CAD Lab Assignment #1: Introduction to Fusion 360

ME 170

Overview

This first CAD lab assignment is designed to get you accustomed to the Fusion 360 CAD program's interface and to manipulating a part model - without having to create it yourself. You will modify the part, examine some aspects of how it was created, and learn some of the basic features of using Fusion. The part has been rather poorly designed, intentionally to demonstrate the importance of good modeling practices to capture logical expectations of design intent and to produce a resulting robust model. This is particularly important when working in industry as an engineer; inevitable future modifications (or engineering revisions) will be made to your design by others and poor design intent/logic will result in time and money being wasted.

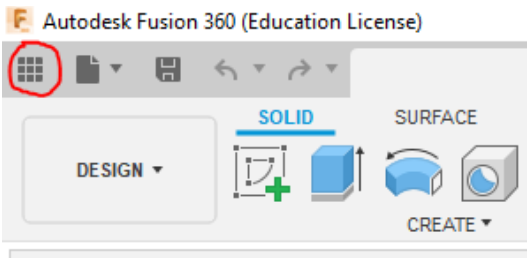
Supplementary Videos: The videos at: https://academy.autodesk.com/software/fusion-360 are a broad introduction to using Fusion, and provide additional help and insights.

Part I. Introduction to Fusion

1) Open Autodesk Fusion 360. If not already installed, go to: https://www.autodesk.com/products/fusion-360/students-teachers-educators

2) Create an ME170 project with a Lab 1 folder.

   a. Open the Data Panel by clicking on the icon in the top-left corner of Fusion. You can close the Data Panel at any time by clicking the icon again.

   b. Select the New Project option. Enter "ME170" as the name and double click to open the project.

   c. In the ME170 project, select the New Folder option and enter "Lab 1" as the name. Double click the folder to open it.

3) Copy the angled bracket to your Lab 1 Folder and open it.

   a. On the class Compass website, in the "CAD Lab Assignments" content area under CADLAB#1, you will find a fusion CAD model of a simple sheet metal bracket. Click on the file, 154bracket.f3d, and download it to your computer

   b. Double click on the 154bracket file to open it. With it open, go to File > Save and choose the location as your newly created Lab 1 folder.

4) Explore the bracket using the various available view options.

   a. Click on the "Display Setting" icon on the

toolbar at the bottom of the screen.

b. Under the "Visual Style" tab click each of the six display style options to see how the model display changes. Note the hotkeys next to each option. These are very helpful for efficient work.

5) Explore spin, pan, and zoom functions using the mouse and with various view options selected.

a. Hold down the middle mouse wheel and move the mouse to pan the camera.

b. Hold down the shift key and the middle mouse button to spin the camera.

c. Twiddle or spin the middle mouse wheel forward and backward to zoom in and out.

Note: these can all be done using the toolbar at the bottom of the screen, but it is much more efficient to use the mouse shortcuts.

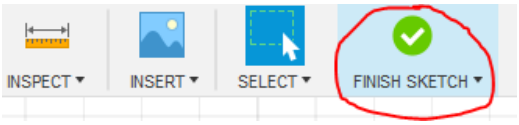
6) Select the default view of the bracket.

a. Hover over the view cube in the top right of the screen and select the home symbol that appears next to it.

7) Enter a sketch.

a. To model in 3 dimensions, we generally begin in 2 dimensions, specifying geometry, before adding the extra dimension. 2-D sketches are located on planes and are used to define "features" which add depth to the sketch.

b. At the bottom-left of the screen is a timeline, detailing the steps taken to produce this part. These include sketches followed by extrusions. An extrusion takes the geometry of a sketch and extends it in the perpendicular dimension for a given length. Double click on Sketch1 to enter the sketch.



Note: You can tell whether you are in a sketch by the "Finish Sketch" option in the top right.

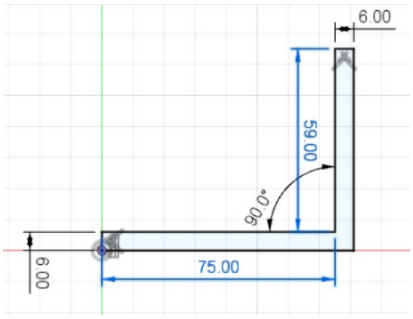
8) Modify the dimensions of the 154deg angle bracket.

a. The bracket should have pre-set dimensions. If you double click any of them, it will allow you to edit the dimension.

b. Double click the 154 degree dimension, and change its value to 90. Then press enter on your keyboard to set the new value.

c. The sketch should automatically update, and once you click "Finish Sketch" in the top-right of the screen, the part will update too. Once you have made sure the part has updated, go back into Sketch1 by double clicking it on the timeline.

9) Repeat the steps listed in section '8' above to change the following dimensions:



10) Change the 59 side length to 41.

   a. Change the 75 side length to 45.

   b. Leave the sketch to observe the changes.

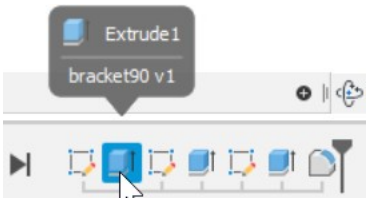
11) Perform the following operations:

   a. From the timeline on the bottom, select Sketch2, which corresponds to the Hole 1 extrusion. This extrusion is a Cut, meaning instead of adding material (like the first extrusion) it removes it.

   b. Edit the 27 distance dimension. Change the value to 20.

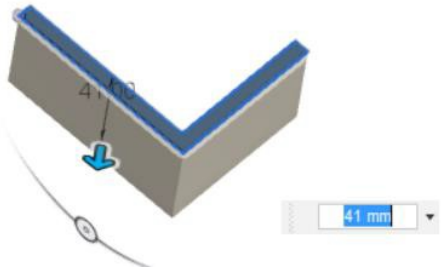
   c. Repeat for Sketch3, corresponding to Hole 2.

   d. Finish Sketch and observe the movement of the holes.



   e. Double click on Extrude1 to modify its dimension. This adds depth to Sketch1, giving the part its width. Change the value from 41 to 80.

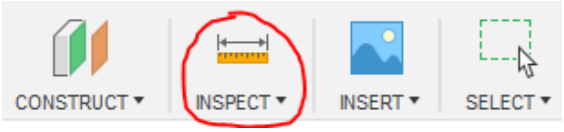
   f. Press enter to finalize the process and note the changes.



   g. **Write down what happens to each hole in a few sentences.** Create a document (Word, Google Docs, etc.) and name the file "Lab#1 responses". At the end of the assignment, you will be asked to submit this through Compass 2g.

   h. **Put a screenshot of what the bracket looks like when the extrusion is changed.** Use the screenshot to help you explain what happens. I ask you to do this, in part, because we will use screenshots quite a bit in this class, so if you don't know how to do this, now is the time to learn. Every computer is a little different, I simply use the "PrtSc" key (F10) on my keyboard but you may need to do a little search with your computer model info. (search "screen shot" or "print screen")

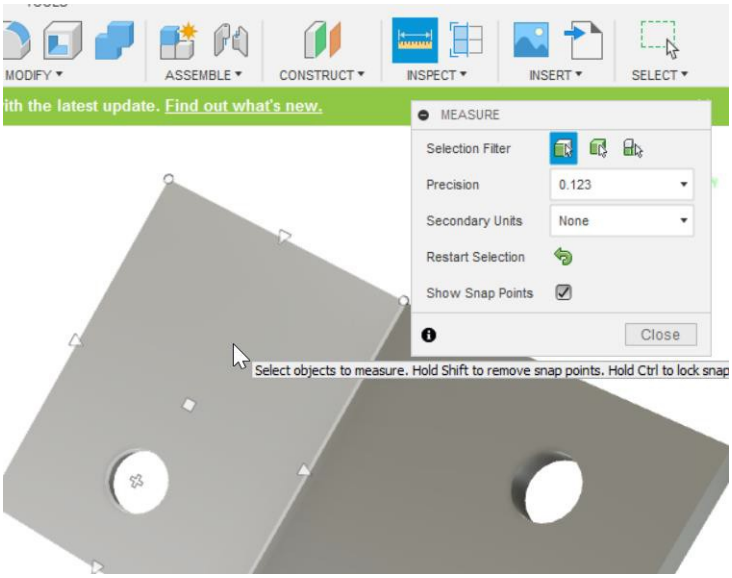
12) Use the measure command to measure the distance from the <u>hole center</u> to the <u>3 outer edges</u> closest to each hole. <u>Do this for both holes</u>.



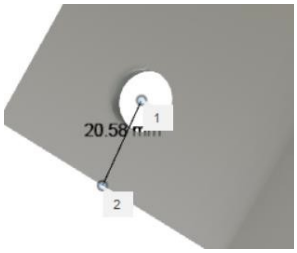
   a. On the top toolbar, under Inspect, choose Measure. Alternatively, you can press i, the hotkey.

   b. First try measuring the width of the part: click on one edge, then an adjacent edge and observe the "Distance" in the Results section of the "Measure" dialog box. It should be 80 mm.

c. Now this bits a little trickier. We want to measure from the center of the hole to an edge of the part. If you just select the hole then the edge of the part, you will get the distance from the edge of the hole to the edge of the part. We want the center of the hole. So you need to click on "show snap points" check mark in the dialog box, then hover over the face and hold the CTRL key down to freeze the snap points for selection.



d. Then click keeping you finger on the CTRL key, click on the center of the hole. Then you can release the CTRL key and click on the edge of the part



e. Type the distance displayed from the center of the hole to each of the 3 edges one at a time by clicking "restart selection" in the dialog box.

f. Repeat this process for the other hole. Be sure to record the three requested dimensions for both holes.

13) Change the bracket width back from 80 to 41 and observe the model.

14) Change the thickness of the bracket from 6 to 10 and observe the model.

a. The thickness of the bracket is governed by two dimensions. Ensure that both thickness dimensions are changed from 6 to 10.

b. Rotate the part using the spin, pan, and zoom commands (section '5') in order to view the inside surfaces of the bracket.

c. Write down what happens to each hole and why in a few sentences.

15) Edit the definition of the blind-depth hole to extend through the entire wall of the bracket.

a. Select the blind-depth hole by selecting the extrusion from the timeline at the bottom-left of the screen.

b. On the Extrusion window under the "Extent" tab, change the input from "Distance" to "All." This will cause the Extrusion to update as the part does, while "Distance" causes it to cut for a set dimension.

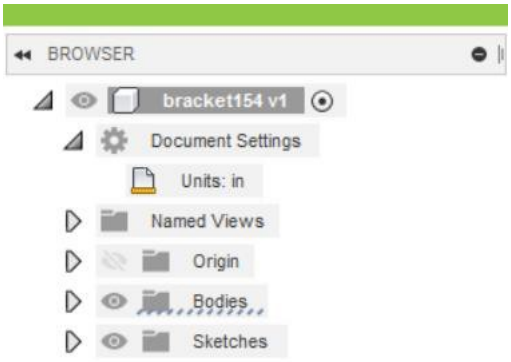
c. Comment on the change to the hole.

16) Change the bracket thickness back from 10 to 6.

17) Measure the radii of the rounds on the inside and outside of the bend in the bracket.

a. Under the Inspect tab select the Measure tool.

b. Select the inner and outer rounds **record both fillet radii**. You may need to zoom in to make sure you select the inner bend.



18) Alter the dimension units for the bracket. Change the units from mm to inches and back to mms again

a. In the top-left of the window, under the "Browser" expand the Document Settings tab by clicking the drop-down arrow.

b. Hover over the units tab, click the "Change Active Units" option, and select inches from the pop-up menu.

Note: this maintains the same absolute measurements, but displays them in a mm. Make sure you are using the correct dimensions every time you start a new part, since Fusion cannot directly change the absolute measurement.

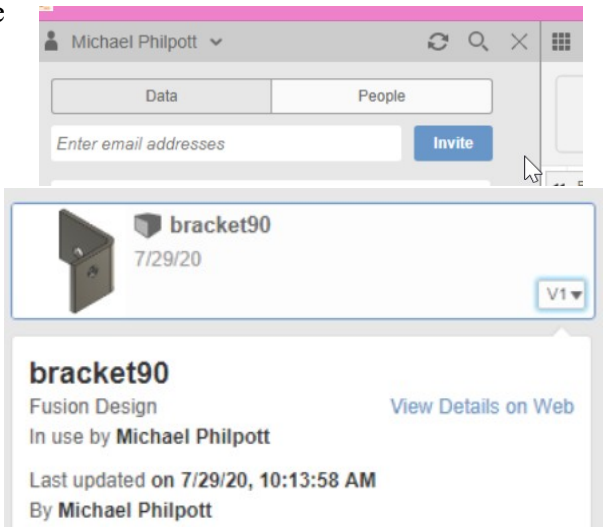
c. Observe, understand the behavior, and switch back again (some CAD systems have the option to "interpret" the numbers in different units or "convert" them).

Part II. Submission Requirements

a. Navigate to File -> Save As and name your new modified bracket with the 90º bend "bracket 90". It should be saved in your Fusion Lab 1 folder ready for grading.

b. Submit your "Lab#1 responses" by clicking directly on the "CADLAB#1" assignment title in Compass and either 'copy and pasting it' using the "Write Submission" button or uploading the file using the "Browse My Computer" button.

c. In Fusion, in the Data Panel, Go to the People tab in the top right of the Data Panel and invite both your TA by entering their emails and pressing "Invite" (their email addresses are on the homepage of the compass website). It is very important that you do this, otherwise they will not be able to access your files and give you credit for your work. Make sure your whole ME170



folder is shared, not just your Lab1 folder, so you don't have to invite them to each Lab.

Remember the due date is the midnight before the start of the next lab; your grader will check the "last updated" date by expanding the file version button; so make sure you don't go back and make any changes after the due date/time.