

2.31 Project 5 Mixing Element Types

Team 1: Costanzo, Goldenshteyn, Peoples
Team 2: Cho, Chau

Due Wed, Nov 14 at 9:30 am

In Assignment 10, you compared the CPU times for the continuum element model and the shell model and saw that using shells can be very effective. Yet, in order to use shells you must have “global” dimensions much larger than the thickness dimension. This means that if the geometry of our U channel is complicated by the profile details shown in the figure, there is no way that we can hope to capture the stress concentration at the fillet radius r (where $r=t$) with a shell model. In general in cases like this you have two alternatives:

- 1) With the “brute force” approach you model the whole structure using continuum elements. This choice makes modeling straightforward, but it can be quite costly.
- 2) The “thrifty” approach requires you to do more work in the modeling stage, but it is much more cost-effective, and it requires the combination of different element types.

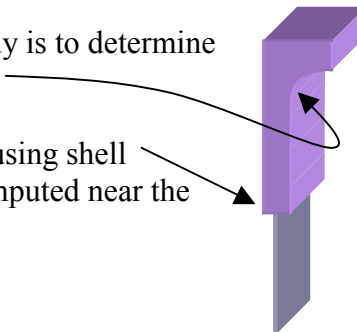
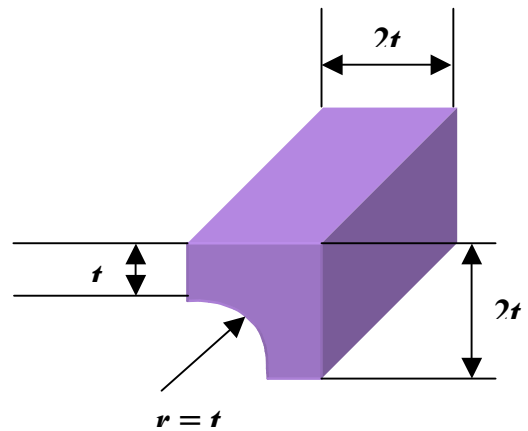
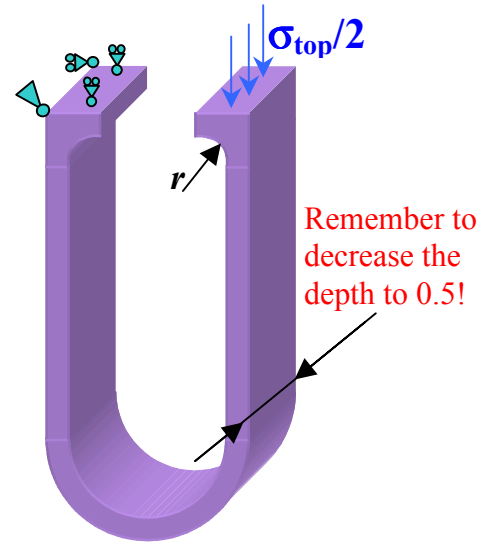
Guess which approach we are going to use.

The geometry of the U channel is the same as in Assignment 10, with the addition of the two corner profiles on the top. Since the area on the top right is now doubled, I have cut the pressure in half, to maintain the same load.

I am attaching at the end of this document a copy of the section of the Cook-book, where issues related to connecting dissimilar elements are discussed.

I would like you to try to model the structure using both continuum elements and shell elements, and imposing the appropriate connection between the two. The objective of the study is to determine the (max) stress at the inner radius of the top left corner section.

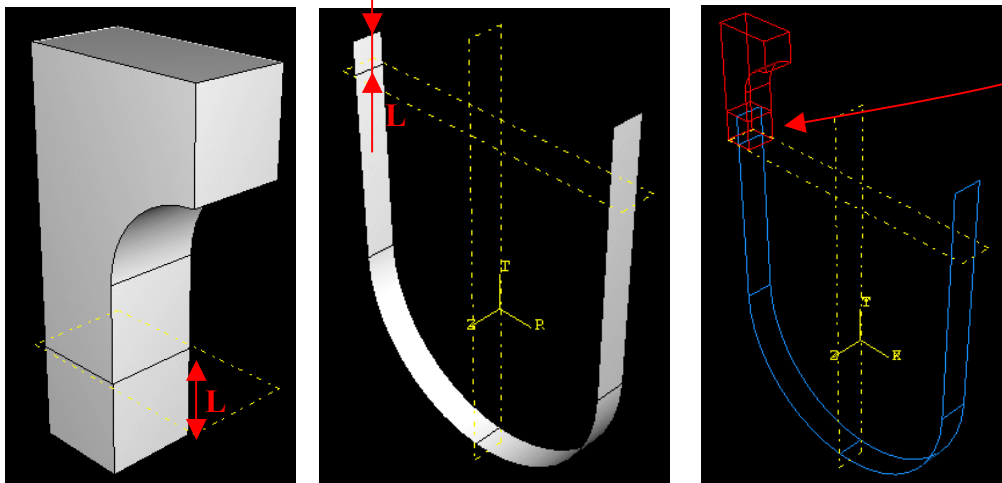
As you decide where to stop using continuum elements and start using shell elements, keep in mind that the stresses will not be accurately computed near the connection.



The easiest way to tie the two kinds of elements together is to use an option in ABAQUS that ties two surfaces together. Essentially we will be using the approach in Figure 8.5-2 (b) of Cook, where nodes are shared by the continuum elements and the shell elements, but the connection will be made between surfaces rather than between nodes.

As we do not expect any problems at the top right corner of the channel, we need to use continuum elements only around the top left corner

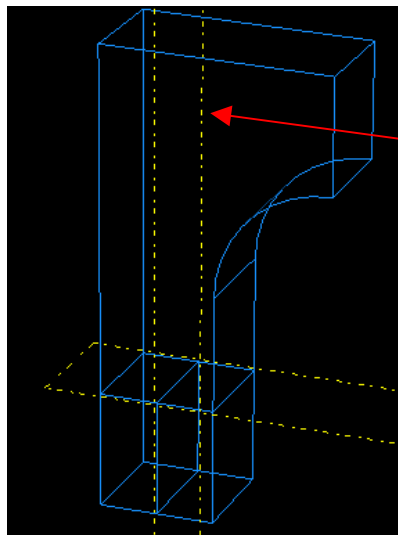
The easiest way to go about this, is to create two parts (U-shells and 3D corner) and then assemble them. Allow for an overlap length L , between the shell and the continuum part.



Partition the overlap regions from the rest of the part.

Your life will be easier if you create the parts in space so that they are correctly positioned with respect to each other when you instance them in the assembly.

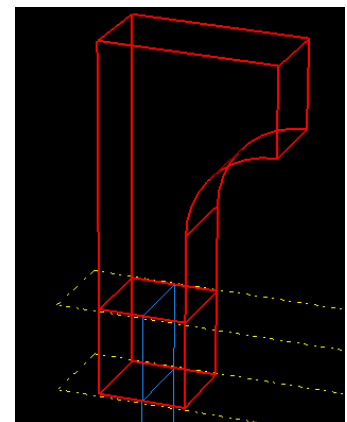
Now you want to tie the two parts so they deform together when loaded. In the overlap region, you want the shell part to stay tied to the midplane of the continuum part.



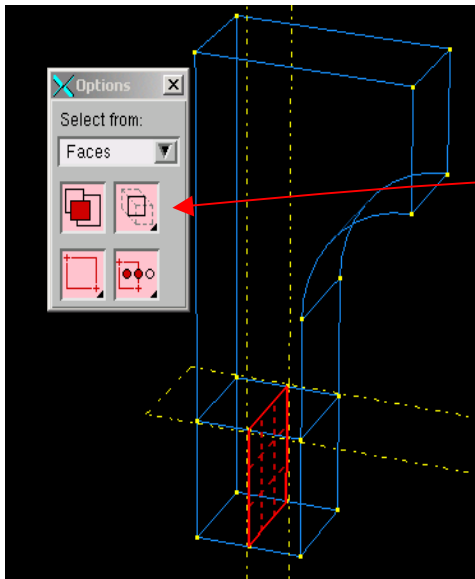
So the first thing to do is to create the midplane of the continuum part by partitioning the overlap region of the part with a datum plane parallel to the YZ reference plane.

As you perform this task, as well as many of the following tasks, it is very useful to be able to display only one part of the assembly (either the shells or the corner). You can find this option in the **View→Assembly Display Options→Instance** tab where you can click individual parts on or off for the current display (you do not see them but they are not deleted from the model, and you can click them back on when you are done).

The tool that allows you to tie the shells and the continuum elements is in the Interaction module.



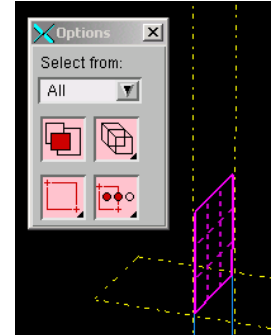
Use the Constraint→Create→Tie option and you will be prompted first for the “master surface”. To select the inner surface of the continuum elements you will have to use the



Select the geometry for the set

Select from: Faces and chose Select Interior Entities.

Then it will prompt you for the slave surface: make the shells visible, and click on the overlap region of the shells. (when prompted, use either magenta or yellow, it does not matter in this case); accept all the defaults for the constraints. You are done. Essentially this constraint automatically forces these two surfaces to stay tied to each other when the structure deforms.



I guess you can see there is a last little problem to take care of before we run this analysis. In the overlap region you have both the continuum elements and the shell elements so you have a doubled stiffness. In order to avoid this, you can simply create an additional material which is half as stiff as the real material, and create corresponding shell and continuum section properties to assign to the continuum elements and to the shell elements in the overlap region. Everything else should be fairly straightforward.

Use S4R shells for the shell elements and the C3D20 quadratic bricks for the continuum elements.

As a check of the mixed elements model, construct a model of the structure with only continuum elements. Compare the results, the CPU time, and the time it took you to setup the model. If you need help, stop by during office hours, or make an appointment to see me (e-mail).

For the presentations, I would like to have:

Team 2 (Cho, Chau) presenting and discussing Assignment 10 and briefly discussing the issues related to connecting continuum elements and structural elements.

Team 1 (Costanzo, Goldenshteyn, Peoples) giving an overview of the mixed element modeling techniques and presenting the results of the study.

Another possible application is shown in Fig. 8.4-1b. Imagine that plane bar B is free to rotate without friction on pin A , which for the time being is to be regarded as rigid. Pin A need not be meshed. D.o.f. of nodes of bar B that lie on the circular boundary can be transformed to the polar coordinate system shown. Then only radial displacements at these nodes are set to zero.

8.5 CONNECTING DISSIMILAR ELEMENTS.

RIGID ELEMENTS

By "dissimilar elements" we mean elements whose d.o.f. are of different type and/or of different location. As examples, we may wish to connect a beam element (having rotational d.o.f.) to a plane element (having translational d.o.f. only), or we may wish to connect two FE meshes whose nodal locations are such that nodes of one mesh cannot be superposed on nodes of the other. The latter situation also arises in *submodeling* and *substructuring* (Sections 10.10 and 10.11, where references are cited).

A way of dealing with these situations is to impose constraints that force d.o.f. of mating elements or meshes to have a prescribed relation to one another. Constraints of this kind can be imposed on a stiffness matrix by the transformation $[k] = [T]^T[k'][T]$. In establishing transformation matrix $[T]$, it is convenient to think in terms of a rigid link or a rigid element that contains the d.o.f. to be related. (Constraints are discussed in a more general way in Chapter 13.)

Simple Examples. In Fig. 8.5-1a, one end of a plane beam element that has both bending and axial stiffness is to be attached at an arbitrary location on one side of a four-node plane quadrilateral that has no rotational d.o.f. The beam element has the stiffness relation $[k']\{d'\} = \{r'\}$, where

$$\{d'\} = [u_5 \quad v_5 \quad \theta_{z5} \quad u_6 \quad v_6 \quad \theta_{z6}]^T \quad (8.5-1)$$

New d.o.f. of the beam element are to be

$$\{d\} = [u_2 \quad v_2 \quad u_3 \quad v_3 \quad u_6 \quad v_6 \quad \theta_{z6}]^T \quad (8.5-2)$$

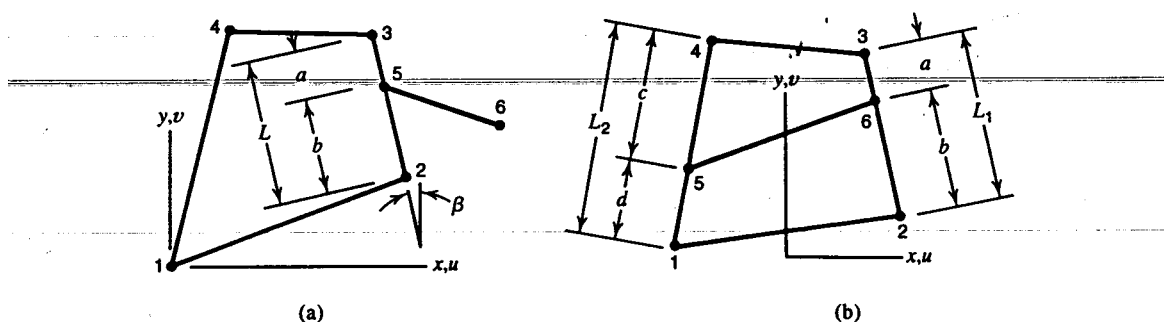


Figure 8.5-1. (a) Plane beam element 5-6 is connected to four-node plane element 1-2-3-4. (b) Two-node bar element 5-6 is connected to four-node plane element 1-2-3-4.

We now require that translation of node 5 be linearly interpolated along side 2-3 from translational d.o.f. at nodes 2 and 3. Rotation at node 5 is defined as the difference between the side-normal displacements at nodes 2 and 3, divided by the distance L between them. Thus, in the relation $\{d'\} = [T]\{d\}$, transformation matrix $[T]$ is

$$[T]_{6 \times 7} = \begin{bmatrix} T_5 & \mathbf{0} \\ \mathbf{0} & \mathbf{I} \end{bmatrix} \quad \text{where} \quad [T_5] = \frac{1}{L} \begin{bmatrix} a & 0 & b & 0 \\ 0 & a & 0 & b \\ \cos \beta & \sin \beta & -\cos \beta & -\sin \beta \end{bmatrix} \quad (8.5-3)$$

and $[\mathbf{I}]$ is the unit matrix $\begin{bmatrix} 1 & 1 & 1 \end{bmatrix}$. The transformed beam element matrix is 7 by 7, and operates on d.o.f. of nodes 2, 3, and 6. After solution for nodal d.o.f., original d.o.f. of the beam element can be recovered by the operation $\{d'\} = [T]\{d\}$, for use in stress calculation.

A second simple example is that of Fig. 8.5-1b. A two-force member, such as a portion of a reinforcing bar in concrete, is to be connected to points arbitrarily located on opposite sides of a four-node plane element. As originally formulated, the 2 by 2 matrix $[k']$ of the bar element operates on d.o.f. $u_5, v_5, u_6,$ and v_6 . By transformation, we seek the 8 by 8 matrix $[k] = [T]^T[k'][T]$ of the bar element, which operates on d.o.f. u_i and v_i of the four-node element, where $i = 1, 2, 3, 4$. Matrix $[T]$ is 4 by 8, and contains terms like those seen in the first two rows of $[T_5]$ in Eq. 8.5-3. References that deal with 3D situations and full bonding between concrete and reinforcement include [8.1].

The examples of Fig. 8.5-1 invoke constraint transformation as a way of connecting elements whose nodes do not coincide. Transformation may not be needed if nodes coincide, even if elements to be connected do not have the same set of nodal d.o.f. Consider Fig. 8.5-2a. The connection at A is a hinge because the beam has a rotational d.o.f. but plane elements do not (unless the plane elements happen to have drilling d.o.f., but then such a connection is not recommended). The hinge is avoided by the ad hoc arrangement in Fig. 8.5-2b, where the beam has been extended into the plane body by adding two beam elements AB and BC . Adding only one beam element, AB , is also plausible [8.2]. Translational d.o.f. are shared by plane elements and beam elements at nodes A, B , and C . Rotational d.o.f. at these nodes are associated with only the beam elements. One should not expect that stresses will be accurately computed near node A .

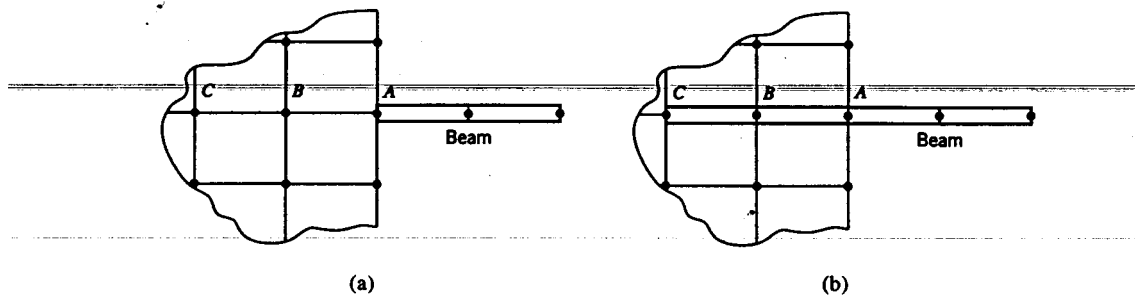


Figure 8.5-2. Connecting a 2D beam element to a mesh of four-node plane elements. (a) Hinge mechanism. No moment is transferred at A . (b) Moment is transferred between the beam and the plane mesh.