LABORATORY # 4: INTRODUCTION TO 3D MODELING
September 22\textsuperscript{nd}, 2006

Let us recapitulate the labs this semester: The first sessions we worked with ‘1-dimension’ line elements. First we created these elements directly by defining nodes and connecting these nodes with elements. So in this sense we were working only with mesh entities (nodes, elements). Progressing, we created geometrical entities (keypoints, lines), ‘mapped’ the lines into a specified number of divisions, and meshed the lines according to our mapping.

In this session we will get started on 3-D finite element modeling. Three-dimensional modeling is not as difficult as you may think. In the case we will consider this week, it is as simple as creating a 2D mesh, and “sweeping” that mesh along a prescribed path, with a specified number of divisions along the way.

HOMEWORK #5: Due on September 29\textsuperscript{th}, 2006

3D Model of a cantilever beam:

Let’s create a 3D model of the steel beam from Homework 3.
Same geometry and loads: $W = 500 \text{ lb/ft, } b = 6 \text{ in, } h = 12 \text{ in, } L = 120 \text{ in}$.

1) Set your jobname and working directory.

2) Construct 3D Volume of Beam Geometry and mesh one cross sectional area.
   2.1) Create a rectangular volume according to the dimensions provided.

   2.2) Turn area numbers on to distinguish the areas associated with the volume. Plot areas.

   2.3) Define 2D element, Quad 4-node (PLANE42). Do not use plane stress w/ thickness. The 2D element is used only for defining the cross-section-mesh, and will be deleted when the 3D mesh is created.

   2.4) Mesh one of the areas through the cross section. (This 2D mesh will be used to create the 3D mesh)

   2.5) Assign divisions as you have done in previous labs to the lines forming the area you have chosen.

   Use a mesh density of 1 element for every 2 inches.

   2.6) Mesh the cross section area that you assigned divisions.
3) **Sweep 2D Mesh to create 3D Mesh.**

3.1) Plot areas.

3.2) Define 3D element, Brick 8-node (SOLID45). No real constants are necessary. *Note:* When sweeping a mesh, you must select compatible 2D and 3D elements. Sweeping a 2D triangle into a cube shaped element will not work. Likewise, sweeping a Quad 8-node (2D) into a Brick 8-node (3D) would not work, because the Quad 8-node has mid-side nodes while the Brick 8-node only has nodes at the vertices.

3.3) Sweep the 2D mesh to create the 3D mesh.

3.4) Meshing>Mesh>Volume Sweep>Sweep Opts. Deselect the auto select source and target areas. Set the number of divisions equal to 60. This will create a division every 2 inches, since the total length of the sweep is 120 inches. This is consistent with the number of divisions assigned on the 2D mesh and will give your elements an aspect ratio of 1:1:1. Leave the spacing ratio set to 0.

3.5) Meshing>Mesh>Volume Sweep>Sweep. First you must select the volume to sweep. So click anywhere on your volume. Hit OK. Now you must select the area to sweep. Select the area that you created the 2D mesh on. Hit OK. Now select the target area. This is the area at the opposite end of the beam from the area you just selected. Hit OK. You should now have a 3D mesh.

*Note:* The 2D elements from step 2.3) no longer exist.

4) **Apply Pressure to Element Faces**

4.1) Choose the apply pressure on elements from the loading menu. Convert the distributed load to $force/length^2$ because you are applying the load on faces of the elements, which have an associated area.

4.2) Select the elements on the upper face of the beam. If you are looking at the side of the beam, you may choose the box option on the pop-up window, and drag a box around the elements you want to select. The select tool will select all elements in the box and through the thickness of the model.

4.3) When you select the elements and continue, there are two options in the pop-up window you need to enter. One is to specify which face of the element to apply the pressure on (load key); the other is the value of pressure. A careful inspection of the coordinate system in conjunction with the ANSYS Element Reference will inform you that the correct element face is “3”, and that for a downward deflection, your pressure value must be entered negative. Plot the pressure as arrows to assure you have applied the pressure correctly. (PlotCtrls>Symbols).

5) **Apply Cantilever Boundary Conditions**

5.1) Choose the apply displacements on nodes from the menu.

5.2) Select the nodes on the left face of the model as described above for applying the pressure on elements. Be sure to only select the nodes on the outermost edge.

5.3) Constrain the nodes in all directions.

6) **Define Material Properties** (for steel, $E = 30e6$ psi, $\nu = 0.3$)

7) **Solve**

8) **Plot the deflection; check that it matches your homework results.** The maximum deflection at free end should be on the order of 0.042 in.

9) **Plot the von Mises stress.**

Turn your report following the format explained before. Include comments about your results.