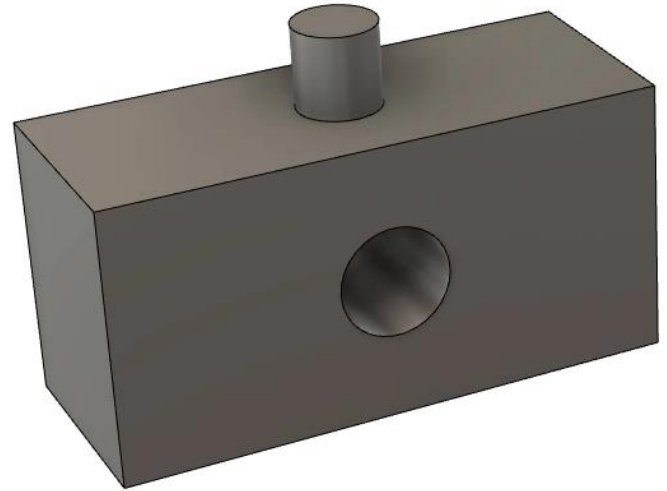


Create parameter-based sketches

In this module, you'll create and modify custom User Parameters to drive sketches and a solid body's dimensions.

Learning objectives:

- Create a user parameter.
- Create a construction plane.
- Use Circle and Rectangle.
- Link sketch dimensions and parameters.



The completed exercise

1. Create a new untitled document in Fusion 360. This module will use millimeters for the units.



Figure 1. Set the document's units to millimeters

2. Click Modify > Change Parameters.

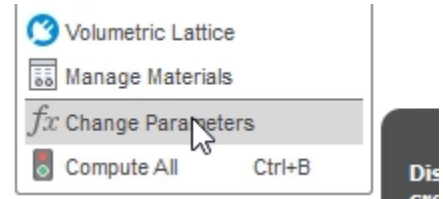


Figure 2. Open the Parameters dialog

3. Click the plus icon next to User Parameters to create a new custom User Parameter.

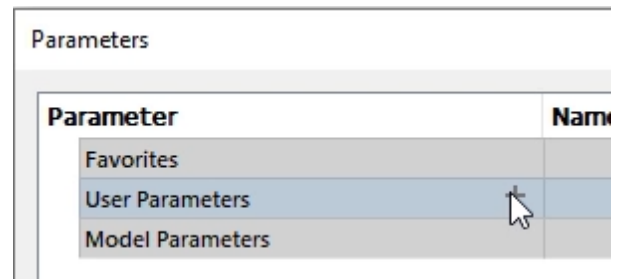


Figure 3. Create a new User Parameter

4. Since Spaces cannot be used in the parameter's name, enter **Plane_Offset** as the new parameter's name. Enter **500** into the Expression box, then OK the dialog.

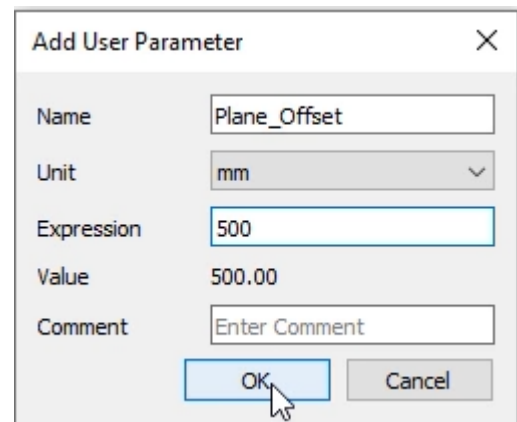


Figure 4. Create the Plane_Offset User Parameter

5. The new custom User Parameter is added to the User Parameters section. OK the Parameters dialog.

Parameter	Name
Favorites	
▼ User Parameters +	
☆ User Parameter	Plane_Offset
Model Parameters	

Figure 5. Inspect the new User Parameter

6. Click Construct > Offset Plane.



Figure 6. Open the Offset Plane tool

7. Choose the YZ plane as the dialog's Plane selection.

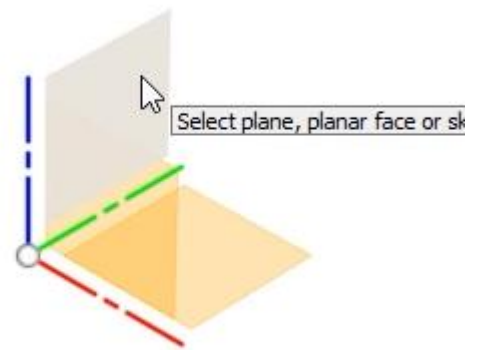


Figure 7. Choose the YZ plane

8. Instead of typing a number into the value box, begin typing Plane_Offset. Choose the Plane_Offset option from the menu. Press Enter. The new plane is created 500 mm away from the origin's YZ plane.

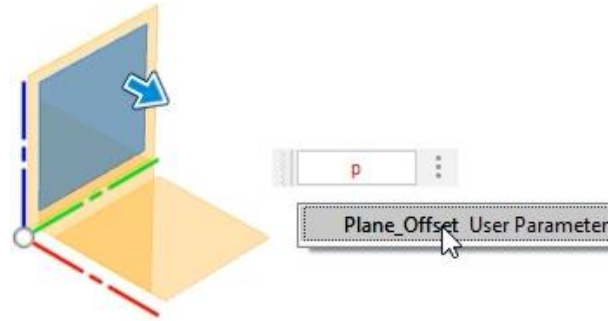


Figure 8. Enter the User Parameter as the value

9. Click Modify> Change Parameters to open the Parameters dialog, then click the plus icon to create a new User Parameter.

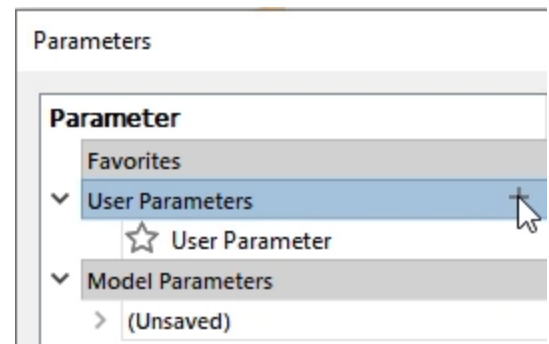


Figure 9. Create a new User Parameter

10. Use the image on the right to create a Length parameter. OK the dialog to create the new parameter.

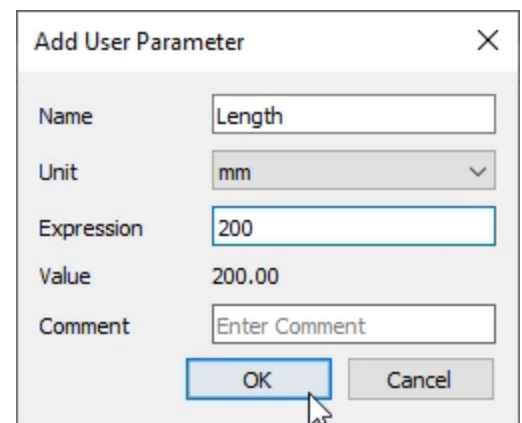


Figure 10. Create a Length parameter

11. Click the plus icon to create another new User Parameter, then use the image on the right to create a Width parameter. OK the dialog.

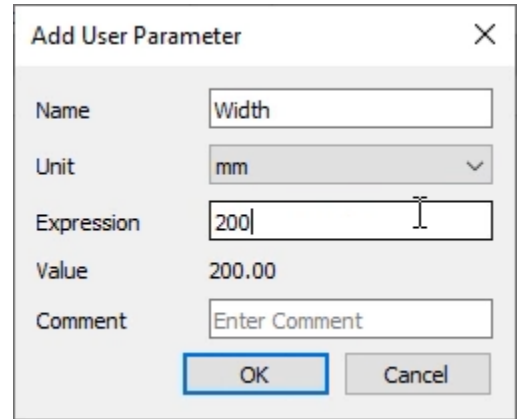


Figure 11. Create a Width parameter

12. Click the plus icon to create another User Parameter, then use the image on the right to create a Hole_dia parameter. This parameter will be used to drive a hole's diameter. OK the dialog. Also OK the Parameters dialog.

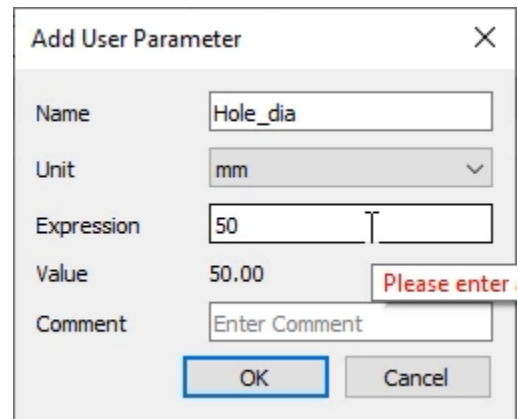


Figure 12. Create a Hole_dia parameter

13. Turn on the visibility for the origin by clicking the eyeball icon next to the Browser's Origin folder.



Figure 13. Show the origin

14. Right-click the origin's YZ plane, then choose Create Sketch from the menu.

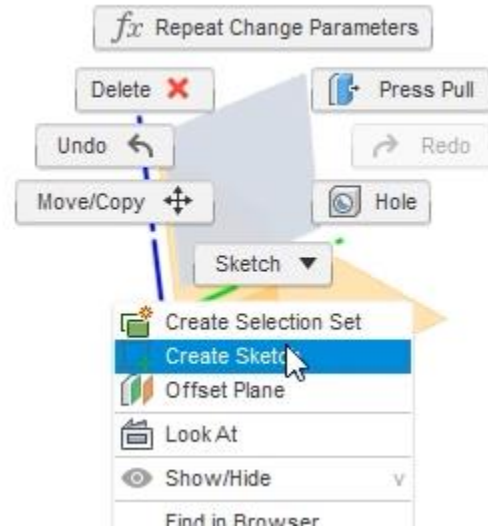


Figure 14. Create a sketch on the YZ plane

15. Click Create > Rectangle > Center Rectangle.

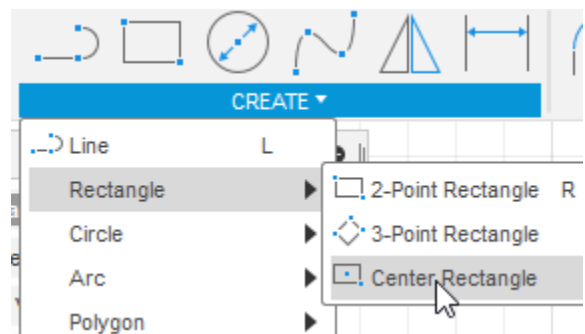


Figure 15. Open the Center Rectangle tool

16. Place the rectangle's center at the sketch's origin but don't click a second time to determine the rectangle's size.

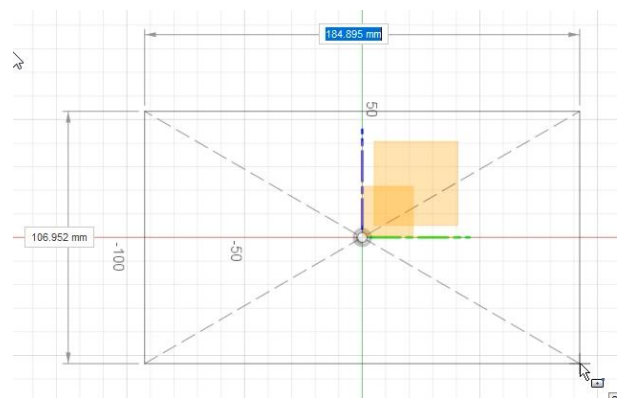


Figure 16. Begin drawing a rectangle

17. Press Tab to navigate to the first dimension box. Begin typing Length, then choose the Length parameter from the menu. Press Enter.

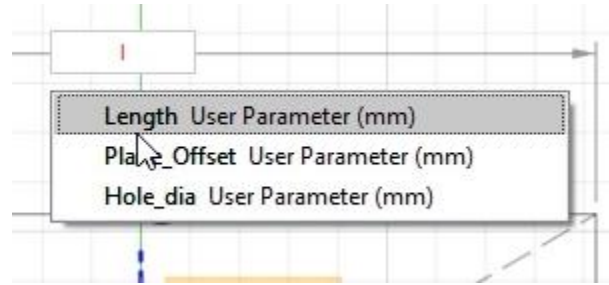


Figure 17. Choose the Length parameter

18. Press Tab to navigate to the next dimension box, then begin typing Width. Choose the Width parameter, then press Enter. Press Enter again to accept the new rectangle.

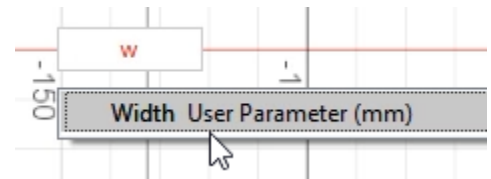


Figure 18. Choose the Width parameter

19. Click Create> Center Diameter Circle.

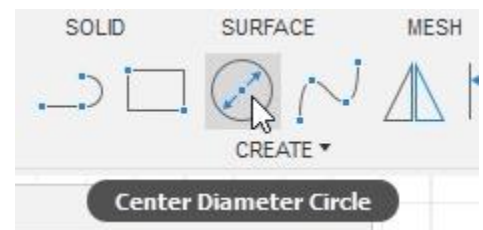


Figure 19. Open the Center Diameter Circle tool

- 20.** Begin drawing a circle at the sketch's origin, but do not specify the circle's size.

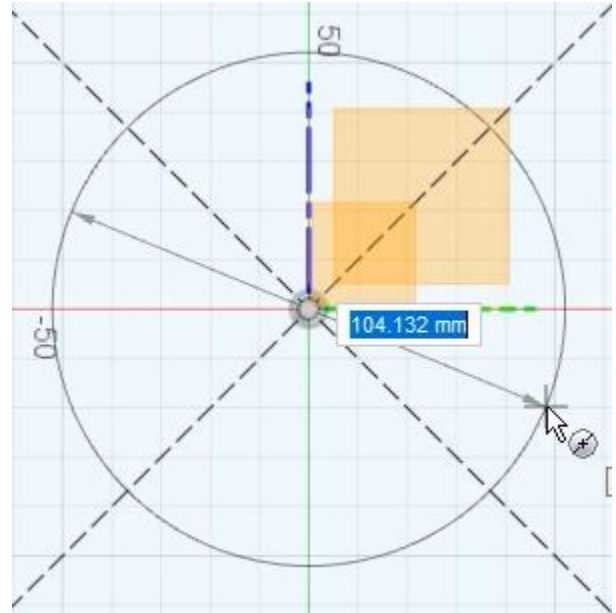


Figure 20. Draw a circle

- 21.** Press Tab to navigate to the circle's dimension box, then use the Hole_dia parameter as the dimension. Press Enter to accept the parameter, then press Enter again to complete the circle.

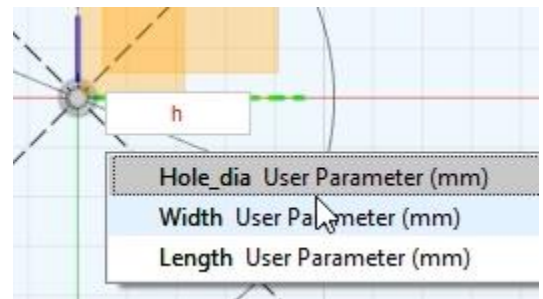


Figure 21. Use the Hole_dia parameter for the new circle

- 22.** Click Finish Sketch> Finish Sketch to end the sketch.



Figure 22. End the sketch

23. Press E to open the Extrude tool, then select the region shown in the image on the right.

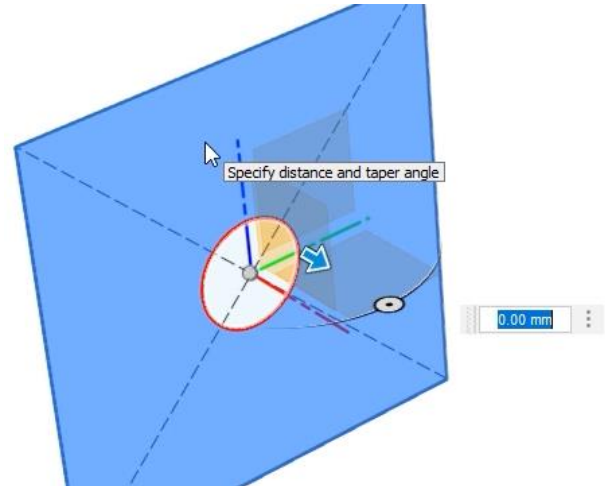


Figure 23. Open the Extrude tool

24. Choose the To Object option from the dialog's Extent Type menu.

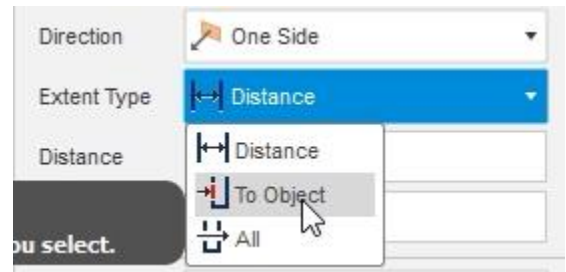


Figure 24. Choose the extent type

25. For the dialog's Object selection, choose the offset plane you created earlier in this module. OK the dialog.



Figure 25. Choose the plane as the Object selection

26. The Plane_Offset parameter could also be entered into the dialog's Distance box as an alternate way to specify the extrude's length.

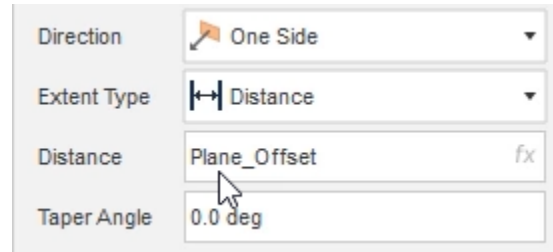


Figure 26. Use a parameter in a dialog's Distance box

27. Click Modify> Change Parameters and reduce the Plane_Offset parameter's expression to 200.

Name	Unit	Expression
Plane_Offset	mm	200
Length	mm	200 mm
Width	mm	200 mm
Hole_dia	mm	50 mm

Figure 27. Change a parameter's expression

28. Notice that the solid body's length automatically updates to match the parameter's new expression.

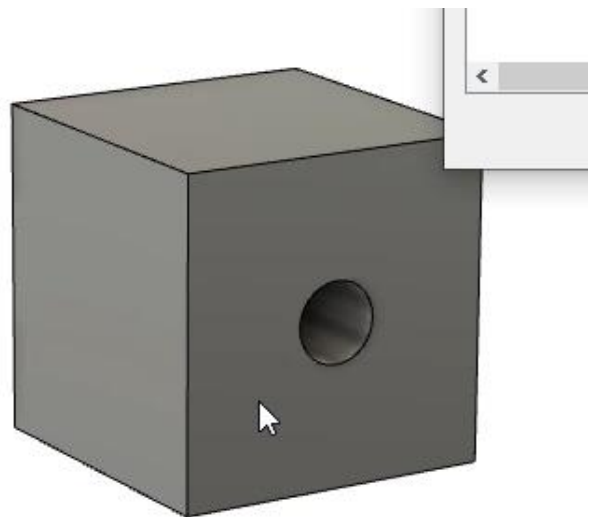


Figure 28. Notice that the solid body is updated

29. Continue to adjust the User Parameters' expressions and notice how the model updates. OK the Parameters dialog after you finish.

Name	Unit	Expression
Plane_Offset	mm	200 mm
Length	mm	500 mm
Width	mm	250 mm
Hole_dia	mm	100 mm

Figure 29. Adjust the User Parameters

30. Right-click the solid body's top face and choose Create Sketch.

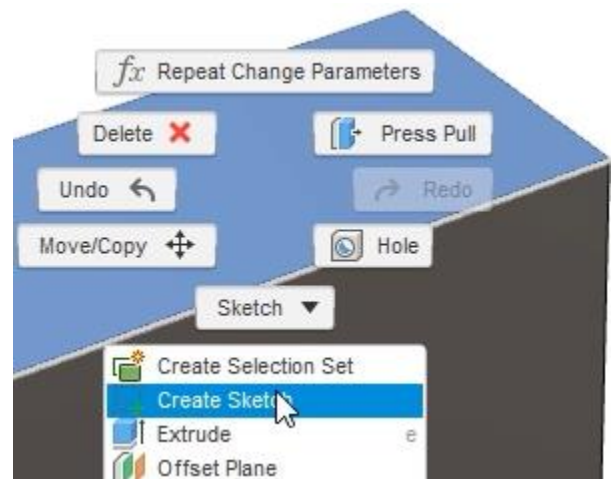


Figure 30. Create a new sketch

31. Press C to open the Circle tool and draw a circle. Click Constraints> Horizontal/Vertical, then add a horizontal constraint between the sketch's origin and the circle's center.

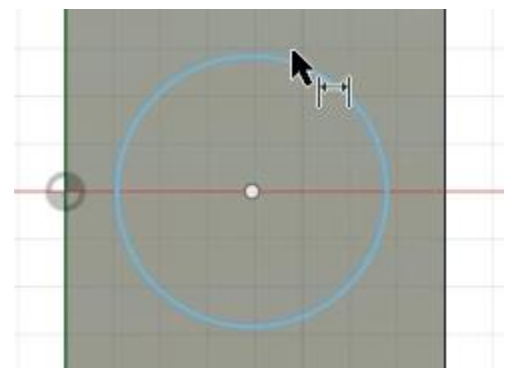


Figure 31. Draw a circle

- 32.** Press D to open the Dimension tool and make the circle's diameter **150 mm**. Also add a dimension between the solid body's edge and the circle's center. Instead of entering a number as the new dimension, enter `Plane_Offset/2` so that the circle's center is exactly half of the offset value. Finish the sketch.

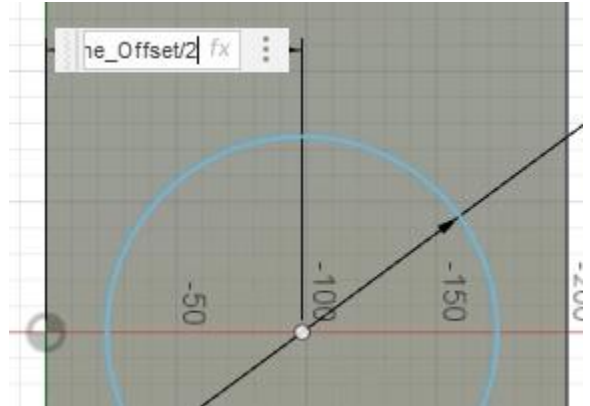


Figure 32. Use a parameter and math operators to modify a parameter

- 33.** Press E to open the Extrude tool, select the circle region you just drew, then extrude it upwards. OK the dialog.

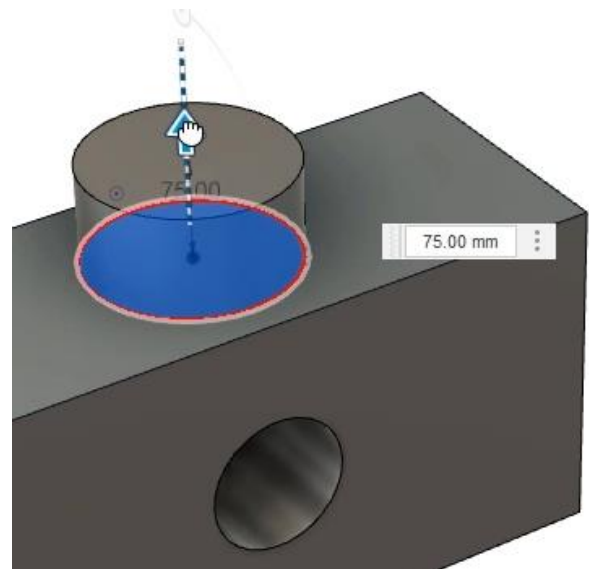


Figure 33. Extrude the new geometry

- 34.** Click Modify > Change Parameters. Expand the Model Parameters, then expand Sketch2. Notice that the Plane_Offset/2 expression is shown here.

Parameter	Name	Expression
Model Parameters		
(Unsaved)		
> Plane1		
> Sketch1		
> Extrude1		
Sketch2		
☆ Diameter Dimension-2	d11	150 mm
☆ Linear Dimension-2	d12	Plane_Offset / 2

Figure 34. Open the Change Parameters tool

- 35.** Parameters can be added to the Favorites section by clicking the star icon next to them.

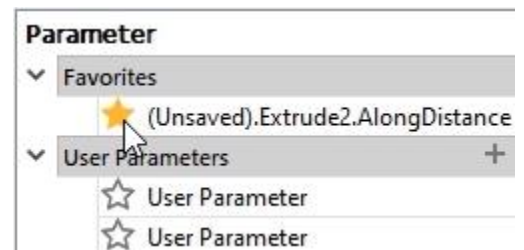


Figure 35. Favorite a parameter

- 36.** Click the plus icon to create a new User Parameter, then use the image on the right to create a Hole_dia2 parameter.

The 'Add User Parameter' dialog box is shown. It contains the following fields:

- Name: Hole_dia2
- Unit: mm (dropdown menu)
- Expression: 75
- Value: 75.00
- Comment: Enter Comment

 At the bottom, there are 'OK' and 'Cancel' buttons. The 'OK' button is highlighted with a blue border.

Figure 36. Create a new User Parameter

37. A parameter's current expression can be changed to use the User Parameters that you have already created. OK the Parameters dialog.

Unit	Expression	Value
mm	h	75.00
mm	Hole_dia User Parameter (mm)	
mm	Hole_dia2 User Parameter (mm)	
mm	hole2 Extrude2.AlongDistance (mm)	

Figure 37. Use an existing parameter as a different parameter's expression

38. Save the file. In the Save dialog, click the arrow to expand the Location options.

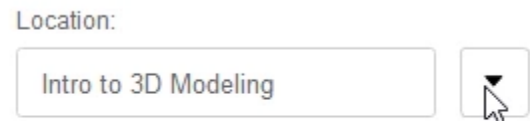


Figure 38. Save the file

39. Create a new folder by clicking New Folder.



Figure 39. Create a new folder

40. Name the new folder **Sketching**, then double-click it to open this location.

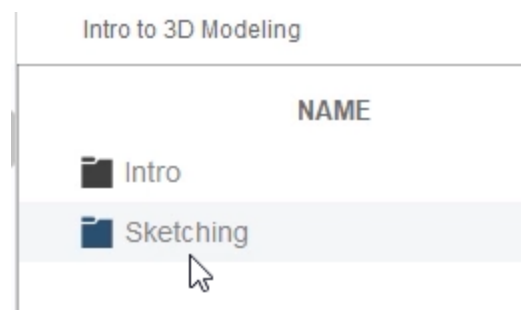


Figure 40. Open the new Sketching folder

41. Name the new design **UserParameterPractice**, then click the Save dialog's Save. Continue to experiment with User Parameters, then continue the next module.

Save

Name:

Figure 41. Save the file