Simulation (FEA) Basics

This Tutorial will walk you through the basics of applying fixtures and loads on a part in order to calculate stresses, strains, and displacements. You may need to use one of the campus computers.

Download the files "Stand" and WeldedCorner" from the ME301 Mindworks page on Day 17. Then **save the parts to a personal folder you can access, such as your OneDrive account (onedrive.uidaho.edu)**.

FEA Tutorial 1 – Easy Points (Hopefully)

Open the part file "**Stand***"*. The part is similar but not the same as the part shown in the images below. Pick similar faces as those illustrated.

1. Click the *Simulation* tab and *New Study*. Under *General Simulation* click *Static* and then click *OK* (green check mark). You should now have a tab at the bottom of your screen named "Static 1". You're now working in this *Static 1* tab, as opposed to the *Model* tab where parts are built.
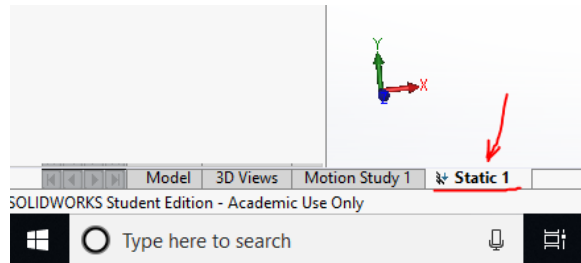

Figure 1

If you cannot find the Simulation tab click on the search bar (make sure it is searching commands) and type "add-ins". From *add-ins* **click on** *Simulation***.** *You may need to use one of the campus computers as not all educational installations enable Simulation by default.*

2. Right-click on *Fixtures* and select *Fixed Geometry*. Then select the face shown in Figure 2 and click *OK*. Your simulation should now look like Figure 3.
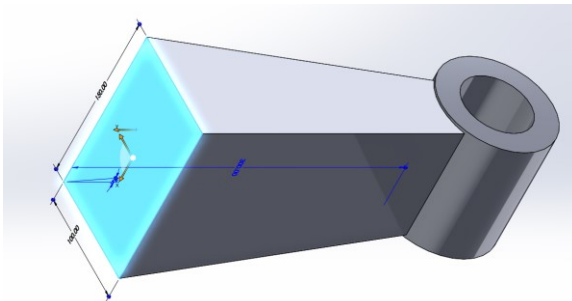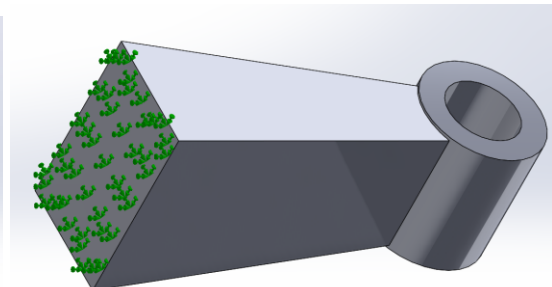

Figure 2

Figure 3

3. Right-click on External Loads and select Force. The direction should be normal, the units should be SI and the Force Value should be set to the value provided in the checksheet

for Tutorial 1. Then select the face shown in Figure 4 for the Faces and Shell edges for Normal Force and click OK. Your simulation should now look like Figure 5.
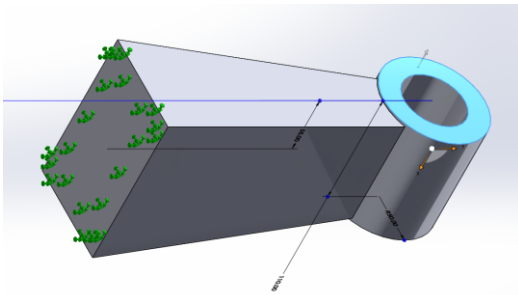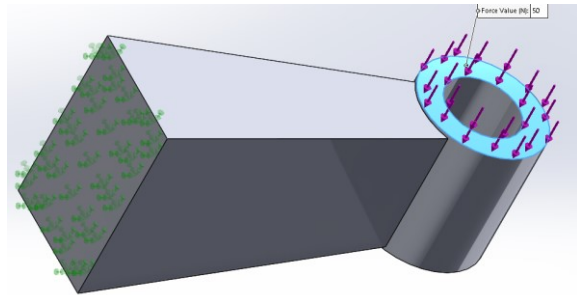

Figure 4   Figure 5

4. In the simulation tree the mechanical body is labeled "Stand", right click on that and select Apply Favorite Materials → Plain Carbon Steel.
5. Right click on the Study name in the simulation tree and select *Run* (the mesh will automatically be made with default settings).
6. When the study is done running, double-click on *Displacement1 (-Res disp-)*. From here you can see the max displacement of the part (in Figure 6, this value is 2.052e-04 mm).
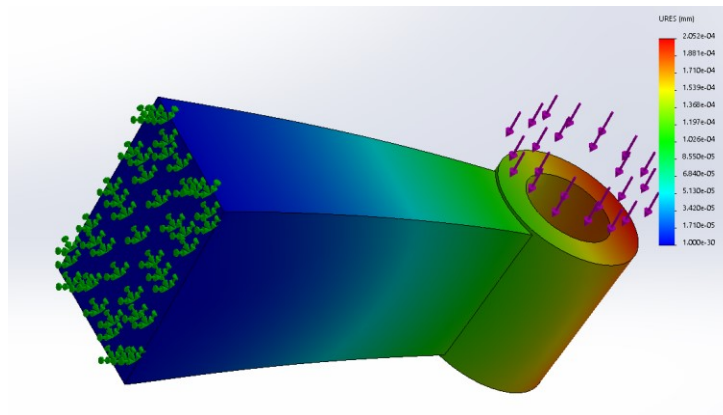

Figure 6

7. Take a similar screenshot of your model (as shown in Figure 6) and include it in your HW10 submission with a brief caption.
8. Double-click on *Stress1 (-vonMises-)* and take another screenshot of your model to include with a brief caption.
9. Go back to the *Model* tab, select feature *Hole1*, hold *SHIFT* and select feature *Hole4*. This will highlight all the features *Hole1* through *Hole4*. Right-click on *Hole4* and unsuppress the highlighted features.
10. Now go back to the *Static1* tab. Right-click on your *Fixed-1* fixture from the Simulation tree and suppress it. Right-click on *Fixtures* and add a *Fixed geometry* constraint to each of the four holes by selected the inner surface of each hole. The face of each hole should now appear in *Faces, Edges, Vertices for fixture*.
11. Re-run the simulation.

12. Take another screen shot similar to Figure 6 of your model and include it in your HW10 submission with a brief caption.

FEA Tutorial 2 – Split Line and Measurement Probes

Open the part file "**WeldedCorner**". The part is similar but not the same as the part shown in the images below. Pick similar faces as those illustrated.

1. Click the *Simulation* tab and *New Study*. Under *General Simulation* click *Static*.
2. In the simulation tree the mechanical body is labeled *WeldedCorner*, right click on that and select Apply Favorite Materials → Plain Carbon Steel.
3. Right-click on *Fixture* in the simulation tree and select *Fixed Geometry*. While in the Fixture tab select *Immovable (No translation)* under *Standard (Fixed Geometry)*. Then select the face shown in Figure 7 and click okay. Your simulaiton should now look like Figure 8.
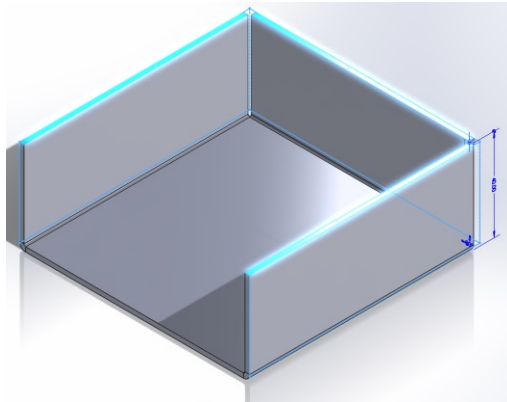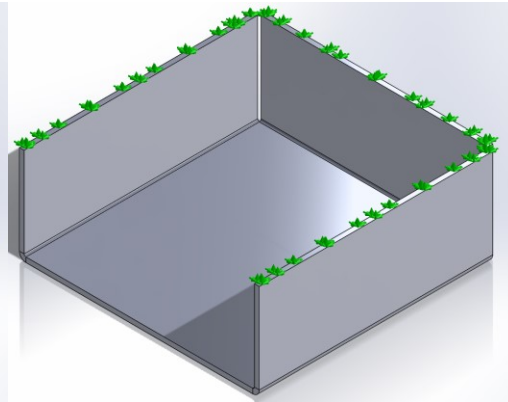

Figure 7	Figure 8

4. Right-click on External Loads and select Force. The direction should be normal, the units should be SI and the Force Value should be set to the value provided in the checksheet for Tutorial 2. Then select the face shown in Figure 9 for the Faces and Shell edges for Normal Force and click okay. Your simulation should now look like Figure 10.

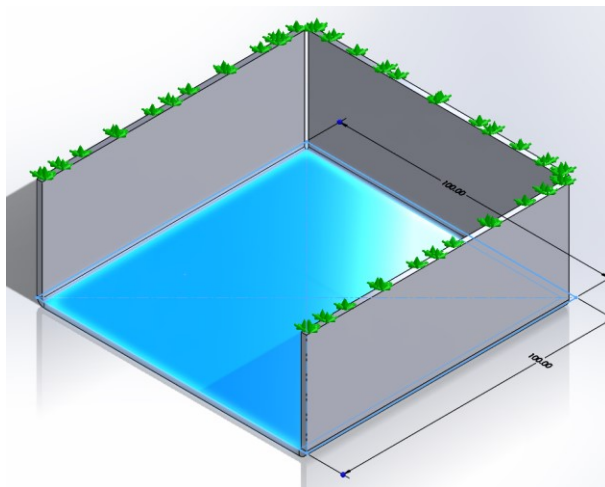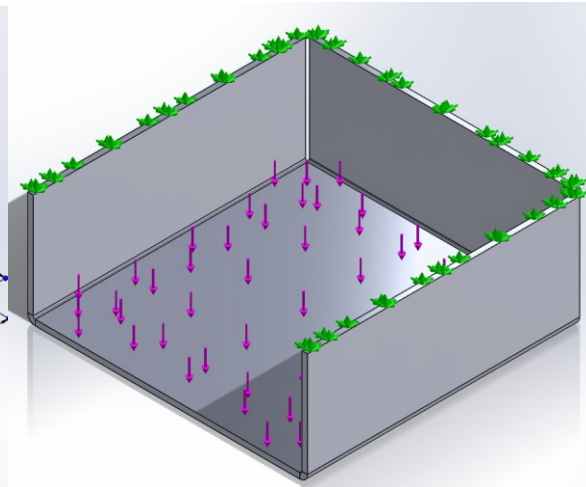| Figure 9	Figure 10

5. Right click on the Study and select Run (the mesh will automatically be made with default settings).
6. When the study is done running double-click on Displacement1 (-Res disp-). From here you can see the max displacement of the part. In Figure 11, this value is 3.379e-02 mm.
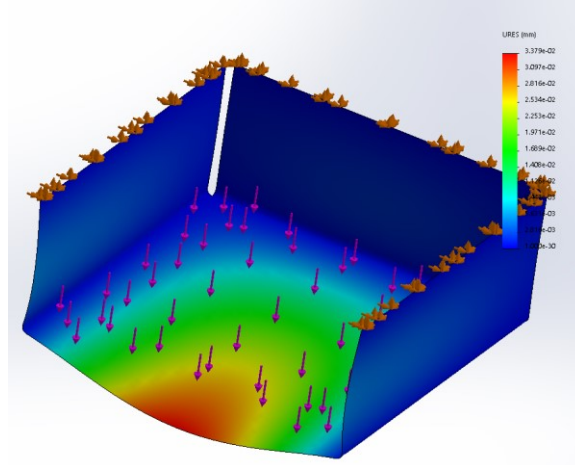

Figure 11

7. Take a screenshot of the model similar to that of Figure 11 and include it in your HW10 submission with a brief caption.
8. Now we want to find the X displacement at the location of the maximum displacement resultant displacement. To do this, we must be able to select the point with the probe tool. Intuitively we know that the maximum displacement occurs at the middle of the front edge on the center plate. To select this point we must edit the solid model. Start by opening the Surfaces tab in SolidWorks. If your computer doesn't have the surfaces tab, you can open the Features tab.
9. From the Surfaces (or Features) tab click *Curves→Split Line*. Under *Type of split* select *intersection*, under *Splitting Bodies/Faces/Planes* select the Right Plane, under *Faces/Bodies to split* select the face indicated in Figure 12. Your simulaiton should now look like Figure 13. Notice the Split line in Figure 14. Re-run the study.
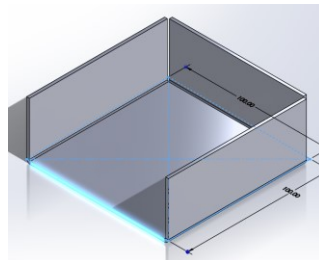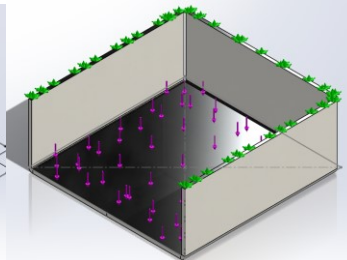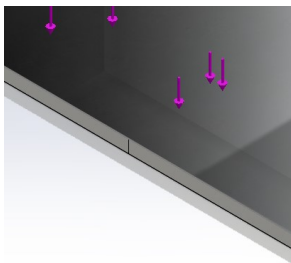

Figure 12   Figure 13

Figure 14

10. Right click on *Displacment1 (-Res disp-)* and select *Edit Definition*. In the definition under *Display* select UX:X displacement. Click OK.
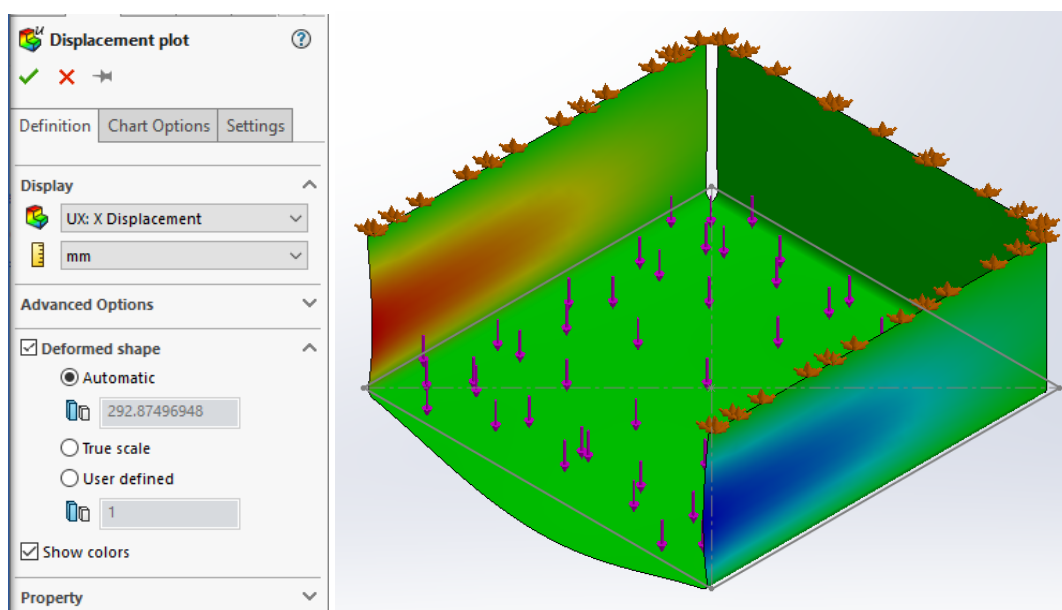

Figure 15

11. Click Plot Tools>Probe. In the Probe results tab select *On selected entities* and click on the node indicated in Figure 16 and click *update* in the Probe Results tab. From here we find the X displacement at this point. In this case, in Figure 16, the X displacement at the selected point is -5.569e-08 mm.
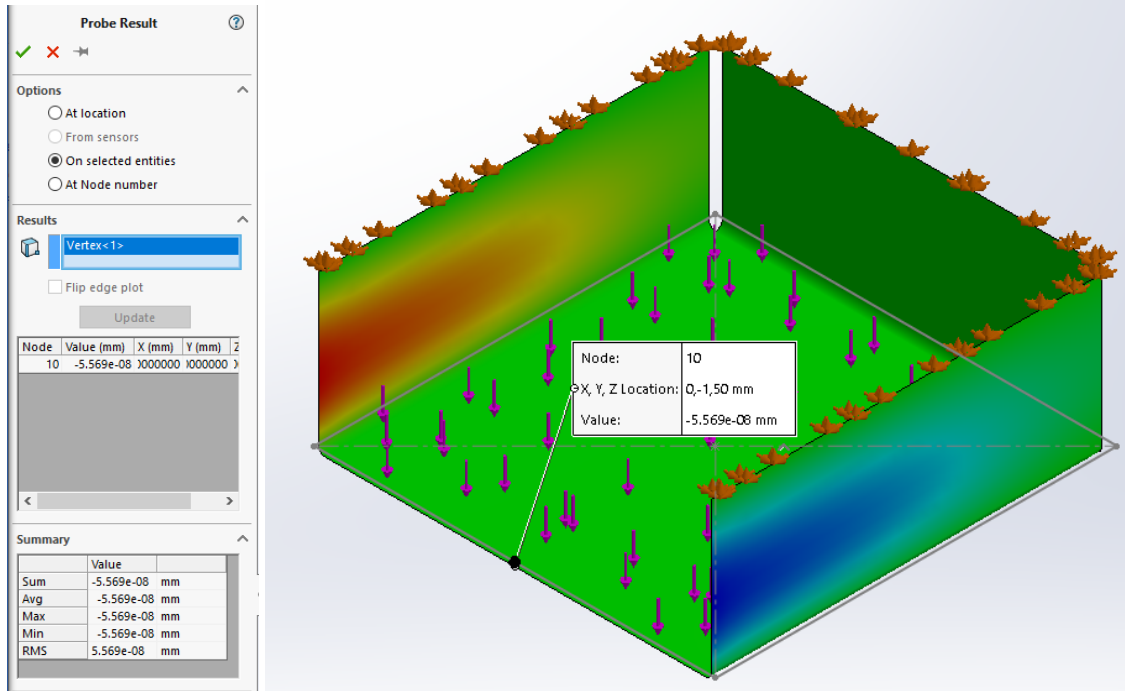
Figure 16

12. Take a similar screen shot of your model (as shown in Figure 16) and include it in your HW10 submission with a brief caption.
13. Include 2-3 lessons learned from this tutorial in your HW submission.