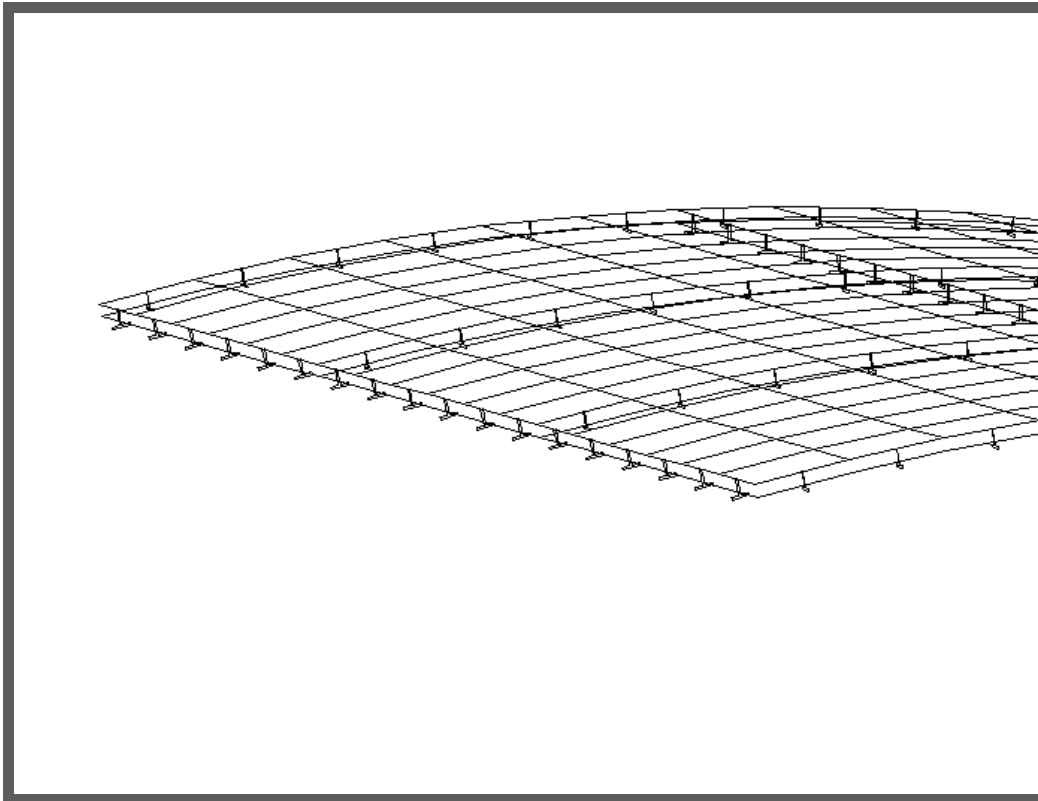


---

**Supplementary  
Exercise - 3**

*Stiffened Plate With Pressure  
Loading*



**Objective:**

- Create geometry and 1/4 symmetry finite element model.
- Create beam elements using shell element edges.



## Model Description:

In this exercise you will create a model of a portion of a curved plate which has T-beams for support. Also, analysis results will be reviewed. Following is a figure depicting the 1/4 shell model with boundary conditions..

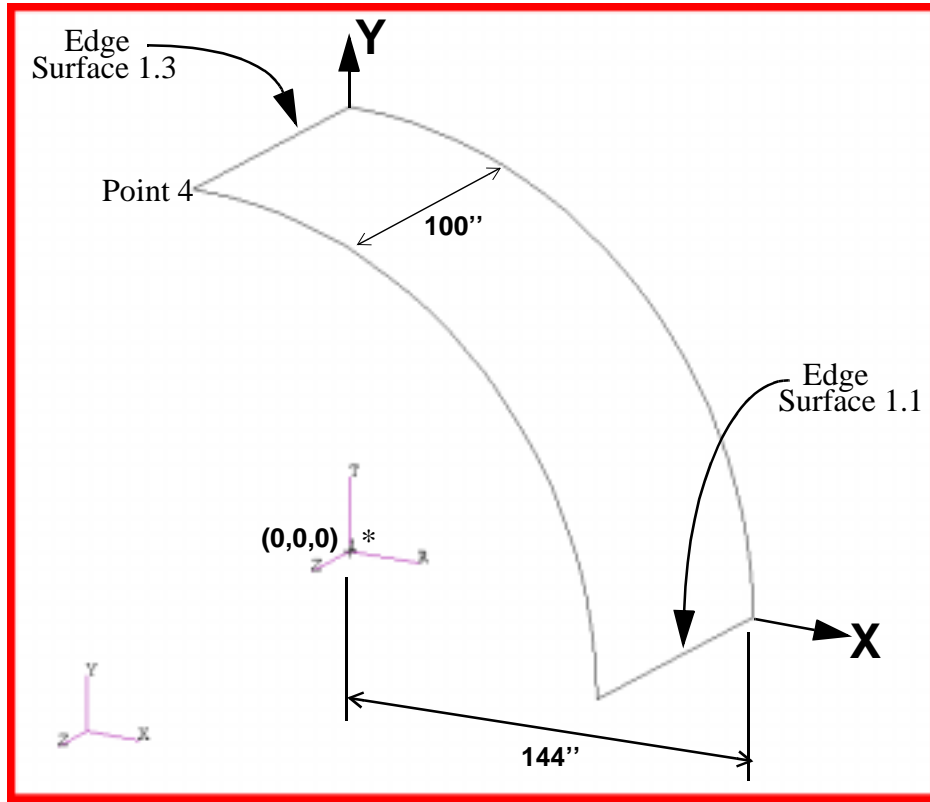


FIGURE 1.

Table 1: Boundary Conditions-Constraints

Set Name	Translations**	Rotations**	Application Region
left_symmetry	< , 0 , >	< 0 , , 0 >	Surface 1.3
bottom_symmetry	< , 0 , >	< 0 , , 0 >	Surface 1.1
end_effects	<>	< , 0 , >	Surface 1
z-constraint	< , , 0 >	<>	Point 4

\* Cylindrical coordinate system.

\*\* Constraints are in the cylindrical coordinate system.

---

## Suggested Exercise Steps:

- Create a new database named **stiffened\_plate.db**.
- Change the Tolerance to **Default** and the Analysis Code to **MSC/NASTRAN**.
- Create the geometry.
- Create the finite element mesh using the appropriate mesh seeding.
- Create beam elements using edges of shell elements.
- Create a cylindrical coordinate frame whose origin is located at [0,0,0] and whose R-, T-, Z-axis are aligned with the X-, Y-, Z-axes respectively of the global coordinate system.
- Create the material properties.
- Create boundary conditions using Table 1.
- Create the pressure load.

## Exercise Procedure:

1. Create a **New Database** and name it **stiffened\_plate.db**.

### File/New Database...

*New Database Name*

stiffened\_plate

OK

2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC/NASTRAN** in the *New Model Preferences* form. Verify that the *Analysis Type* is **Structural**.

### New Model Preference

*Tolerance*

◆ Default

*Analysis Code:*

MSC/NASTRAN

*Analysis Type:*

Structural

OK

3. Create the geometry of the curved plate shown in Figure 1.

### ◆ Geometry

*Action:*

Create

*Object:*

Curve

*Method:*

Revolve

*Axis*

Coord 0.3

*Total Angle*

90

*Point List*

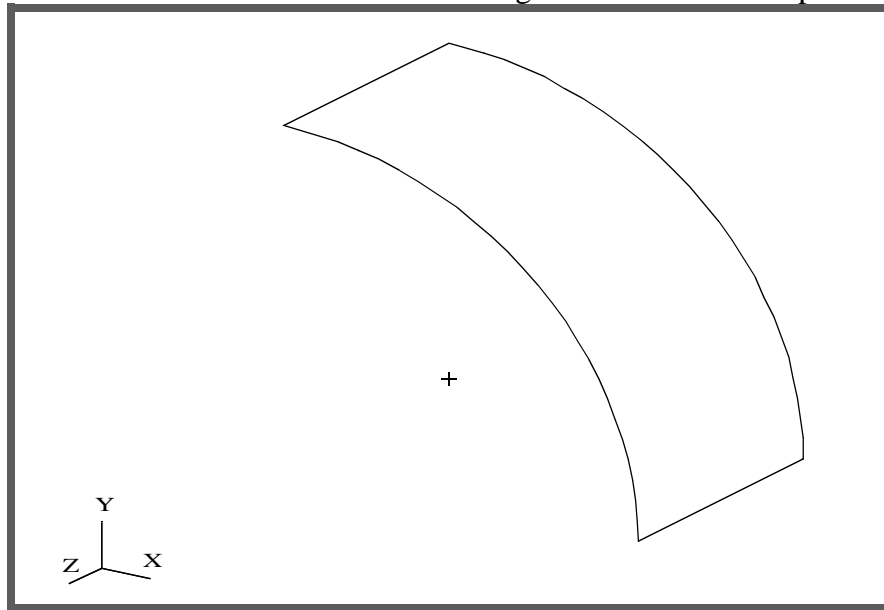
[144, 0, 0]

Apply

Create the  
Curved Plate  
model

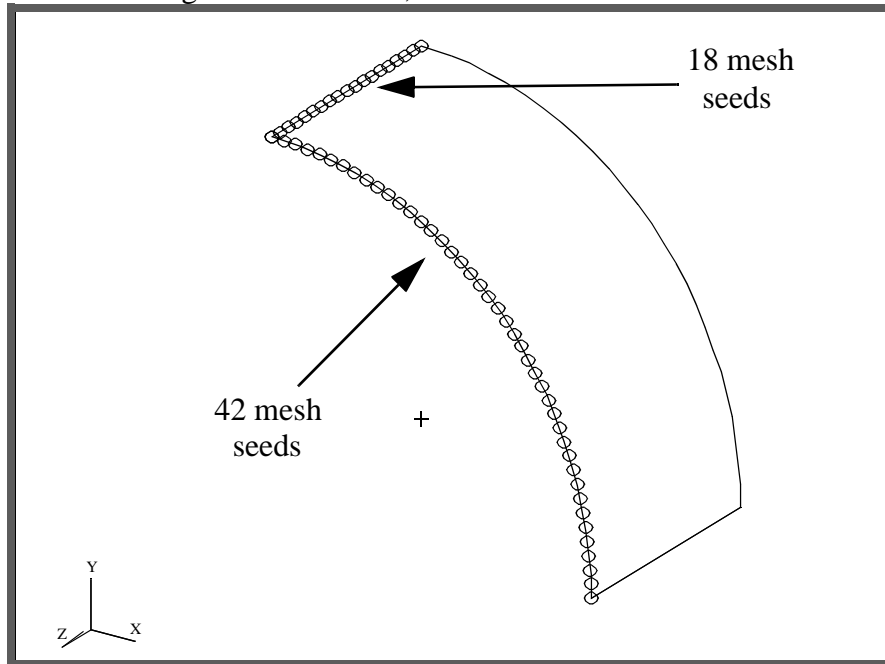
<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Surface</b>
<i>Method:</i>	<b>Extrude</b>
<i>Translation Vector</i>	<b>&lt; 0, 0, 100 &gt;</b>
<i>Curve List</i>	<b>Curve 1</b>
<b>Apply</b>	

The model should look like the figure below when completed.



# Stiffened Plate With Pressure Loading

4. Create the finite element mesh seeds, as displayed in the figure below. Then, create a mesh on the surface .



**Mesh the Model**

**◆ Finite Elements**

<i>Action:</i>	<input type="button" value="Create"/>
<i>Object:</i>	<input type="button" value="Mesh Seed"/>
<i>Type:</i>	<input type="button" value="Uniform"/>
<i>Number of Elements</i>	<input type="text" value="42"/>
<i>Curve List</i>	<input type="text" value="Surface 1.2"/>
<input type="button" value="Apply"/>	

<i>Action:</i>	<input type="button" value="Create"/>
<i>Object:</i>	<input type="button" value="Mesh Seed"/>
<i>Type:</i>	<input type="button" value="Uniform"/>
<i>Number of Elements</i>	<input type="text" value="18"/>
<i>Curve List</i>	<input type="text" value="Surface 1.3"/>
<input type="button" value="Apply"/>	

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Mesh</b>
<i>Type:</i>	<b>Surface</b>
<i>Global Edge Length</i>	<b>0.1</b>
<i>Element Topology</i>	<b>Quad 4</b>
<i>Surface List</i>	<b>Surface 1</b>
<b>Apply</b>	

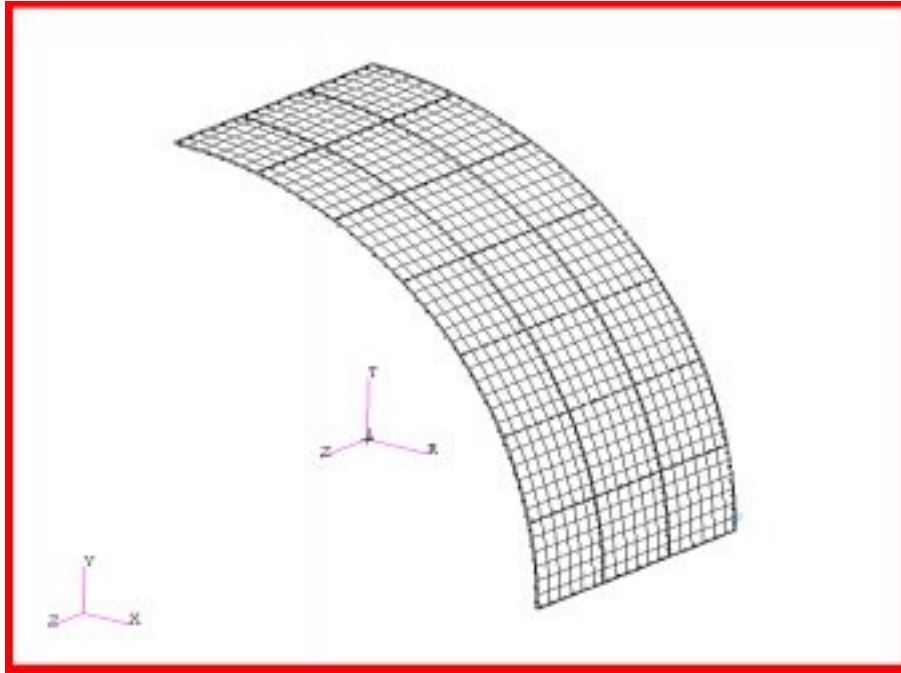
5. Create bar2 elements from the shell element edges. There are to be six rows of shell elements between adjacent rows of beam elements. Carefully select correct element edges, rotating model as necessary.

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Element</b>
<i>Method:</i>	<b>Edit</b>
<i>Shape:</i>	<b>Bar</b>
<i>Pattern:</i>	<b>Elem Edge</b>

Use the Edge of Element icon  to pick edges of shell elements.

<i>Edge</i>	<b>Select element edges as shown in following diagram</b>
<b>Apply</b>	

Create the beam elements so they are arranged like the ones shown below.



6. Create a cylindrical coordinate frame whose origin is located at  $[0,0,0]$  and whose R-, T-, Z-axis are aligned with the X-, Y-, Z-axes respectively of the global coordinate system.

**Create a  
Cylindrical  
Coordinate  
Frame**

### ◆ Geometry

Action:

Create

Object:

Coord

Method:

3Point

Type:

Cylindrical

Origin

$[0, 0, 0]$

Point on Axis 3

$[0, 0, 1]$

Point on the Plane 1-3

$[1, 0, 0]$

Apply


## Create Boundary Conditions

7. Boundary conditions for the 1/4 symmetry model should now be created using values from Table 1.

### ◆ Loads/BCs

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Displacement</b>
<i>Type:</i>	<b>Nodal</b>
<i>New Set Name</i>	<b>left_symmetry</b>
<b>Input Data...</b>	
<i>Translational</i>	<b>&lt;,0,&gt;</b>
<i>Rotations</i>	<b>&lt;0, ,0&gt;</b>
<i>Analysis Coordinate Frame</i>	<b>Coord 1</b>
<b>OK</b>	
<b>Select Application Region...</b>	

### ● Geometry

Use the Curve or Edge icon  to pick the appropriate surface edge.

<i>Select Geometric Entities:</i>	<b>Surface 1.3</b>
<b>Add</b>	
<b>OK</b>	
<b>Apply</b>	

Repeat the above steps for each boundary condition listed in Table 1.

8. Inspect boundary conditions by using markers.

<i>Action:</i>	<b>Plot Markers</b>
<i>Assigned Load/BC Sets</i>	<b>Displ_left_symmetry</b>
<i>Select Groups:</i>	<b>default_group</b>
<b>Apply</b>	

Check each constraint set using Plot Markers.

## Inspect boundary conditions

# Stiffened Plate With Pressure Loading

9. Create the isotropic material.

Specify the Material

◆ **Materials**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Isotropic</b>
<i>Method:</i>	<b>Manual Input</b>
<i>Material Name</i>	<b>material</b>
<b>Input Properties...</b>	
<i>Elastic Modulus</i>	<b>10E6</b>
<i>Poisson Ratio</i>	<b>0.3</b>
<b>Apply</b>	
<b>Cancel</b>	

10. Create the model's shell element properties. Use the name prop\_shell for your element property definitions.

Specify the Physical Properties

◆ **Properties**

<i>Action:</i>	<b>Create</b>
<i>Dimension:</i>	<b>2D</b>
<i>Type:</i>	<b>Shell</b>
<i>Property Set Name</i>	<b>prop_shell</b>
<b>Input Properties...</b>	
<i>Material Name</i>	<b>m:material</b>
<i>Thickness</i>	<b>0.0625</b>
<b>OK</b>	
<i>Application Region</i>	<b>Surface 1</b>
<b>Add</b>	
<b>Apply</b>	

## 11. Create beam element section properties.

<i>Action:</i>	<b>Create</b>
<i>Dimension:</i>	<b>1D</b>
<i>Type:</i>	<b>Beam</b>
<i>Property Set Name</i>	<b>prop_beam</b>

**Input Properties...**

<i>Material Name</i>	<b>m:material</b>
<i>Bar Orientation</i>	<b>&lt;-1 0 0 Coord 1&gt;</b>
<i>Offset@Node 1</i>	<b>&lt;-0.704 0 0 Coord 1&gt;</b>
<i>Offset@Node 2</i>	<b>&lt;-0.704 0 0 Coord 1&gt;</b>

 **Associate Beam Section****Create Sections...**

<i>New Section Name</i>	<b>section_tee</b>
-------------------------	--------------------

**Select T shape**

<i>W:</i>	<b>1</b>
<i>H:</i>	<b>1</b>
<i>t1:</i>	<b>0.125</b>
<i>t2:</i>	<b>0.125</b>

**Calculate/Display****Close****OK****OK***Application Region*

Use Beam element icon  for picking all beam elements.

*Select Members***Select all beam elements****Add****Apply**

# Stiffened Plate With Pressure Loading

12. Display beam sections with offsets.

### Display/Loads/BC/Elem. Props...

*Beam Display*

**2D: Mid-Span + Offsets**

**Apply**

**Cancel**

13. Create pressure load and name it “pressure\_shell”.

### ◆ Loads/BCs

*Action:*

**Create**

*Object:*

**Pressure**

*Type:*

**Element Uniform**

*New Set Name*

**pressure\_shell**

*Target Element Type:*

**2D**

**Input Data...**

*Bot Surf Pressure*

**0.25**

**OK**

**Select Application Region...**

*Select Surfaces or Edges*

**Surface 1**

**Add**

**OK**

**Apply**

14. Check to see if the current load case “Default” is correct.

### ◆ Load Cases

*Action:*

**Show**

*Existing Load Cases:*

**Default**

Look at the type of loads and boundary conditions (lbc), lbc names, scale factors, and lbc priorities.

15. Check to see if solution is set to Linear Static and the Analyze model.

## Analysis

### ◆ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name

stiffened\_plate

#### Translation Parameters...

Output2 Format:

Text

Nodal Coordinates:

reference frame

MSC.Nastran Version:

70.5

OK

#### Solution Type...

OK

#### Subcase Create...

Available Subcases: (Click on this)

Default

Available Load Cases:

Default

Apply

Cancel

#### Subcase Select...

Subcases For Solution  
Sequence:101

Default

Subcases Selected:  
(Do not click)

Default

OK

Apply

### ● Linear Static

# Stiffened Plate With Pressure Loading

An MSC.Nastran input file(contains the executive, case, and bulkdata section data) named stiffened\_plate.bdf is written to the Patran working directory.

Read results from MSC.Nastran.

<i>Action:</i>	<b>Read Output2</b>
<i>Object:</i>	<b>Result Entities</b>
<i>Method:</i>	<b>Translate</b>
<b>Select Results File...</b>	

- After reading in the results, plot the deformation and von mises stress for the plate.

## Results

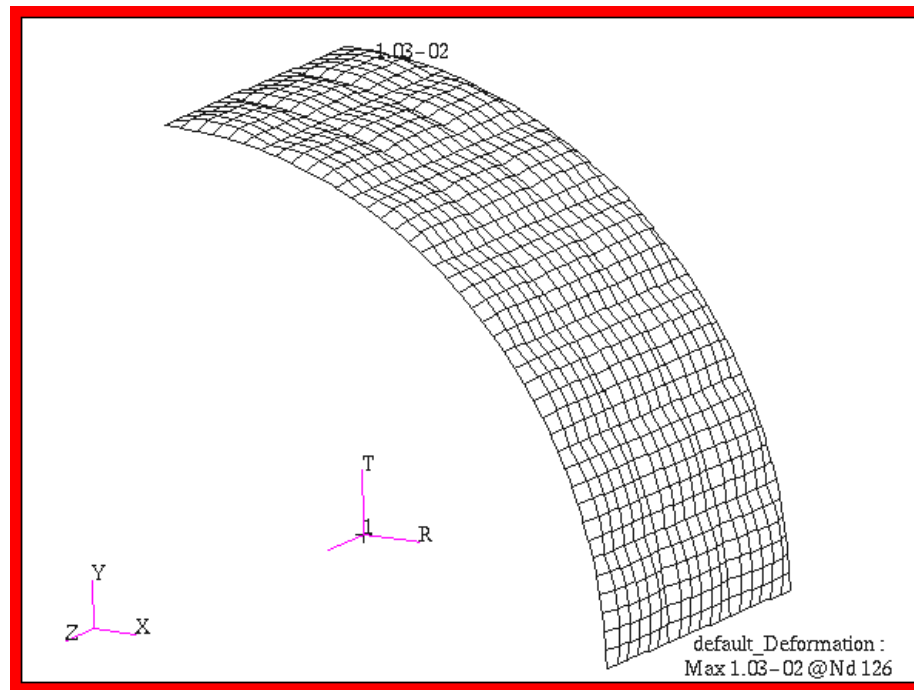
### ◆ Results

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Deformation</b>
<i>Select Result Case(s):</i>	<b>Default,Static Subcase</b>
<i>Select Deformation Result:</i>	<b>Displacements,Translational</b>

Pick Display Attributes icon . Specify the scale factor to 0.025 .

<i>Scale Factor</i>	<b>0.025</b>
<input type="checkbox"/> <b>Show Undeformed</b>	
<b>Apply</b>	

The deformation plot is shown below.



Reset graphics or unpost deformation plot for better view of fringe.

*Action:*

**Create**

*Object:*

**Fringe**


*Select Result Case(s):*

**Default,Static Subcase**

*Select FringeResult:*

**Stress Tensor**

For a better look at the fringe results turn off the element edges in Display Attributes.,

Pick Display Attributes icon .

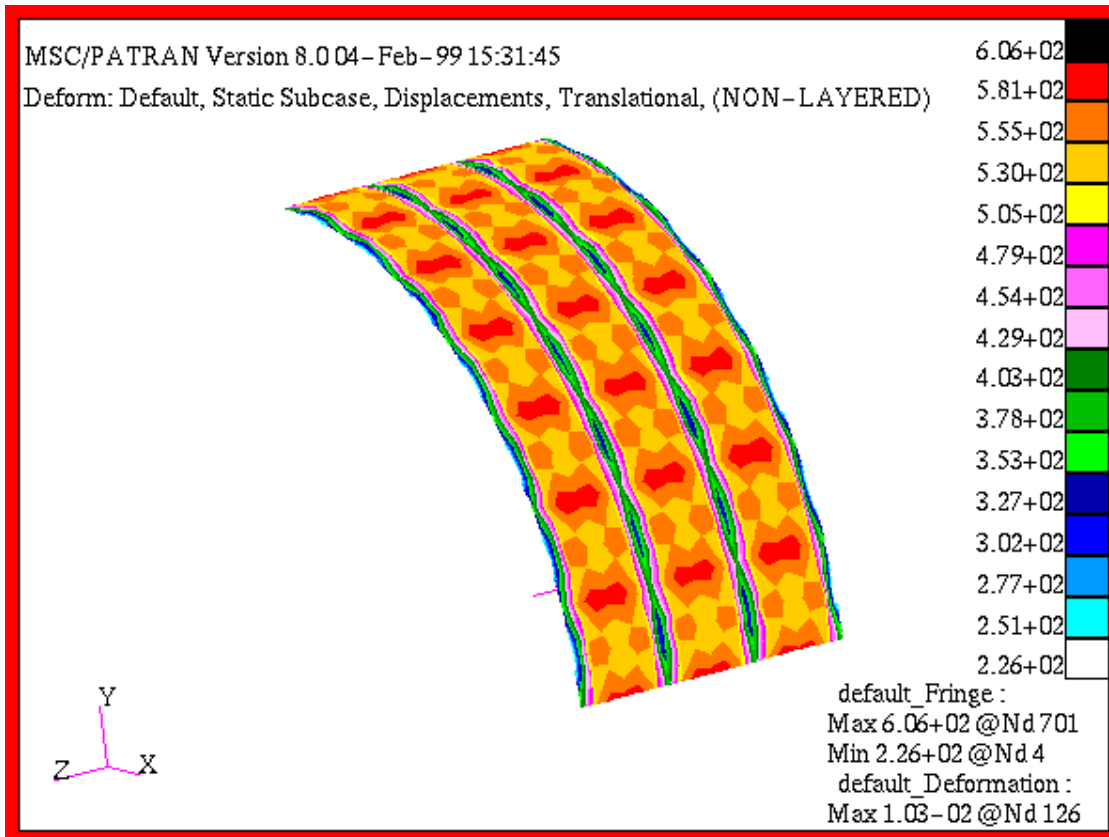
Fringe Edges

*Display:*

**No Edges**

**Apply**

The stress plot is provided below.



17. Quit MSC.Patran, closing the database

**File/Quit**

