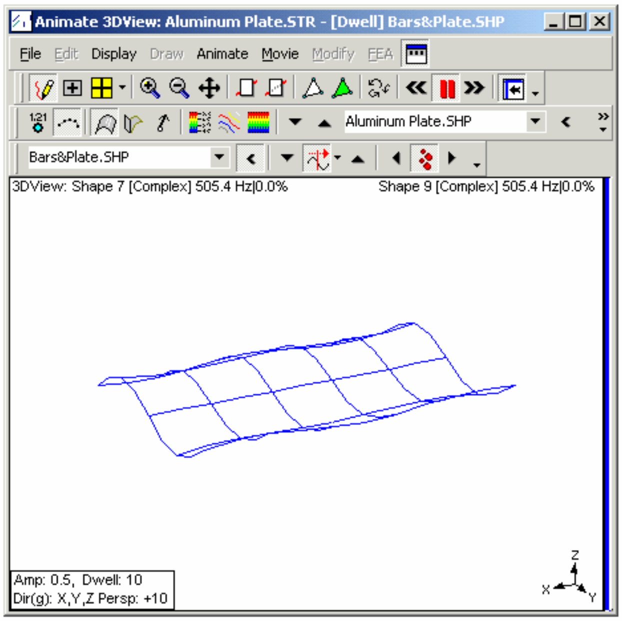
Execute: **Animate | Comparison | Overlay**.



Overlay comparison of two different modes.

Sometimes a comparison will become clearer if you invert the *phase* of one trace. For example, the last figure shows two very similar shapes at **505.4 Hz** from before stiffening and after the **Bar** addition.

Execute: **Animate | Comparison | Flip Phase**.



Previous comparison overlay with Phase Flipped.

It is now blatantly apparent that adding a stiffening rib does not change the shape of the mode with *2nd bending along width* shape!

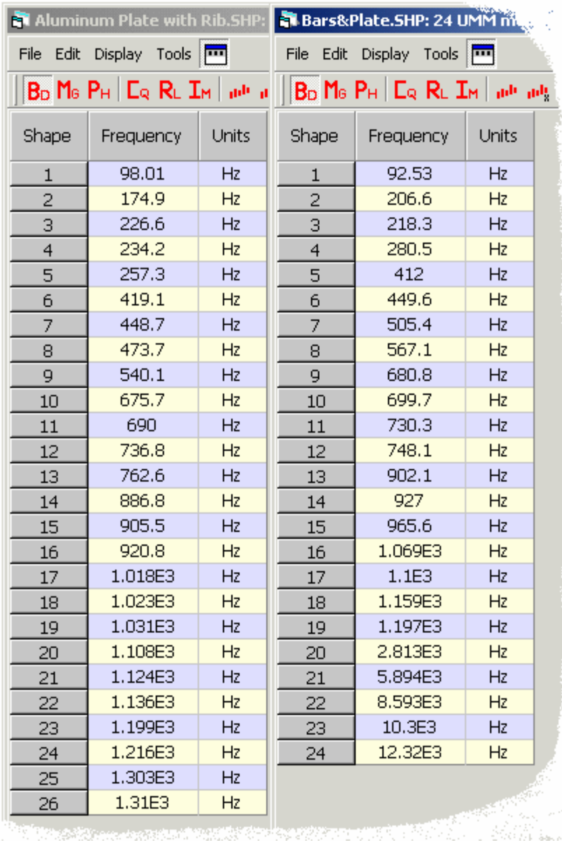
COMPARING SDM AND FEA RESULTS

Now let's compare our modified-structure results with those of the NASTRAN model of the plate with a stiffener rib.

- Close the **Aluminum Plate.SHP** window.
- Double-click on **Aluminum Plate with Rib.SHP** in the *navigation pane*.

Use all of the same comparison methods just described to view the similarities and differences between these two models. Note that modes 1, 2 and 3 of both models exhibit very similar shapes, but occur at different frequencies. The fourth modes are completely different in shape from one another.

In fact, our fourth SDM mode matches well in shape with the *fifth* NASTRAN mode. This "one mode offset" persists over higher frequency modes.



Shape	Frequency	Units
1	98.01	Hz
2	174.9	Hz
3	226.6	Hz
4	234.2	Hz
5	257.3	Hz
6	419.1	Hz
7	448.7	Hz
8	473.7	Hz
9	540.1	Hz
10	675.7	Hz
11	690	Hz
12	736.8	Hz
13	762.6	Hz
14	886.8	Hz
15	905.5	Hz
16	920.8	Hz
17	1.018E3	Hz
18	1.023E3	Hz
19	1.031E3	Hz
20	1.108E3	Hz
21	1.124E3	Hz
22	1.136E3	Hz
23	1.199E3	Hz
24	1.216E3	Hz
25	1.303E3	Hz
26	1.31E3	Hz

Shape	Frequency	Units
1	92.53	Hz
2	206.6	Hz
3	218.3	Hz
4	280.5	Hz
5	412	Hz
6	449.6	Hz
7	505.4	Hz
8	567.1	Hz
9	680.8	Hz
10	699.7	Hz
11	730.3	Hz
12	748.1	Hz
13	902.1	Hz
14	927	Hz
15	965.6	Hz
16	1.069E3	Hz
17	1.1E3	Hz
18	1.159E3	Hz
19	1.197E3	Hz
20	2.813E3	Hz
21	5.894E3	Hz
22	8.593E3	Hz
23	10.3E3	Hz
24	12.32E3	Hz

NASTRAN FEM (left) compared with Bar element SDM.

COMPARING SHAPES USING MAC

The Modal Assurance Criterion (MAC) is an analytical method for comparing two mode shapes. MAC values are always between 0.0 and 1.0.

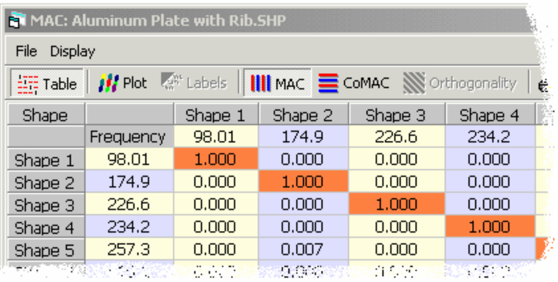
NOTE: A MAC value less than **0.9** means that two shapes are different from one another. A MAC value above **0.9** means that the shapes are very similar. A MAC value of **1.0** means that the two shapes are *identical* to one another.

It is clear from the **Shape Headers**, above, that the modes of **Aluminum Plate with Rib.SHP** and **Bars&Plate.SHP** don't share common frequencies. Nevertheless, MAC will tell us which **Bars** shape most closely matches each **Rib** shape.

- Close the **Aluminum Plate.STR** window.

Execute: **Display | MAC**.

A **MAC** window will open, presenting an Auto-MAC, a comparison between all of the NASTRAN modes contained in **Aluminum Plate with Rib.SHP.**



Shape		Shape 1	Shape 2	Shape 3	Shape 4
	Frequency	98.01	174.9	226.6	234.2
Shape 1	98.01	1.000	0.000	0.000	0.000
Shape 2	174.9	0.000	1.000	0.000	0.000
Shape 3	226.6	0.000	0.000	1.000	0.000
Shape 4	234.2	0.000	0.000	0.000	1.000
Shape 5	257.3	0.000	0.007	0.000	0.000

Portion of Auto-MAC matrix with diagonals highlighted.

The MAC values are presented in a matrix. Each row and each column represent a single mode **Shape**. The MAC value at the intersection of a row and column is measure of similarity between the row **Shape** and the column **Shape**.

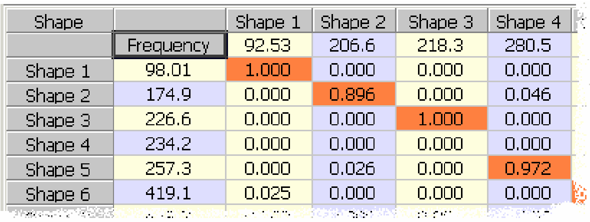
Note the **1.000** values running down the diagonal (highlighted in orange). These represent the comparisons of a mode with itself. The remaining (off-diagonal) elements describe the similarity of one mode to another. In an Auto-MAC, the matrix is always *square* and *symmetrical* because the row and column shapes are the same modes.

In a Cross-MAC, row and column mode shapes are different sets of data. Hence, the matrix need not be square or symmetric. More importantly, the maximum values *do not have to appear on the diagonal* and they *may be considerably less than* **1.000**.

Let's compare our Bar-modeled stiffener to the NASTRAN results.

Execute: **Display | Cross MAC** in the **Aluminum Plate with Rib.SHP** window.

- Select **Bars&Plate.SHP** in the **Shape Table Selection** dialog that opens and click **OK**.



Shape		Shape 1	Shape 2	Shape 3	Shape 4
	Frequency	92.53	206.6	218.3	280.5
Shape 1	98.01	1.000	0.000	0.000	0.000
Shape 2	174.9	0.000	0.896	0.000	0.046
Shape 3	226.6	0.000	0.000	1.000	0.000
Shape 4	234.2	0.000	0.000	0.000	0.000
Shape 5	257.3	0.000	0.026	0.000	0.972
Shape 6	419.1	0.025	0.000	0.000	0.000

MAC between NASTRAN with stiffener and SDM Bars.

The resulting cross-MAC display has one *row* for each mode in **Aluminum Plate with Rib.SHP** and one *column* for each mode in **Bars&Plate.SHP**. Note the maximum values on or near the diagonal. Most of these are greater than **0.9**. Mode **2** is lower (**0.896**) and *no match* is found for NASTRAN **Shape 4**.

MODELING THE RIB WITH QUAD ELEMENTS

The **Bars&Plate.SHP** modes are not fully satisfactory. A plausible reason for this is the simplicity of the rib representation used. The NASTRAN FEM modeled this area of the structure using 20 quadrilateral plates (480 **DOFs**); we used 10 **Bars** with a total of only 120 **DOFs**. Further, since only **Z** direction translations were used in the SDM, the active **DOFs** were reduced to 20.

Let's try a more resolute representation of the rib. Rather than simply increasing the number of **Bar** elements, we will use **10** quadrilateral plate elements. Since each of these has **6 DOFs** *at each corner*, so we will be doubling the number of constraint equations applied to the plate.

- Close the **Bars&Plate.SHP** and **Plate with Rib.SHP** windows.
- Open **Aluminum Plate.STR**.

Adding a Top Edge as a SubStructure

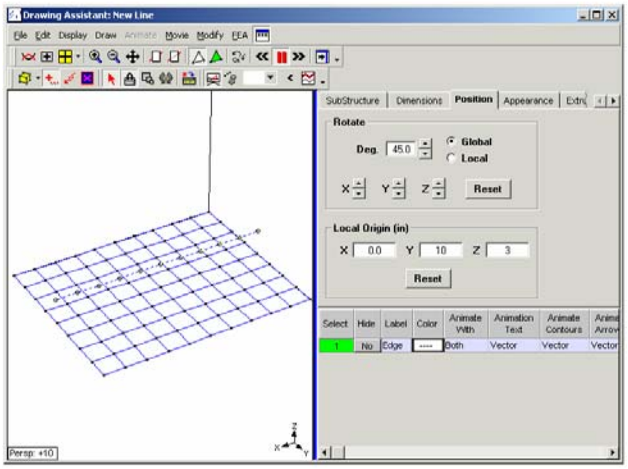
In order to add Quad elements, additional Points must first be added to the model to define the top edge of the rib. Then, the **Quad** elements can be added between the top of the rib and the plate centerline.

We will add a row of 11 Points above the centerline of the plate to model the top edge of the rib. The rib is 25 inches long and 3 inches high. The row of Points on the top edge will be added as a new **SubStructure**.

Execute: **Draw | Drawing Assistant**.

The **Drawing Assistant** window will open.

- On the **SubStructure** tab, double-click on **Line** and edit the **Label** to **edge**.
- On the **Dimensions** tab, enter **Length (in) = 25**, and **Points = 11**.
- On the **Position** tab, enter **Rotate Deg. = 45**, select **Global** axis rotation, and click on the **Z** up-arrow *twice* to align the new line in the **X** direction.
- Also on the **Position** tab a enter **Local Origin (in)** of **X = 0.0**, **Y = 10**, **Z = 3**.
- On the **Appearance** tab, select only **Points**, and click on **OK**.



Deg. 45.0 ● Global ○ Local

Local Origin (in) X 0.0 Y 10 Z 3

Select	Hide	Label	Color	Animate With	Animation Text	Animate Contours	Anim Arrow
1	No	Edge	----	Both	Vector	Vector	Vector

Using the Drawing Assistant to add the top-of-rib Points.

The model is now ready to add the **FE Quads**. Before adding them, the previously entered **Bar** elements must be hidden so they will not be used in the SDM.

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

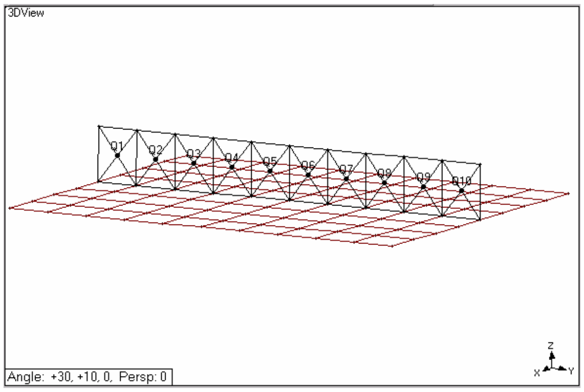
- In the **Bars** spreadsheet, double-click on the **Hide** column heading. A dialog box will open. Select **Yes** and then click on **OK** to hide the Bars.

Adding Quad Elements

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

Select: **Edit | Add Object**.

add the **Quad** elements by clicking on the four corners of each element, until all **10 Quads** have been added, as shown below.



Adding 10 Quad elements between edge points and plate.

Deselect: **Edit | Add Object**.

Quad Properties

Now, the properties of the **Quad** elements are entering into their respective spreadsheet cells.

- Make sure all of the elements are *selected*.
- Double-click on each respective column header listed below, and enter the following properties into the opened dialog box.

  Thickness = 0.375 in
  Elasticity = 10E6 lb/in^2
  Poissons Ratio = 0.33
  Density = 0.101 lb/in^3

Calculating the New Modes

The model is now ready to apply Structural Dynamic Modifications. To perform the modifications:

- Open the **Aluminum Plate.SHP** by double-clicking on it in the *navigation pane*.

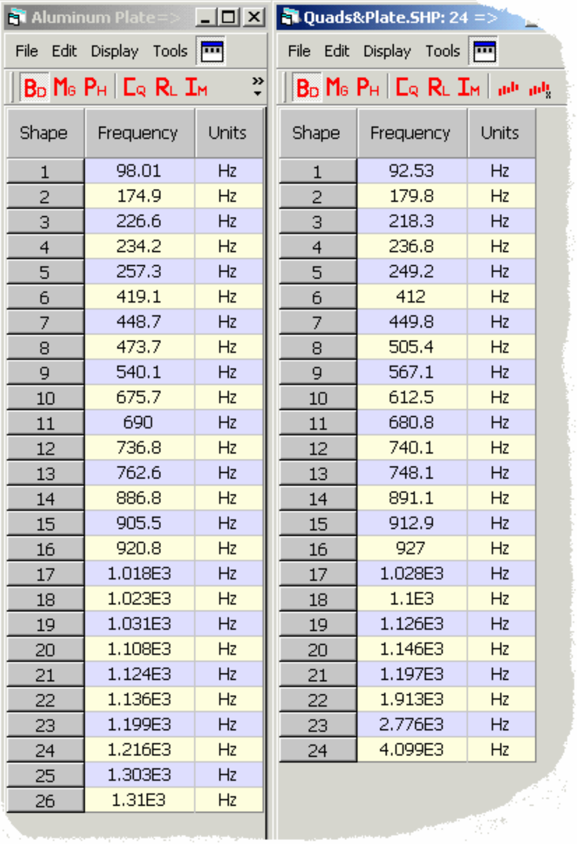
Execute: **Modify | Calculate New Modes**. A dialog box will open.

- Verify that new modes will be calculated using **10 FE Quads**, and click on **Yes**.
- Select the **Aluminum Plate.SHP** (the unmodified plate mode shapes), and click on **OK**.

After the new modes have been calculated, a dialog will open asking you to name the resulting **Shape** file.

- Enter the name **Quads& Plate** and click on **OK**.

The new **Quads&Plate.SHP** window will open, listing the new modal frequencies as shown below, with the NASTRAM FEM results as a reference.

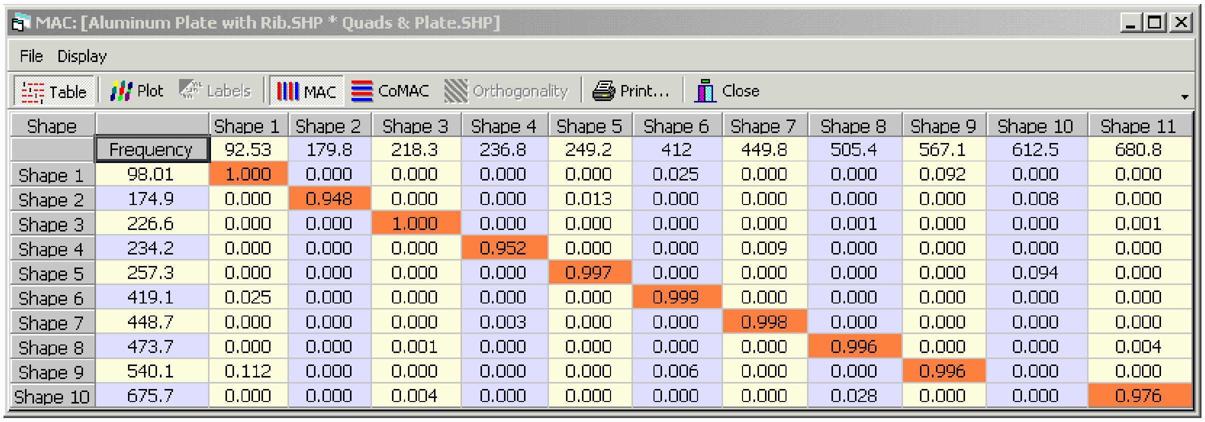


Shape	Frequency	Units
1	98.01	Hz
2	174.9	Hz
3	226.6	Hz
4	234.2	Hz
5	257.3	Hz
6	419.1	Hz
7	448.7	Hz
8	473.7	Hz
9	540.1	Hz
10	675.7	Hz
11	690	Hz
12	736.8	Hz
13	762.6	Hz
14	886.8	Hz
15	905.5	Hz
16	920.8	Hz
17	1.018E3	Hz
18	1.023E3	Hz
19	1.031E3	Hz
20	1.108E3	Hz
21	1.124E3	Hz
22	1.136E3	Hz
23	1.199E3	Hz
24	1.216E3	Hz
25	1.303E3	Hz
26	1.31E3	Hz

Shape	Frequency	Units
1	92.53	Hz
2	179.8	Hz
3	218.3	Hz
4	236.8	Hz
5	249.2	Hz
6	412	Hz
7	449.8	Hz
8	505.4	Hz
9	567.1	Hz
10	612.5	Hz
11	680.8	Hz
12	740.1	Hz
13	748.1	Hz
14	891.1	Hz
15	912.9	Hz
16	927	Hz
17	1.028E3	Hz
18	1.1E3	Hz
19	1.126E3	Hz
20	1.146E3	Hz
21	1.197E3	Hz
22	1.913E3	Hz
23	2.776E3	Hz
24	4.099E3	Hz

NASTRAN FEM (left) compared with Quad element SDM.

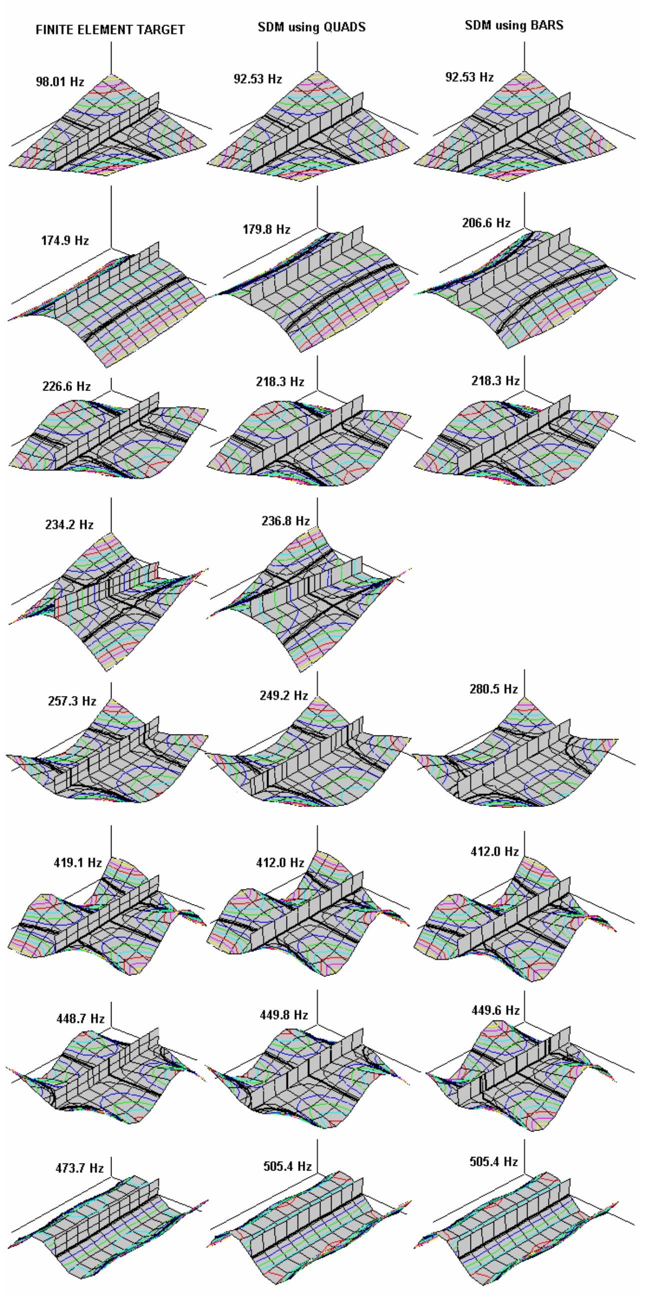
Note the close agreement in natural frequencies between the lists above. Note the first nine *diagonal* elements of the MAC matrix below; each is well over **0.9**. In all, *17 modes* meet this matching criterion. The problematic fourth mode, missing from the bar-based modification, is realistically captured here.



Shape		Shape 1	Shape 2	Shape 3	Shape 4	Shape 5	Shape 6	Shape 7	Shape 8	Shape 9	Shape 10	Shape 11
	Frequency	92.53	179.8	218.3	236.8	249.2	412	449.8	505.4	567.1	612.5	680.8
Shape 1	98.01	1.000	0.000	0.000	0.000	0.000	0.025	0.000	0.000	0.092	0.000	0.000
Shape 2	174.9	0.000	0.948	0.000	0.000	0.013	0.000	0.000	0.000	0.000	0.008	0.000
Shape 3	226.6	0.000	0.000	1.000	0.000	0.000	0.000	0.000	0.001	0.000	0.000	0.001
Shape 4	234.2	0.000	0.000	0.000	0.952	0.000	0.000	0.009	0.000	0.000	0.000	0.000
Shape 5	257.3	0.000	0.000	0.000	0.000	0.997	0.000	0.000	0.000	0.000	0.094	0.000
Shape 6	419.1	0.025	0.000	0.000	0.000	0.000	0.999	0.000	0.000	0.000	0.000	0.000
Shape 7	448.7	0.000	0.000	0.000	0.003	0.000	0.000	0.998	0.000	0.000	0.000	0.000
Shape 8	473.7	0.000	0.000	0.001	0.000	0.000	0.000	0.000	0.996	0.000	0.000	0.004
Shape 9	540.1	0.112	0.000	0.000	0.000	0.000	0.006	0.000	0.000	0.996	0.000	0.000
Shape 10	675.7	0.000	0.000	0.004	0.000	0.000	0.000	0.000	0.028	0.000	0.000	0.976

MAC between NASTRAN model with rib stiffener and SDM approximation using Quad elements.

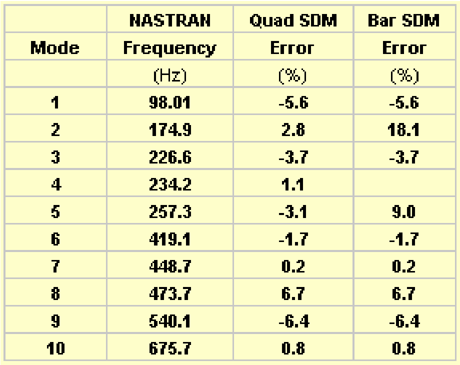
Use the various methods of graphical and analytical comparison previously discussed to investigate the similarity between these two analytic results. Reflect on the fact that a successful SDM result was found using a modal model *with only translation in the Z direction*. This is a subset, which is commonly measured in the testing laboratory today.



Graphic comparison of Quad and Bar SDM with FEM.

NASTRAN and NASTRAN for Windows are trademarks of MacNeal Schwendler Corp.

FEMAP is a trademark of Enterprise Software Products, Inc.



Mode	NASTRAN Frequency (Hz)	Quad SDM Error (%)	Bar SDM Error (%)
1	98.01	-5.6	-5.6
2	174.9	2.8	18.1
3	226.6	-3.7	-3.7
4	234.2	1.1	
5	257.3	-3.1	9.0
6	419.1	-1.7	-1.7
7	448.7	0.2	0.2
8	473.7	6.7	6.7
9	540.1	-6.4	-6.4
10	675.7	0.8	0.8

Frequency deviations of SDM models from FEM.

CONCLUSIONS

Bar and Quad elements in the **ME'scopeVES** *Visual SDM* option were used to model a rib stiffener attached to the centerline of an aluminum plate. These results were compared with FEA modes for the plate with rib.

The bar-implemented study provided only modest correlation. The more resolute quad-based modification provided very robust correlation over a 10-mode span.

The FEA modal data for the unmodified plate structure was "truncated" by deleting the rotational DOFs, and the translational DOFs in the X & Y directions. This was done to simulate experimental modal data, which is typically measured only in the surface-normal direction for plate-like structures. If these additional DOFs had been left in the unmodified modal data, perhaps the rib modification would have been modeled even more accurately.

Notice in the lists of frequencies for the Bar and Quad cases, that the highest frequency modes are at much higher frequencies, far outside of the analysis range. These are *"computational modes"* that shifted to higher frequencies to compensate for the truncated model of the unmodified structure.

These modes resulted because all of the modes of the unmodified structure (above 1200 Hz) were left out of the modal model. This is called modal truncation. The real structure has more (higher frequency) modes than those included in **Aluminum Plate.SHP**. The computational modes "absorbed" the effect of the modification in place of the absent higher frequency modes.

To truncate the modal data set even further, re-run the above cases using only the *first 10 modes* in **Aluminum Plate.SHP**. A truncated set of 10 or fewer modes is quite realistic, since this number is often the upper bound of valid modes found in experimental data.