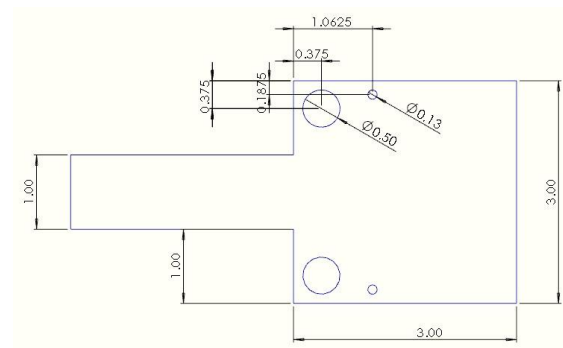
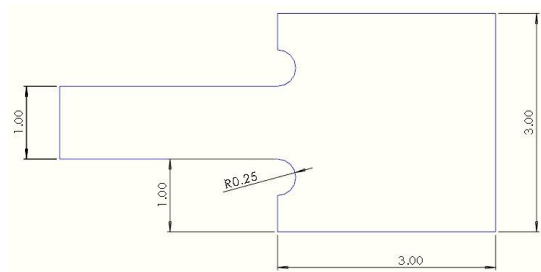
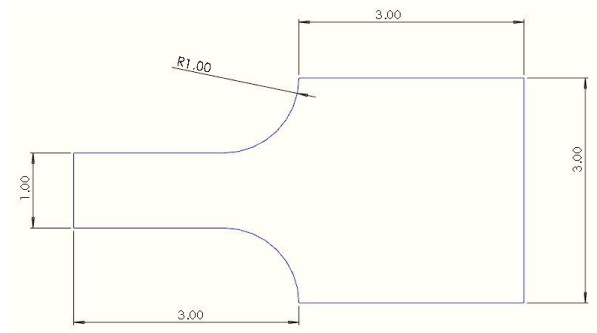
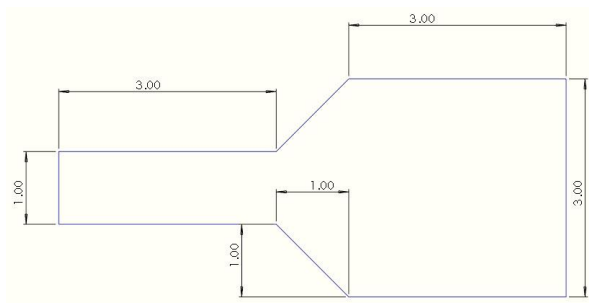
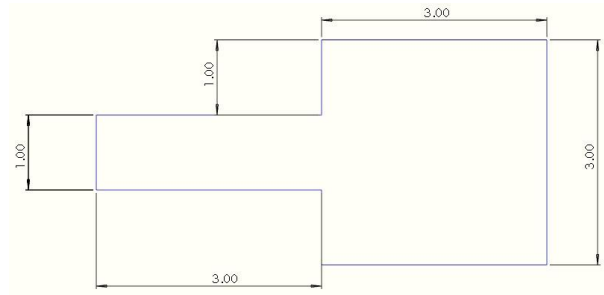


Introduction to SolidWorks and ANSYS Workbench

1 Problem

For the various parts shown below (where all units are in inches), determine the design which best relieves stress. Make the original part have a small radius of 0.1 in. at the inside re-entrant corners. Place a uniform pressure load of 1000 psi on the right end of each part and fix the left end. All units shown are taken in inches. Let the material be structural steel ($E = 29 \times 10^6 \text{ lb/in}^2$, $\nu = 0.3$) and the thickness of all designs be 0.5 in.



2 Importing into and Modeling in ANSYS

1. Save part/assembly in SolidWorks, then import to ANSYS.
2. Connect SolidWorks imported Geometry to your chosen study's Geometry in ANSYS Project Schematic.
3. Select or add new materials via Engineering Data.
4. In ANSYS Mechanical, choose each part (if multiple) from the side menu and select its material.
5. Select the study type's name in the menu and choose the appropriate B.C.'s and Loads.
6. Alter the mesh parameters accordingly. (Note: maximum of 256,000 *nodes*)
7. Solve.
8. Post-process by selecting Solution and choosing the appropriate parameter to plot. You may need to right-click on newly added parameters and select Evaluate All Results.
9. Note: if you edit a part in SolidWorks, save it then right-click Geometry from the tree in ANSYS Mechanical and select Update Geometry from Source.