Tutorial W16 x 26 simply supported beam with/without a lateral brace under constant moment. Beam
20 ft. long. A36 Steel

Starting the program
1. On the Windows Start menu, select Programs > BASP1.4 > BASP1.4.
2. Click a mouse button on the window showing license information.

Creating a beam
1. Select Beam/Geometry from the menu bar.
2. Input the geometry of W16 x 26 as shown below.
3. Select End 2, and then click Set End 2 = End 1
4. Select Material, and then input 29000 for Modulus of Elasticity and 0.3 for Poisson' Ratio.
5. Select Mesh and input 80 for Longitudinal and 5 for Transverse.
6. Click Create Finite Element Mesh, then the finite element mesh shows.
7. Click Angled View from the toolbar. The beam is shown below.

Adding boundary conditions
1. Select Elements/Boundary to change the editing mode to BOUNDARY. The editing mode is shown in the status bar.
2. Select Zoom All from the toolbar.
3. Select nodes on the left end of the beam as shown below.
4. Hold down the shift key, and select nodes on the right end of the beam as shown below
5. Click Set Property from the toolbar.
6. Set Z displacement to be zero as shown below, and click Apply.
7. Select the node at the lower left corner of the beam, and set X and Y displacement to be zero as well as Z displacement.
8. Select the node at the lower right corner of the beam, and set Y displacement to be zero as well as Z displacement.
9. Select View > View Options from the menu bar, and then uncheck Show Plate and Show Stiffeners. Click Apply and Close.
10. Click Angled View from the toolbar. The beam with the imposed boundary conditions is shown below.
11. Select View > Actual/Fit View from the menu bar. The View Mode in the status bar has changed to Fit View, and the scale in Y direction has increased as shown below. Now, the boundary conditions are more clearly seen.

Applying Loads
1. Select Elements > Load to change the editing mode to LOAD. The editing mode is shown in the status bar.
2. Select Zoom All from the toolbar.
3. Select the node at the upper left corner, click Set Property, and the input 1 as the load in X direction. Click Apply.
4. Select the node at the lower left corner, click Set Property, and the input -1 as the load in X direction. Click Apply.
5. Select the node at the upper right corner, click Set Property, and the input -1 as the load in X direction. Click Apply.
6. Select the node at the lower right corner, click Set Property, and the input 1 as the load in X direction. Click Apply.

7. Double click the node at the lower left corner. A window shows X = 0 and Y = 0, which is the coordinate of the node. Click Close.
8. Double click the node at the upper let corner. A window shows X = 0 and Y = 15.345. Since the difference in Y coordinates of the coupling loads at left end is 15.345, the loading condition will impose a constant moment of 15.345 on the beam. Click Close.
9. Select View > View Options from the menu bar, and then uncheck Show Boundary Conditions. And, check Show Values next to Show Load Conditions. Click Apply and Close.
10. If the loading condition is not seen clearly, select View > View Options from the menu bar. Select Marker, increase the value of Property Marker Size, and click Apply.

Running Analysis

Select Analysis > Run. Try 0 as Estimated. Click Run. The eigenvalue is 32.3. Click Close.

Viewing Buckled Shape

1. Select View > Actual/Fit View from the menu bar. The View Mode in the status bar has changed back to Actual View.
2. Select View > View Options from the menu bar, and then uncheck Show Load Conditions. Check Show Plate and Show Stiffeners. Click Apply and Close.
3. Select Results > Show Buckled Shape from the menu bar. The buckled shape is shown below. The shape looks like the first buckling mode. Therefore, 32.3 times 15.345 is 495.6 k-in., which is the buckling moment.

Viewing Stress Plot

1. Select Results > Stress Plot from the menu bar. Select Sx and Stiffeners option and click Apply.
2. Move the mouse over the stress plot, the stress value at the mouse location shows.
3. Select Property > Show Property Window. Click Stress. Stresses in bottom flange are mostly 0.394. Stresses at the ends of the bottom flange are higher than 0.394. This is because of the stress concentration under the concentrated load. The model is not under pure bending. If you plot Sxy, you can see shear stresses at the ends of the beam. Since the shear is restricted at the ends only, we will disregard the shear stresses. Using the stress of 0.394 and the eigenvalue of 32.3, the stress is 0.394 times 32.3, which is 12.7. This stress is less than 36 ksi. Therefore, the beam buckles in elastic range. Click Save to File. Give a file name of BeamStress, The file is saved as BeamStress.txt. Select the rows listing ID number from 1 to 10, and then click Print. Only the selected text is printed. Click Show Property on the toolbar to close the property window.
4. Close the stress plot toolbar by clicking Close.
5. Select File > Save Data, and save the file with the file name of Unbraced Beam. The file is saved as Unbraced Beam.inp.

Checking the bending stress

Using M = 15.345 and I = 301, bending stress at the center of the bottom flange is 15.345 * 7.67 / 301 = 0.391 ksi. This is very close to 0.394, the stress output from the program.

Adding a brace

1. Select Elements > Brace to change the editing mode to Brace.
2. Select the node at top flange in the mid-span, click Set Property, and the input 1.6 as the brace stiffness in Z direction. Click Apply.

3. Run the analysis using 0 as an estimate. If we have a negative number, try a number larger than 0 as an estimate. Try 15 as Estimated and run the analysis. If you still get a negative value, try a larger value. Using 32 as Estimate gives an eigenvalue of 104.5.
4. Plot the buckled shape. As shown in the figure below, the buckled shape look like a second mode.
5. Check the stress. The stress plot of the model is same as the one without the brace since the stress plot only shows the stresses after in-plane analysis. Sx stress is 0.394. The stress is 0.394 times 104.5 = 41.17. This stress is larger than the yield stress. Therefore, the beam yields before the buckling occurs.
6. Click Close
7. Select File > Save Data, and save the file with the file name of Braced Beam. The file is saved as Braced Beam.inp.

Miscellaneous
1. Select File > Read Data, and select Unbraced Beam.inp.
2. Select Results > Show Buckled Shape from the menu bar. Then, select View > Clipping. Set the viewable range to the right half of the beam by moving the slider. The result is shown below. Click Reset and Close.
3. Select File > Create Bitmap File. The operation mode changes to Create BMP. Select a region enclosing the model on the screen. Give a file name of Half Span. The file is saved as Half Span.bmp.
4. Select View > View Option > Result, and change Out-of-plane Magnification Factor to 30. Click Apply and Close. The buckled shape shows more clearly.
5. Select View > Rotate. The Operation Mode shown in the status bar is now Rotate. Press mouse button, drag the mouse to the right, and release the button. The model has rotated.
6. Select File > End. This ends the program execution.