

Modeling Problem 5.31 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 **Elements**, select **Create, Node**.

Uncheck both **Associate with Geometry** and **Auto Execute**.

Enter in **Node Location List**: [0 0 0]

Press **Apply**.

Enter in **Node Location List**: [180 180 0]

Press **Apply**.

Enter in **Node Location List**: [180 0 0]

Press **Apply**.

Enter in **Node Location List**: [300 180 0]

Press **Apply**.

Click **Display**, select **Finite Elements**.

Move **Node Size** slider to middle; check **Node Label**.

Press **Apply**.

Click **Viewing**, select **Fit View**.

Click **Elements**, select **Create, Element**; select: **Bar, Bar2**; uncheck **Use existing midnodes** and **Auto Execute**.

Click in **Node 1** = box; click on Node 1.
Click in **Node 2** = box, click on Node 2.

Press **Apply**.

Repeat to create elements from Node 2 to 3, and 2 to 4.

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

Fill in **Material Name**: Steel

Click **Input Properties**.

Fill in **Elastic Modulus**: 30e+6. Click **OK**.

Press **Apply**.

Click **Properties**, select **Create, 1D, Beam**.

Fill in **Property Set Name**: Bottom Left Beam

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

Fill in **Bar Orientation**: < 0 1.414 0 >

Fill in **Area**: 8

Fill in **[Inertia 1,1]**: 100

Click **OK**.

Press **Select Application Region**.

Click in **Select Members** box.

Click on **Beam element** select button (button shows a little line between two nodes).

Click on bottom left element; press **Add**.

Click **OK**.

Press **Apply**.

We continue with **Properties** form.

Fill in **Property Set Name**: Bottom Right Beam

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

Fill in **Bar Orientation**: $\langle 1 -1 0 \rangle$

Fill in **Area**: 8

Fill in **[Inertia 1,1]**: 100

Click **OK**.

Press **Select Application Region**.

Click in **Select Members** box.

Click on bottom right element; press **Add**.

Click **OK**.

Press **Apply**.

Fill in **Property Set Name**: Top Beam

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

Fill in **Bar Orientation**: $\langle 1 1 0 \rangle$

Fill in **Area**: 8

Fill in **[Inertia 1,1]**: 100

Click **OK**.

Press **Select Application Region**.

Click in **Select Members** box.

Click on top element; press **Add**.

Click **OK**.

Press **Apply**.

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

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

Click **Input Data**.

Fill in **Translations**: $\langle 0 0 0 \rangle$

Fill in **Rotations**: $\langle 0 0 0 \rangle$

Click **OK**.

Click **Select Application Region**.

Select **FEM**.

Click in **Select Nodes**.

Click on Node 1. Click **Add**.

Click on Node 3. Click **Add**.

Click on Node 4. Click **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**: Unnecessary DOF's

Click **Input Data**.

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

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

Click **OK**.

Click **Select Application Region**.

Select **FEM**.

Click in **Select Nodes**.

Click on Node 2. Click **Add**.

Click **OK**.

Press **Apply**.

Click **Loads/BCs**. Select **Create, Distributed Load, Element Uniform**.

Enter in **New Set Name** box: Top Beam Load

Select for **Target Element Type**: 1D.

Click **Input Data**. Fill in **Distr Load**: < 0 -41.667 0 >

Click **OK**.

Click **Select Application Region**.

Click in **Select 1D Elements**. Click on top element.

Press **Add**.

Click **OK**.

Press **Apply**.

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

Press **Subcases**.

Enter for **Subcase Name**: Subcase 1

Press **Output Requests**.

Under **Select Result Type**, click **Element Forces**.

Click **OK**.

Press **Apply**.

Press **Cancel**.

Press **Subcase Select**.

Check **Unselect All**.

Select: Subcase 1

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 create a customized report, click **Analysis** and select **Access Results, Attach XDB, Result Entities**.

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

Click **OK**.

Click **Translation Parameters**. Check **Rotational Nodal Results**.

Click **OK**.

Press **Apply**.

Click **Results** and select **Create, Report, Overwrite File**.

Select in **Select Report Result** box - Displacements, Translational - and then press **Apply**.

Change **Overwrite File** to **Append File**.

Repeat for: Displacements, Rotational; Bar Forces, Translational; Bar Forces, Rotational; Constraint Forces, Translational; Constraint Forces, Rotational.

You should now have a patran.rpt file in your folder with these results.

Explanation of **Bar Orientation** vector: As input, the vector is defined in the *global* axes at the *beginning* node of the beam element. In the *local* elemental axes, \hat{x} and \hat{y} , the vector we entered is always $\hat{v} = 1\hat{i} + 1\hat{j}$. This input arrangement creates an \hat{x} - \hat{y} system for each beam element as we defined it in class. This arrangement also makes $\hat{I}_z = [\text{Inertia } 1,1]$. To display the \hat{y} axis for each beam element, click **Properties**, select **Show**. Under **Select Property**, select Bar Orientation. Under **Display Method**, select **Vector Plot**, and press **Apply**.

The distributed load is input in the local \hat{x} - \hat{y} axes.