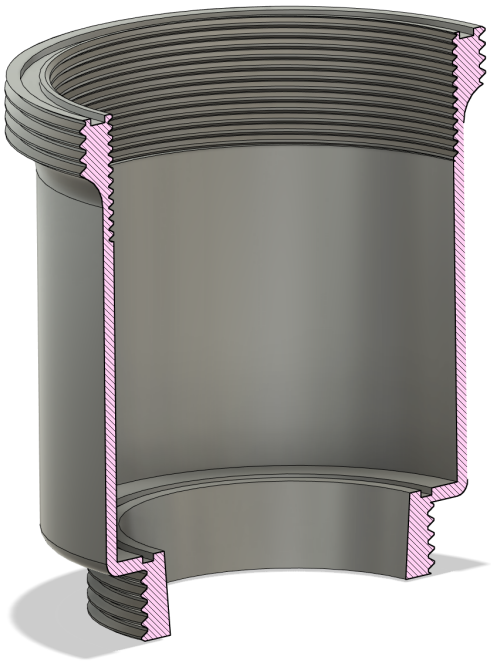
Section duration: ~15 minutes

Create 2D sketches and use Revolve and Thread tools

Use your knowledge of sketches to begin the project and create a simple profile as well as sketch constraints and dimensions. Then you'll revolve the sketch to create a solid and add thread features.

Learning Objectives:

- Create fully constrained sketches.
- Use centerlines to dimension sketches with diameters.
- Add sketch constraints to geometry.
- Create and modify user parameters.
- Use the Revolve tool to create components.
- Apply thread features to create internal and external threads.



The completed phase of the project.

1. Create a new design and name the file **Filter**.

   In the Autodesk Account Preferences, verify that the Default Modeling Orientation is set to Z Up.

   Change the document units to millimeters for this design.

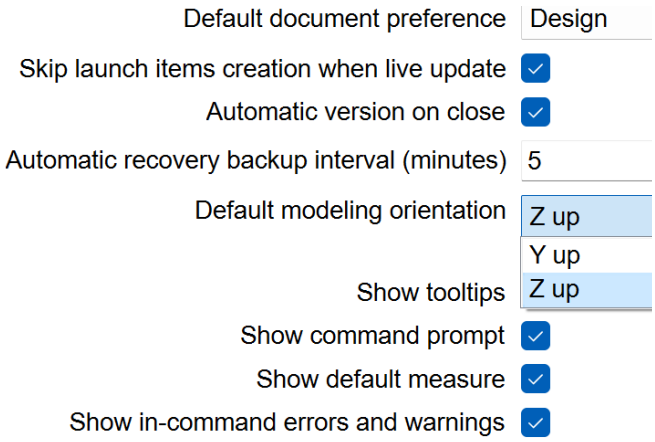


Figure 1. Confirming that model orientation is Z up.

2. Setting some parameters at the beginning of the design will help allow for easier manipulation of dimensions throughout the modeling process.

   In the Solid tab toolbar, click Modify > Change Parameters.

   Create the user parameters by clicking Add User Parameter (the plus sign). Input parameters as shown in Figure 2b.

   These sizes are a good starting point for the design but are intended to be modified by you as the designer.

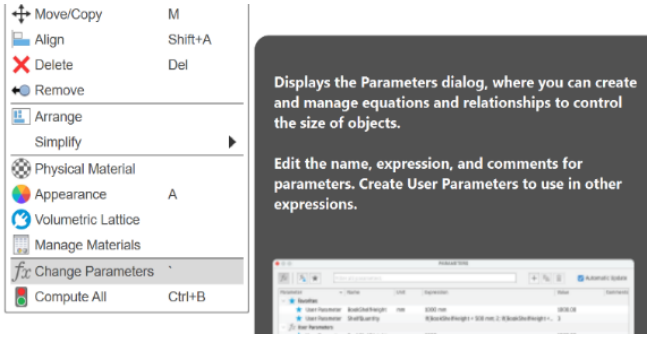


Figure 2a.Selecting Change Parameters.

Name	Un	Expression	Value	Comments
TopSeal_ID	in	2.612 in	2.612	Square O Ring Top Seal ID
TopSeal_OD	in	2.818 in	2.818	Square O Ring Top Seal OD
ThreadFaceOffset	mm	0.1 mm	0.10	Allowance for thread faces

Figure 2b. Adding User Parameters with comments.

3. Creating the outline of a main component in the first sketch is an efficient way to model a design.

   To begin, create a sketch on the XZ plane and sketch a center line along the vertical axis beginning at the origin. This will be used as an axis of rotation for a revolved feature.

   Use lines and rectangles to create the profile shown in Figure 3. Lines will be located from the center line in the next step. Constraints are not visible in Figure 3 to make the geometry easier to see.

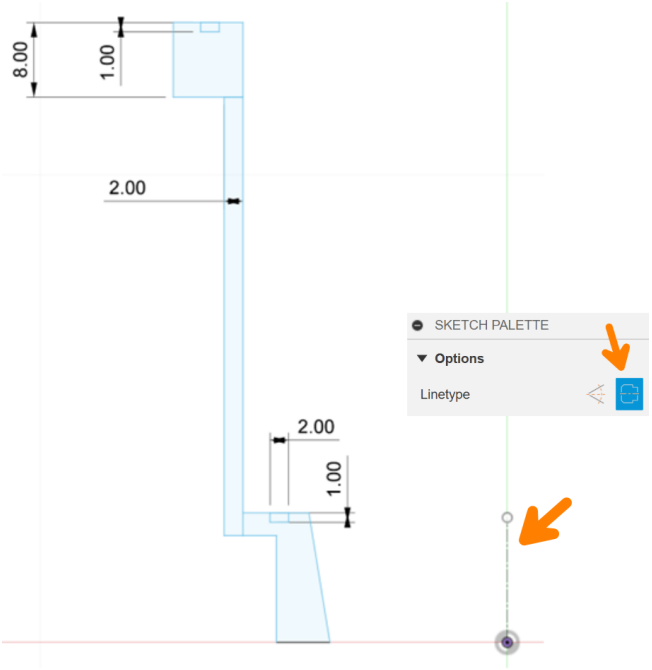


Figure 3. Sketching a profile for to be revolved.

4. Add the dimensions shown in Figure 4 by selecting the centerline as one of the selections. This will automatically create a diameter dimension.

   Use parameters TopSeal_ID and TopSeal_OD as the values for the diameters that are shown as functions.

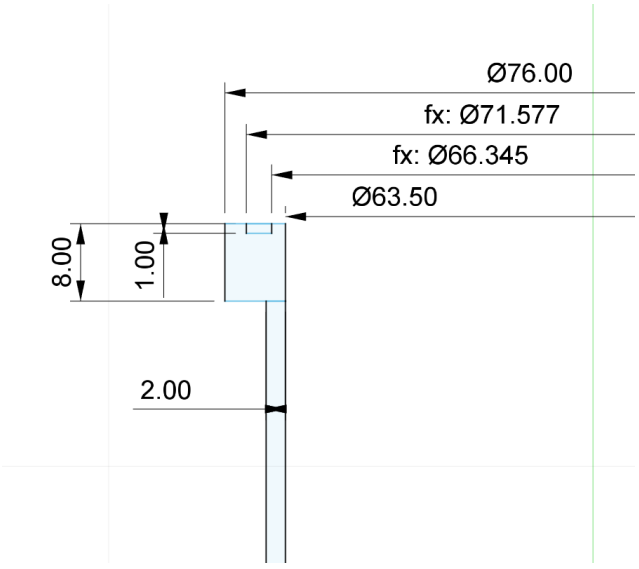


Figure 4. Entering user parameters as dimension values.

5. Add the diameter dimensions shown in Figure 5 by selecting the centerline as one of the selections.

   Apply linear dimensions as shown.

   Ensure that the bottom line is horizontal with the origin.

   All of the lines in Figure 5 should be fully constrained.

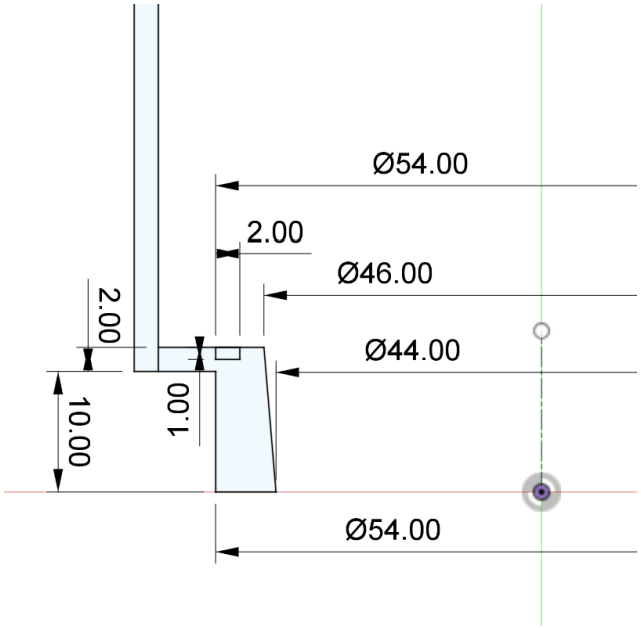


Figure 5. Locating geometry by numeric and geometric constraints.

6. Add the 65 mm dimension as shown in Figure 6 and check to make sure that the entire sketch is fully constrained.

   Click Finish Sketch.

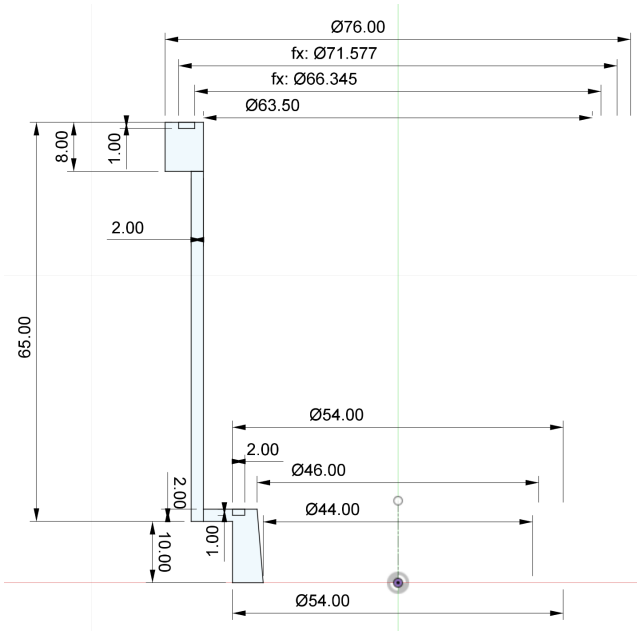


Figure 6. Adding dimensions to fully constrain a sketch.

7. Now you'll revolve the housing of the filter from the profile you just created.

   In the Create group, click the Revolve tool. Select each of the closed profiles except for the two small rectangles.

   TIP: You may need to press and hold the Ctrl key (Command on Mac) to select multiple profiles.

   It is likely that the Axis will be automatically selected because of the use of the centerline type.

   Click OK or press Enter.

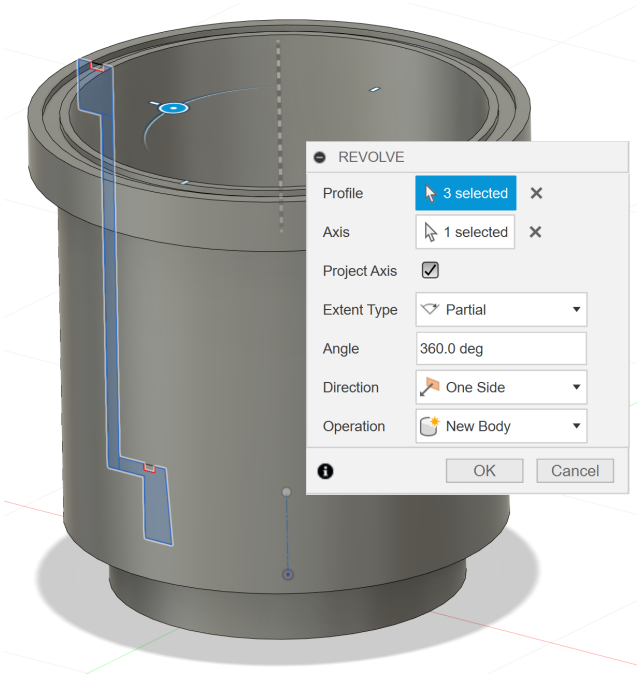


Figure 7. Previewing a full revolution.

8. Viewing the cross-section of a body with internal features can make it much easier to select faces when adding features.

   In the Inspect group, click Section Analysis. Select the XZ plane. You may have to click the Flip button if the portion removed from view is facing away from you. Click OK or Press Enter.

   An analysis does not alter solids or sketches; it temporarily hides solids from view between your line of sight and the cut plane. Analyses can be toggled on and off in the Browser.

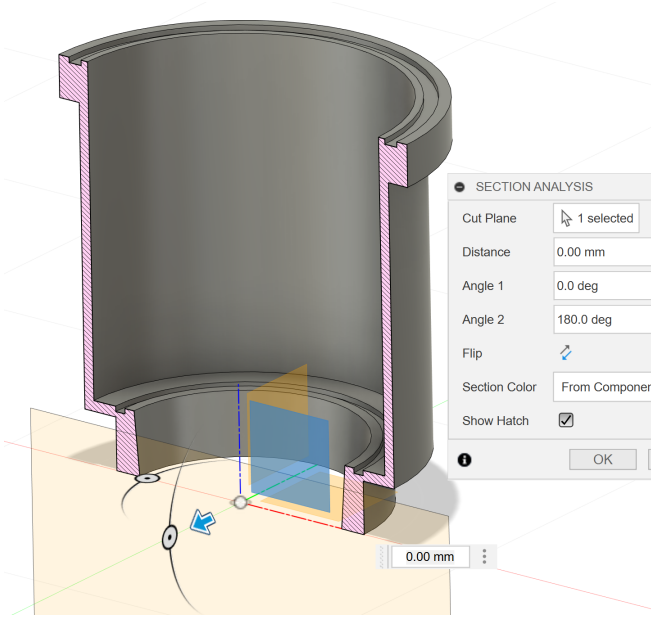


Figure 8. Previewing a full section analysis.

9. A few sets of threads need to be added to the housing to provide a method for attaching adjacent parts. In the Create group, click Thread and select the cylindrical face at the bottom of the revolution.

   Select the checkbox for Modeled so that the thread profile will be added to the solid.

   Set the Designation to M55x2.

   Click OK or press Enter.

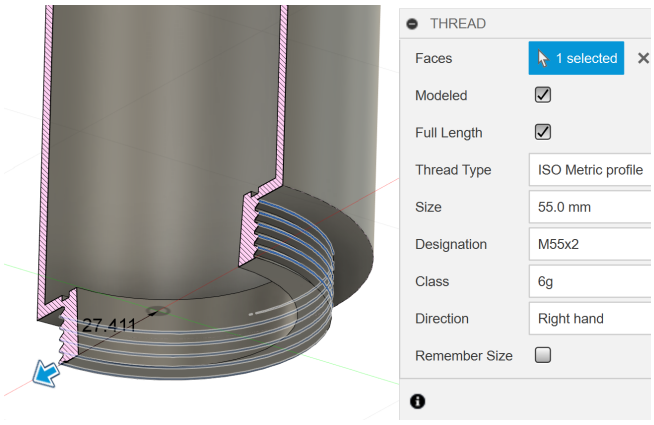


Figure 9. Previewing modeled threads.

10. Use the Thread tool to add internal threads to the top of the housing.

Threads should be modeled and applied only to the top 18 mm of the inside of the housing.

The Designation should be set to M65x1.5.

NOTE:

Metric thread designations end with a value that denotes the pitch, or distance between threads. Selecting a pitch of 1.5 mm or greater will result in good quality 3D printed prototype.

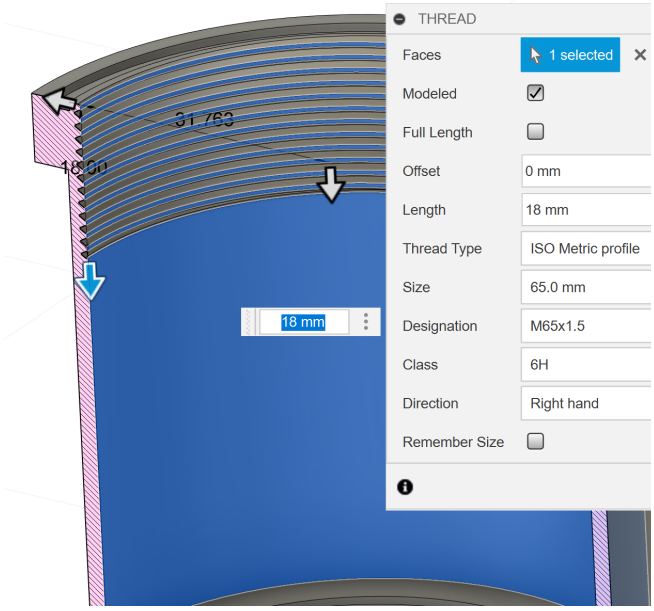


Figure 10. Previewing internal modeled threads.

11. Use the Thread tool to add external threads to the top of the housing.

Threads should be modeled and applied the full length of the cylindrical surface selected.

The Designation should be set to M46x2.

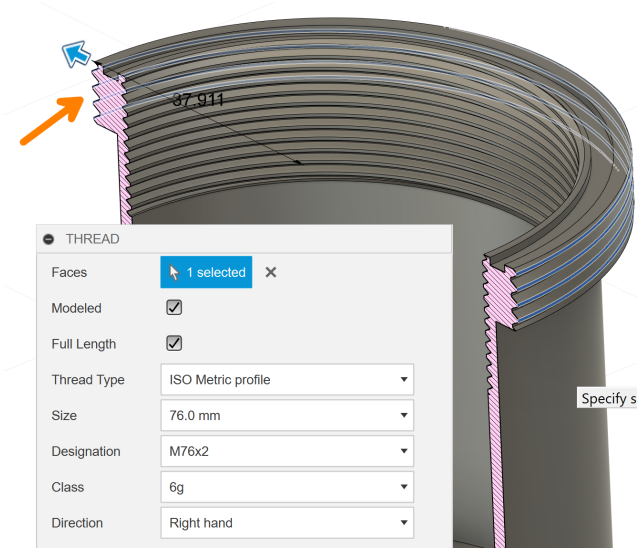


Figure 11. Previewing modeled external threads.

12. Adding a fillet to an inside corner strengthens the part and avoids the risk of stress fractures developing over time. Adding a fillet beneath an overhang makes an object easier to 3D print as well.

Apply a 5 mm fillet to the inside corner as shown in Figure 12.

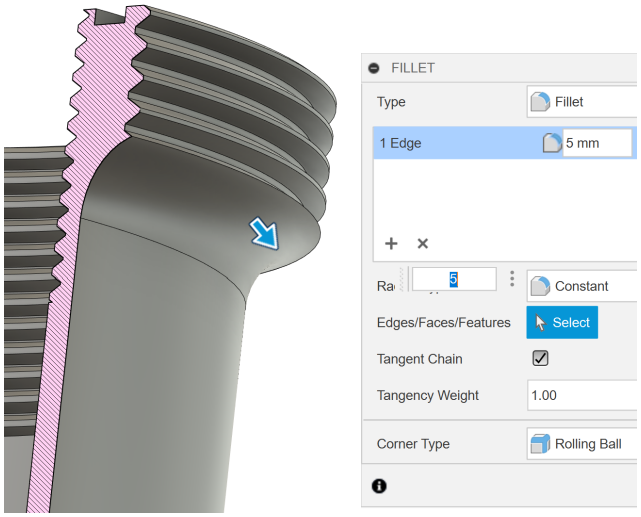


Figure 12. Previewing an edge fillet.

13. Apply a 2 mm fillet to the outside corner as shown in Figure 13.

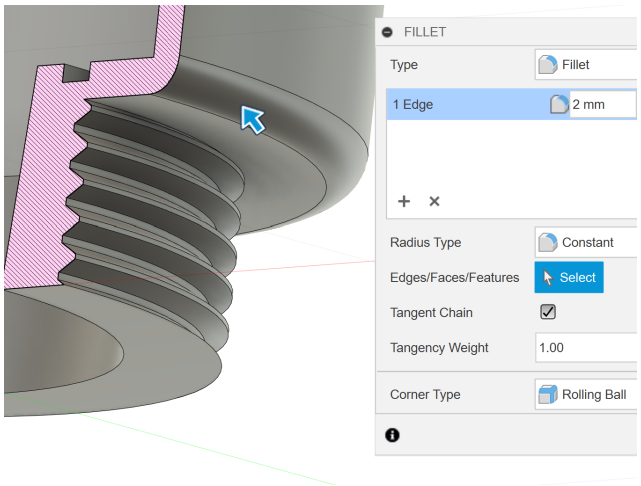


Figure 13. Previewing the second edge fillet.

14. Congratulations! You have completed the housing of the filter.

Save your work with a description before moving on.

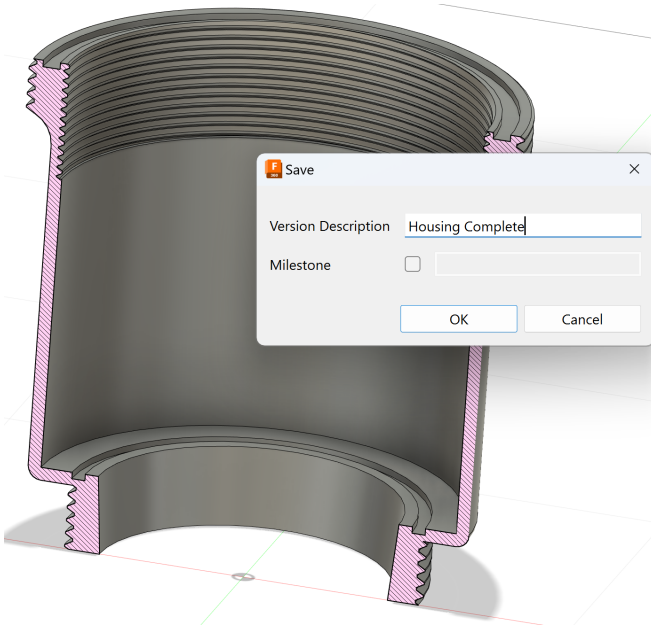


Figure 14. Saving progress with a version description.