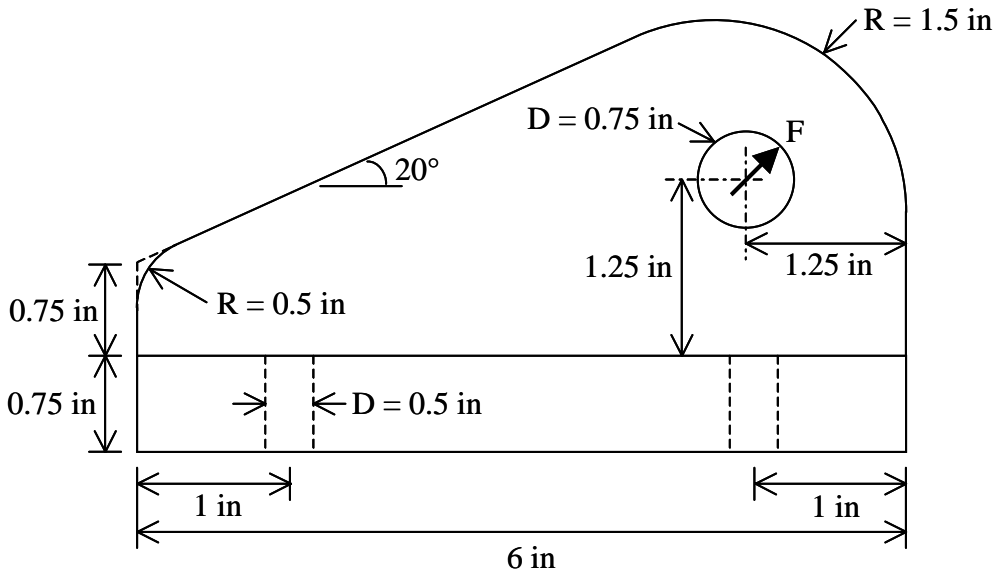
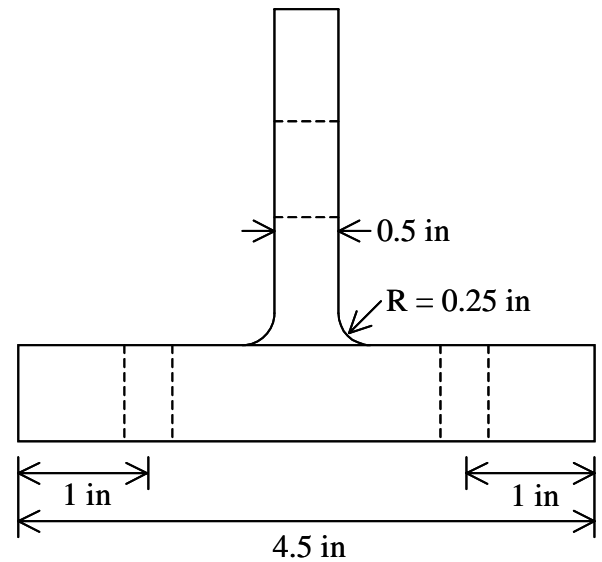


ANSYS Example: Linear Elastic Stress Analysis of a 3D Bracket

The bracket shown below is bolted to a beam at the four bolt holes located in the base. Washers with an outer diameter of 0.7 in are used with the 0.5 in bolts. The bracket is then loaded by a force F of magnitude 2500 lb, at a 45° angle to the horizontal through the hole in the flange. The bracket is made of steel ($E = 30$ Msi, $\nu = 0.3$).



In this example, ANSYS will be used to analyze the stresses and deflections in the bracket due to the applied force. Since the bracket is three-dimensional, a full 3D stress analysis will be performed using 10 node tetrahedral elements (Solid 92). The bracket, constraints, and loading are symmetric about a plane that bisects the flange; therefore, only half the bracket will be modeled. The bolts and washers will be assumed to be rigid and frictionless, so radial displacements of nodes on the surface of the bolt holes will be constrained. In addition, nodes lying along the top surface of the base within a radius of 0.35 in from the center of the bolt holes (underneath the washers) will be constrained from displacing in the vertical direction to simulate the restraint provided by the washers. The force applied to the flange will be distributed as a pressure applied to the upper right quadrant of the hole. The magnitude of the pressure is calculated by dividing the force by the projected area over which it acts (bearing area). Since the bearing area only includes one quadrant of the bolt hole in this example, the pressure is



$$p = \frac{F}{(0.7071d)(t)} = \frac{2500}{(0.7071)(0.75)(0.5)} = 9,428 \text{ psi}$$

ANSYS Analysis:

Start ANSYS Product Launcher, set the Working Directory to C:\temp, define Job Name as '3D_Bracket', and click Run. Then define Title and Preferences.

Utility Menu → File → Change Title... → Enter '3D Linear Elastic Stress Analysis of a Bracket' → OK

ANSYS Main Menu → Preferences → Preferences for GUI Filtering → Select 'Structural' and 'h-method' → OK

Enter the Preprocessor to define the model geometry:

Define Element Type and Material Properties.

ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete → Add... → Structural Solid Tet 10 node 92 (define 'Element type reference number' as 1) → OK → Close

ANSYS Main Menu → Preprocessor → Material Props → Material Models → Double Click Structural → Linear → Elastic → Isotropic → Enter 30e6 for EX and 0.3 for PRXY → OK → Click Exit (under 'Material')

Begin creating the geometry by defining a Block (Volume) for the base of the bracket. Since symmetry will be used, we will only model half the bracket (depth).

ANSYS Main Menu → Preprocessor → Modeling → Create → Volumes → Block → By Dimensions → Enter 0 and 6 for X-coordinates, 0 and 0.75 for Y-coordinates, and 0 and -2.25 for Z-coordinates → OK

Change to an Isometric View using the Plot Controls Menu (right side of Graphics Window).

The bolt holes will be created by defining Cylindrical Volumes, and subtracting these from the block we just created. The Cylinders are easily defined by rotating the WorkPlane (WP X-Y plane) so that the normal to the plane (Z direction) is directed along the axis of the Cylinder (simply rotate the WorkPlane about the X-axis by -90°).

Utility Menu → WorkPlane → Offset WP by Increments... → *** Move the Slider above Degrees to 90 *** → Click once on the 'X-⊖' button (Check the orientation of the WX-WY-WZ axes with respect to the X-Y-Z axes on the plot.) → Click OK in the 'Offset WP' window

ANSYS Main Menu → Preprocessor → Modeling → Create → Volumes → Cylinder → Solid Cylinder → Enter 1, 1, 0.25 and 0.75 for WP X, WP Y, Radius and Depth, respectively → Apply → Enter 5, 1, 0.25 and 0.75 for WP X, WP Y, Radius and Depth, respectively → OK

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Volumes → Select (with the mouse) the rectangular block → OK → Select the cylinders we just created → OK

Utility Menu → Plot → Replot

The flange will now be created by again moving the WorkPlane. First, reset the WorkPlane to the Global Coordinate System (active system), and then move it (without rotating it) to the lower left corner of the flange.

Utility Menu → WorkPlane → Align WP with → Active Coord Sys

Utility Menu → Plot → Replot

Utility Menu → WorkPlane → Offset WP to → XYZ Locations + → Type '0,0.75,-2' in the Command Line of the 'Offset WP' window (Global Cartesian coordinates) → Apply → OK

The front surface of the flange will be defined by creating Keypoints along the perimeter, and then defining Lines between the Keypoints.

ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → On Working Plane → Click on ‘WP Coordinates’ in the ‘Create KPs on WP’ window → Type ‘0,0,0’ in the Command Line of this window → Apply → Type ‘0,0.75,0’ in the Command Line → Apply **Utility Menu → Plot → Multi-Plots** (*this makes it easier to visualize*)

Type ‘6,0,0’ in the Command Line → Apply → Type ‘6,4,0’ in the Command Line → Apply → OK

ANSYS Main Menu → Preprocessor → Modeling → Create → Lines → Lines → Straight Line → Define a horizontal Line along the base of the flange and two vertical Lines along the left and right sides, by picking the appropriate Keypoints (with the mouse) → OK

ANSYS Main Menu → Preprocessor → Modeling → Create → Lines → Lines → At angle to line → Pick (with the mouse) the vertical Line on the right side of the flange → OK → Pick the upper Keypoint on the vertical Line on the left side → OK → Enter 290 or 110 (depending on how the vertical line was created) for ‘Ang in degrees’ (*angle measured CCW from downward or upward vertical*) → OK

ANSYS Main Menu → Preprocessor → Modeling → Delete → Line and Below → Click on the Line above the intersection point on the right side → OK

Now create the two Line Fillets on the flange and the circular Area in the flange. Create the Area that defines the flange, and then subtract the circular Area from the main flange Area.

ANSYS Main Menu → Preprocessor → Modeling → Create → Lines → Line Fillet → Select (with the mouse) the Lines bounding the small Fillet (left side) → OK → Enter 0.5 for Fillet radius → Apply → Select the Lines bounding the large Fillet (right side) → OK → Enter 1.5 for Fillet radius → OK

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle → Enter 4.75 for WP X, 1.25 for WP Y, and 0.375 for Radius → OK

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → By Lines → Select (with the mouse) the Lines and Fillets along the flange perimeter → OK

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas → Select (with the mouse) the main flange Area (polygon) → OK → Select the Circle → OK

We can now extrude the flange Area into a Volume of depth 0.25 in.

ANSYS Main Menu → Preprocessor → Modeling → Operate → Extrude → Areas → By XYZ Offset → Select (with the mouse) the flange Area → OK → Enter -0.25 for DZ → OK

Reset the WorkPlane to the Global Origin, and then add the two Volumes (flange and base) together.

Utility Menu → WorkPlane → Offset WP to → Global Origin

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Volumes → Select ‘Pick All’ → OK

Finally, we must create the Area Fillet between the flange and base. Additional Areas on each side of the Fillet must then be defined so that the Volume within the Fillet can be created and added to the main bracket Volume.

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Area Fillet → Select (with the mouse) the two Areas (flange and base) → OK → Enter 0.25 for Fillet radius → OK

Use the 'Ctrl' key and right mouse button to rotate the model so that one end of the Fillet can be seen, and zoom in on the end of the Fillet

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → By Lines → Select (with the mouse) the Fillet and Lines at the end of the Fillet → OK

Use the 'Ctrl' key and right mouse button to rotate the model so that the other end of the Fillet can be seen, and zoom in on this end of the Fillet

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → By Lines → Select (with the mouse) the Fillet and Lines at this end of the Fillet → OK

Utility Menu → PlotCtrls → Numbering... → Click Line numbers Off, Keypoint numbers Off and Area numbers On → OK

ANSYS Main Menu → Preprocessor → Modeling → Create → Volumes → Arbitrary → By Areas → Select (with the mouse) the Areas surrounding the Fillet (*you may need to rotate or zoom in on the model to select the Areas*) → OK

Utility Menu → PlotCtrls → Numbering... → Click Volume numbers On → OK

Utility Menu → Plot → Volumes

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Volumes → Select 'Pick All' → OK

Since the geometry is irregular (typical for three-dimensional structures), we will use the Free Meshing option. Note that, when Free Meshing Volumes, only Tetrahedral Elements can be used (ANSYS cannot Free Mesh Volumes with Hexahedral Elements). Because of this restriction, it is wise to use 10 node Tetrahedrons rather than 4 node Tetrahedrons. In this example, we will choose an average Element edge length of 0.17 in (note all Elements will be approximately the same size).

ANSYS Main Menu → Preprocessor → Meshing → MeshTool → Under 'Size Controls: Global' click Set → Enter 0.17 for 'Element edge length' → OK → Under 'Mesh:' select Volumes, Tet and Free → Click Mesh → Select (with the mouse) the Volume → OK

Another helpful meshing tool available in ANSYS is the "Smart Size" feature. When this feature is enabled, ANSYS will automatically adjust the size of the elements to place smaller elements (finer mesh) in regions of higher stress gradients (near holes, fillets, etc.) and place larger elements (coarser mesh) in regions of more uniform gradients. This is a very useful feature which should be used when free-meshing objects. To try it, first clear the existing mesh and then remesh with Smart Sizing enabled.

ANSYS Main Menu → Preprocessor → Meshing → MeshTool → Click Clear → Select the Volume → OK

Utility Menu → Plot → Volumes

ANSYS Main Menu → Preprocessor → Meshing → MeshTool → Click 'Smart Size' On → Move the Slider on the Scroll Bar to 2 (near Fine) → Under 'Mesh:' select Volumes, Tet and Free → Click Mesh → Select (with the mouse) the Volume → OK

Enter the Solution Menu to define boundary conditions and loads and run the analysis:

ANSYS Main Menu → Solution → Analysis Type → New Analysis → Select Static → OK

Symmetry Boundary Conditions can be defined along the backside Areas of the bracket (rear Areas of the flange and base), and on the Areas inside the bolt holes to restrict the radial displacements of nodes there.

ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → Symmetry B.C. → On Areas → Select (with the mouse) the rear Areas of the flange and base and the Areas inside the bolt holes → OK

To restrict the vertical displacements of Nodes underneath the washers, we must first select the appropriate Nodes (i.e., Nodes on the top surface of the base within a radius of 0.35 in of the center of each bolt hole). This can be done by defining a Local Cylindrical Coordinate System at the center of each hole on the top surface, oriented with the Z-axis along the hole axis. In a Cylindrical Coordinate System, the X-direction represents the radial direction and the Y-direction represents the tangential direction. First define a Local Coord. Sys. at the left bolt hole ($X = 1, Y = 0.75, Z = -1$), and rotate it so that the Z-axis is along the hole axis.

Utility Menu → WorkPlane → Local Coordinate Systems → Create Local CS → At Specified Loc + → Enter the coordinates '1, 0.75, -1' (Global Cartesian) in the Command Line of the 'Create CS at Location' window → OK → Enter 11 (any number greater than 10) for 'Ref number of new coord sys' (KCN) → Choose Cylindrical for 'Type of coordinate system' (KCS) → Enter -90 for 'Rotation about local X' (THYZ) → OK

The new Coordinate System becomes active when it is defined. Now select the Nodes with a Z-coordinate of 0 and an X (radial) coordinate between 0 and 0.35, and apply Displacement Constraints in the Global Y direction to these Nodes.

Utility Menu → Select → Entities... → Select 'Nodes' and 'By Location' → Click on 'Z coordinates', enter '0, 0' for Min and Max values, click on 'From Full' → Apply → Click on 'X coordinates', enter '0, 0.35' for Min and Max values, click on 'Reselect' → OK

Utility Menu → Plot → Nodes (*only the selected nodes should appear*)

Utility Menu → WorkPlane → Change Active CS to → Global Cartesian

ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Nodes → Select 'Pick All' → Select UY for 'DOFs to be constrained' → OK

Utility Menu → Select → Everything

Utility Menu → Plot → Replot

Repeat this procedure for the right bolt hole ($X = 5, Y = 0.75, Z = -1$).

Utility Menu → WorkPlane → Local Coordinate Systems → Create Local CS → At Specified Loc + → Enter the coordinates '5, 0.75, -1' (Global Cartesian) in the Command Line of the 'Create CS at Location' window → OK → Enter 12 (any number greater than 10, but don't duplicate previous number) for 'Ref number of new coord sys' (KCN) → Choose Cylindrical for 'Type of coordinate system' (KCS) → Enter -90 for 'Rotation about local X' (THYZ) → OK

Utility Menu → Select → Entities... → Select 'Nodes' and 'By Location' → Click on 'Z coordinates', enter '0, 0' for Min and Max values, click on 'From Full' → Apply → Click on 'X coordinates', enter '0, 0.35' for Min and Max values, click on 'Reselect' → OK

Utility Menu → Plot → Nodes (*only the selected nodes should appear*)

Utility Menu → WorkPlane → Change Active CS to → Global Cartesian
ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement
→ On Nodes → Select 'Pick All' → Select UY for 'DOFs to be constrained' → OK
Utility Menu → Select → Everything
Utility Menu → Plot → Replot

Finally, apply the pressure along the upper right quadrant of the hole in the flange.

ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Areas → Select (with the mouse) the Area inside the hole where you want to apply the Pressure → OK → Enter 9,428 for 'Load PRES value' → OK

Save the Database and initiate the Solution.

ANSYS Toolbar → SAVE_DB
ANSYS Main Menu → Solution → Solve → Current LS → OK → Close the information window when solution is done → Close the /STATUS Command window

Enter the General Postprocessor to examine the results:

Plot the Deformed Shape and Stress Contour Plots of interest, and List the Nodal Reaction Forces to verify the loading is correct.

ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape → Select Def + undef edge → OK

ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solu → Select 'Stress' and 'von Mises stress' (or any other component) → OK

ANSYS Main Menu → General Postproc → List Results → Reaction Solu → Select 'All items' → OK

Etc. . . .