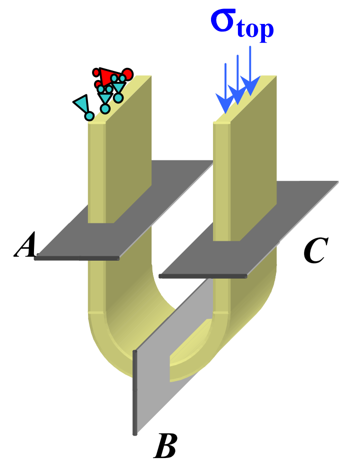
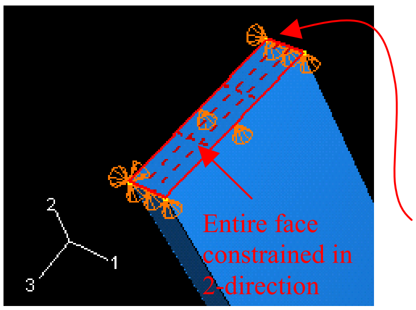
2.31 Assignment 10

Due Wed, Nov 14 at 9:30 am

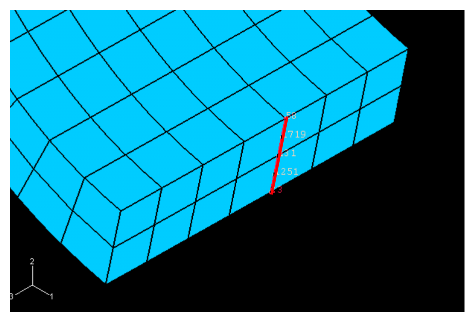


We are going to model the same channel considered for Assignment 9, but this week we will use a shell formulation.

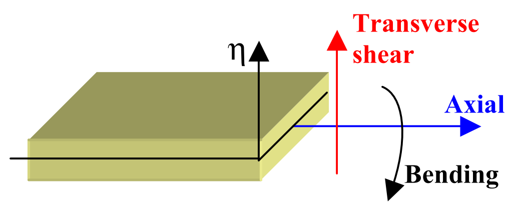


Geometry and loading conditions are exactly the same, but we have to change the boundary conditions on the top left face to eliminate the free rotation around the 2-axis: simply add a constraint in the 1-direction to the back corner on the top face. We want to compare FE predictions for continuum elements and for shell elements, as well as the relative computational and modeling costs.

1) Continuum Elements

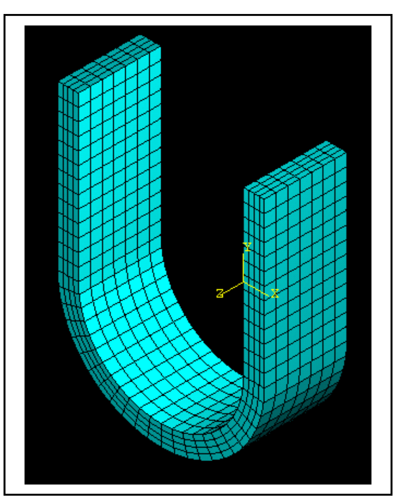


Repeat the simulation done for Assignment 9. From the .dat file, obtain the CPU time it took to run the model. Also, using the path tool, obtain and plot profiles of axial stress and transverse shear stress across the thickness for sections A, B, and C. Note that these are quadratic elements, so there are 5 nodes through the thickness.



When you studied beam bending, you learned that the profile of transverse shear stress varies quadratically through the thickness (zero at the surfaces, max at the midplane).

Why is the FE profile of shear stress at section B varying linearly within each of the two elements?



Refine the mesh by changing the local seed on the thickness edges so as to have four elements through the thickness. Rerun the job. Record the CPU time, and obtain new profiles for the axial and transverse shear stresses across the thickness for section B only (9 nodes in the path). Does the shear profile seem more like what you expected? Why?

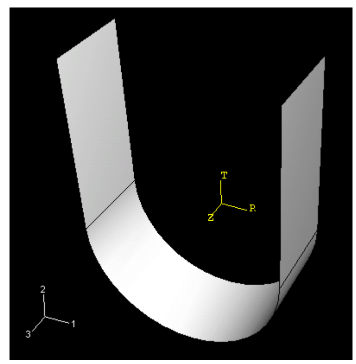
2) <u>Shell Elements</u>

Create an FE model of the component using shell elements.

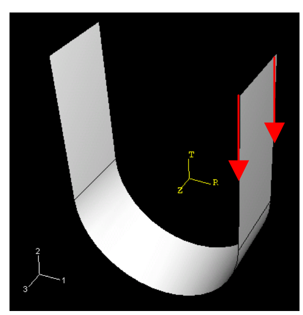
Things to keep in mind as you set up the model:

PART: You want to create a 3D Deformable <u>Shell</u> Part using Extrusion (Approx size :20). Create the U profile of the shell keeping in mind that you are creating the shell midplane (the radius is 2+0.25). Your life will be easier later on if you sketch your part so as to have the center of curvature of the bottom round part at (0,0).

PROPERTY : In material properties you have to input only the Mechanical props (E, ν). In section property you want to create a *<u>shell</u>* *<u>homogeneous</u>* section of thickness 0.5. Because it is a linear analysis, you can perform all the section <u>integration before the analysis</u>→ click the corresponding diamond. Assign the section to the entire shell. Now you want to assign material orientations which are consistent along the shell. The procedure of assigning material orientations is outlined in Handout 11. In this case, it is easy to obtain a consistent local 1-axis if you first define a cylindrical datum coordinate system: use the Tool→Datum→CSYS→default→cylindrical tool to create a cylindrical coordinate system. The datum CSYS will be automatically positioned at the origin, with the z-axis at the center of the round section of the shell (this is why you wanted the center of curvature at (0,0). Now you can use the Assign→material orientation tool: select the entire part, and choose the newly created CSYS. Select the radial direction (Axis-1) as the direction that defines the approximate shell normal (with zero additional rotation), and click OK to confirm the change. Use the Query→Material Orientation tool to check if now you have a consistent 1-axis. Print the plot and attach it to your assignment.



STEP : Choose static, linear perturbation. Use the Field Output Request Manager to edit the requests and change the default for the quantities written to the .odb file: add *Section Forces and Moments* to the required forces/reactions output quantities, and *transverse shear stress* to the desired stress output.



LOAD : You will have to apply the vertical force on the top left edge as a series of concentrated forces at points along the edge. For a first simple model, you can try splitting the total load evenly between the two end points. Apply the appropriate boundary conditions to the top left edge. Be careful to be consistent with what you had in the continuum model. Keep in mind that by default the translation and rotations that you are constraining are interpreted in the GLOBAL coordinate system (e.g., UR3 means rotation about the global 3-axis). You can change the CSYS for the boundary conditions if you need to, but in this case it will not be necessary.

MESH : Remember to partition the shell by the vertical symmetry plane. Follow what you get by default: (0.4 seed, SR4 elements).Mesh the whole thing.

Submit the job: note the CPU time from the .dat file.
Look at the results in the VISUALIZATION module of ABAQUS/CAE:

1. Is the mesh refined enough to avoid creating folds in the U sections? How can you tell?
2. Plot the normals on the undeformed shape model. Which surface is going to be SPOS? Which surface is going to be SNEG? (Yes, you have to read Handout 11 ☺)
3. Plot non-zero Section Forces and Moments. Compare and contrast with your calculations from Assignment 9. Use the Query tool to get a better estimate of the values at the three sections. What are the effects of using a very drastic approximation for the loading conditions?
4. Plot the stress contours. Start with S11. For what section point (i.e., location η through the thickness → see handout) is S11 plotted by default? Use the Query tool to obtain the value of S11 at sections A,B,C. Are these values consistent with the continuum element model? To look at S11 at other locations (η) through the thickness you have to change the section point: in the Field Output box, click on the Section Points… button. In the box that comes up, highlight the only category available in this model (shell general….) and in the Available Section Points in Cross-section list, you will see that you have only two points available (on the top and bottom of the shell). <u>This is the default when the through thickness integration is done before the analysis</u> (you selected this option in the Property module). In other words, with the choices we have made so far in setting up the model, we can retrieve stresses only on the top and bottom surface of the shell. Click on SPOS to obtain the S11 contours on the other surface. Which surface is this? Are the results at A,B, C consistent with the continuum elements data? Repeat the same thing for the transverse shear stress (TSHR13). What value do you get? Why?
5. Clearly you need additional through-thickness data if you want to look at transverse shear effects. In order to get values of stress at all 5 section points, you need to change two things in the model: 1) In the Property module use the section manager to edit the properties of your shell section: click on Section Integration: ✦During analysis.
2) You have to require the additional output at the other section points→ in the Step module, use the Field Output Request Manager to edit the requests: right underneath the variable list, change the Output at shell, beam, and layered section points to ✦Specify: 1,2,3,4,5. This saves stress values at all 5 section points to the .odb file.
Rerun the job and look at the results: obtain values of S11 and transverse shear stress at all the points through the shell thickness for sections A,B,C, and compare them with continuum elements results. Comment on differences & computer time saved.