

ANSYS - 3D Curved Beam step 9

Problem Specification

1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. **Validate the results**

Step 9: Validate the results

Simple Checks

Does the deformed shape look reasonable and agree with the applied BCs? We checked this in step 8.

Do the reactions at the supports balance the applied forces for static equilibrium? To check this, select

Main Menu > General Postproc > List Results > Reaction Solu

Select **All struc forc F** for **Item to be listed** and click **OK**.

There are no applied forces in this problem, so the total reaction force should be zero for equilibrium. The total reaction force in the radial direction (F_X) is 4.6 N which is close to zero. We can lower it even further by refining our estimate of r_c . The total reaction forces F_Y in the circumferential direction and F_Z in the axial direction are small but not zero. This is possible because F_X is small but not zero. So the structure is in equilibrium to a reasonable degree of approximation.

Refine Mesh

Let's repeat the calculations on a mesh with twice the no. of mesh divisions in the radial and axial directions while retaining a single division on AC and BD. We need to reset $NDIV$ and $SPACE$ on the following lines:

Line no.	$NDIV$	$SPACE$
L2,L4,L8, L12	10	1
L7,L9,L11	16	0.3
L5	16	1/0.3

Let's use a different jobname for the refined mesh case. Change jobname: **Utility Menu > File > Change Jobname**

Enter cbeam2 as the **New jobname** and click **OK**.

Main Menu > Preprocessor > Meshing > MeshTool

Delete the current mesh: Select **clear** under **Mesh:** and **Pick All** in the *pick* menu. The mesh is deleted.

Utility Menu > Plot > Lines

Under **Size Controls** and **Lines**, click **Set**. This brings up a pick menu.

Pick lines L2,L4,L8, and L12 and click **OK** in the pick menu. Enter 10 for **No. of element divisions**, leave **Spacing Ratio** blank and click **Apply**.

Pick lines L7,L9, and L11 in the *Graphics* window and click **OK** in the pick menu. Enter 16 for **No. of element divisions**, 0.3 for **Spacing Ratio** and click **Apply**.

Pick line L5 in the *Graphics* window and click **OK** in the pick menu. Enter 16 for **No. of element divisions**, 1/0.3 for **Spacing Ratio** and click **OK**.

Select Volumes for **Mesh:** and Hex for **Shapes:**, then click **Mesh**.

Since we applied the BCs to the finite-element model rather than the solid geometry model, the BCs were deleted along with the mesh. So we have to reapply the BCs again. Repeat step6 to reapply the BCs. It might feel like a chore but consider it as good practice. Since the *vface2* table for applying the BC on face 2 already exists, you need not recreate the function or the table.

Save your work: **Toolbar > SAVE_DB**

This will create the file *cbeam2.db* in your working directory.

After reapplying the BCs, solve the problem as in step7.

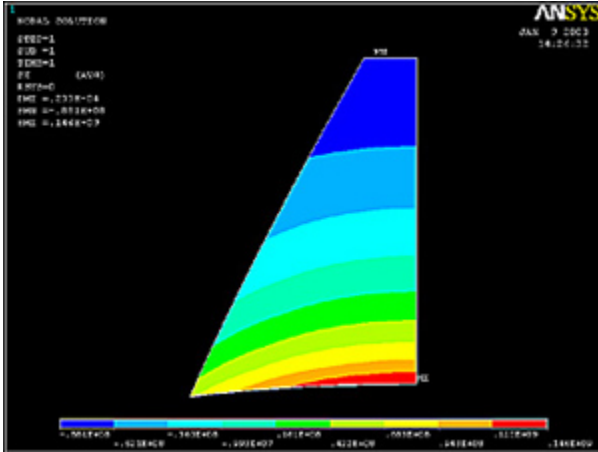
Plot Circumferential Stress

Display the stress distribution over face 1:

Utility Menu > PlotCtrls > Pan,Zoom,Rotate > Right

Main Menu > General Postproc > Plot results > Contour Plot > Nodal Solu

Select **Stress** from the left list, **Y-direction SY** from the right list and click **OK**.



(Click Picture for Larger Image)

Compare this result with the plot obtained on the coarser mesh. The results on the two meshes compare well indicating that the coarse mesh provides good resolution. Similarly, compare the von Mises stress results on the two meshes.

Exit ANSYS

Utility Menu > File > Exit

Select **Save Everything** and click **OK**.

Reference

Cook, R.D., Malkus, D.S., Plesha, M.E., and Witt, R.J., *Concepts and Applications of Finite Element Analysis*, Fourth Edition, John Wiley and Sons, Inc., 2002.

[Back to Problem Specification](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)