

Modeling the Tapered Bar Problem with Patran/Nastran

Start MD R2 Patran.

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

Fill in **File name** – the extension will be *.db (Note in which folder your file will be stored.)

Click **OK**.

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

Click **OK**.

Click **Viewing** and select **Angles**; enter in **Angles**: 90. 90. 0.

Press **Apply**.

Click **Geometry**.

Select **Create, Point, XYZ**.

Uncheck **Auto Execute**.

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

Press **Apply**.

Enter in **Point Coordinate List**: [2. 0 0]

Press **Apply**.

Enter in **Point Coordinate List**: [0 0 12.]

Press **Apply**.

Enter in **Point Coordinate List**: [1. 0 12.]

Press **Apply**.

Click **Point Size**.

Click **Geometry**.

Select **Create, Curve, Point**.

Uncheck **Auto Execute**.

Click in **Starting Point List** box, and click on bottom-left point.

Click in **Ending Point List** box, and click on top-left point.

Press **Apply**.

Click in **Starting Point List** box, and click on bottom-right point.

Click in **Ending Point List** box, and click on top-right point.

Press **Apply**.

Click **Elements**, select **Create, Mesh Seed, Uniform**.

Select **Number of Elements**, and enter in **Number =** box: 4

Uncheck **Auto Execute**.

Click in **Curve List** box, and click on left vertical line.

Press **Apply**.

Click in **Curve List** box, and click on right vertical line.

Press **Apply**.

Click **Elements**, select **Create, Mesh, 2 Curves**.

Select **Tria, IsoMesh, Tria6**.

Uncheck **Auto Execute**.

Click in **Curve 1 List**, and click on left vertical line.

Click in **Curve 2 List**, and click on right vertical line.

Uncheck **Automatic Calculation**, and enter in **Value** box: 1.0

Press **Apply**.

Click **Materials**, select **Create, Isotropic, Manual Input**.

Fill in **Material Name**: Aluminum

Click **Input Properties**.

Fill in **Elastic Modulus** =: 10e+6

Fill in **Poisson Ratio** =: 0.3

Click **OK**.

Press **Apply**.

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

Fill in **Property Set Name**: Tapered Bar

Under **Options** select **Axisymmetric** and **Standard Formulation**.

Click **Input Properties**. Select "Aluminum" from **Mat Prop Name**.

Click **OK**.

Press **Select Application Region**.

Select **Entities**.

Press **Tri Element** button (this button shows a small triangle with nodes at its corners).

Click in **Select Members** box.

Select all the elements of the model by creating a box around the entire model by clicking the mouse at the top left of the model and dragging the mouse to the bottom right of the model.

Press **Add**.

Click **OK**.

Press **Apply**.

Click **Point Size**.

Click **Node Size**.

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

Fill in **New Set Name**: Wall Support

Click **Input Data**.

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

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

Click **OK**.

Click **Select Application Region**.

Select **FEM**.

Click in **Select Nodes** box.

Select all the nodes along the left end of the model by boxing them in with the cursor.

Press **Add**.

Click **OK**.

Press **Apply**.

To show the constraints, *if they don't*, click **Display, Loads/BC/Elem. Props.**, select **Show All**, and press **Apply**.

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

Fill in **New Set Name**: Centerline

Click **Input Data**.

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

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

Click **OK**.

Click **Select Application Region**.

Select **FEM**.

Click in **Select Nodes** box.

With the exception of one node at the wall, select all the nodes along the bottom edge of the model by boxing them in with the cursor.

Click **Add**.

Click **OK**.

Click **Apply**.

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

Enter in **New Set Name** box: Bar Loading

For **Target Element Type** select **2D**.

Click **Input Data**. Fill in **Edge Pressure**: -12732

Click **OK**.

Click **Select Application Region**.

Click on **Edge of Element** button (this button shows a quadrilateral with left side highlighted).

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

Click on the vertical edge of the top triangle at the far right end. Press **Add**.

Click on the vertical edge of the triangle below the one above. Press **Add**.

Click on the vertical edge of the triangle below the one above. Press **Add**.

Click on the vertical edge of the triangle below the one above. Press **Add**.

Click **OK**.

Press **Apply**.

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

Press **Solution Type** and verify it is LINEAR STATIC.

Click **OK**.

Press **Apply**.

Go to the folder you are working in, and look under **Name** for your file name, and under **Type** for the F06 file. This file will be your first clue that the model ran successfully! The output is pretty much self-explanatory.

To see the results graphically, click **Analysis** and select **Access Results, Attach XDB, Result Entities**.

Click **Select Results File**. Select file_name.xdb.

Click **OK**.

Press **Apply**.

Click **Results**.

Under **Select Fringe Result**, select: Displacements, Translational

Under **Quantity**, select: Z Component

Press **Apply**.

You may want to turn off the node and constraint symbols.

Under **Select Fringe Result**, select: Stress Tensor

Under **Quantity**, select: Z Component

Press **Apply**.