

### Modeling Problem 7.23 with Patran/Nastran

*We will take advantage of symmetry and model only half of the structure.*

Start MD R2 Patran.

Click **File**, select **New**.

Fill in **File name** – the extension will be \*.db.

Click **OK**.

Look at **Model Preference** form – select **MSC.NASTRAN** and **Structural**.

Click **OK**.

Click **Geometry**, select **Create, Curve, Point**.

Under **Option**: choose **2 Point**.

Uncheck **Auto Execute**.

Enter in **Starting Point List**: [0 0 0]

Enter in **Ending Point List**: [0.225 0 0]

Press **Apply**.

Select **Create, Curve, 2D ArcAngles**.

Under **Radius** enter 0.025

Under **Start Angle** enter 0.

Under **End Angle** enter 180.

Uncheck **Auto Execute**.

Under **Center Point List** enter [0.25 0 0]

Press **Apply**.

Select **Create, Curve, Point**.

Enter in **Starting Point List**: [0.275 0 0]

Enter in **Ending Point List**: [0.75 0 0]

Press **Apply**.

Enter in **Starting Point List**: [0.75 0 0]

Enter in **Ending Point List**: [0.75 0.05 0]

Press **Apply**.

Enter in **Starting Point List**: [0.75 0.05 0]

Enter in **Ending Point List**: [0.5 0.05 0]

Press **Apply**.

Enter in **Starting Point List**: [0.5 0.05 0]

Enter in **Ending Point List**: [0.25 0.15 0]

Press **Apply**.

Enter in **Starting Point List**: [0.25 0.15 0]

Enter in **Ending Point List**: [0 0.15 0]

Press **Apply**.

Enter in **Starting Point List**: [0 0.15 0]

Enter in **Ending Point List**: [0 0 0]

Press **Apply**.

Click **Create, Surface, Trimmed**

Under **Option**: choose **Planar**.

Click **Auto Chain...**, uncheck **Auto Execute**.

Click in **Select a Start Curve** box; click on one of the curves you created.

Press **Apply**.

Answer **Yes** to **Do you wish to delete the original curves?**

Click **Cancel**.

Click in **Outer Loop List** box.

Click on model outline.

Press **Apply**.

Answer **Yes** to **Do you wish to delete the original curves?**

Press **Apply**.

Click **Elements**, select **Create, Mesh, Surface**.

Select **Tria, Paver, Tria3**.

Click in **Surface List** box and click on model outline.

Under **Global Edge Length** enter 0.01.

Press **Apply**.

Click **Materials**, enter the data as usual, and press **Apply**.

Click **Properties**, select **Create, 2D, Membrane**.

Fill in **Property Set Name**: Plate

Click **Input Properties**. Press **Mat Prop Name**, and click on "Steel".

Fill in **Thickness**: 0.010

Click **OK**.

Press **Select Application Region**.

Click in **Select Members** box and click on the model; press **Add**.

Click **OK**.

Press **Apply**.

Click **Loads/BCs**, select **Create, Displacement, Nodal**.

Fill in **New Set Name**: Two-dimensional constraints

Click **Input Data**.

Fill in **Translations**: < , , 0 >

Fill in **Rotations**: < 0 0 0 >

Click **OK**.

Press **Select Application Region**.

Select **Geometry** under **Geometry Filter**.

Click on model when cursor is in **Select Geometry Entities**.

Click **Add**.

Click **OK**.

Press **Apply**.

Fill in **New Set Name**: Symmetry constraints

Click **Input Data**.

Fill in **Translations**: < , 0 0 >

Fill in **Rotations**: < 0 0 0 >

Click **OK**.

Press **Select Application Region**.

Select **FEM** under **Geometry Filter**.

Click in **Select Nodes**. Create a box that encloses the nodes along the bottom edge of the model. You start the box at the top left corner by left clicking and then dragging the mouse to the bottom right corner of the box. Don't include the bottom left node of the model.

Click **Add**.

Click **OK**.

Press **Apply**.

***You may want to turn off the constraints on the model if they show.***

We continue with **Loads/BCs**, select **Create, Displacement, Nodal**.

Fill in **New Set Name**: Support at wall

Click **Input Data**.

Fill in **Translations**: < 0 0 0 >

Fill in **Rotations**: < 0 0 0 >

Click **OK**.

Press **Select Application Region**.

Select **FEM** under **Geometry Filter**.

Click in **Select Nodes**. Create a box that encloses the nodes along the left edge of the model.

Click **Add**.

Click **OK**.

Press **Apply**.

Click **Loads/BCs**, select **Create, Pressure, Element Uniform**.

Enter in **New Set Name** box: End load.

Under **Target Element Type** choose **2D**.

Click **Input Data**.

Fill in **Edge Pressure** box: -50000 (*This is in N/m by NASTRAN convention!*)

Click **OK**.

Press **Application Region**.

Be sure the **Edge of element** button is pressed.

Click in **Select 2D Elements or Edges** box.

Click on the edge of each element at the right end of the model following each click with **Add**.

Click **OK**.

Press **Apply**.

Click **Analysis**. Select: **Analyze, Entire Model, Full Run**.

Press **Apply**.

Look for file\_name.f06. This file will tell you if the model ran successfully!

Click **Analysis** and select **Access Results, Attach XDB, Both**.

Click **Select Results File**. Select file\_name.xdb.

Click **OK**.

Press **Apply**.

Click **Results** and select **Create, Fringe**.

Under **Select Fringe Result**, select **Stress Tensor**, and for **Quantity:**, select **Max Principal 2D**.

Press **Apply**.