

**Modeling with Patran/Nastran the rigid frame problem done in class,  
which is a modification of Example 5.1**

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 **Associate with Geometry** and **Auto Execute**

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

Press **Apply**

Enter in **Node Location List**: [0 120 0]

Press **Apply**

Enter in **Node Location List**: [120 120 0]

Press **Apply**

Enter in **Node Location List**: [120 0 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 3 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**: Left Beam

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

Fill in **Bar Orientation**: < -1 1 0 >

Fill in **Area**: 10

Fill in **[Inertia 1,1]**: 200

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 left element; press **Add**

Click **OK**

Press **Apply**

We continue with **Properties** form.

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

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

Fill in **Bar Orientation**: < 1 1 0 >

Fill in **Area**: 10

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

Click **OK**

Press **Select Application Region**.

Click in **Select Members** box

Click on horizontal element; press **Add**

Click **OK**

Press **Apply**

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

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

Fill in **Bar Orientation**: < 1 -1 0 >

Fill in **Area**: 10

Fill in **[Inertia 1,1]**: 200

Click **OK**

Press **Select Application Region**.

Click in **Select Members** box

Click on right element; press **Add**

Click **OK**

Press **Apply**

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

Fill in **New Set Name**: 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**

Click on Node 1. Click **Add**

Click on Node 4. Click **Add**

Click **OK**

Click **Apply**

To show the constraints, *if they don't*, click **Display, Loads/BC/Elem. Props.** and 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 on Node 3. Click **Add**

Click **OK**

Click **Apply**

Click **Loads/BCs**, select **Create, Force, Nodal**

Enter in **New Set Name** box: Consistent Load on Node 1

Click **Input Data**. Fill in **Force**: <1563 0 0 >

Fill in **Moment**: <0 0 -56250>

Click **OK**

Click **Select Application Region**

Click in **Select Nodes** box. Click on Node 1.

Press **Add**

Click **OK**

Press **Apply**

Enter in **New Set Name** box: Consistent Load on Node 2

Click **Input Data**. Fill in **Force**: <8438 0 0 >  
Fill in **Moment**: <0 0 168800>

Click **OK**

Click **Select Application Region**

Click in **Select Nodes** box. Click on Node 2.

Press **Add**

Click **OK**

Press **Apply**

We continue with **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 -20.833 0 >

Click **OK**

Click **Select Application Region**

Click in **Select 1D Elements**. Click on horizontal 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.