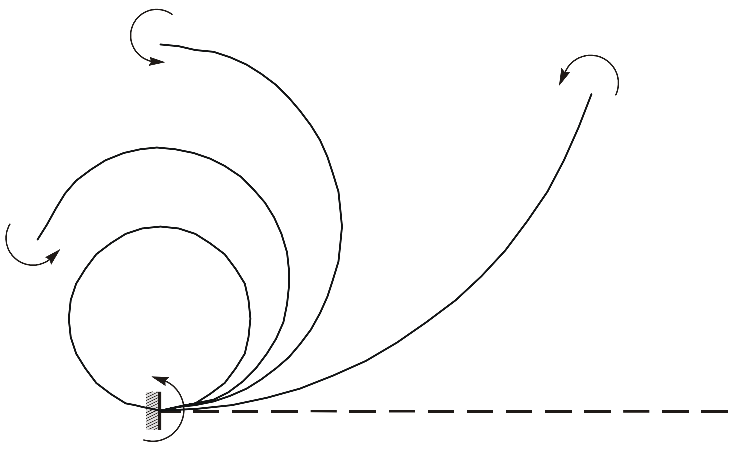
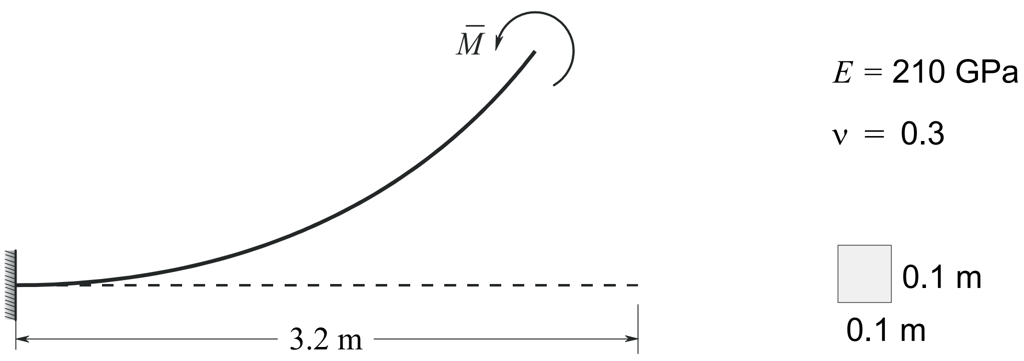
Workshop 10

Nonlinear Static Analysis of a Cantilever Beam



This example uses a simple cantilever beam model to demonstrate the geometric nonlinear effects of large rotations. A linear solution method would not be adequate for this case as equilibrium can only be satisfied in the deformed configuration due to the high applied loading. The example is extensively studied in the literature to demonstrate the efficiency of numerical methods and the large rotation capability of beam, plate and shell elements.

Model Description



The model consists of a cantilever beam: 3.2 m long, divided into 15 elements, subjected to an end moment of $\bar{M} = 3436117$ N m at the free end, which forces the beam to curl into a complete circle. The beam has a 0.1 m square cross section.

Exercise Procedure

1. Start up **MSC/NASTRAN for Windows 4.5** and begin to create a new model.

   Double click on the icon for the **MSC/NASTRAN for Windows V4.5**.

   On the *Open Model File* form, select **New Model**.

   Turn off the workplane:

   Tools / Workplane (or **F2**) / ☐ **Draw Workplane / Done**

   View / Regenerate (or **Ctrl G**).

2. Create a material called **mat_1**.

   From the pulldown menu, select **Model / Material**.

Title	**mat_1**
Young's Modulus	**2.1e11**
Poisson's Ratio	**0.3**

   Select **OK** / **Cancel**.

   NOTE: In the *Messages Window* at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

3. Create a property called **prop_1** to apply to the members of the beam.

   From the pulldown menu, select **Model / Property**.

Title	**prop_1**
Material	**mat_1**

Elem / Property Type

Change the property type from **Plate** element (default) to **Beam** element.

Line Elements	**Beam**

Select **OK**.

To select the cross-sectional shape, click on **Shape**:

Shape	**Rectangular Bar**
H	**0.1**
Width	**0.1**
Orientation Direction (y)	✓ **Up**

Select **OK / OK / Cancel**.

4. Create the beam model (15 elements along the beam).

We will do a bit different from Workshops 7 and 8.

From the pulldown menu, select **Geometry / Curve-Line / Coordinates**.

	X:	Y:	Z:	
Point 1	**0**	**0**	**0**	**OK**
Point 2	**3.2**	**0**	**0**	**OK / Cancel**

To fit the display onto the screen, select **View / Autoscale / Visible** (or **Ctrl A**).

Now define de mesh.

From the pulldown menu, select **Mesh / Mesh Control / Custom Size Along Curve**.

Select the curve on the screen using the mouse and, then, **OK**.

Num Elements	**15**	**OK / Cancel**

Mesh the curve.

From the pulldown menu, select **Mesh / Geometry / Curve**.

Select the curve on the screen using the mouse and, then, **OK**.

Property	**prop_1**	**OK**

Now, specify the orientation vector for the beam elements.

	X:	*Y*:	*Z*:	
Base	***0***	***0***	***0***	**OK**
Tip	***0***	***1***	***0***	**Preview / OK**

NOTE: In MSC/NASTRAN, the way to construct the element coordinate system is by defining an *orientation vector*, as explained in Workshop 2. The element lies on the local x axis and the moments of inertia I_y and I_z are related to the bending about the local y and z axes, respectively. So, be certain that you understand the assumed beam orientation.

5. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

Model / Constraint / Set

Title	**constraint_1**

Select **OK**.

Now, define the relevant constraint for the model.

Model / Constraint / Nodal

Select **Node 1 / OK**.

On the *DOF* box, select all 6 boxes (or just *fixed*)

✓	**TX**	✓	**TY**	✓	**TZ**
✓	**RX**	✓	**RY**	✓	**RZ**

Select **OK** / **Cancel**.

Notice that the constraint appears on the screen at Node 1, fixing the 1, 2, ..., 6 directions (corresponding to TX, TY, TZ, RX, RY and RZ).

6. Create the model loading.

Like the constraints, a load set must first be created before creating the appropriate model loading.

Model / **Load** / **Set** (or **Ctrl F2**)

Title **load_1**

Select **OK**.

Now, define the 3436117 N m applied moment.

Model / **Load** / **Nodal**

Select **Node 16** / **OK**.

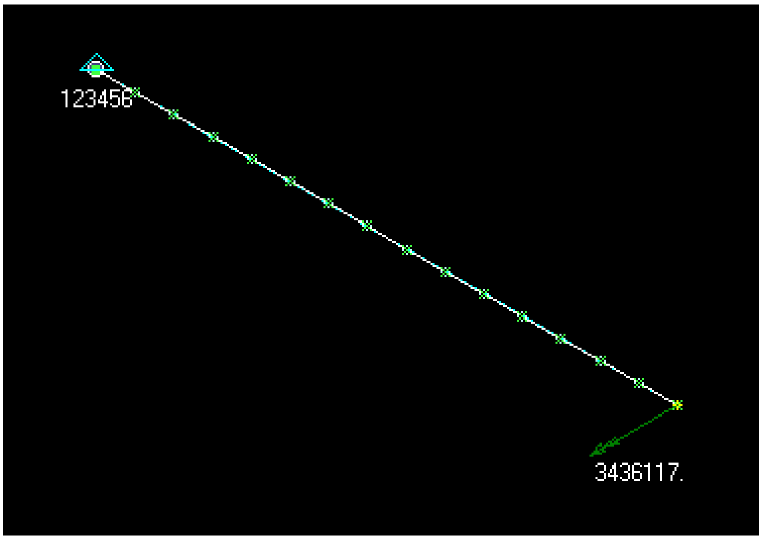
Highlight **Moment**

Load **MZ** ✓ **3436117**

Select **OK** / **Cancel**.

To obtain a better view, do the following:

View / **Rotate** (or **F8**) / **Isometric** / **OK**.



7. Set the nonlinear solution parameters.

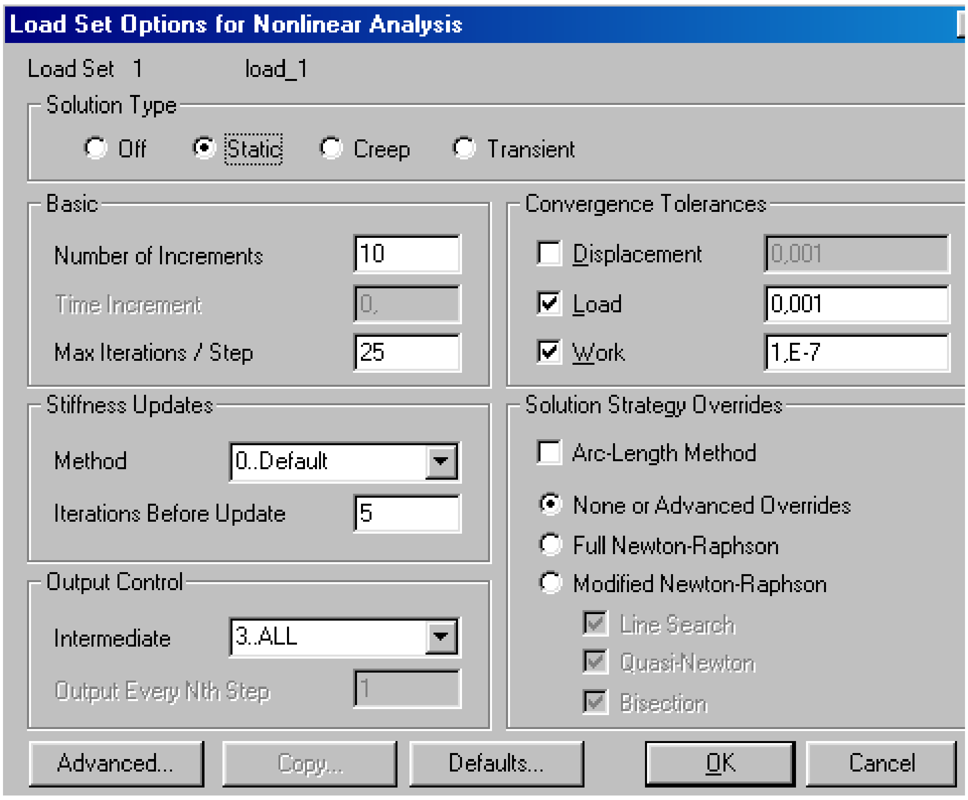
Model / Load / Nonlinear Analysis

Solution Type ✓ **Static**

Click the **Defaults** button to apply the default values for the nonlinear solution control parameters.

The number **10** is a reasonable value for the **Number of Increments** at this analysis.

If results are desired at every increment, then the **Output Control Intermediate** box should be changed to **ALL**.

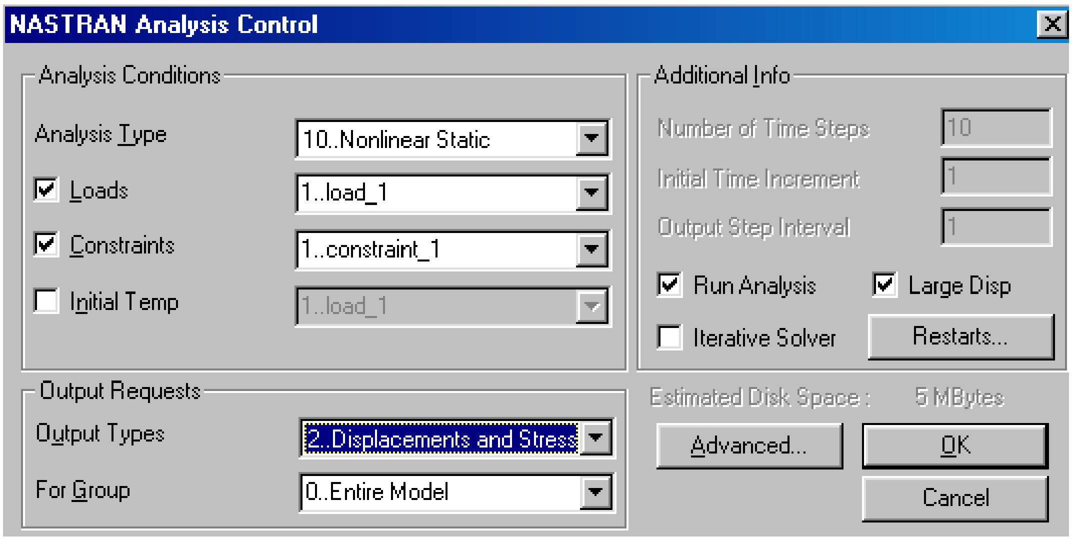


Now click **OK** to accept the chosen selections.

It is possible to change the nonlinear solution parameters by using **Solution Strategy Overrides**, for instance, but it is not usually necessary. For certain problems, however, the default settings may not lead to a solution covergence.

8. Run the analysis.

   File / Analyze



Analysis Type	**Nonlinar Static**
Loads	✓ **load_1**
Constraints	✓ **constraint_1**
	✓ **Run Analysis**
	✓ **Large Disp**
Output Types	✓ **Displacements and Stresses**

Select **OK**.

When asked if you wish to save the model, respond **Yes**.

Be sure to set the desirable working directory.

File Name	**work_10**

Select **Save**.

When the MSC/ NASTRAN manager is through running, MSC/ NASTRAN for Windows will be restored on your screen, and the *Message Review* form will appear.

To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time. Then select **Continue**.

When asked: *OK to Read Nonlinear Stresses and Strains?* Respond **Yes**.

9. Processing the results.

View / Select (or **F5**)

Deformed Style	**Deform**
	Deformed and Contour Data
Output Set	**Case 20 Time 1**
Output Vectors / Deformation	**Total Translation**

Select **OK / OK**.

Notice that the automatic iteration control process of MSC/NASTRAN for Windows has automatically added some additional load increments on top of the 10 selected in the nonlinear parameter settings. In the analysis phase, 10 iterations clearly were not enough to satisfy the convergence criteria so the program automatically added a sufficient number to obtain a solution.

View / Options (or **F6**)

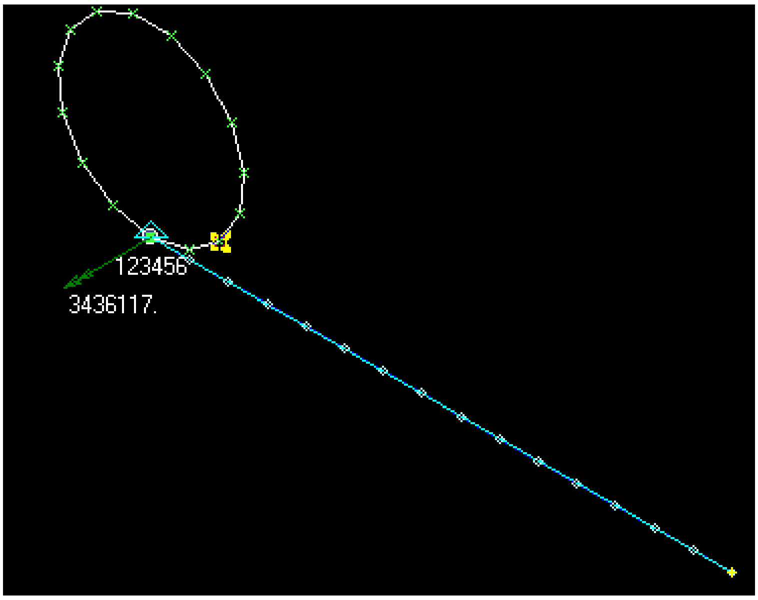
Category ✓ **PostProcessing**

Highlight **Deformed Style**

% of Model (Actual) ☐

Scale Act ***1*** **OK**

This ensures that the deformed shape is set to the actual scale.



The above shows the actual deformation of the cantilever due to the applied end moment loading.

This concludes the exercise.

File / Save

File / Exit.