2.1: Sketching

Many features that you create in Fusion 360 start with a 2D sketch. In order to create intelligent and predictable designs, a good understanding of how to create sketches and how to apply dimensions and geometric constraints. Fusion does support 3D sketches although, in this module we will cover basic sketching tools to create and edit a 2D sketch. In this lesson we will be building the housing for a hypocycloidal gearbox. Visit robotarm.org to learn more about it.

Lesson 1: Creating a sketch

Learning Objectives

1. Create a 2D sketch
2. Create geometry in a sketch
3. Use constraints to position geometry
4. Use dimensions to set the size of geometry

Datasets Required

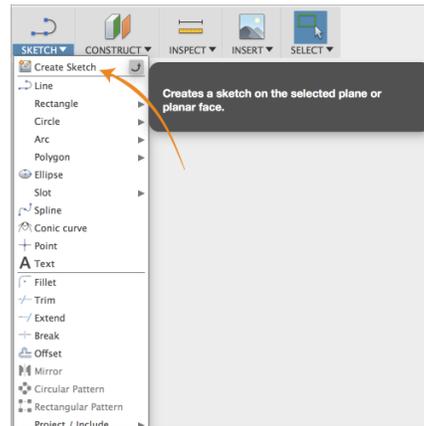In Samples section of your Data Panel, browse to:

Fusion 101 Training > 02 – Sketching > 02_Sketching

Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides
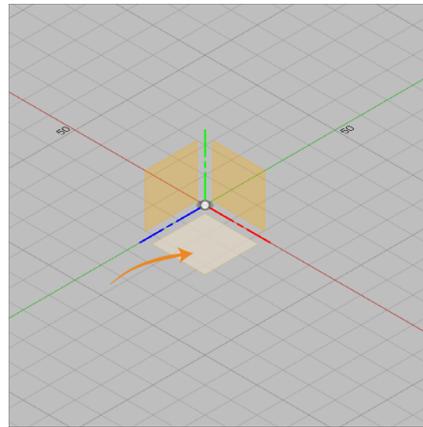
Step 1: – Start the **Sketch** command

1. Select **Sketch** > **Create Sketch**.

Step 2: – Select the sketch plane

1. You are now prompted to select a "plane" to sketch on.
2. Select the "Front" (XY) plane.

Note: Aside from the origin planes, you can create sketches on one of the 3 default planes, on a custom construction plane or on an existing model face, more on this later.

Step 3: – Create a circle

1. Select **Sketch > Circle > Center Diameter Circle**
2. Now hover over the **origin** (center) of the sketch. You should see the cursor "snap" to this location.
3. Click once to begin placing the circle.
4. Move the mouse away from the center to define the size.
5. Click again to place the circle.

Note: In Fusion 360 it is important to "snap" entities to the origin when possible. This accurately grounds the objects and ensures they will behave as expected.
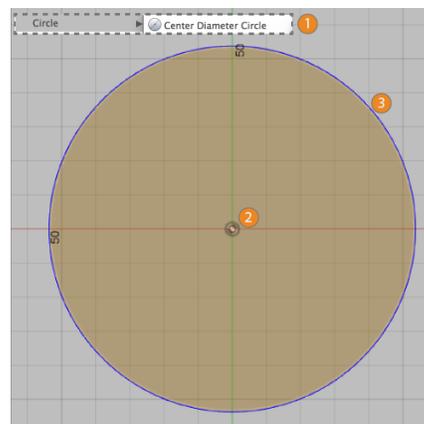
**Step 4:** – Dimension the circle

1. Select **Sketch > Sketch Dimension.**
2. Select the circle sketch.
3. Click again to place the dimension.
4. Type in a value of 62 mm.
5. Press **Enter** to accept the value.
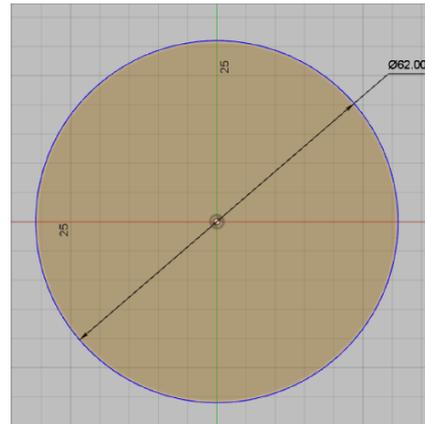
Step 5: – Finish the **Sketch** command

1. **Finish Sketch** by clicking on Stop Sketch.
2. Select the home view icon to the let of the view cube.

Note: You must "stop" a sketch before you can continue building geometry since Sketch is a mode that you enter and exit within workspaces.
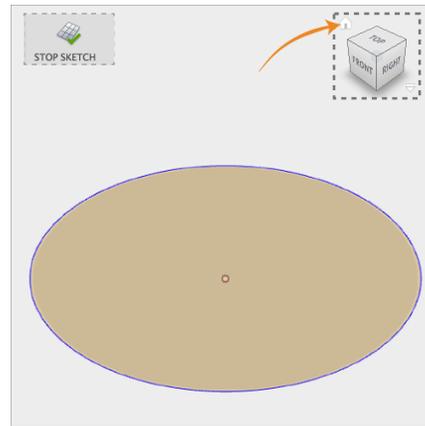
Step 6: – Extrude the Circle

1. Select **Modify > Press Pull.**
2. Select the Profile.
3. Drag the arrow up or type in a value of **8 mm** to set the depth.
4. Press **OK to finish.**

Note: In Fusion 360 you need to select the shaded area in the middle of the circle, NOT the edge of the circle to define profile for the extrude. Later we will look more closely at sketch profiles but for now simply select the shaded area.
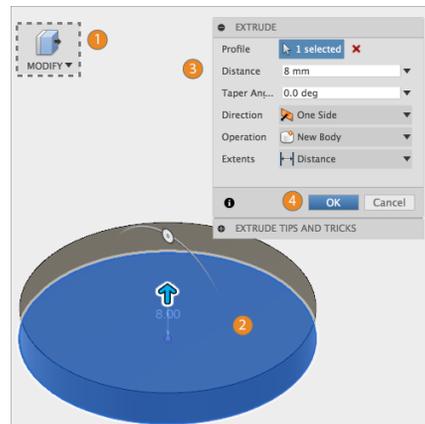
Step 7: – Create another sketch

1. Select **Sketch > Create Sketch.**
2. Select the top of the cylinder.

Note: This starts a new sketch on the top face of the cylinder. You will see the outer edge of the cylinder is already captured in the sketch.
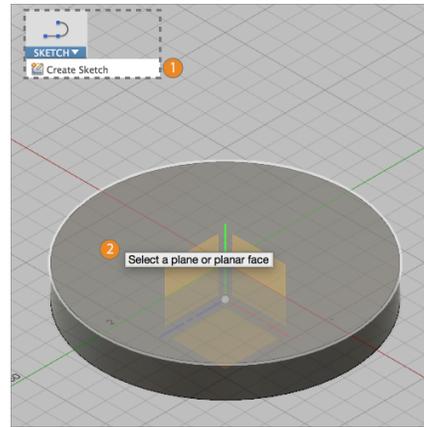
Step 8: – Create an inner circle.

1. Select **Sketch > Circle > Center Diameter Circle.**
2. Select the center point of the top face.
3. Start dragging the circle.
4. Type a value of **56 mm.**
5. Press **Enter** twice.
6. Select **Stop Sketch.**

Note: We can also create the inner circle with the Sketch > Offset command. Click on the outer circle and offset it to 56 mm. If you type in a value when creating sketch geometry the dimension will be automatically added. Otherwise you can apply a dimension like in the previous step.
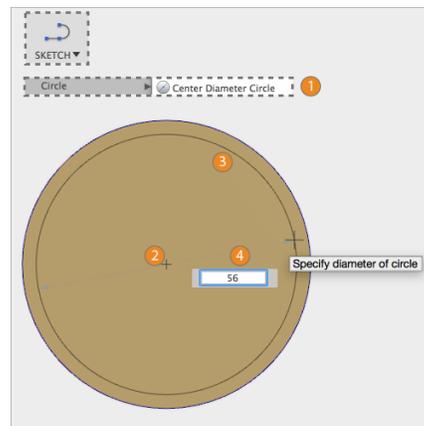
Step 9: – Extrude the inner circle

1. Select **Modify > Press Pull.**
2. Select the inner circle profile.
3. Drag the arrow up or type in a value of **22 mm** to set the depth.
4. Press **OK.**

Note: One sketch may contain multiple profiles. In this case there is a profile inside the circle you drew and a profile between the outer edge and the inner circle. You can select multiple profiles for an extrude feature. In this case you will only select the outer profile.
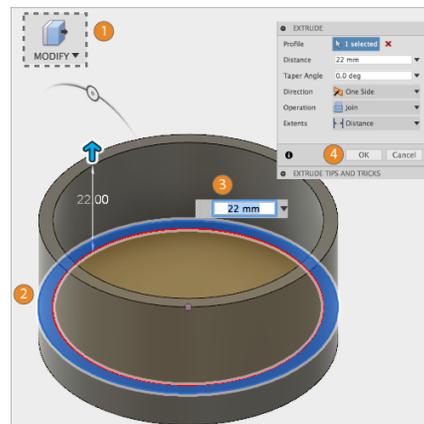
Step 10: – Create another sketch

We're now going to start creating smaller circle profiles for the holes so we can pattern them later.

1. Select **Sketch > Create Sketch.**
2. Select the top of the cylinder.
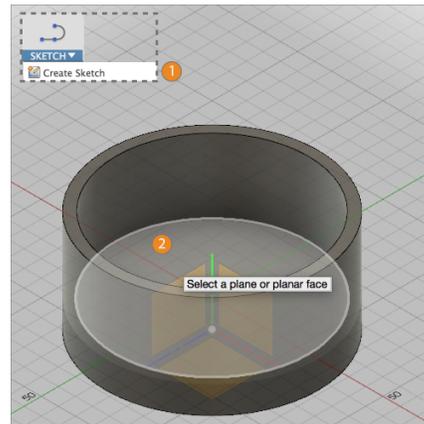
Step 11: – Create a circle

1. Select **Sketch > Circle > Center Diameter Circle.**
2. Select a point in the top part of the inner circle.
3. Start dragging the circle.
4. Type a value of **4 mm.**
5. Press **Enter** twice to accept the value.
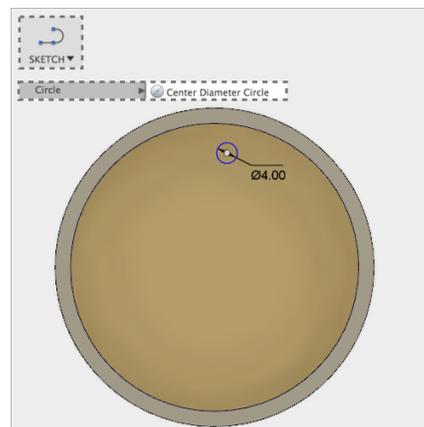
Step 12: – Constrain the Circle

1. Select **Horizontal/Vertical** constraint from the Sketch Palette.
2. Select the center point of the new circle and the center point of face.
3. Press **esc** to exit the command.

Note: Constraints create relationships in your sketch. By saying that these two points are "vertical" determines how the will be aligned in the sketch. These relationships are persistent, meaning that if the center of the face moves, the sketch will move also.

Step 13: – Dimension the circle

1. Select **Sketch > Sketch Dimension.**
2. Select the center of the circle.
3. Select the center of the face.
4. Click to place the dimension.
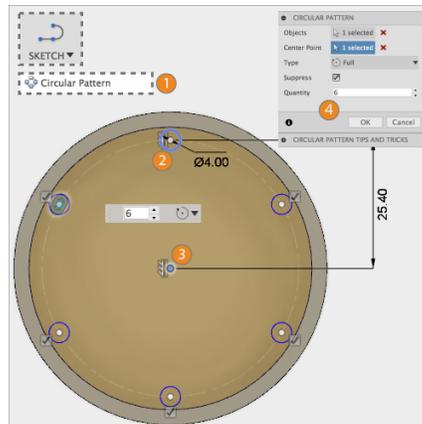5. Type in a value of 25.4 mm.
6. Press **Enter** to accept the value.

Note: This now has "fully" constrained the circle. This means that the geometry is fully locked down.
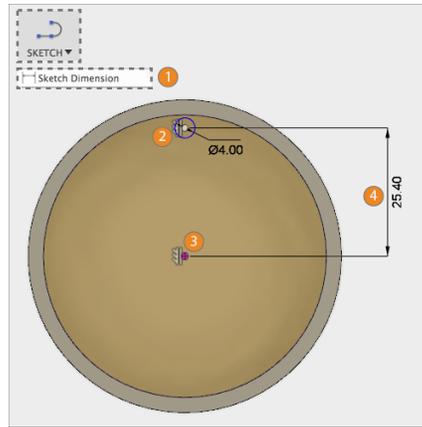
Step 14: – Pattern the Circle

1. Select **Sketch > Circular Pattern.**
2. In **Objects,** select the circle.
3. In **Center Point**, select the face center.
4. Change the **Quantity** to **6.**
5. Press **OK.**
6. Select **Stop Sketch** to end the sketch mode and return back into the modeling environment**.**

Note: Make sure you initiate what you want to select by click on the "no selection" box before trying to pick the center point for the pattern. Each box should then say "1 Selected." You can press the red "X" next to the box if you want to clear the current selections.

Step 15: – Extrude the Circles.

5. Select **Modify > Press Pull.**
6. Select the 6 Profiles.
7. Drag the arrow up or type in a value of **6 mm** to set the depth.
8. Press **OK.**

Note: It may help to zoom in a little bit to make sure you are grabbing the correct profiles. Make sure you have six profiles selected. If you need to add or remove profiles later hold ctrl (for Windows) or command (for Mac) and select the profile.
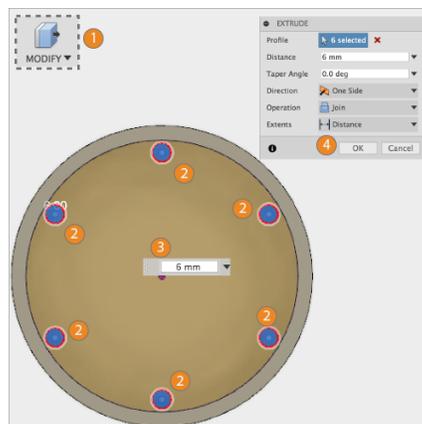
Step 16: – Create another sketch

We're almost done with our design. We need to create the pockets for the round gear teeth.

3. Select **Sketch > Create Sketch.**
1. Select the top of the cylinder again.
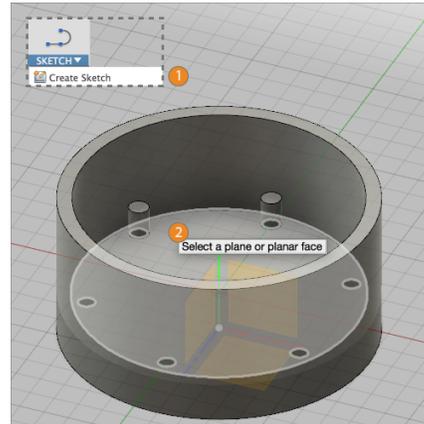
Step 17: – Create an arc

1. Select **Sketch > Arc > 3 Point Arc.**
2. Select the center of one circle.
3. Select an adjacent circle center point.
4. Click the third point near the edge.
5. Press **esc** to exit the command.
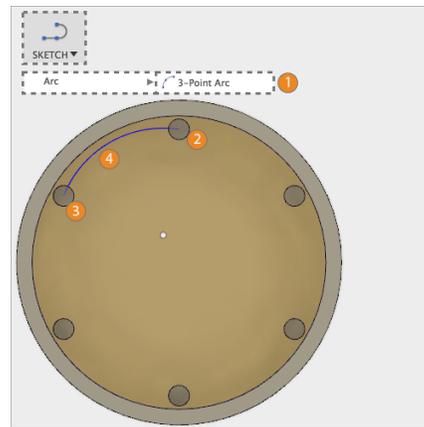
Step 18: – Constrain the Arc

1. Select the **Tangent** constraint from the Sketch Palette.
2. Select the new Arc and the inner edge so that they are constrained.
3. Press **esc** to exit the command.
4. Select **Stop Sketch** to return back to the modeling environment.
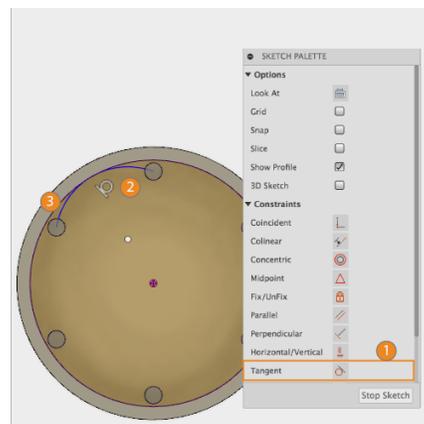
Step 19: – Pattern the Arc

1. Select **Sketch > Circular Pattern.**
2. In **Objects,** select the arc.
3. In **Center Point**, select the face center.
4. Change the **Quantity** to **6.**
5. Press **OK.**
6. Select **Stop Sketch.**
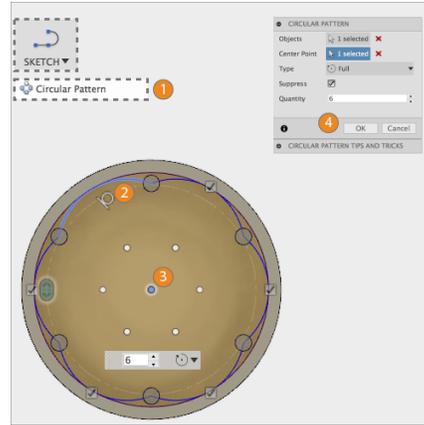
Step 20: – Extrude the Arcs

1. Select **Modify > Press Pull.**
2. Select the 6 Profiles.
3. Drag the arrow up or type in a value of **6 mm** to set the depth.
4. Press **OK.**

Note: Notice that by simply creating the arcs, the profiles are automatically created. The geometry we are extruding is the space between the existing geometry and the new arcs.
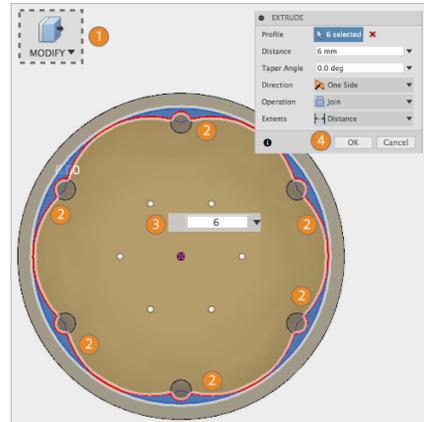
Step 21: – All Done

1. Congratulations you have finished this lesson and have learned the basics of how to create sketch geometry, dimension sketches, constrain sketches, as well as creating solid models based on sketches. Next we will look at more ways to define geometry in a sketch.
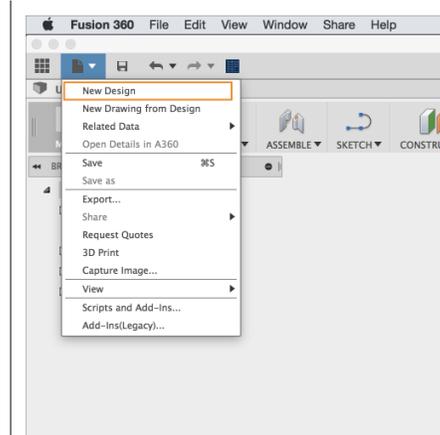
Lesson 2: Creating a sketch

Learning Objectives

1. Create a 2D sketch
2. Use construction geometry
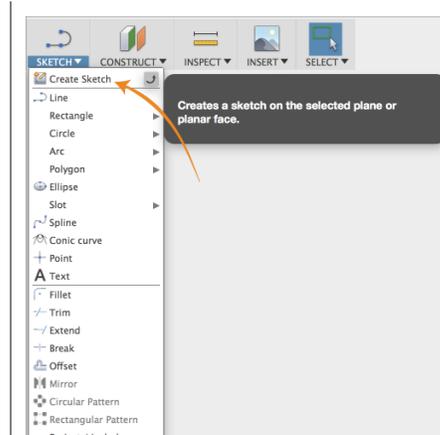3. More advanced use of constraints

Step 1: Create new Design - Let's start with creating a new design. We're going to use this to create a new part.

1. Launch Fusion 360.
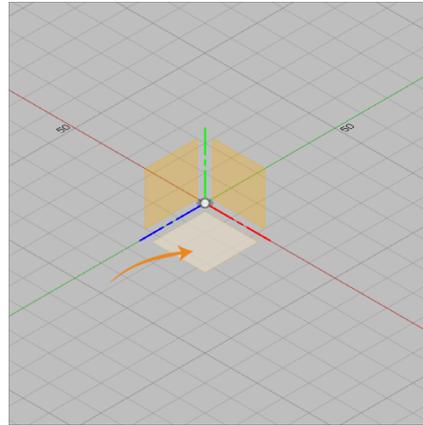2. Start a new design.

Step 2: – Start the **Sketch** command

1. Select **Sketch** > **Create Sketch**.

Step 3: – Select the sketch plane

1. You are now prompted to select a "plane" to sketch on.
2. Select the "Front" (XY) plane.

Note: Aside from the origin planes, you can create sketches on one of the 3 default planes, on a custom construction plane or on an existing model face, more on this later.

Step 4: – Create Lines

1. Select **Sketch > Line.**
2. Select the sketch origin.
3. Click to end the line
4. Continue sketching lines as follows
5. Press **esc** to exit the command

Note: Note as you place the lines some constraints are "automatically" created. If you do not get exactly the same ones don't worry. Also try to make sure your first line is roughly 500mm it is good practice to sketch shapes "close" to the correct size.
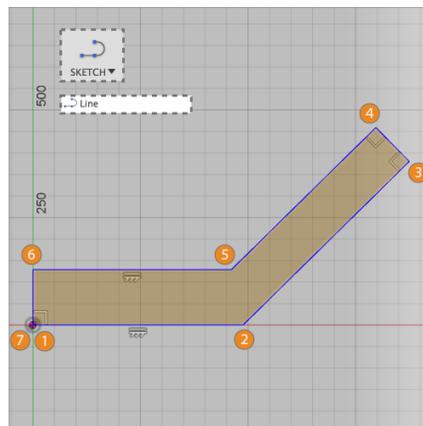

Step 5: – Create Constraints

1. Select the **Perpendicular** constraint
2. Select two lines
3. Repeat 3 times (1)
4. Select **Horizontal/Vertical**
5. Select the two lower lines (2)
6. Press **esc** to exit the command

Note: The constraint commands are in the Sketch Palette on the right side of the screen. Some of these constraints may already be created. If they are don't bother recreating them.
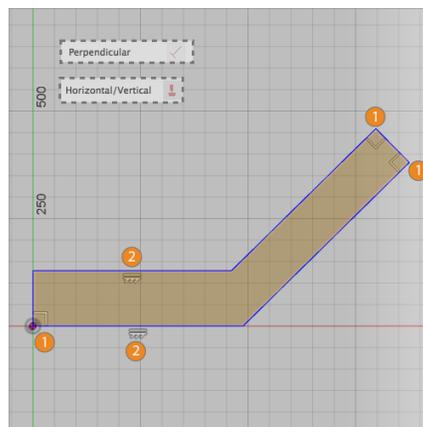
Step 6: – Create Equal Constraint

1. Select **Equal constraint**
2. Select the two pairs of lines shown
3. Press **esc** to exit the command
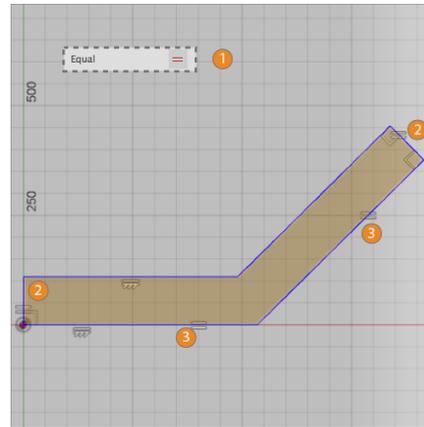
Step 7: – Create Dimension

1. Select **Sketch > Sketch Dimension.**
2. Select the line
3. Click again to place the dimension.
4. Type in a value of 500 mm.
5. Press **Enter** to accept the value.

Note: The dimension command is still active and you can go right into placing the next dimensions.
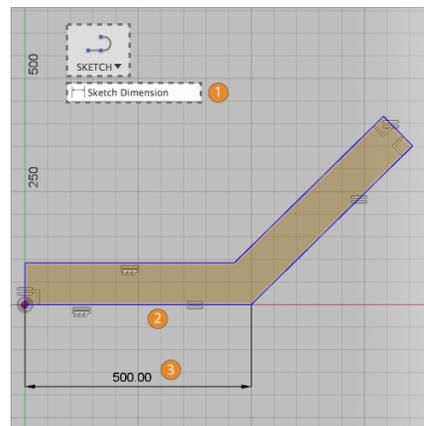

Step 8: – Create Angle Dimension

1. Select **Sketch > Sketch Dimension.**
2. Select the bottom line
3. Select the angled line
4. Place the dimension
5. Type 45
6. Press **Enter** to accept the value

Note: The dimension command is still active and you can go right into placing the next dimensions.
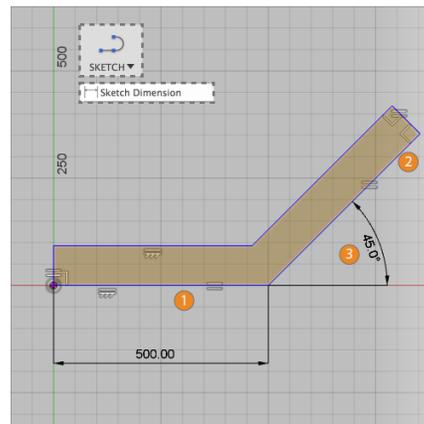
Step 9: – Aligned Dimension

1. Select **Sketch > Sketch Dimension.**
2. Select the upper right line
3. Move just away from the line
4. Notice a small icon on the cursor
5. Select in space again to begin an aligned dimension
6. Type 100
7. Press **Enter** to accept the value
8. Press **esc** to exit the command
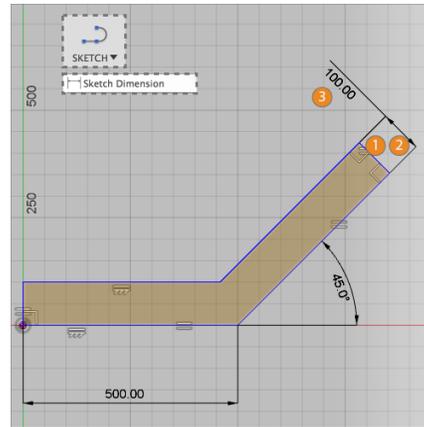
Step 10: – Construction Geometry

1. Select Upper line
2. Hold **Shift,** select second line
3. Press **Right Mouse Button**
4. Select **Normal/Construction**

Note: Construction geometry is not considered when looking for profiles. Use construction geometry for reference when creating sketches. It will show as dashed lines to indicate it is construction geometry.
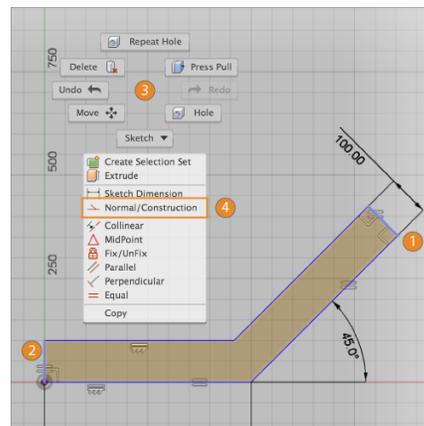
Step 11: – Create 2 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the bottom line midpoint
3. Select the intersection shown
4. Repeat at the top edge
5. Press **esc** to exit the command

Note: Make sure you "snap" the geometry in place. You should see a midpoint (triangle) constraint created. Try dragging the circles, if they move you missed a snap.
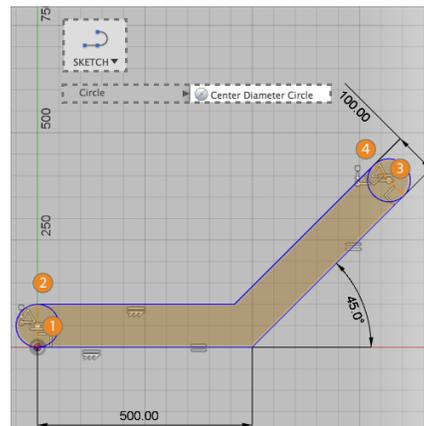
Step 12: – Sketch Fillet

1. Select **Sketch > Fillet**
2. Select the intersection point
3. Select the other intersection point
4. Enter a value of 100 mm
5. Press **Enter** to confirm

Note: All fillets created at the same time will have an equal radius. Create them separately to have different radius values. Note in many cases creating a fillet in a sketch isn't the best choice. As you see here it deletes the 500 mm dimension.
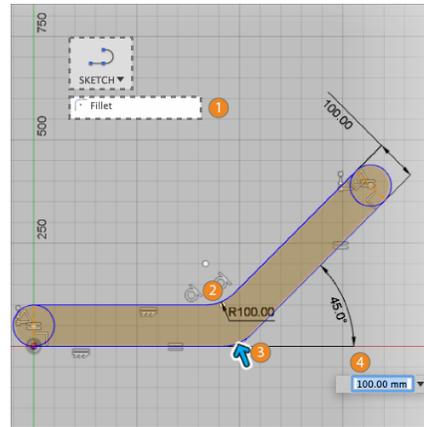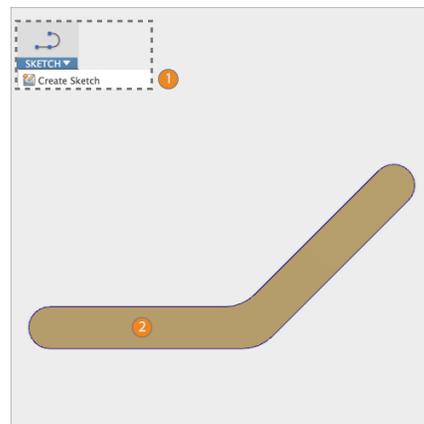

Step 13: – Extrude the profile

1. Select **Modify > Press Pull.**
2. Select the 3 profiles
3. Drag the arrow up or type in a value of **50 mm** to set the depth.
4. Press **OK.**

Note: One sketch may contain multiple profiles. Here we limited the number of profiles by using construction geometry.
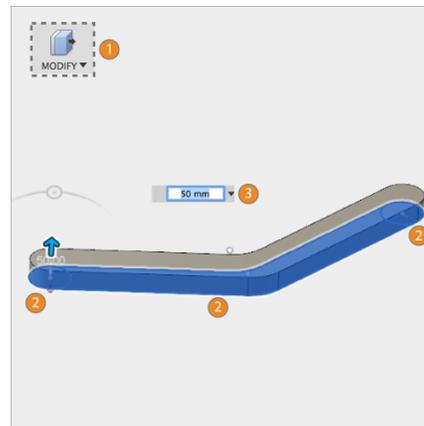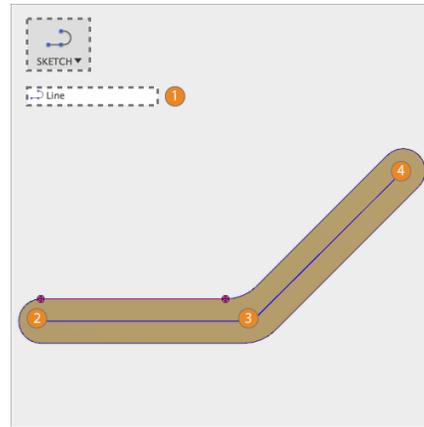

Step 14: – Create another sketch

1. Select **Sketch > Create Sketch.**
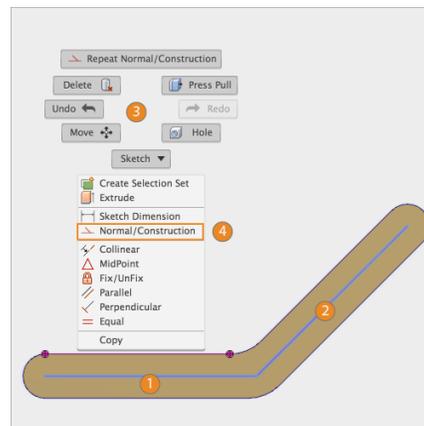2. Select the top of the bracket.

Step 15: – Sketch Lines

6. Select **Sketch > Line**
7. Select the left center point
8. Select near the middle
9. Select the upper center point
10. Press **esc** to exit the command

Step 16: – Construction Geometry

1. Select one line
2. Hold **Shift,** select second line
3. Press **Right Mouse Button**
4. Select **Normal/Construction**

Step 17: – Create Parallel Constraints

1. Select **Parallel**
2. Select lower line
3. Select lower edge
4. Select angled line
5. Select Angled edge
6. Press **esc** to exit the command

Note: This actually fully constrains these lines. It is worthwhile to learn the different strategies you can take to fully define the sketch lines.
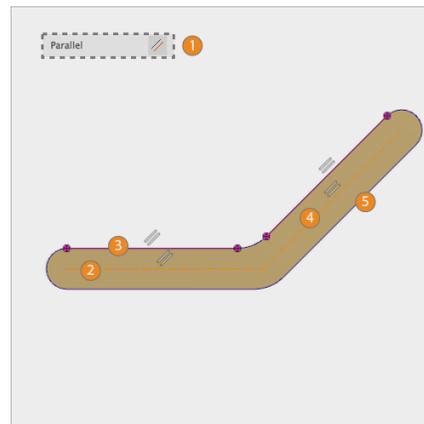
Step 18: – Create 3 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the lower left center point
3. Type **50mm**
4. Press **Enter**
5. Draw 2 more circles

Note: Make sure to "snap" to the end points of the lines when placing the circle center points.
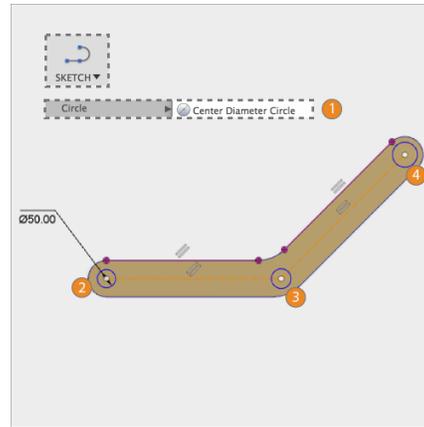
Step 19: – Create 2 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the lower line midpoint
3. Click to place the circle
4. Draw 1 more circle at the midpoint of the angled line

Note: Make sure to "snap" to the midpoints of the construction lines. You should see a small triangle appear to indicate your circle is locked to the midpoint.
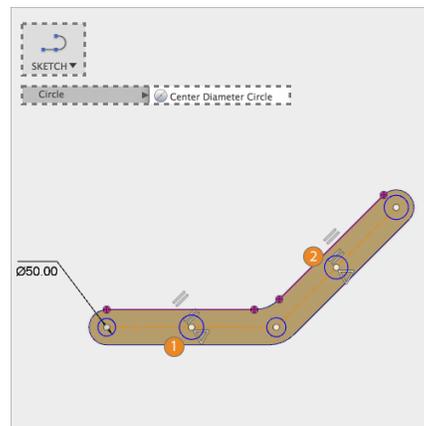
Step 20: – Create Equal Constraints

1. Select **Equal**
2. Select 2 circles
3. Repeat until you have all circles equal
4. Press **esc** to exit the command

Note: You have to select the circles in pairs. One equal constraint is always applied to two circles. So for each constraint you need to select two circles.
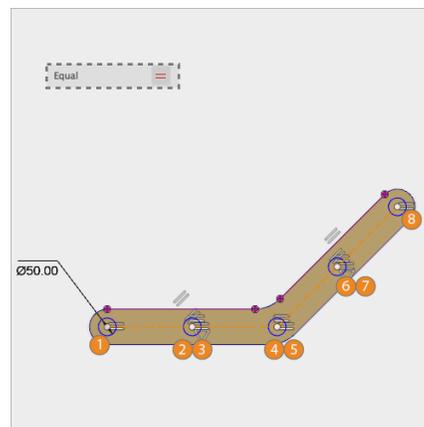
Step 21: – Extrude Circles

1. Select **Modify > Press Pull.**
2. Select the 3 profiles
3. Drag the arrow into the part.
4. Make sure **Cut** is selected
5. Select **All** from the menu
6. Press **OK.**

Note: One sketch may contain multiple profiles. Make sure to select the 5 circles. You can add or remove profiles from a selection later by using the **CMD** key on mac and **CTRL** key on windows.
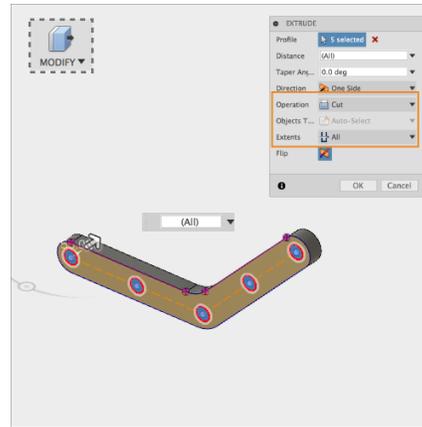
Step 22: – Lesson complete!

Congratulations you have finished this lesson and have learned more ways to create relationships in a sketch using constraints and construction geometry. In the next lesson, you'll go through the fundamentals of how to further develop 3D models using various modeling tools.