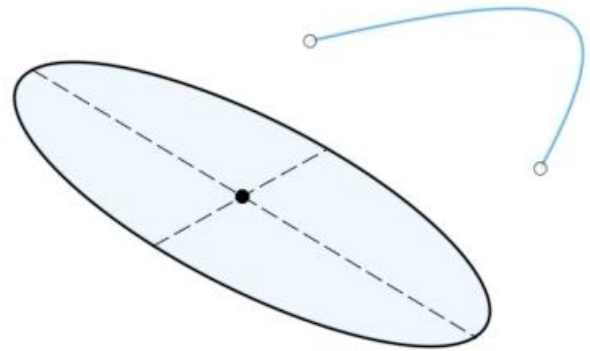# Sketch splines and slots

In this module, you'll explore ways to create and modify various types of sketch geometry.

Learning objectives:

- Create a sketch slot.
- Create a spline.
- Create a conic.

The completed exercise

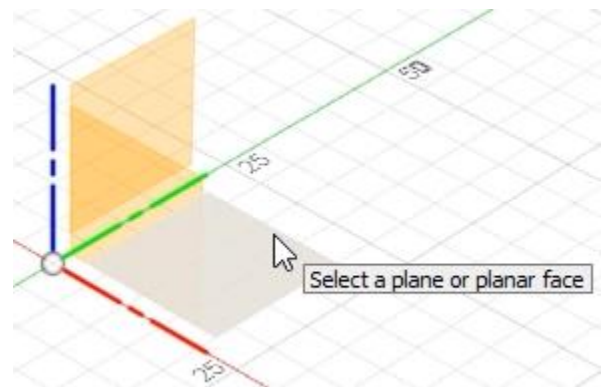1. Create a new untitled design inside Fusion 360. Create a new sketch on the XY plane.

Figure 1. Create a sketch on the XY plane

2. Click Create> Slot> Center Point Slot.

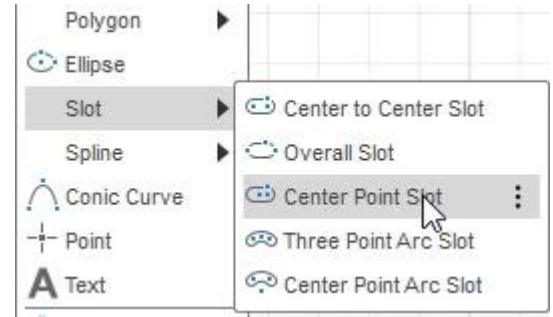| Polygon | ▶ |
⊙ Ellipse	
Slot	▶	⊙ Center to Center Slot
Spline	▶	� Overall Slot
⌒ Conic Curve		⊙ Center Point Slot ⋮
⊣ Point		⊛ Three Point Arc Slot
A Text		⊙ Center Point Arc Slot

Figure 2. Open the Center Point Slot tool

3. Click to place one of the line's ends at the sketch's origin, then click to place the second end roughly 45° away from the origin.
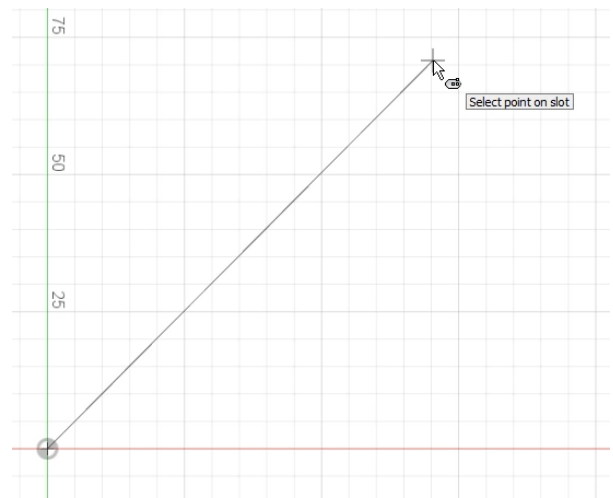

Select point on slot

Figure 3. Place the line's two ends

4. Click a third time to determine the slot's width. Press Esc to leave the Slot tool.
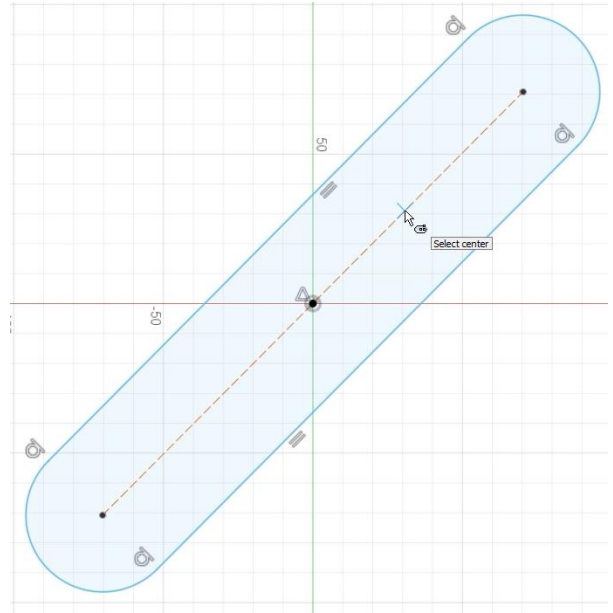

Figure 4. Determine the slot's width

5. The slot is underdefined because no dimensions or constraints are added. Click and drag one of the slot's points to a new position and notice that the slot is allowed to adjust from its original geometry.
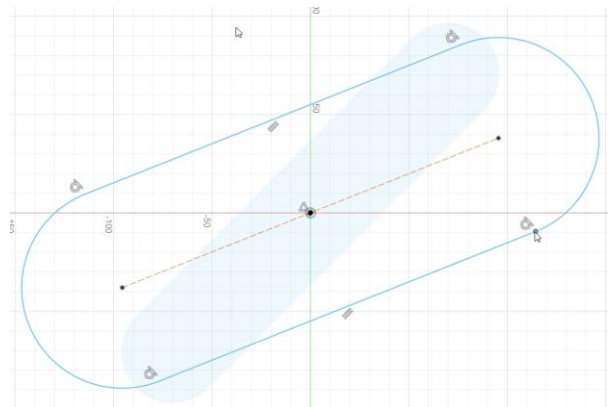

Figure 5. Adjust the slot

6. Select the slot's dashed construction line and click Constraints> Horizontal/Vertical. Depending on whether the line is closer to horizontal or vertical, a horizontal or vertical constraint will be applied to it.
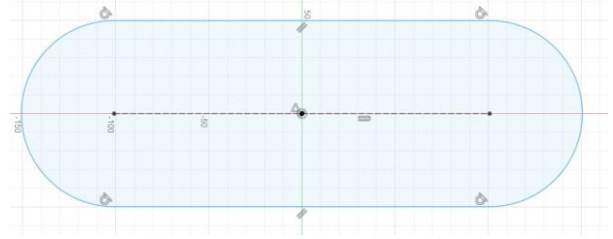

Figure 6. Add a horizontal constraint to the slot's construction line

7. Press D to open the Dimension tool and add a **150 mm** dimension to the slot's construction line. Add a **25 mm** dimension for the slot's radius. Notice that the constraints and dimensions have fully defined the slot's geometry and it turns black.

Figure 7. Dimension the slot

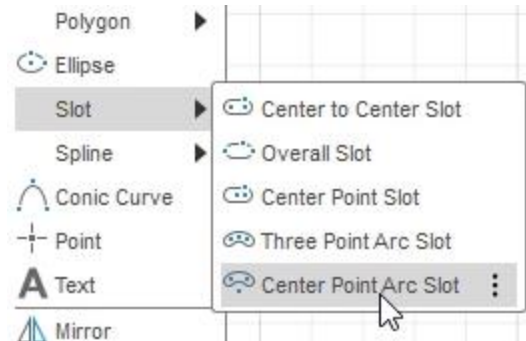**8.** Click Create> Slot> Center Point Arc Slot.

Figure 8. Open the Center Point Arc Slot tool

9. First select the sketch's origin as the center of the new slot's arc, then select the arc's start point and endpoint. Click a force time to determine the slot's radius.
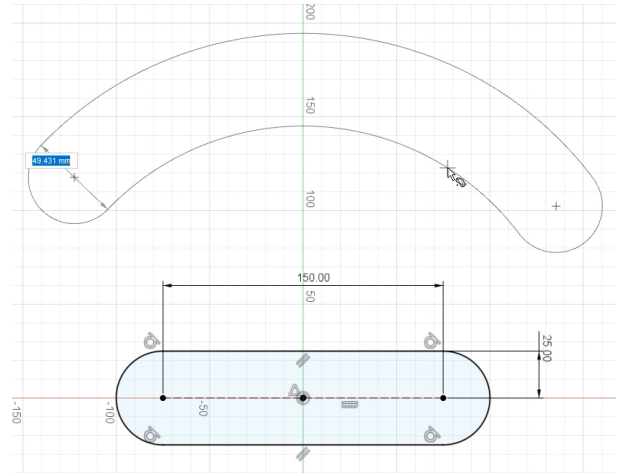

Figure 9. Create the new arced slot

10. Press D to open the Dimension tool and specify an arc of 150 mm for the slot's curve. Next, add a 15 mm dimension to the arc's radius. Use the Horizontal/Vertical constraint tool to add a horizontal constraint between the arc's two endpoints.
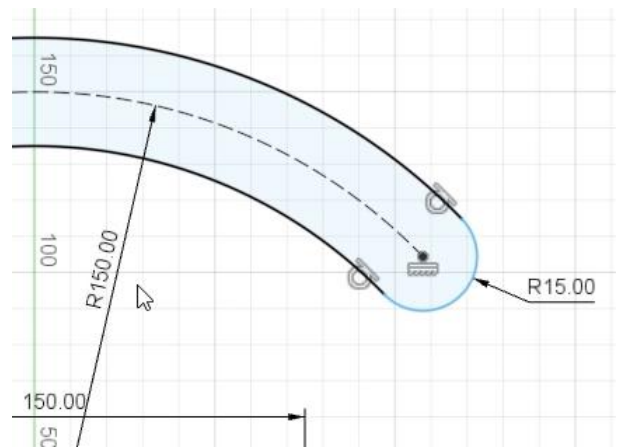

Figure 10. Add dimensions and constraints to the new arc

11. Press L to open the Line tool and draw a line connecting the end of the arced slot's construction line to the sketch's origin. Press Esc to end the Line tool. Select the line you just drew then press X to convert it to construction geometry.
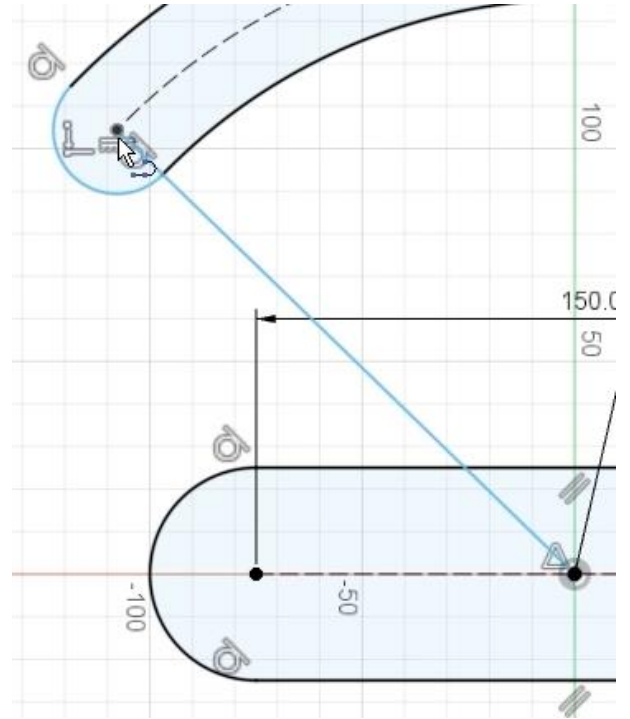

Figure 11. Draw a line

12. Press D to open the Dimension tool and add a **135°** dimension between the new construction line and the original slot's construction line.

Figure 12. Add an angle dimension between the two construction lines

13. Both slots should now be fully defined. Finish the sketch by clicking Finish Sketch> Finish Sketch.
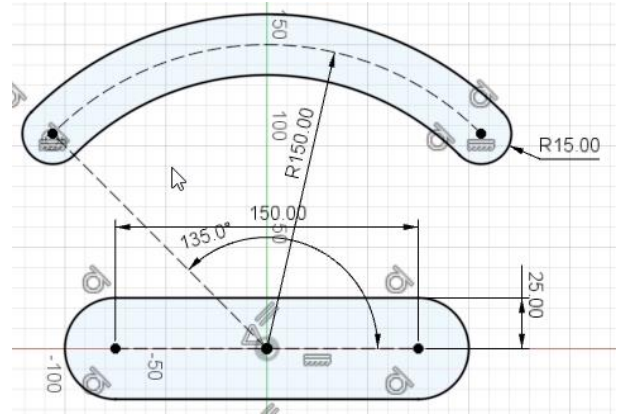

Figure 13. Inspect the result

14. Hide Sketch1 by expanding the Sketches folder and clicking the eyeball icon next to Sketch1.
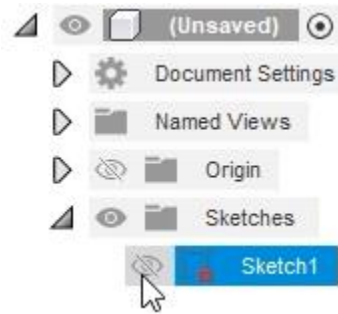

Figure 14. Hide Sketch1

15. Create a new sketch on the XY plane, then click Create> Spline> Fit Point Spline.
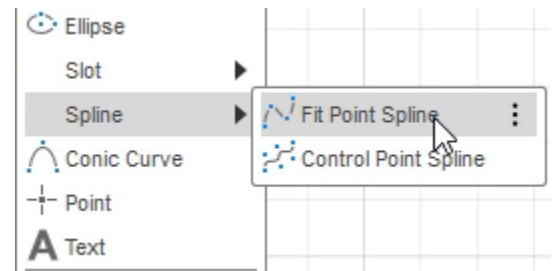

Figure 15. Open the Fit Point Spline tool

16. Click in three random places to place the spline's three fit points, then click the green checkmark. Press Esc to leave the Spline tool.
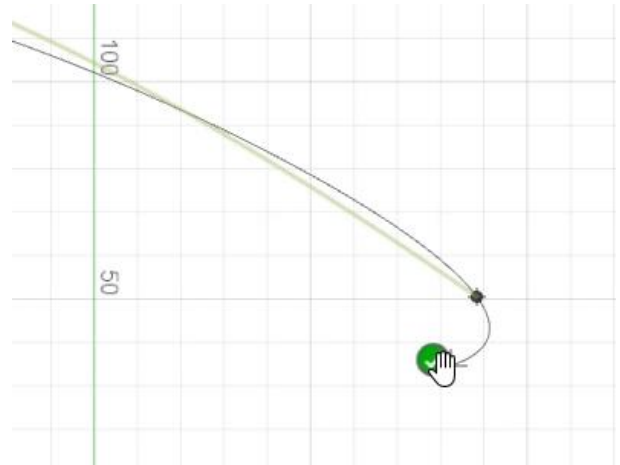
Figure 16. Place the spline's fit points

17. Note that each fit point you placed has a pair of green handles. These handles control the spline's tangency direction and weight.

Figure 17. Note the green handles

18. A spline's fit point can be deleted by selecting it and pressing Delete.

Figure 18. Delete a spline's fit point

19. The fit point can be added to the spline by selecting the spline, right-clicking it, then choosing Insert Spline Fit Point. Click the spline where you want to add the new fit point.
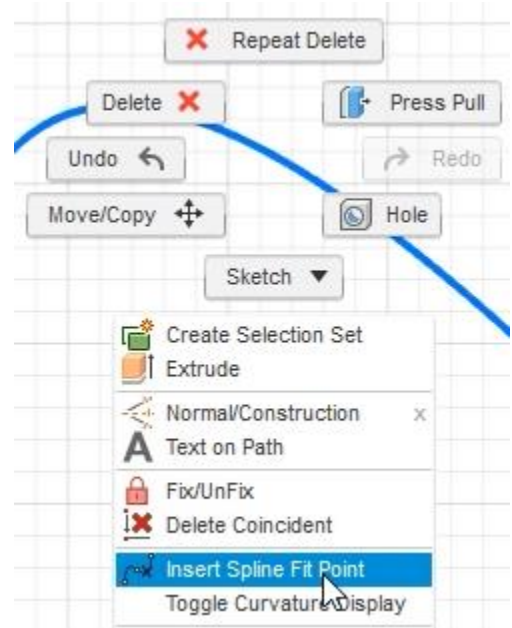

Figure 19. Insert a new fit point

20. Click and drag the green handle's endpoint to adjust the spline's geometry.
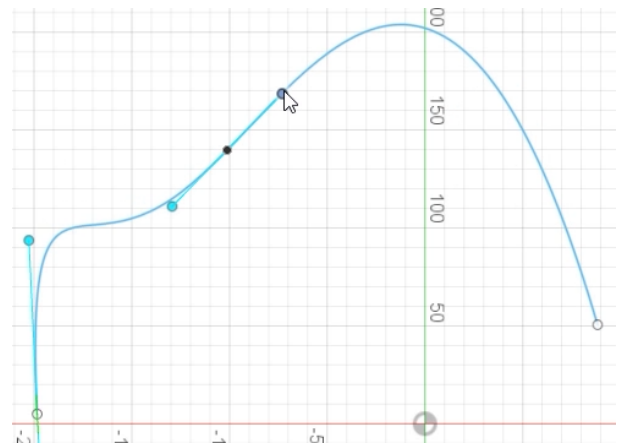

Figure 20. Adjust the spline

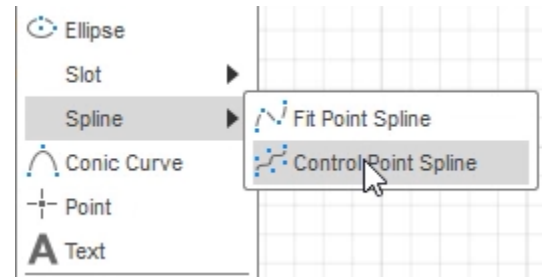**21.** Click Create> Spline> Control Point Spline.

Figure 21. Open the Control Point Spline tool

22. Click three times in the Canvas area to place the three control points, then click the green checkmark. Press Esc to leave the Control Point Spline tool.
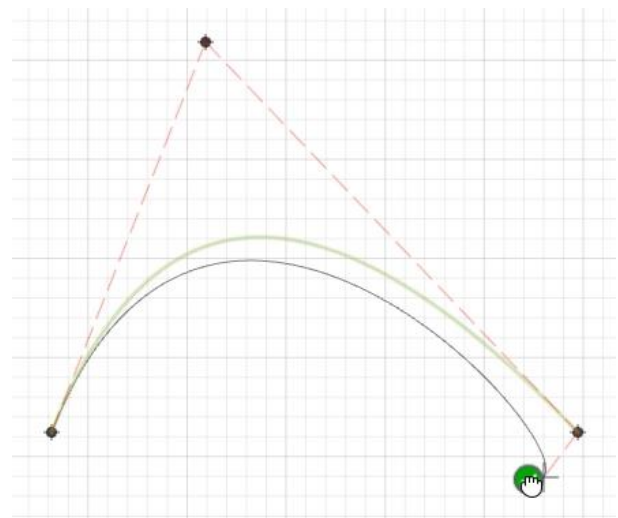

Figure 22. Place the spline's three control points

23. Notice that an external cage controls the spline's geometry. Click and drag the cage's control point to adjust the spline's curvature. When creating Fit Point Splines, using the fewest possible number of fit points is important if you want to produce clean, smooth geometry. When creating Control Point Splines, additional control points can help finesse the curve's transitions.
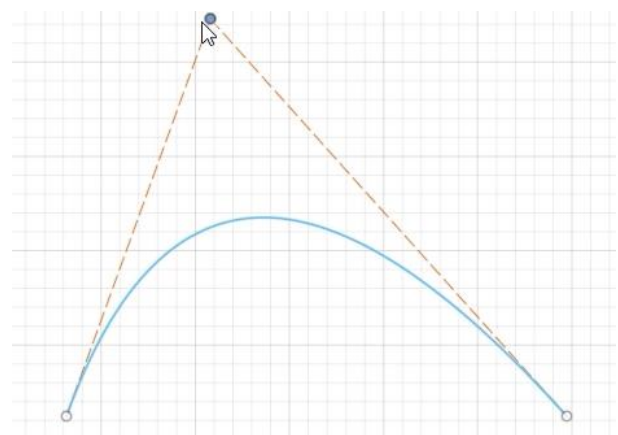

Figure 23. Adjust the spline's curvature

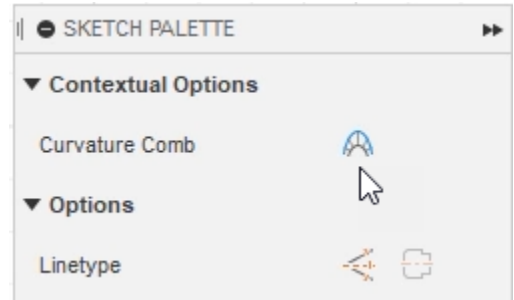**24.** Select one of the splines and activate the Sketch Palette dialog's Curvature Comb option.

Figure 24. Activate the curvature combs

25. Curvature combs are added to the spline and help illustrate the smoothness of the spline's curvature. Finish the sketch by clicking Finish Sketch> Finish Sketch.
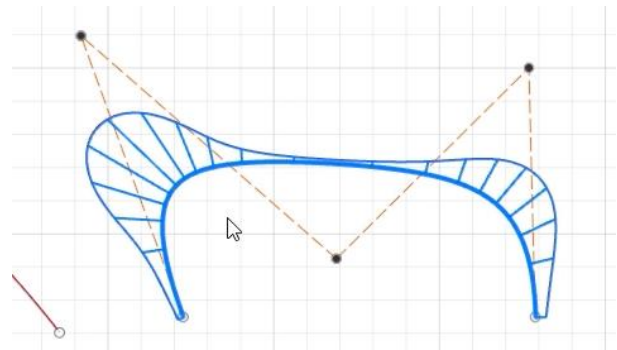

Figure 25. Inspect the curvature combs

26. Use the Browser to turn off the visibility for Sketch2, then create a new sketch on the XY plane. Click Create> Conic Curve.
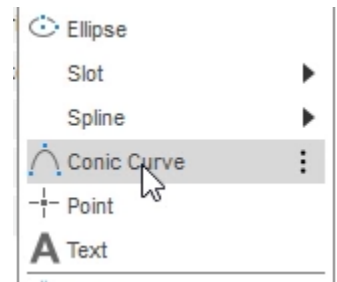

Figure 26. Open the Conic Curve tool

27. Click three times in the Canvas area to place the curve's three points, then notice that an arrow appears near the middle point. This arrow allows you to adjust the Rho value. The closer the Rho value is to 1.0, the closer the curve is to a cone shape.
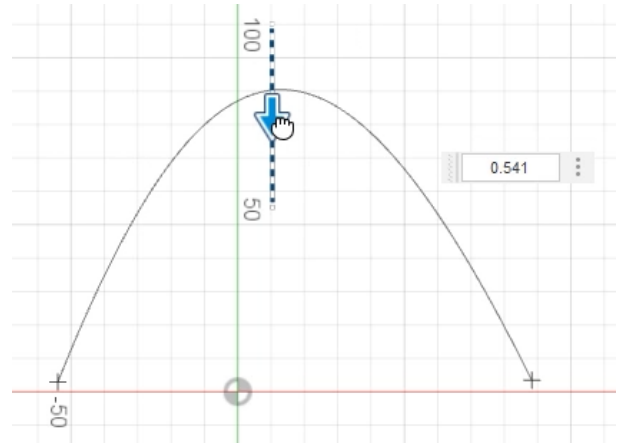
Figure 27. Adjust the curve's Rho value

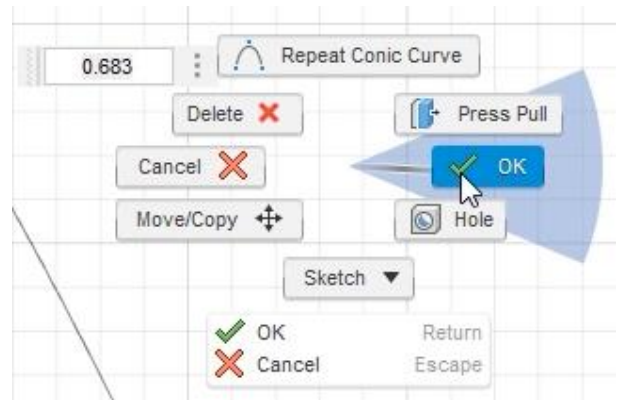**28.** Right-click in the Canvas area and choose OK from the Marking Menu.

Figure 28. OK the conic curve
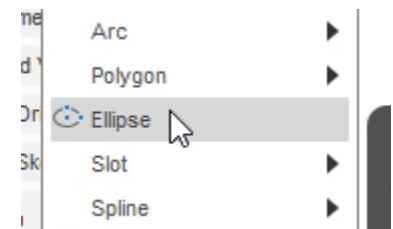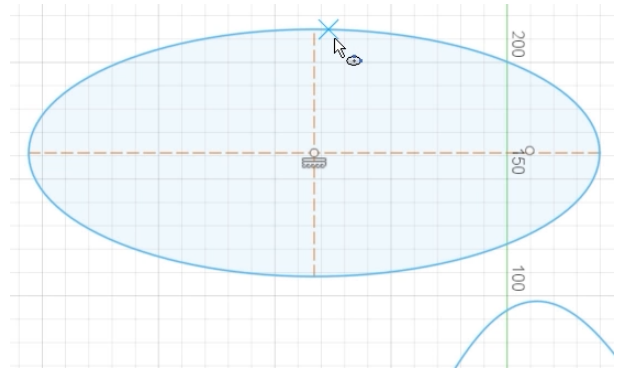
29. Click Create> Ellipse.

Figure 29. Open the Ellipse tool

30. Click three times in the Canvas area to draw the ellipse. The first click will determine the center point, the second click will determine the width, then the third will determine the height. Press Esc to leave the Ellipse tool.

Figure 30. Draw the ellipse

31. Just like any other sketch entity, you can add dimensions and constraints to ellipses. Click Finish Sketch> Finish Sketch. Save the file and continue to the next module.
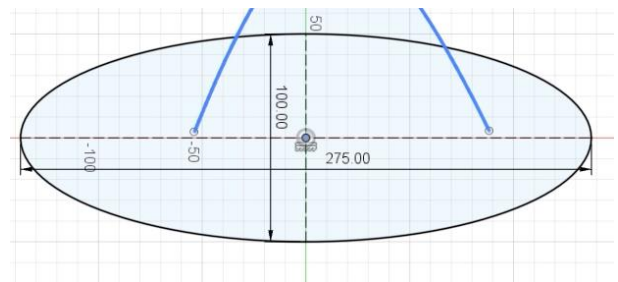
Figure 31. Dimension and constrain the ellipse