

This Lab deals with the following:

1. *Importing a solid model (iges file) into ANSYS*
2. *Creating and meshing midplane surfaces*
3. *Using MPC's (multipoint constraints) to transmit forces in a joint*

We will be working with the structure shown below. The model consists of 3 plates that are pinned to ground at points A, B, and D, and which are pinned together at point C. The structure supports a load of 40,000 on the plate ABC that is loaded evenly over an area of 20 x 10 (the thickness). We will be using symmetry about the XY plane to reduce the problem size.

Material Properties: $E=200000$; $\nu=0.3$

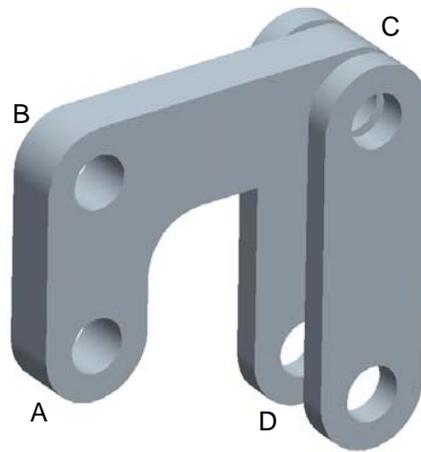


Figure 1. Isometric View of the Plate Structure

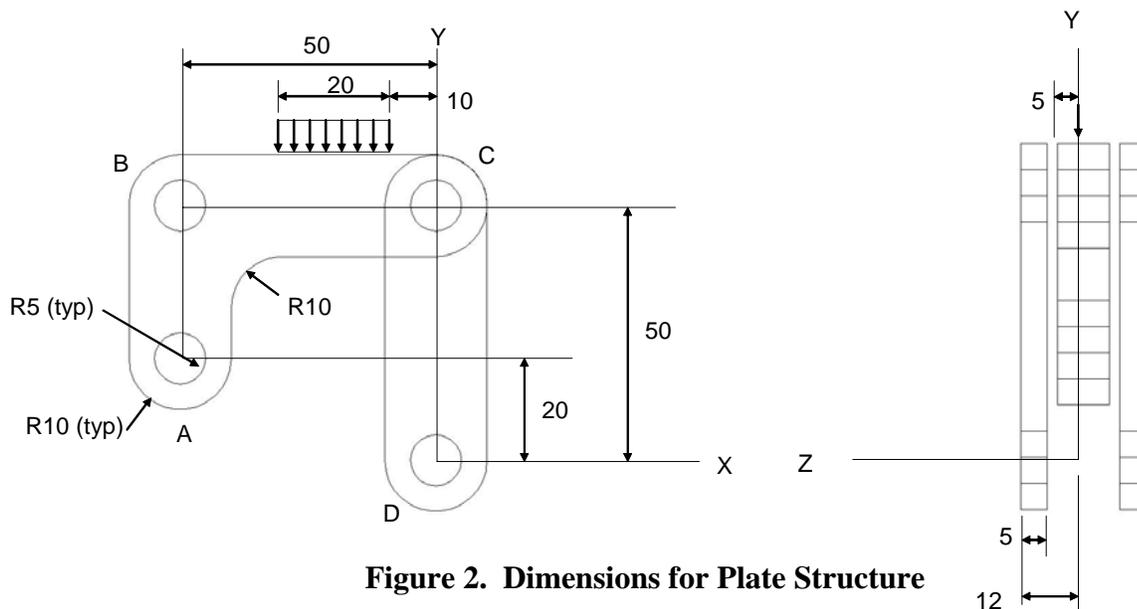


Figure 2. Dimensions for Plate Structure

Step 1: Import the model into Ansys

Download the iges file of the part shown above from the course website and save in your CADE directory (home directory or other). Start ANSYS and then do the following:

File→*Import*→*IGES*→(window)leave the default settings and select OK→(window) browse to find the iges part where you saved it (the part is called “lab_part.iges”) and open it into ANSYS. Note you can use the mouse buttons to dynamically orient the model if you pick the upper right view menu button (hold cursor over button and it should say “Dynamic Model Mode”). Also, use Plot and PlotCtrls to display the keypoints, lines, and volumes that are automatically brought in with the iges model.

Step 2: Modify the model to make use of symmetry

Workplane→highlight the button “Display Working Plane”

Workplane→*WP Setting...* →(window) click Grid and Triad. You may want to adjust the size, minimum, and maximum values to 5, 20, and 20, respectively, to help display the working plane→click OK.

Preprocessor→*Modeling*→*Operate* →*Booleans*→*Divide* →*Volume by wrkplane*→(window) pick the center plate volume→OK→Partitions the volume down the center into 2 volumes.

Preprocessor→*Modeling*→*Delete* →*Volume and below*→ (window) pick the volumes forward of the XY plane (i.e. the front plate and the front half of the center plate), then hit OK→volumes are deleted.

Step 3: Partition the rear vertical plate to create a midsurface (area)

Workplane→*offset WP by increments*→(window) type 0,0,-9.5 in the X,Y,Z offsets box and hit OK.

Preprocessor→*Modeling*→*Operate* →*Booleans*→*Divide* →*Volume by wrkplane*→(window) pick the rear vertical plate volume, then hit OK→Partitions the volume down the center into 2 volumes.

Step 4: Select and Display only the areas (midplanes) we will be concerned with

Select→*Entities*→(window) choose Areas and By Num/Pick and then hit OK→(window) pick the exposed midsurface of the center plate and the midsurface of the rear vertical plate which you just created.

Plot the areas (or simply type “aplot” in the command prompt); you should see the only the 2 midplanes we are concerned with.

Step 5: More Work Plane manipulation and partitioning of the areas for meshing and loading purposes. You may want to look at the figure below as a guide for what the final partitions should look like.

Workplane→*Align WP with* →*Global Cartesian* (you may need to replote to see effect).

Workplane→*offset WP by increments*→(window) adjust the degrees sensitivity to 90 by sliding button, then hit the +Y button once, and then hit the X- button once.

Preprocessor→*Modeling*→*Operate* →*Booleans*→*Divide* →*Area by wrkplane*→(window) pick the rear vertical plate area, then hit OK.

Workplane→*offset WP by increments*→(window) type 0,0,10 in the X,Y,Z offsets box and hit OK.

Preprocessor→*Modeling*→*Operate* →*Booleans*→*Divide* →*Area by wrkplane*→(window) pick the rear vertical plate area, then hit OK.

Workplane→*offset WP by increments*→(window) type 0,0,30 in the X,Y,Z offsets box and hit OK.

Preprocessor→*Modeling*→*Operate* →*Booleans*→*Divide* →*Area by wrkplane*→(window) pick the rear vertical plate area, then hit OK.

ETC....

Keep partitioning the areas until you have what you see below. Remember that you can always realign the work plane to the global coordinate system in case you loose track of the Work Plane orientation.

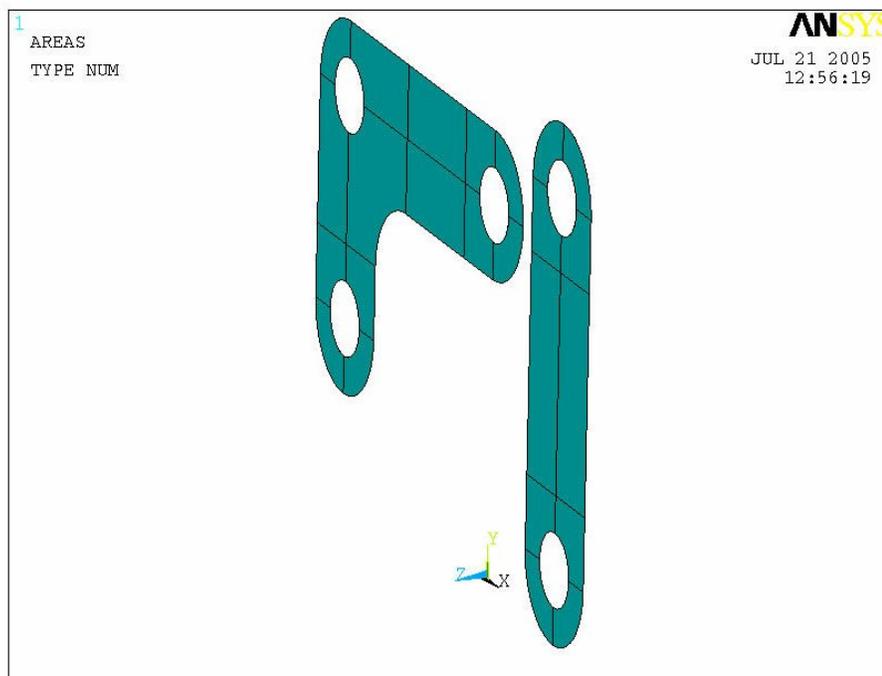


Figure 3. Partitioned Midplane Areas

Step 6: Define Material Properties

Preprocessor→*Material Properties*→*Material Models*→(window) double click *Structural/Linear/Elastic/Isotropic*→(window) input modulus and Poisson's ratio→*OK*→(close Material Model window).

Step 7: Define Element Type 1 (for shell elements)

Preprocessor→*Element Type*→*Add/Edit/Delete*→(window) *Add...* →(window) highlight *Shell63*→*OK*→*CLOSE*.

Step 8: Define Physical Property Set 1 (thickness for shell63 elements)

Preprocessor→*Real Constants*→*Add/Edit/Delete* →(window) *Add...* →(window with element Shell63 highlighted) *OK*→(window) input thickness of 5 (because we have constant thickness, only the first value is required)→*OK*→*CLOSE*.

(Note that because we are using symmetry, the middle plate will be the same thickness as the rear plate).

Step 9: Select and Display only the lines we will be concerned with

Select→*Entities*→(window) choose Lines and Attached to, then select Areas and hit *OK*→(window) pick the areas on screen or just select *Pick All* (since we previously filtered our areas to just these midsurfaces) and then hit *OK*.

Step 10: Use the Mesh Tool to set up element divisions on lines and mesh the areas

Preprocessor→*Meshing*→*Mesh Tool* →(window) under element attributes, set the button to Area, then hit *Set*→(window) select the *Pick All*→(window) shows material properties, real constant sets, and element type that we are going to be applying to the areas; hit *OK*. Now the *Mesh Tool* should still be up (if not just select it again from commands). Use the *Set* button under the *Size Controls*, *Lines* option to assign the element divisions shown in Figure 4 below:

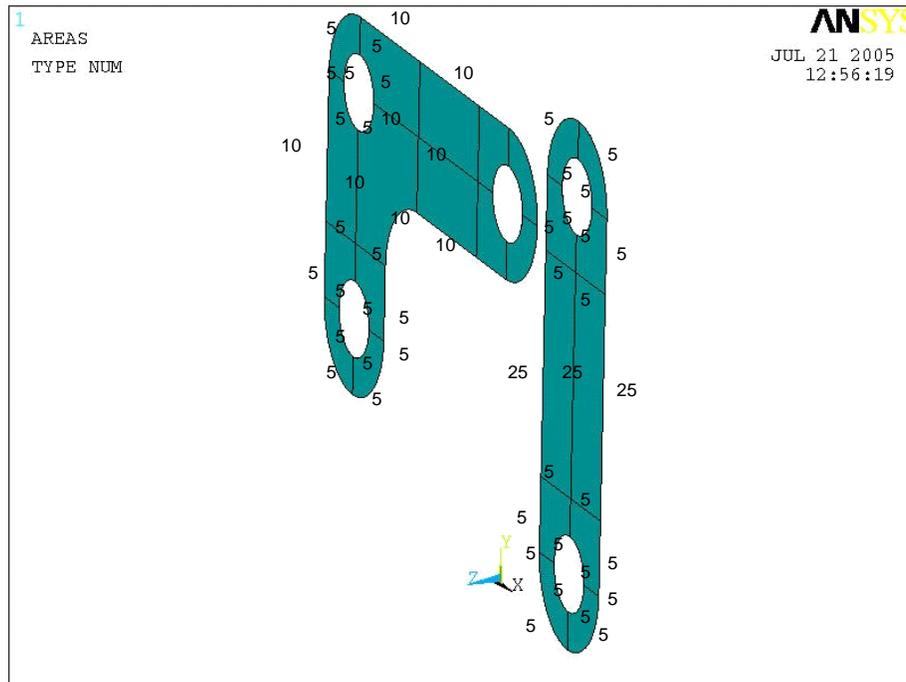


Figure 4. Divisions Applied to Lines for Mesh Control

After you have assigned all the element divisions to the lines, go back to the Mesh Tool and select that you want to mesh *Areas*, using a *Quad* shape and *Free*, then select *Mesh*→(window) hit the *Pick All* button to mesh all the areas. The resulting mesh should look something like this:

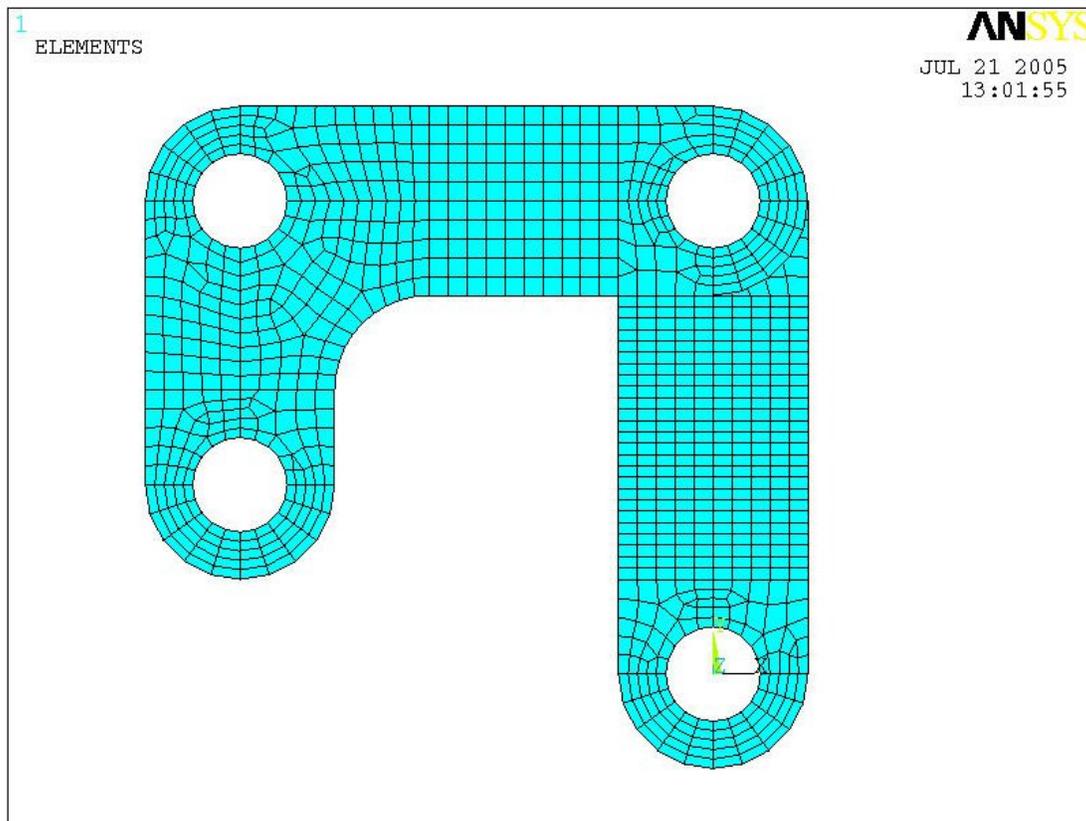


Figure 5. Mesh of the Plate Midsurfaces

Step 11: Define the MPC (Multi-Point Constraint) Element (Type 2)

Preprocessor→*Element Type*→*Add/Edit/Delete*→(window) *Add...* →(window) highlight *Constraint*, *MPC184*→*OK*→select the *Options* button→(window) set the element behavior to be a *Rigid Beam*, then hit *OK*→*CLOSE*.

(Note that there are not real constants or materials associated with an MPC)

Step 12: Define the Beam 44 Element (Type 3)

Preprocessor→*Element Type*→*Add/Edit/Delete*→(window) *Add...* →(window) highlight *Beam*, *tapered 44 (Beam44)*→*OK*→(Back to window, make sure *Beam44* is highlighted) select *Options*→(window) change “member force + moment output” to *Include Output*; select buttons *Rotx* for stiffness release at either node I or J, but not both; then select *OK*→*CLOSE*.

Step 13: Define Physical Property Set 2 (for the Beam 44)

Preprocessor→*RealConstants*→*Add/Edit/Delete* →(window) *Add...*→(window with element Type Beam4 highlighted) *OK*→(window) input properties for beam element*→*OK*→*CLOSE*.

* Use the following inputs for the beam element:

A= 76.5 (at both ends)

I=490.87 (at both ends in both directions)

J=981.75 (at both ends in both directions)

Step 14: Define Nodes at Centers of the Pin Holes

Create the following nodes with the given coordinates:

| Node | x | y | z |
|-------|-----|----|------|
| 10000 | 0 | 0 | -9.5 |
| 10001 | 0 | 50 | -9.5 |
| 10002 | 0 | 50 | 0 |
| 10003 | -50 | 20 | 0 |
| 10004 | -50 | 50 | 0 |

Step 15: Build MPC's Between the Hole Center Nodes to Hole Perimeter Nodes

Preprocessor→*Modeling*→*Create*→*Elements*→*Element Attributes*→(window) change Element No. to 2, then select *OK*.

(you may want to zoom in on a pin hole and display only the nodes before doing the next step)

Preprocessor→*Modeling*→*Create*→*Elements*→*Auto Numbered*→*Thru Nodes*→(window) select the center node first and then a node from the ring of perimeter nodes around it, then hit *Apply*. Repeat until you have created a “pinwheel” of MPC elements from the center node out to the hole perimeter nodes. Then repeat this process for the other holes in the model. After you are finished, your model should look as follows:

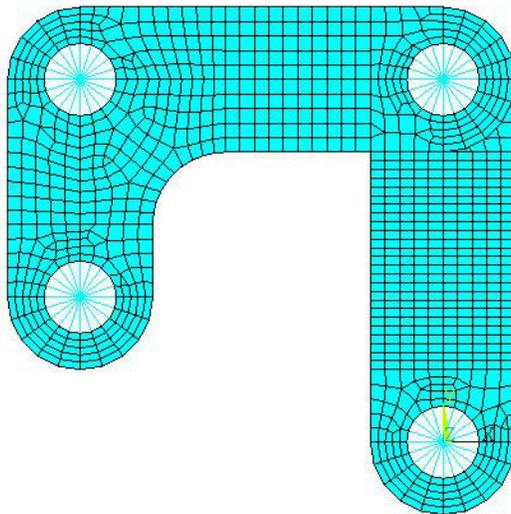


Figure 6. Mesh Including MPC's at the Holes

Step 16: Build the Beam 44 Element Between the Plates

Preprocessor → *Modeling* → *Create* → *Elements* → *Element Attributes* → (window) change Element No. to 3, then select *OK*.

Preprocessor → *Modeling* → *Create* → *Elements* → *Auto Numbered* → *Thru Nodes* → (window) select the center nodes from both plates at joint C, then select *OK*.

Step 17: Apply Bounday Conditions

Preprocessor → *Loads* → *Define Loads* → *Apply* → *Structural* → *Displacement* → *On Nodes* → (window) pick nodes 10002 and 10003, then select *OK* → (window) highlight *Ux* and *Uy*; make sure it shows *Apply As: Constant Value*; enter value as 0, select *OK*.

Preprocessor → *Loads* → *Define Loads* → *Apply* → *Structural* → *Displacement* → *On Nodes* → (window) pick node 10000, then select *OK* → (window) highlight *Ux*, *Uy* and *Uz*; make sure it shows *Apply As: Constant Value*; enter value as 0, select *OK*.

Type the following in the command prompt to apply symmetry boundary conditions to the nodes on the XY plane:

```
nsl,s,loc,z,0,0  
dsym,symm,z  
nsl,all
```

WARNING: We have constrained the model correctly, but if we tried to run it with the BC's we've applied, ANSYS would produce a fatal error resulting from Zero Pivots in the stiffness matrix (often associated with models that are not constrained properly and have rigid body modes). The reason for this is that we have currently overconstrained the MPC elements. One way to think of this is that the hole center nodes are like "master" nodes that control the "slave" nodes around the hole perimeter. Whatever dof's we constrain at the master node, we automatically constrain at the slave nodes. However, because we selected all the nodes along the XY plane to apply the symmetry BC's, we have applied constraints to both the master and slave nodes and have therefore overconstrained the MPC's. This causes problems for the MPC element and will produce errors unless we do the following:

Preprocessor → *Loads* → *Define Loads* → *Delete* → *Structural* → *Displacement* → *On Nodes* → (window) pick all the hole perimeter nodes for the three holes on the center plate, then select *OK* → (window) highlight *All DOF*, then hit *OK*.

Step 18: Apply Loading to the Model

Preprocessor → *Loads* → *Define Loads* → *Apply* → *Structural* → *Pressure* → *On Lines* → (window) pick the line representing the width of pressure area (which we created by partitioning previously), then select *OK* → (window) enter 200, then select *OK*.

Step 19: Solve

Solution → *Solve* → *Current LS* → (asks you to review summary info) select *OK* → ANSYS will begin solving the problem and will post a message “Solution is done!” when it has finished. Close message windows and go to next step.

Step 20: Inspect Results

1. Check to make sure the torque in the beam44 element is zero (i.e. select only the beam44 element and use *etable,torque,smisc,4* to get the torque in the element).
2. Determine the reactions at nodes 10000, 10003, and 10004 (i.e. where the structure is grounded) by doing the following:

First, select only nodes 10000, 10003, and 10004 (either by using *Select* → *Entities* in the GUI or using the *nselect* command directly). Then list the reactions; the GUI path is as follows:

General Postproc → *List Results* → *Reaction Solution* → (window) highlight *All Items* and then select *OK*. You should get the following

| NODE | FX | FY | FZ | MX | MY | MZ |
|-------|--------------|--------|--------------|--------------|-------------|----|
| 10000 | -0.11344E-04 | 1653.0 | 38.545 | | | |
| 10003 | 1245.1 | 868.85 | -0.15373E-11 | -0.67314E-11 | 0.53342E-11 | |
| 10004 | -1245.1 | 1478.2 | 0.12860E-12 | -0.13020E-11 | 0.29280E-11 | |

3. A contour plot of the von Mises stress should produce the following plot:

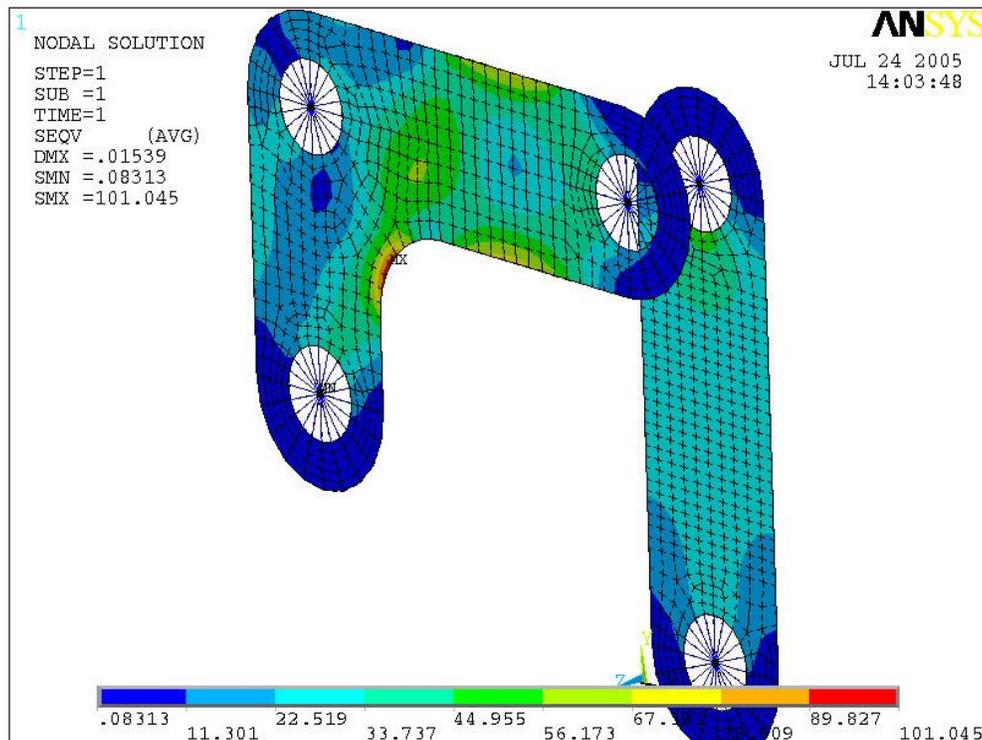


Figure 7. Contour Plot of von Mises Stress in the Plates