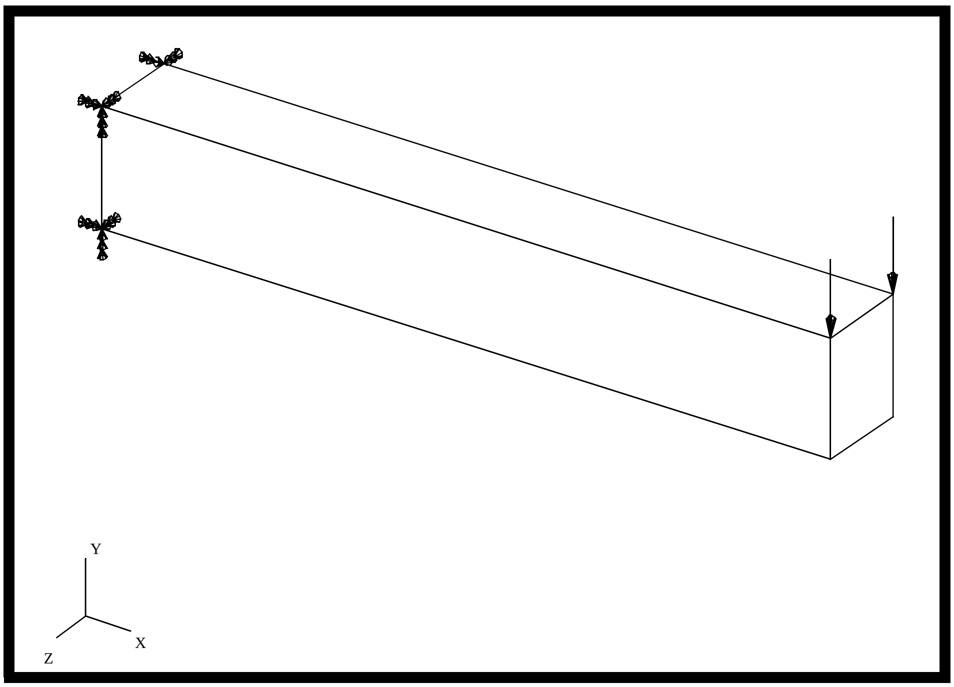
WORKSHOP 18

CBAR Element
Shear Factor, K



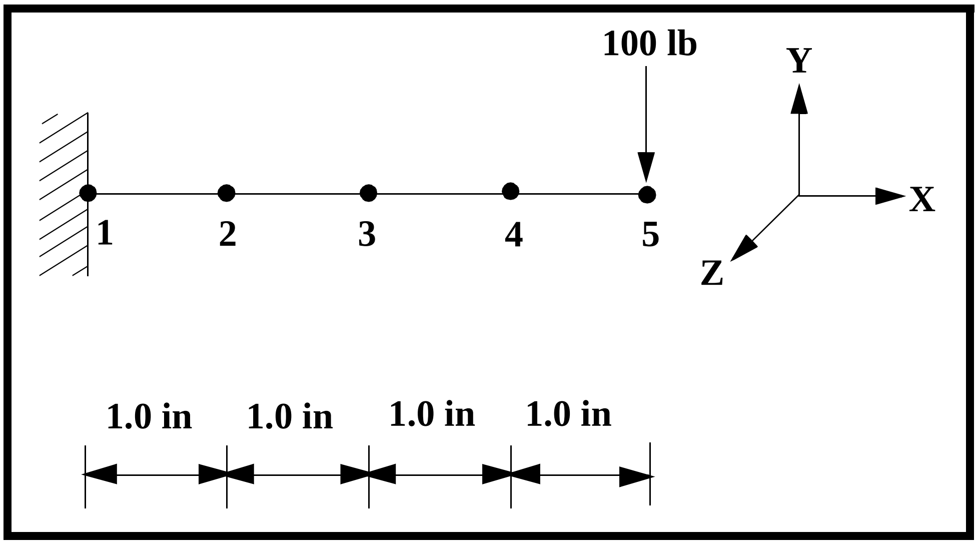
Objectives:

- Model a loaded cantilever beam with CBAR elements, including shear factors in element properties.
- Create a revised model which does not include shear factors.
- Compare both results with theory.

Model Description:

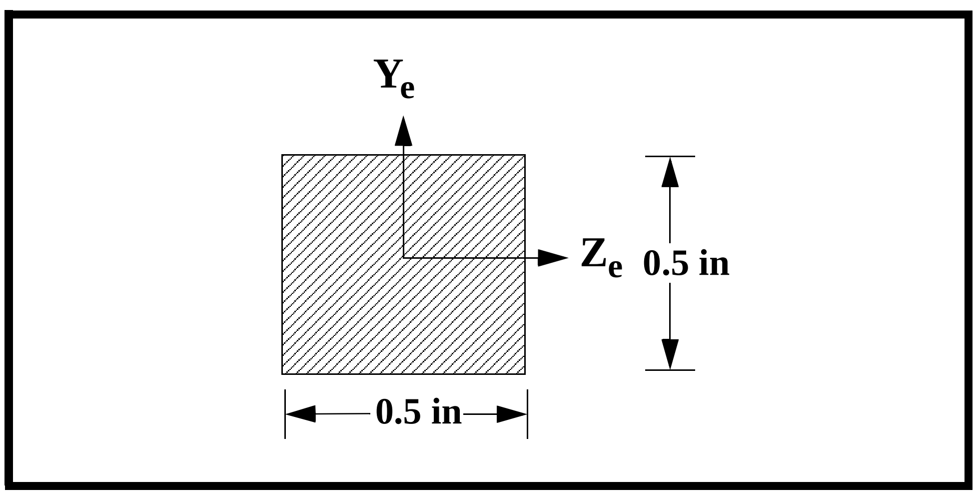
Illustrate the effect of the shear factor, K, on a cantilever beam under a transverse load.

Figure 18.1



Modeling the CBAR elements with an orientation vector of <0., 1., 0.> results in the cross section:

Figure 18.2



Since the cross-section is square, **K = 5/6 = 0.8333**.

Table 18.1 - Model Properties

Material:	**Aluminum**
Youngs Modulus:	**10E+06 psi**
Poisson's Ratio:	**0.3**
Cross-Sectional Area:	**0.25 in^2**
I_{aa}:	**0.0052 in^4**
I_{bb}:	**0.0052 in^4**
J:	**0.0088**

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to create a new model.

   Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

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

Open Model File:	**New Model**

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

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

   Model/Material...

Title:	**mat_1**
Youngs Modulus:	**10E6**
Poisson's Ratio:	**0.3**

   OK

   Cancel

3. Create a property called **prop_1** for the bar elements of the model.

   Model/Property...

Title:	**prop_1**
Material:	**1..mat_1**

   Elem/Property Type...

   Change the property type from plate elements (default) to bar elements.

Line Elements:	● **Bar**

   OK

Area, A:	**0.25**
Moments of Inertia, I1 or Izz:	**0.0052**

I2 or Iyy:	**0.0052**
Torsional Constant, J:	**0.0088**
Y Shear Area:	**0.20833**

OK

Cancel

4. Create the MSC.Nastran geometry for the beam.

Mesh/Between...

Property:	**1..prop_1**
Mesh Size/ #Nodes/ Dir 1:	**5**

OK

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

OK

	X:	*Y:*	*Z:*
Corner 2:	**4**	**0**	**0**

OK

Now, specify the orientation vector for the bar elements.

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

OK

To fit the display onto the screen, use the **Autoscale** feature.

View/Autoscale...   <Ctrl+A>

Turn off the workplane.

Tools/Workplane... **<F2>**

☐ **Draw Workplane**

Done

View/Regenerate... <Ctrl+G>

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**

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 select the toggle button "Fixed".

Fixed

OK

Cancel

6. Create the model loading.

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

Model/Load/Set...

Title: **load**

OK

Create the tip load.

Model/Load/Nodal...

<Select **Node 5**>

OK

Highlight **Force.**

	Force
FY ☒	**-100**

OK

Cancel

7. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Format/Type:	**1..Static**

OK

Change the directory to **C: \temp**.

File Name:	**shear**

Write

Additional Info:	☒ **Run Analysis**

OK

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

Yes

File Name:	**shear**

Save

When the MSC.Nastran manager is through running, MSC.Nastran 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 successfully, we will not bother with the details this time.

Continue

8. List the results of the analysis.

To list the results, select the following:

List/Output/Unformatted...

ID:	**1**

OK

Unselect <**All Vectors, or**> and instead select **T2 Translation**.

☐ **All Vectors, or**

3..T2 Translation

OK

NOTE: You may want to expand the message box in order to view the results.

Answer the following questions using the results. The answers are listed at the end of the exercise.

What is the tip deflection at Node 5?
Y Disp @ Node 5 = _______

Compare this value to the theoretical value.

9. Redo this exercise without the Shear Factor.

Modify the existing model and take out the Y Shear Area by following the following steps.

Modify/Edit/Property...

ID:	**1**

OK

Y Shear Area: **0**

OK

Do step 7 again and rerun the analysis. But remember to write to a new file name in the **C:\temp** directory to differentiate the two cases.

File Name: **no_shear**

To list the new results, follow step 8 but select ID #2. (should be the default).

ID: **2**

Without the shear factor, what is the tip deflection at Node 5?
Y Disp @ Node 5 = _______

Compare this value to the model with shear factor and the theoretical values.

This concludes this exercise.

File/Save

File/Exit

	Tip Deflection
With Shear Factor:	**-0.041525**
Without Shear Factor:	**-0.041026**

Results:

The shear factors Ky and Kz define the shear displacements V_{ys} and V_{zs} in the element coordinate system. The total displacement of the reference axis is

$$V_y = V_{yb} + V_{ys}$$

where V_{yb} = displacement due to bending.

From hand calculations, the predicted maximum displacement due to bending is:

$$\frac{PL^3}{3EI} = \frac{100(4)^3}{3(10.E6)(0.0052)} = 0.04102564 \text{ in}$$

The maximum displacement due to shear is:

$$\frac{VL}{KAG} = \frac{100(4)}{0.833(0.25)(3.846E6)} = 0.000499 \text{ in}$$

Total displacement = 0.04102564 + 0.000499 = 0.041525 in

The following represent first, the beam modeled with shear factors, and second, the beam modeled without shear factors.

	Tip Deflection
Model w/ shear factors	-0.04153
Model w/o shear factors	-0.04103
Theory	-0.04153