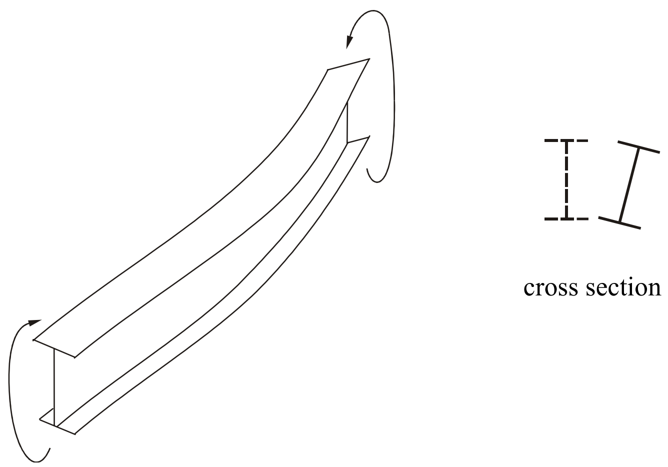
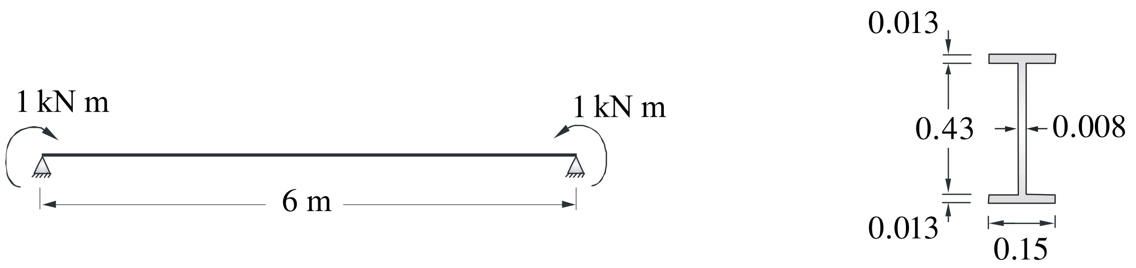
Workshop 8

Lateral Buckling



A transversely loaded member that is bent about its major axis may buckle sideways if its compression flange is not laterally supported. The reason buckling occurs in a beam is that the compression flange, which is in effect a column on an elastic foundation, becomes unstable. At the critical loading there is a tendency for the compression flange to bend sideways and for the remainder of the cross section, which is stable, to restrain it from doing so. The net effect is that the entire section rotates and moves laterally, as shown above. Lateral buckling, or as it is sometimes called *lateral-torsional buckling*, of a beam is thus a combination of twisting and lateral bending brought about by the instability of the compression flange. The present workshop ilustrates an interesting case of lateral buckling of an I beam in pure bending.

Model Description



cross section
(all dimensions in m)

Above is the beam shown on the title page loaded by 1 kN m opposite end moments in the plane of the web. The ends are assumed to be simply supported as far as bending about the principal axes of inertia of the cross section is concerned. In addition, the ends are prevented from rotating about the axial direction but are free to warp. In linear buckling analysis we solve for the eigenvalues which are scale factors that multiply the applied load (unit in this case) in order to produce the buckling load. The material properties are listed below.

Young's Modulus	7.1×10^{10} kN/m^2
Poisson's Ratio	0.33

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	**7.1e10**
Poisson's Ratio	**0.33**

   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**.

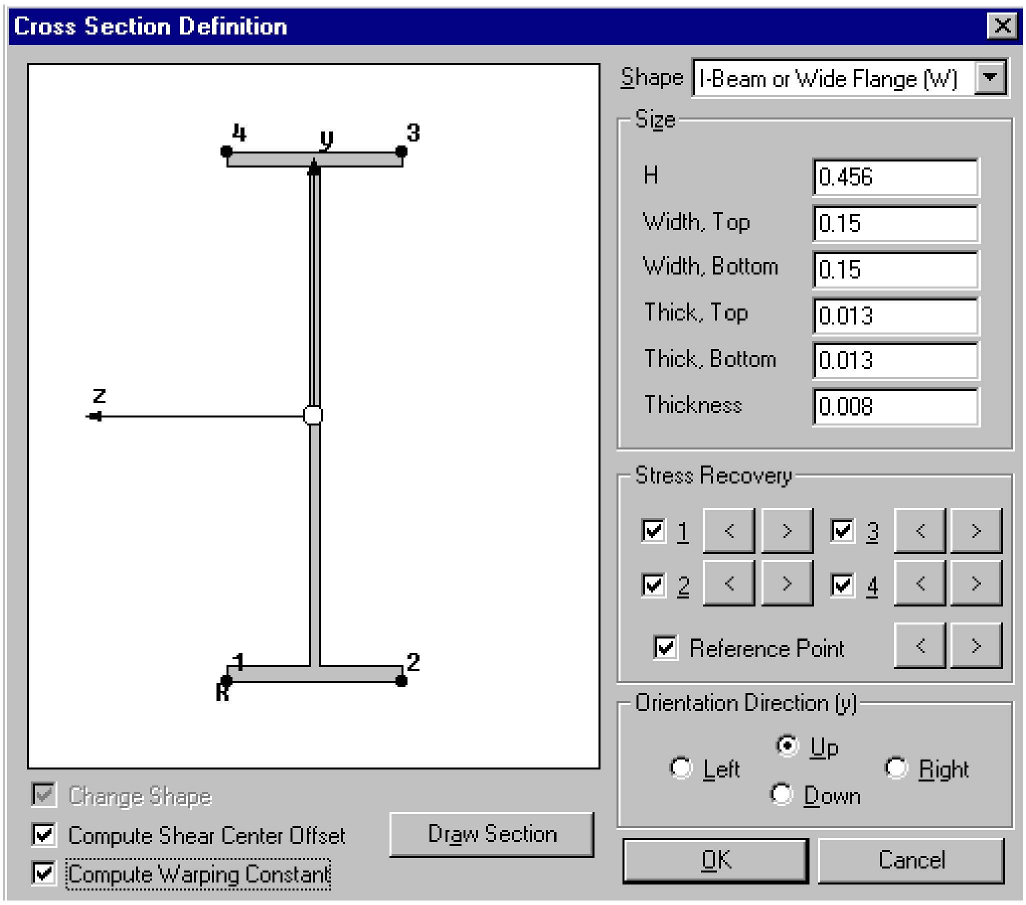
Title	**prop_1**
Material	**mat_1**
	Elem / Property Type

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

Line Elements **Beam**

Select **OK**.

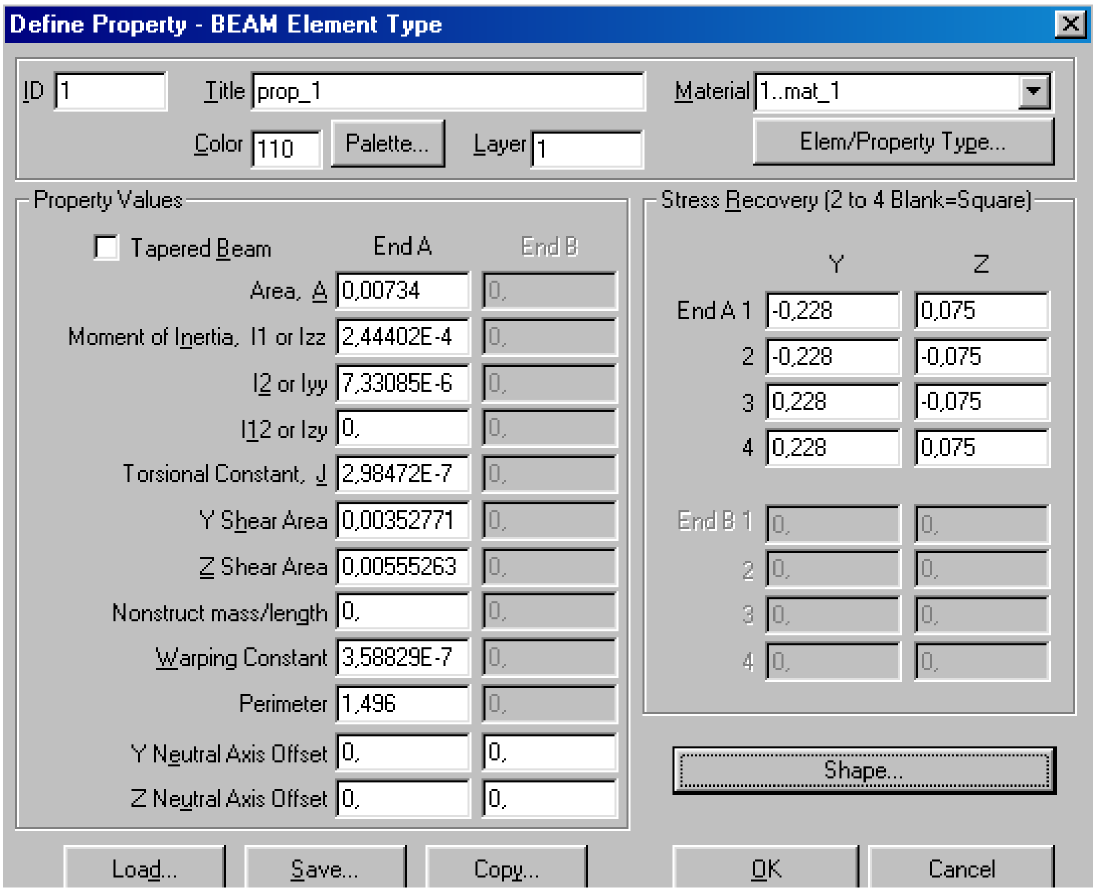
Instead of entering the section properties manually, do so automatically by selecting **Shape**



Shape	**I-Beam or Wide Flange (W)**
H	**0.456**
Width, Top	**0.15**
Width, Bottom	**0.15**
Thick, Top	**0.013**
Thick, Bottom	**0.013**
Thickness	**0.008**

Orientation Direction (y)	✓	**Up**
	✓	**Compute Warping Constant**

Select **OK**.



Select OK / Cancel.

4. Create the MSC/NASTRAN model for the beam (10 elements).

From the pulldown menu, select **Mesh / Between** (or **Ctrl B**).

Property	**prop_1**
Mesh Size / #Nodes / Dir. 1	**11**

Select **OK**.

	X:	*Y*:	*Z*:	
Corner 1	*0*	*0*	*0*	**OK**

	X:	Y:	Z:	
Corner 2	***6***	***0***	***0***	**OK**

Now, specify the orientation vector for the bar elements.

	X:	Y:	Z:
Base	***0***	***0***	***0***
Tip	***0***	***1***	***0***

Select **OK**.

NOTE: In MSC/NASTRAN, the way to orient the element coordinate system xyz in the global XYZ space 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 bar orientation.

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

Display the node and element numbers:

View / Options / Quick Options (or **Ctrl Q**) **/ Labels On / Done / OK**.

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 / Node 11 / OK**.

On the *DOF* box, select

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

Select **OK / Cancel**.

Notice that the constraint appears on the screen at Nodes 1 and 11, fixing the 1, 2, 3 and 4 directions (corresponding to TX, TY, TZ and RX).

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 1 kN m opposite end moments.

Model / Load / Nodal

Select **Node 1 / OK**.

Highlight **Moment**

Load **MZ** ✓ **-1** **OK**

Select **Node 11 / OK**.

Highlight **Moment**

Load **MZ** ✓ **1** **OK**

Select **Cancel**.

7. Run the analysis.

   File / Analyze

Analysis Type	**Buckling**
Loads	✓ **load_1**
Constraints	✓ **constraint_1**
Number of Eigenvalues	**2**
	✓ **Run Analysis**

   Select **OK**.

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

   Be sure to set the desirable working directory.

File Name	**work_8**

   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 detail this time. Then select **Continue**.

8. What is the first two eigenvalues?

   In general, only the lowest buckling load is of interest, since the structure will fail before reaching any of the higher-order buckling loads. Therefore, usually only the lowest eigenvalue needs to be computed. Here, the first two eigenvalues are requested to illustrate a subtle point. The eigenvalues are identical in magnitude but different in sign. Since both the structure and loading are symmetric, it does not matter whether the applied moments are as shown on page 8-2 or inverted. Thus, the negative value is legitimate both from a physical and mathematical standpoint.

Use **View / Select** (or **F5**) / **Deformed and Contour Data / Output Set** or **List / Output / Query / Output Set**.

9. Display the deformed plot on the screen.

Finally, you may now display the first buckling mode which corresponds to the positive or negative eigenvalue. You may want to remove the node and boundary constraint markers.

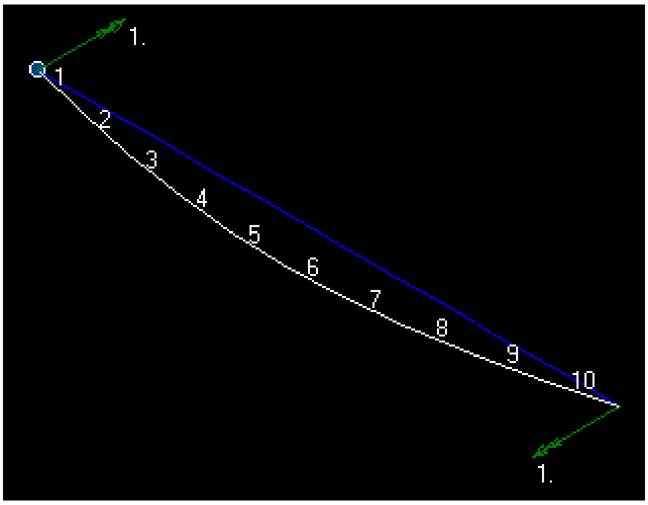
View / Options / Quick Options (or **Ctrl Q**) / ☐ **Node** / ☐ **Constraint / Done / OK**.

View / Select (or **F5**)

Deformed Style **Deform**

Select **Deformed and Contour Data / Output Set / OK / OK**

View / Rotate / Isometric / OK



This concludes the exercise.

File / Save

File / Exit.

Answer

Eigenvalue 1	33841.04
Eigenvalue 2	-33841.04

From the theory (Chajes, A., 1974, *Principles of Structural Stability Theory*, Prentice-Hall, Englewood Cliffs), the critical moment is given by

$$\begin{aligned}\bar{M}_{cr} &= \frac{\pi}{L}\sqrt{EI_y\left(GJ + E\Gamma\frac{\pi^2}{L^2}\right)}\\ &= \frac{\pi}{6}\sqrt{7.1\times10^{10}\left(7.33085\times10^{-6}\right)\left[\frac{7.1\times10^{10}}{2\left(1+0.33\right)}2.98472\times10^{-7} + 7.1\times10^{10}\left(3.58829\times10^{-7}\right)\frac{\pi^2}{6^2}\right]}\\ &= 46189.72 \text{ kN m},\end{aligned}$$

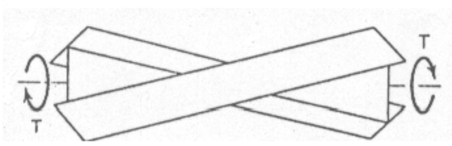
in which J is the *torsional constant* and Γ is the *warping constant.* Apparently, the MSC/NASTRAN results converge toward

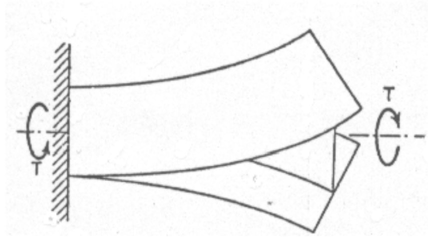
$$\bar{M}_{cr} = \frac{\pi}{L}\sqrt{EI_yGJ} = 33716.70 \text{ kN m}.$$

Is this program version unable to activate the warping contribution? The result $\pm$ 46447.41 would be given by **MSC/Visual Nastran for Windows 2003**.

NOTE:

- If the longitudinal displacements that produce warping are allowed to take place freely, then longitudinal fibers do not change length, and no longitudinal stresses are induced as a result of warping. This type of resistance to twisting consisting solely of shear is called *St. Venant* or *uniform torsion* and is related to the *torsional rigidity* GJ.





However, certain support or loading conditions will prevent longitudinal displacement from taking place freely. For example, the built-in end of the cantilever beam in figure is completely restrained against warping, while the unsupported end is allowed to warp freely. As a consequence, longitudinal fibers change in length, and axial stresses are induced in the member. Comparison of the beam on the right of figure, in which warping is partially restrained, with the beam on the left, which is free to warp, indicates that a restraint of warping deformation results in a differential bending of the flanges. One flange bends to the right and one to the left. This type of resistance to twisting is called *nonuniform torsion* and is related to the *torsional rigidity* GJ in addition to the *warping rigidity* $E\Gamma$.

- Since lateral buckling is a combination of twisting and lateral bending, it is not surprising that the critical moment involves EI_y, GJ and, in this specific problem with warping restraint, $E\Gamma$.

- The magnitude of the critical moment does not depend on the flexural rigidity EI_z of the beam in the vertical plane if the deflection in this plane is small, which is justifiable when the flexural rigidity EI_z is very much greater than the rigidity EI_y. If the rigidities are of the same order of magnitude, the effect of bending in the vertical plane may be of importance and should be considered.

- The results of numerous theoretical studies have demonstrated that the above expression for the critical moment can be made valid for other loading and boundary conditions by applying appropriate correction factors.