Lesson: Fully Define a Complex Sketch

In this lesson, you’ll add dimensions and constraints to fully define a sketch.

Learning Objectives

- Apply sketch dimensions.
- Apply sketch constraints.
- Identify graphic changes to fully defined sketch entities.

The completed exercise

1. Upload and open the supplied file “Attachment Bracket.f3d”.

2. Expand the Sketches folder inside the Browser. Notice Sketch2, Sketch3, and Sketch4 have red lock icons. These lock icons indicate these sketches are fully defined. Sketch1 has a pencil icon because the sketch is underdefined.
3. Right-click on Sketch1 and choose the Edit Sketch option from the drop-down menu.

4. Some of the sketch’s entities are blue and some are black. The black sketch entities are fully defined and the blue entities are underdefined.

5. Select one of the blue sketch entities and drag it to a new location. If dragged far enough, the sketch’s intent will be ruined. Fully defining sketches will keep this from accidentally happening. Press Ctrl + Z to undo the modification.

6. Click Create> Sketch Dimension. Pressing the D key on the keyboard will also open the Sketch Dimension tool.
7. Click on the lower radius and drag the dimension to an open area of the Canvas. Inside the value field, type 50 then press the Enter key. Assigning a dimension to this radius fully defines the entire sketch. Every sketch element turns black.

8. The new radius value can be edited by double-clicking on the dimension. Double-click on the dimension and change the radius to 40. Press the Esc key to leave the Dimension tool.

9. Right-click on the dimension and select the Toggle Diameter option from the Marking Menu’s drop-down menu. This will toggle the dimension from being a driving dimension to a driven dimension. The new driven dimension will be inside parentheses to indicate that it is driven. A driven dimension simply measures a current value instead of dictating the intended value.
10. Because the radius of the sketch element is not identified, the sketch returns to being underdefined and some entities are blue again. As the blue sketch elements are modified, the driven radius value changes to indicate the new radius.

11. Adding dimensions is not the only way to fully define a sketch; constraints can also be added. Click Constraints> Concentric.

12. Select the two arcs as the entities to make concentric. Notice the center point for the radii are now in the same location and all the sketch entities turn black. The sketch is once again fully defined. Press the Esc key to leave the Constraint tool.

13. Just because a sketch is fully defined does not mean it cannot be edited. Double-click on the dimension describing the 120° angle between the edges in the image on the right. Change this dimension to 130°. All the sketch entities update to reflect the new angle.
14. The two holes in the image on the right are black to indicate they are fully defined. The top hole has a dimension of 8 mm applied and there is an equal constraint between the top hole and the bottom hole; this means the bottom hole is also 8 mm. The location of the holes is defined by an equal constraint on the construction lines next to the holes. The bottom hole has a 10 mm construction line driving the distance between the hole and the bottom edge. The top hole’s construction line has an equal constraint with the bottom hole’s 10 mm line. These constraints fully define the two holes while using only a couple dimensions.

15. Some sketches have many dimensions and constraints that can clutter the view. The dimensions and constraints can be hidden using the options in the Sketch Palette.

16. With the dimensions and constraints hidden, selecting a sketch element will reveal only the associated dimensions. After you have finished exploring the constraints and dimensions, click the Finish Sketch button in the Sketch Palette.
17. The design rebuilds to accommodate the updated dimensions in the sketch. Notice in the timeline that a chamfer feature turns red because it was not able to rebuild properly. Drastic changes to sketches sometimes will cause a feature to fail. Right-click on Chamfer1 and select the Edit Feature option.

18. Notice there are a few areas where the part’s edges not selected for chamfer. Click on the missing edges to make sure all edges are selected for the chamfer feature. Even after the selections are made, the Edit Feature control panel’s OK button stays grayed out because something else is wrong.

19. Because of the updated geometry, one of the features did not cut all the way through the part. A small, thin wall remains over the ends of one of the slots. Press the cancel button inside the Edit Feature control panel.

20. The Extrude3 feature needs to be adjusted so the part is created correctly. Drag marker at the end of the timeline to the point after the Extrude3 feature. Right-click the Extrude3 feature and select the Edit Feature option.
21. The selected profile needs to cut upwards slightly to remove the material from the radiused edge.

22. In the Edit Feature control panel, change the Direction option to Symmetric. This will cut the profile through all the part’s material in both directions. Click the OK button.

23. Now that the Extrude3 feature cuts completely through the part as intended, the chamfer feature can be fixed. Roll the timeline marker to the end of the timeline, right-click the Chamfer1 feature, then select the Edit Feature option.

24. Select all the part’s external edges to make sure everything is selected. You may need to hold down the Ctrl key to select edges. Once the selection is complete, click the OK button in the control panel; the chamfer is applied to the edges.
25. Notice in the Browser's Sketches folder, all sketches are now fully defined and have the red lock icon. Save the file and continue to the next module.