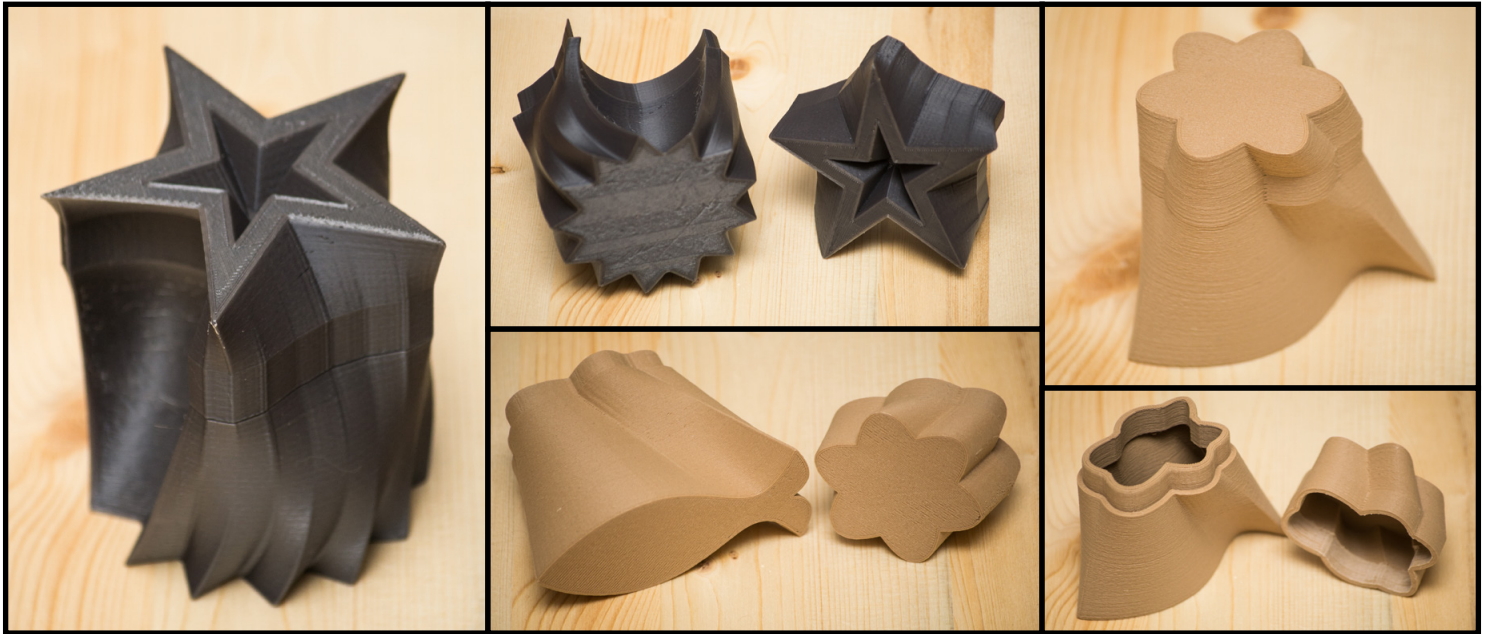


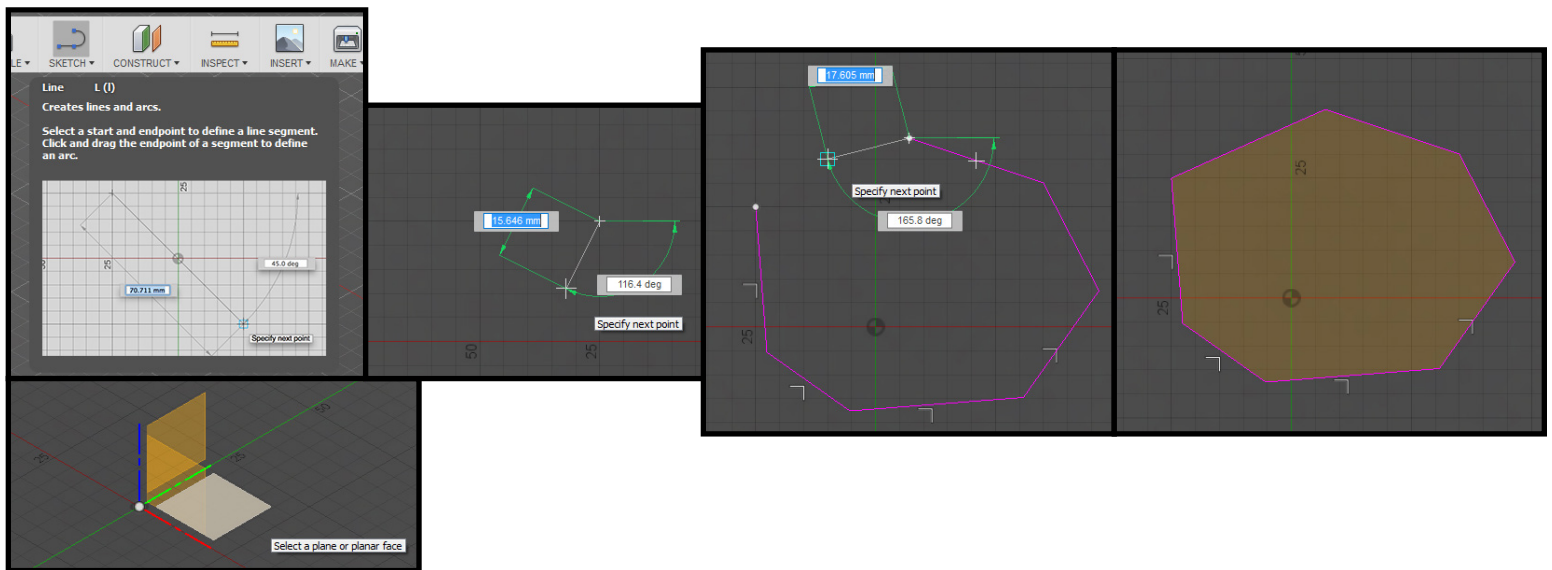
# CUSTOM CONTAINER IN FUSION 360

(Fusion 360 Tutorial - Beginner Level - Written by Maz a.k.a. Rose M.)



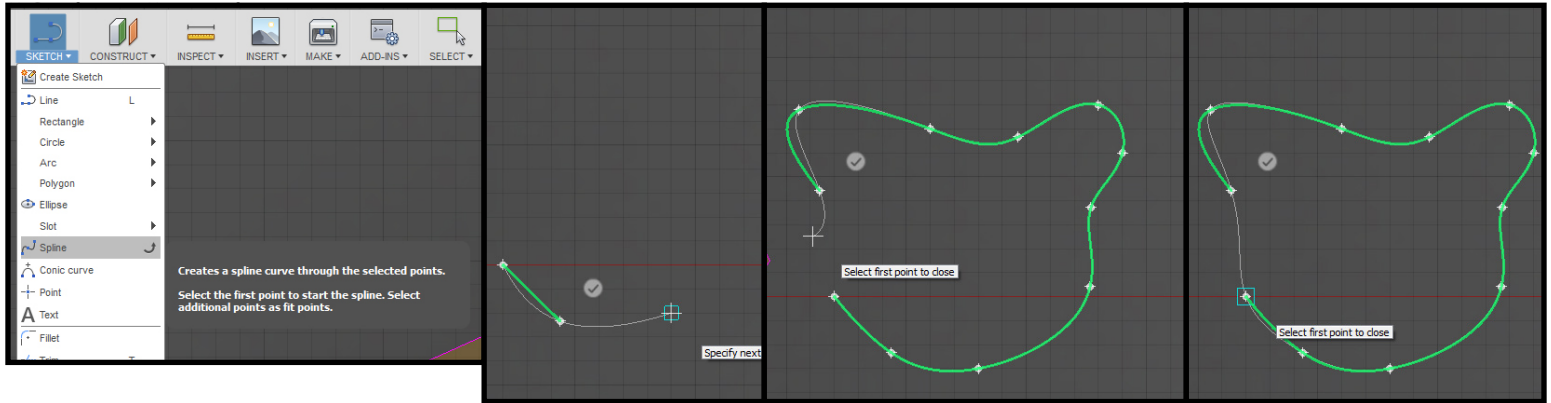
## STEP 1: CREATE PROFILES FOR CONTAINER SHAPE

- Create bottom profile for the container using sketch tools such as spline or line. In the tool bar at the top, click the hooked line with three dots to activate the "Line" tool (or press "L" on the keyboard). Select the floor plane to sketch onto.



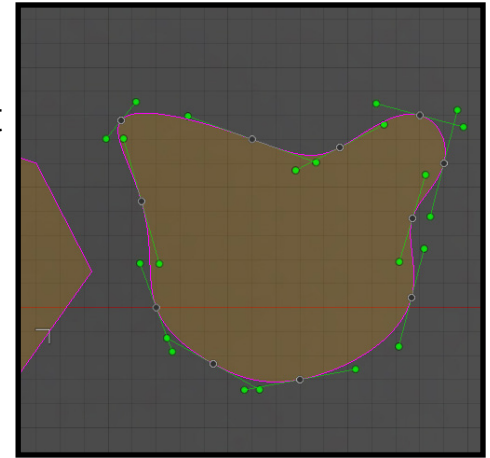
- Click in the workspace to place the first point. The next point can be set by moving the mouse and clicking, or entering distance and angle values. To stop making lines click the check mark in the circle (or press "ESC" on the keyboard).

- In the tool bar at the top, click “Sketch ▼” to expand the menu, and click



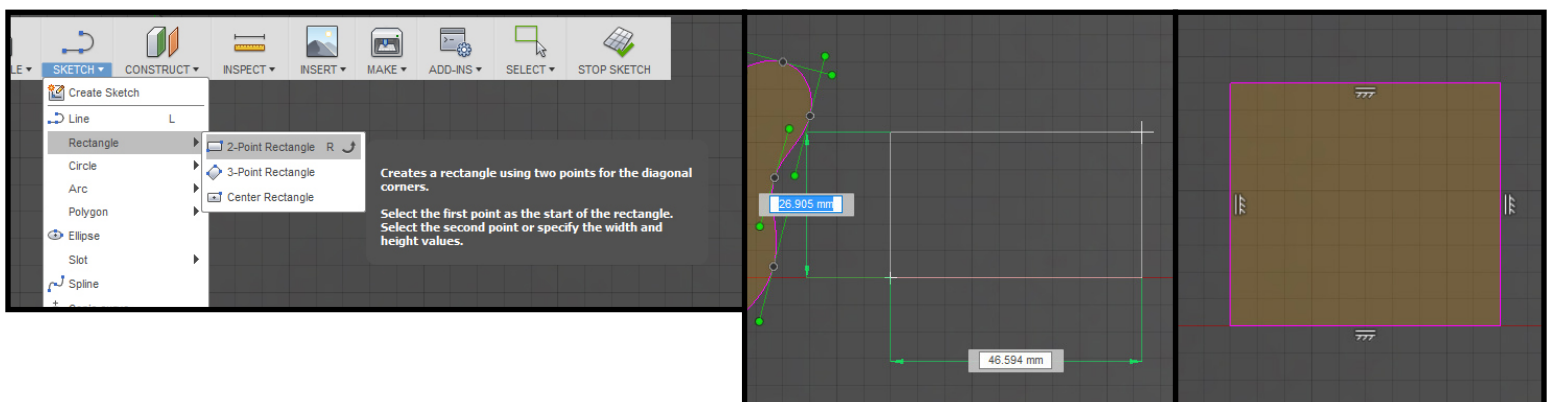
Click in the workspace to place the first point. The next point can be set by moving the mouse and clicking.

The spline tool will automatically create a curve between points once a second point is set. A continuous spline will alter curves with every point added to the line. To create a sharp spike in the spline, reactivate the spline tool while still in an active spline.

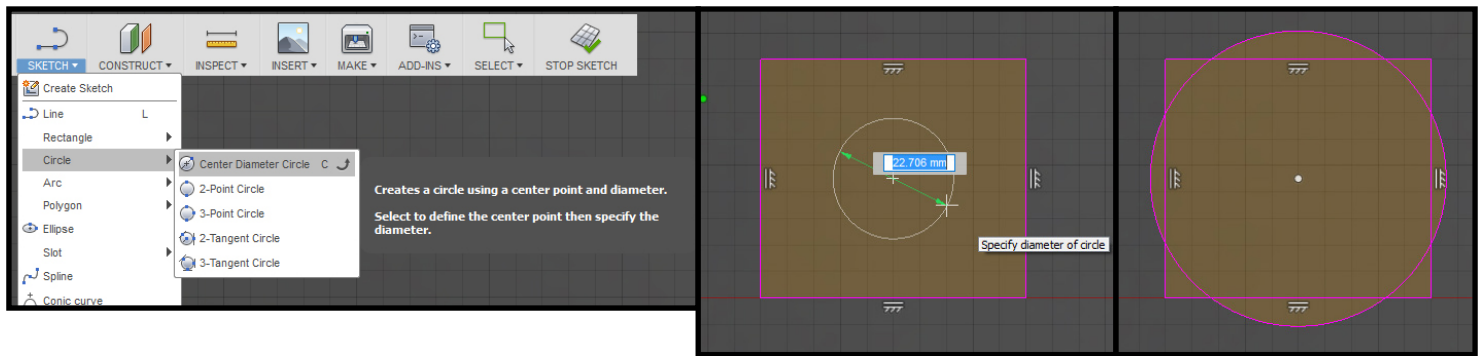


To end a spline click the check mark in the circle.

Note: Spline curves can be edited by clicking the spline to activate selection, curves with handles will appear, click + drag on handle to change curve.

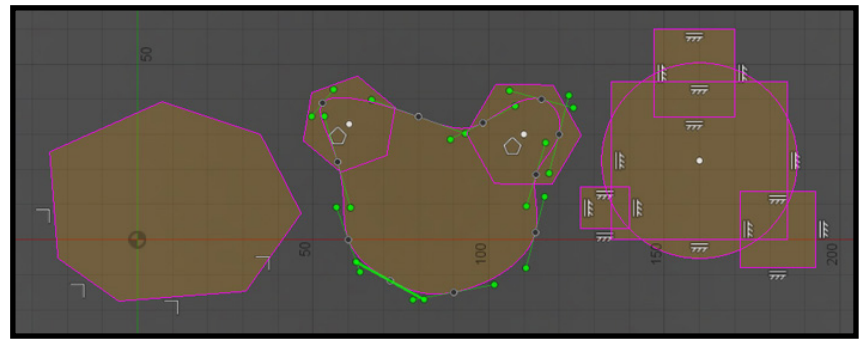


In the tool bar at the top, click “Sketch ▼” to expand the menu, hover over “Rectangle ►”, and click “2-Point Rectangle” (or press “R” on the keyboard). Click to set the first point, move mouse away and rectangle will appear. Eyeball your desired size or use the input boxes to set specific values and press enter/return to accept value for each.



• In the tool bar at the top, click "Sketch ▾" to expand the menu, hover over "Circle ►", and click "Center Diameter Circle" (or press "C" on the keyboard). Click to place center point of circle, move mouse, set diameter value by entering a value in the floating text box and press enter/return to accept, or move mouse out to desired size and click to place (or press enter/return twice to place circle and exit circle sketch tool).

Use any of these sketch tools to create three (3) sketch profiles for the container.



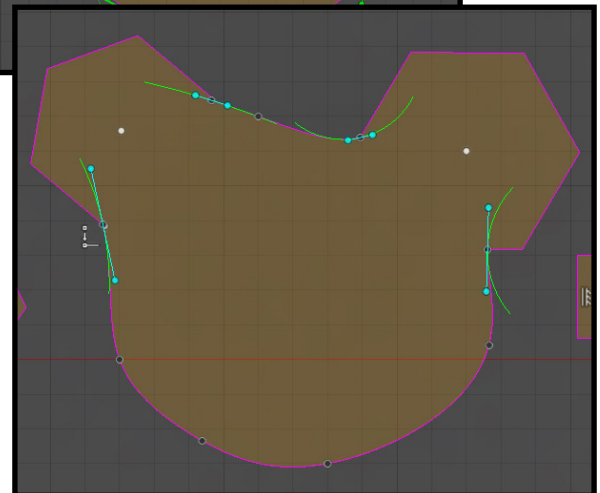
## STEP 2: CLEAN SKETCH PROFILES FOR LOFT TOOL

In order for the "Loft Tool" to work the profile sketch needs to be a single line as shown below. Use the "Trim Tool" to clean up the sketch profiles.

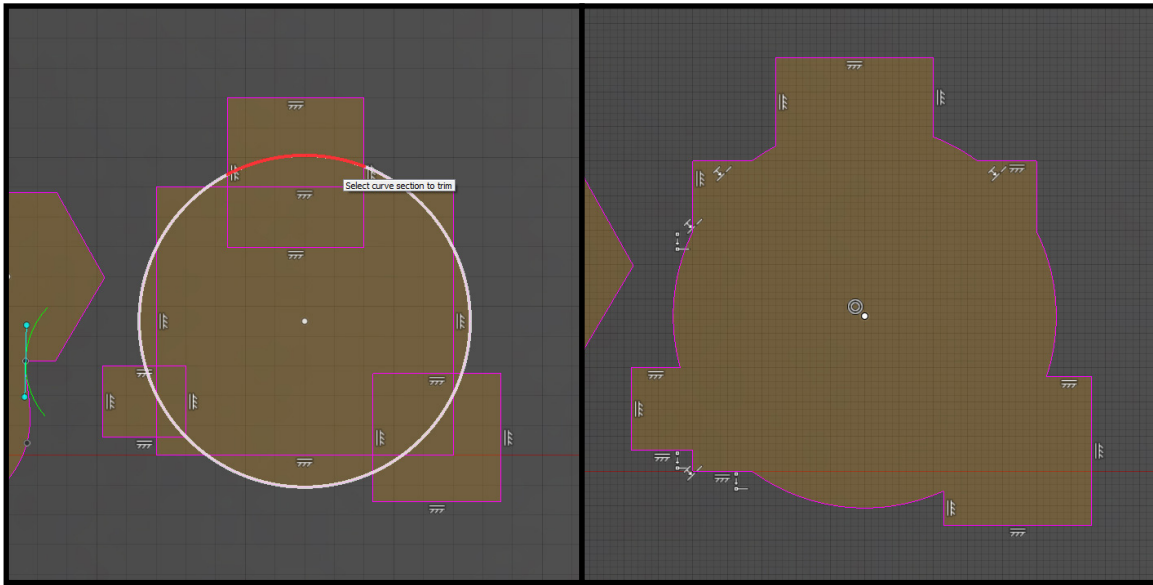


• In the tool bar at the top, click "Sketch ▾" to expand the menu, and click "Trim" in the menu (or press "T" on the keyboard). Hover over a section in sketch that is unnecessary, the line turns red, and click to remove.

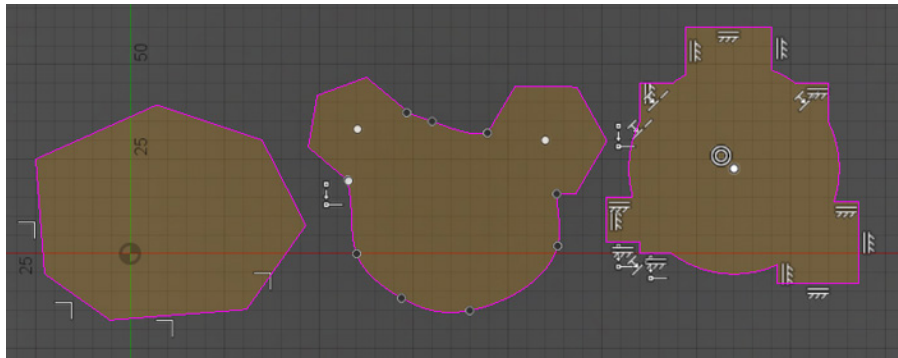
Continue removing unnecessary line segments in sketch profiles.





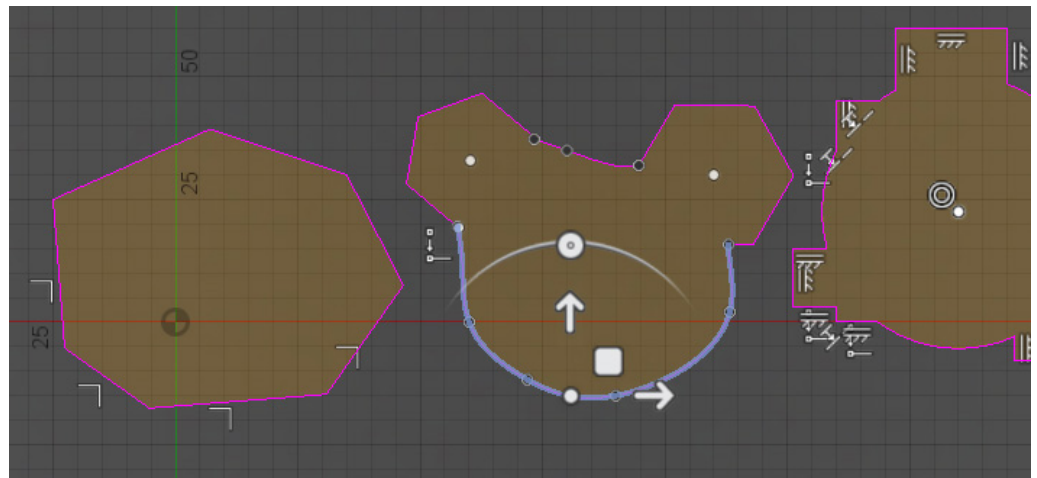
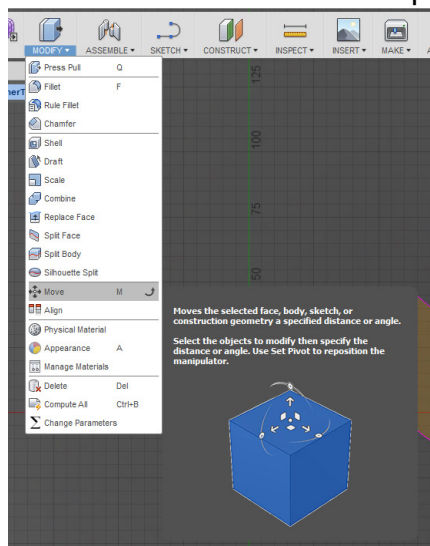


Note: Sometimes sketch restrictions need to be removed first in order to remove a line segment (e.g. angle, radius, diameter, etc.).



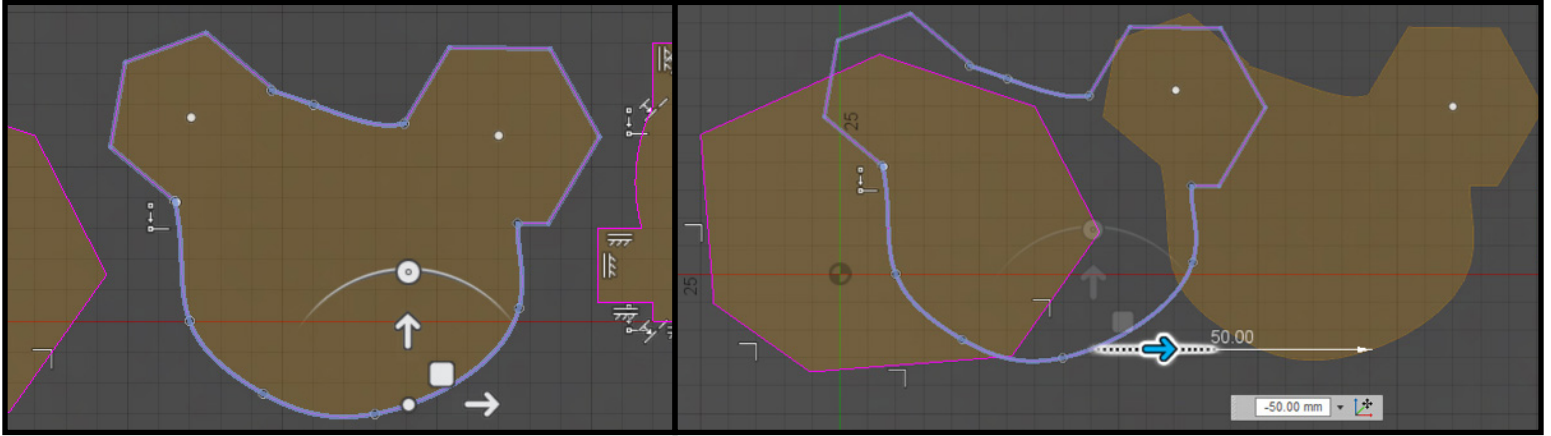
### STEP 3: POSITION SKETCH PROFILES TO PREPARE FOR LOFT TOOL

In order for the “Loft Tool” to create our container from the sketch profiles, they need to be stacked on top of each other with some distance in between. To do this use the move tool to position sketch profiles.

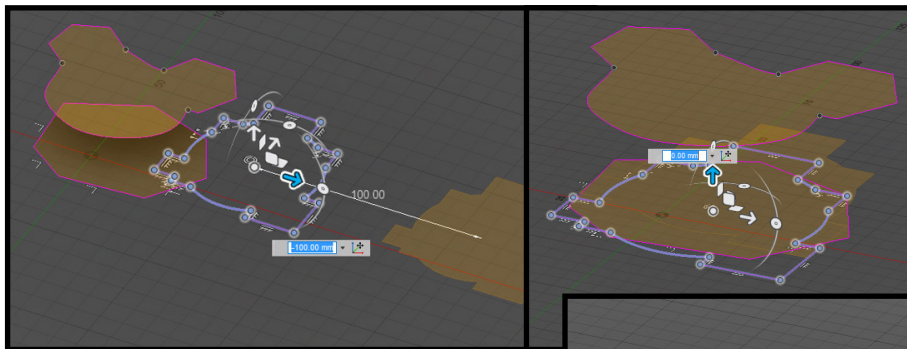
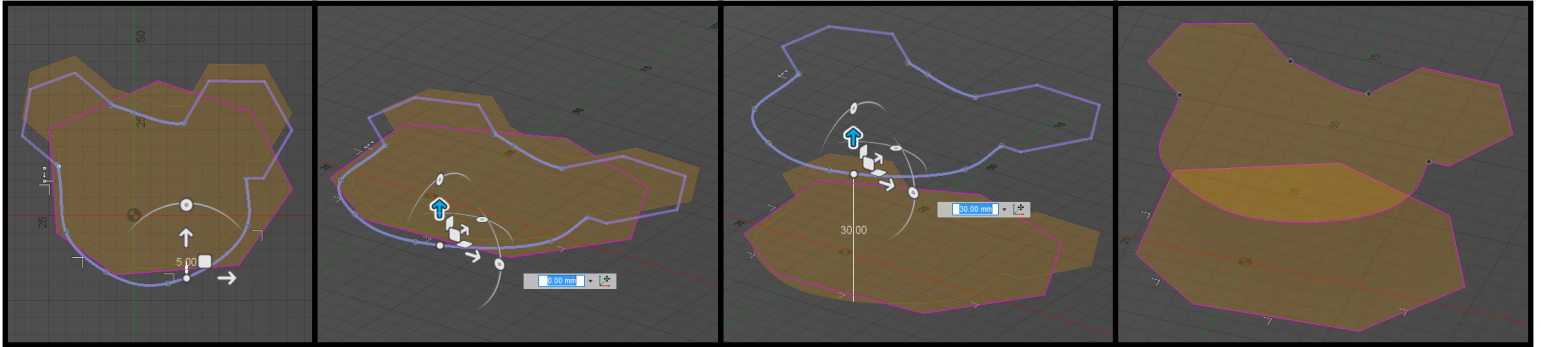


- In the tool bar at the top, click “Modify ▼” to expand the menu, click “Move” (or press “M” on the keyboard), and click the second sketch profile.

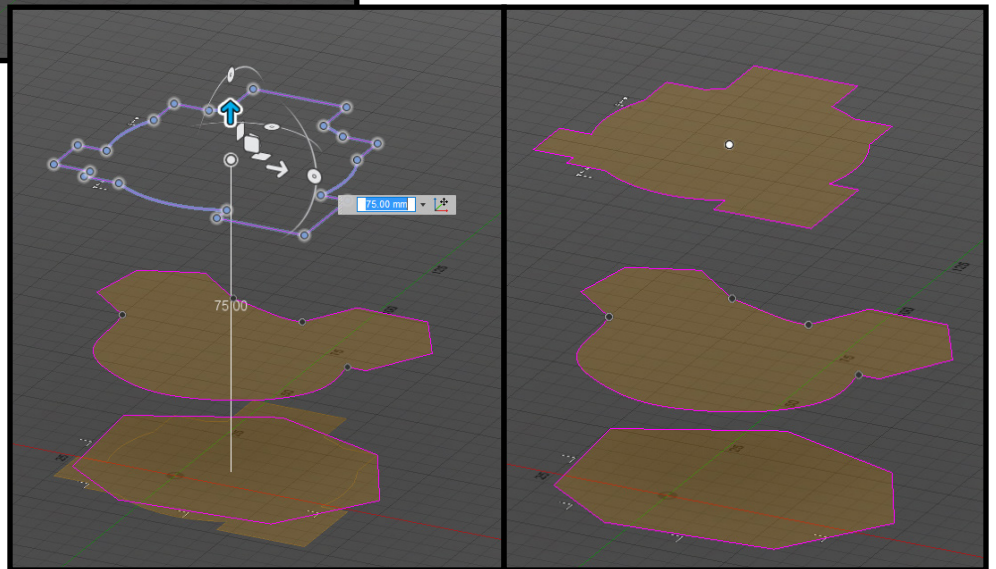
If multiple sketch lines make up the single profile, hold shift and click additional lines to select. (Dragging a selection box over the entire second profile to select will work also.)



Use the arrows to move sketch into place above first profile. Once positioned as desired press enter/return to accept or click okay in the dialog box on the right. Do the same for the third profile.

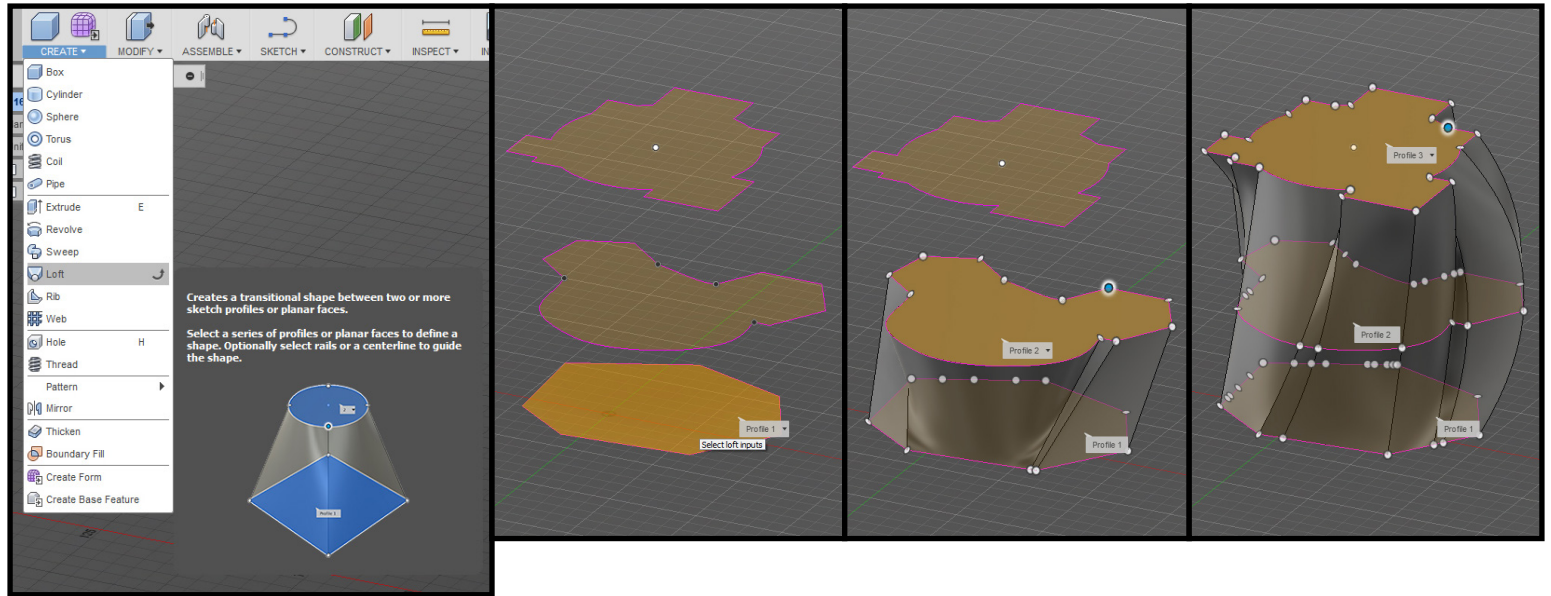


Note: Sometimes sketch profiles get broken when moved off the original sketch plane. In order to fix this create an offset of the profile, copy, and then move.



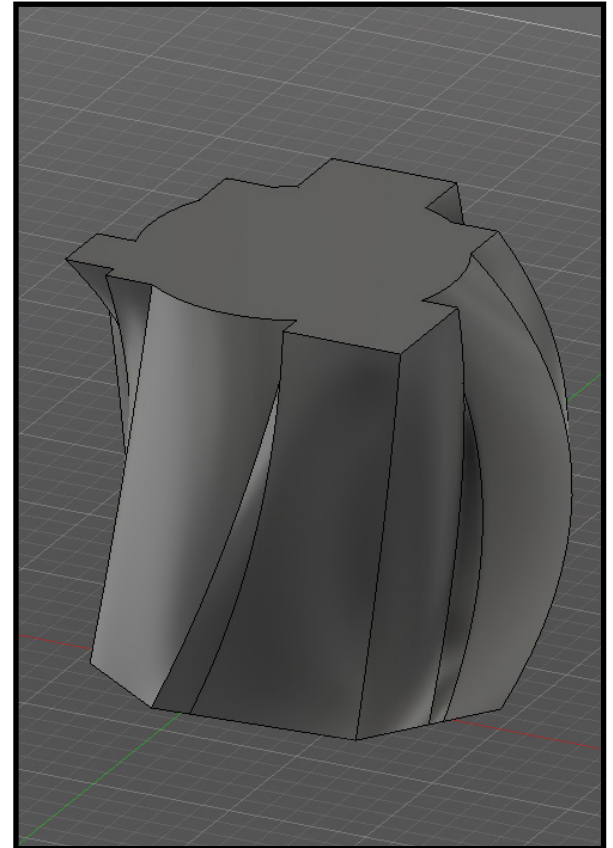
## STEP 4: CREATE BODY OF CONTAINER

Using the "Loft Tool" create a solid body by combining all three (3) sketch profiles.



- In the tool bar at the top, click "Create ▼" to expand the menu, and click "Loft". Position workspace view to be able to see all three (3) sketch profiles at an angle.

- Hover over inside of first sketch profile.
- Sketch will turn orange, click to select.
- Hover over inside of second sketch profile.
- Sketch will turn orange, click to select, solid geometry appears.
- Hover over inside of third sketch profile.
- Sketch will turn orange, click to select, solid geometry will change.

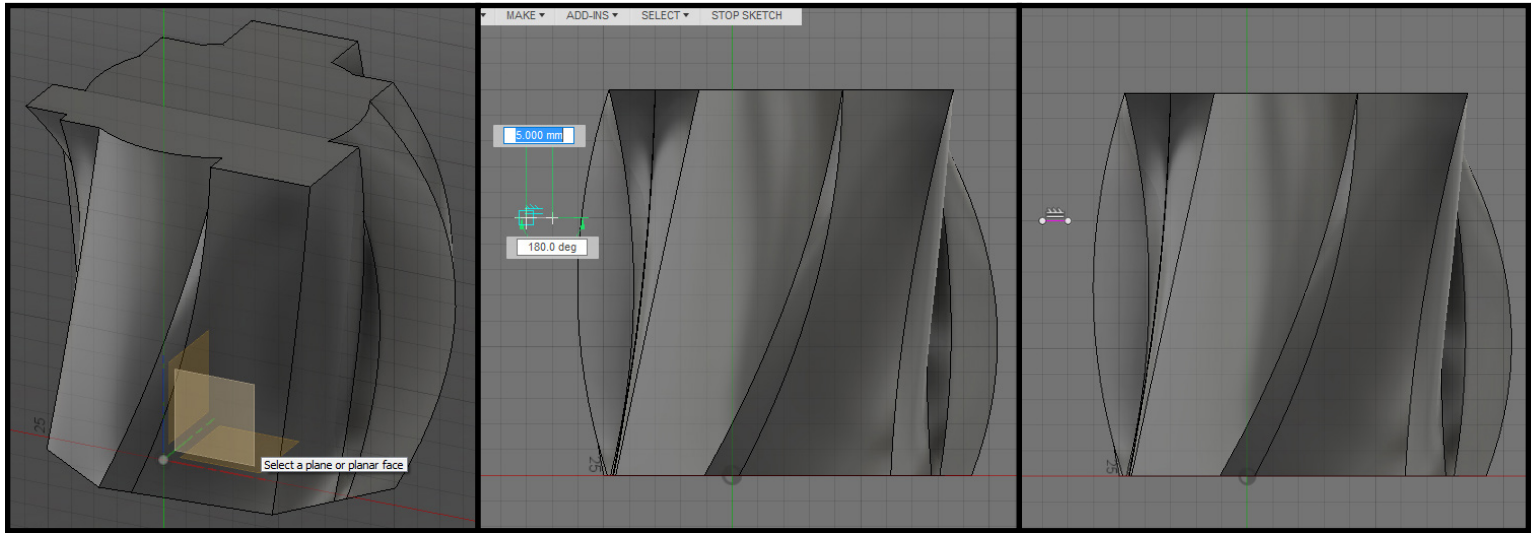


In the dialog box on the right there are options that change the loft. Do not worry about using these and leave them set to defaults. Click okay to accept or press enter/return on the keyboard.

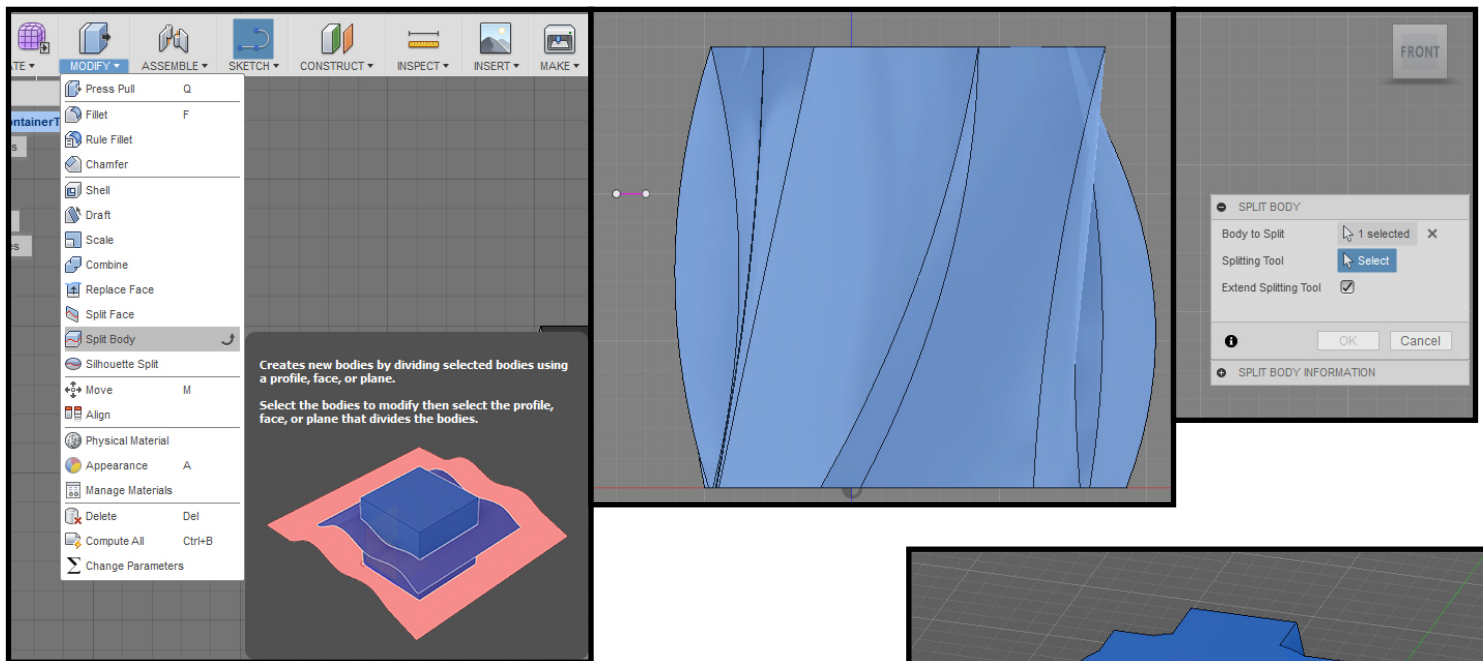


## STEP 5: SPLIT CONTAINER INTO LID & BOTTOM SHAPES

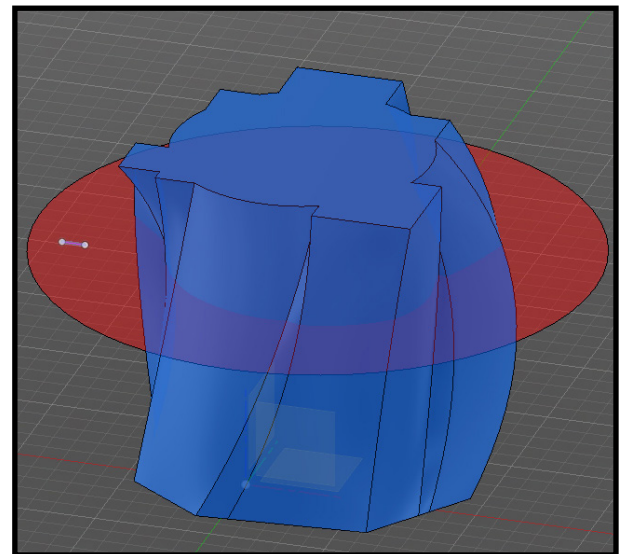
Use the "Split Body" tool to separate lid and bottom shapes of container. In order to do this a sketch line for cut needs to be made.



- Use the line tool to create a straight 180°/0° line segment in position desired for separation.

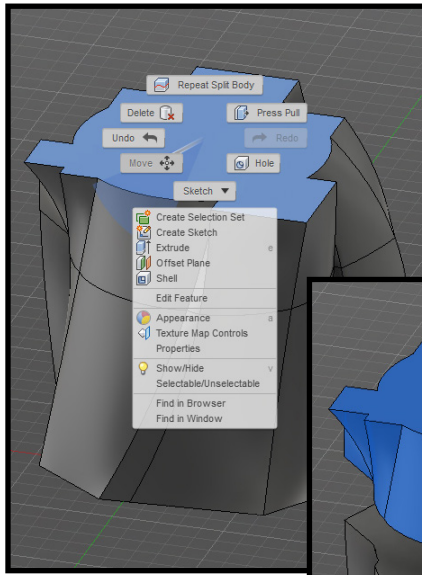


- In the tool bar at the top, click "Modify ▼" to expand the menu, and click "Split Body". Select the container body first, in the dialog box on the right click gray box with "Select" near "Splitting Tool", click line segment to select, and click okay or press enter/return on the keyboard.

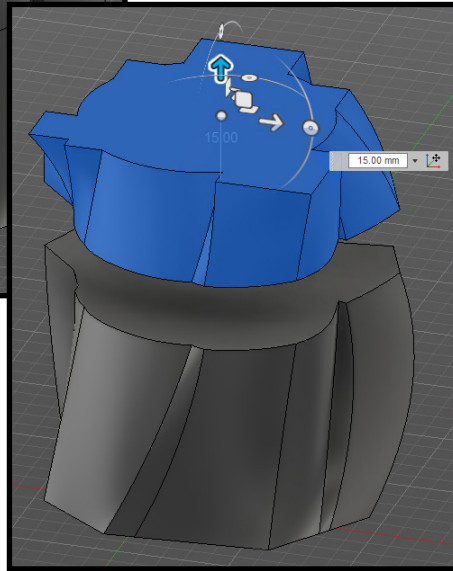


## STEP 6: REPOSITION LID SHAPE & EXTEND LID BOTTOM

For the next step to work the lid needs to be moved up 15.00mm on the Z axis.



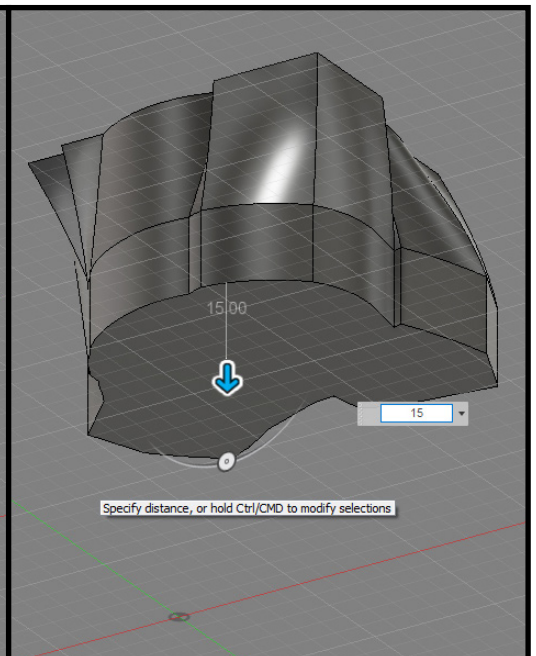
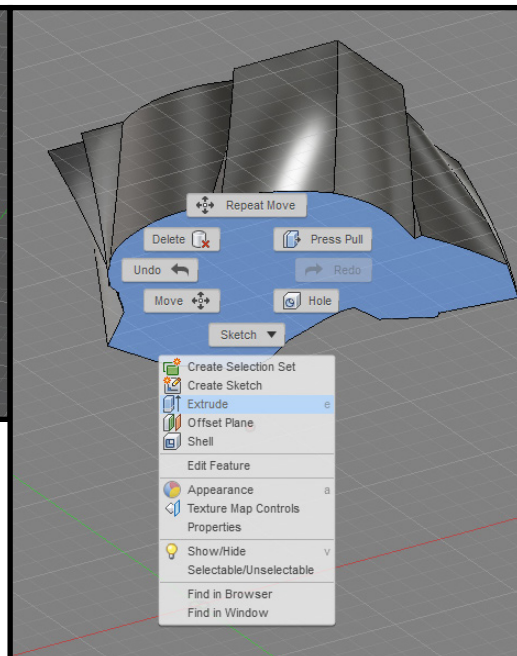
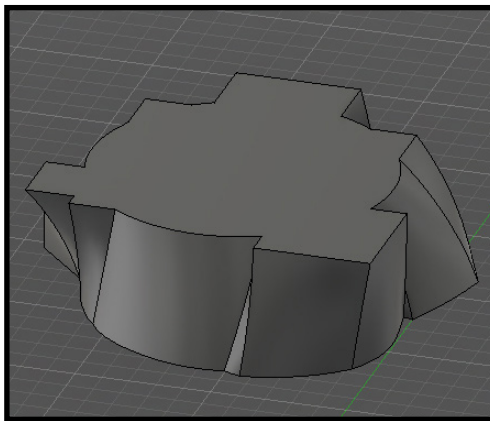
- Use the move tool to position lid "15.00mm" above top of container bottom on the Z axis. Use arrow to drag up or enter value.



- In the browser tree on the left, hide the container bottom.

- Select the bottom face of the lid and extrude down 15.00mm as a join. To do this, in the tool bar at the top, click "Create ▼" to expand the menu, and click "Extrude" (or press "E" on the keyboard).

With the bottom face of lid selected extrude down "15.00mm", click okay or press enter/return to accept.

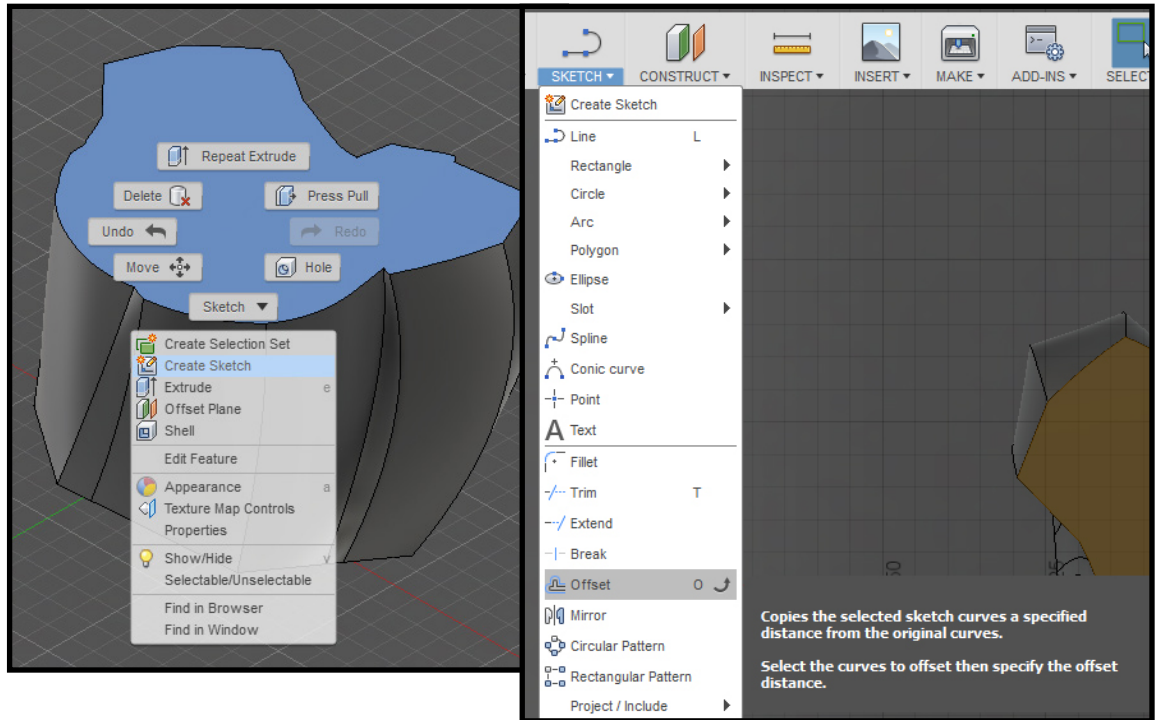


- Hide the lid body for now.
- Unhide the container bottom.

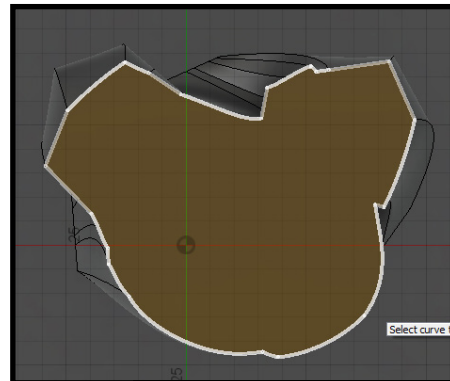


## STEP 7: CREATE BEGINNING OF LIP FOR CONTAINER BOTTOM

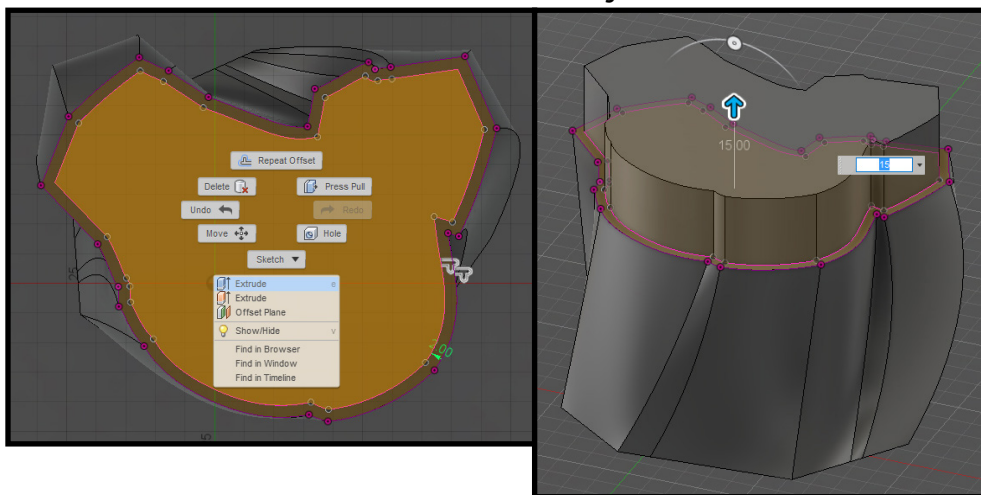
To create a lip for container bottom to keep lid from sliding off use offset sketch tool and extrude.



- Select the top face of the container bottom and create sketch. In the tool bar at the top, click "Sketch ▼" to expand the menu, and click "Offset" in the menu (or press "O" on the keyboard). Hover over top face sketch edge and click to select. Click and drag "<|>" to offset sketch loop "2.00mm" inside top face profile edge (or enter "2.00mm" value into text box) and press enter/return to accept or click okay in the dialog box on the right.

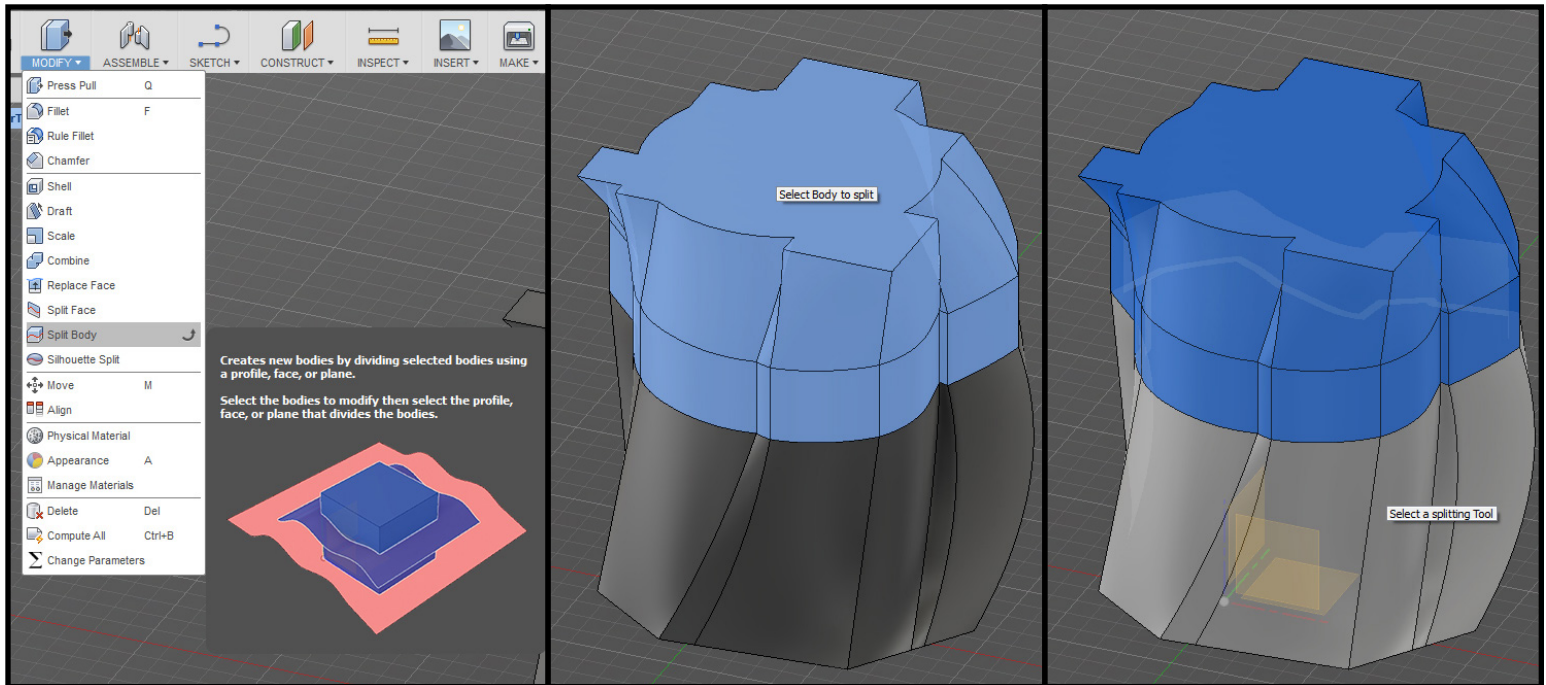


- Extrude the inside sketch 15.00mm.
- Unhide container lid body.

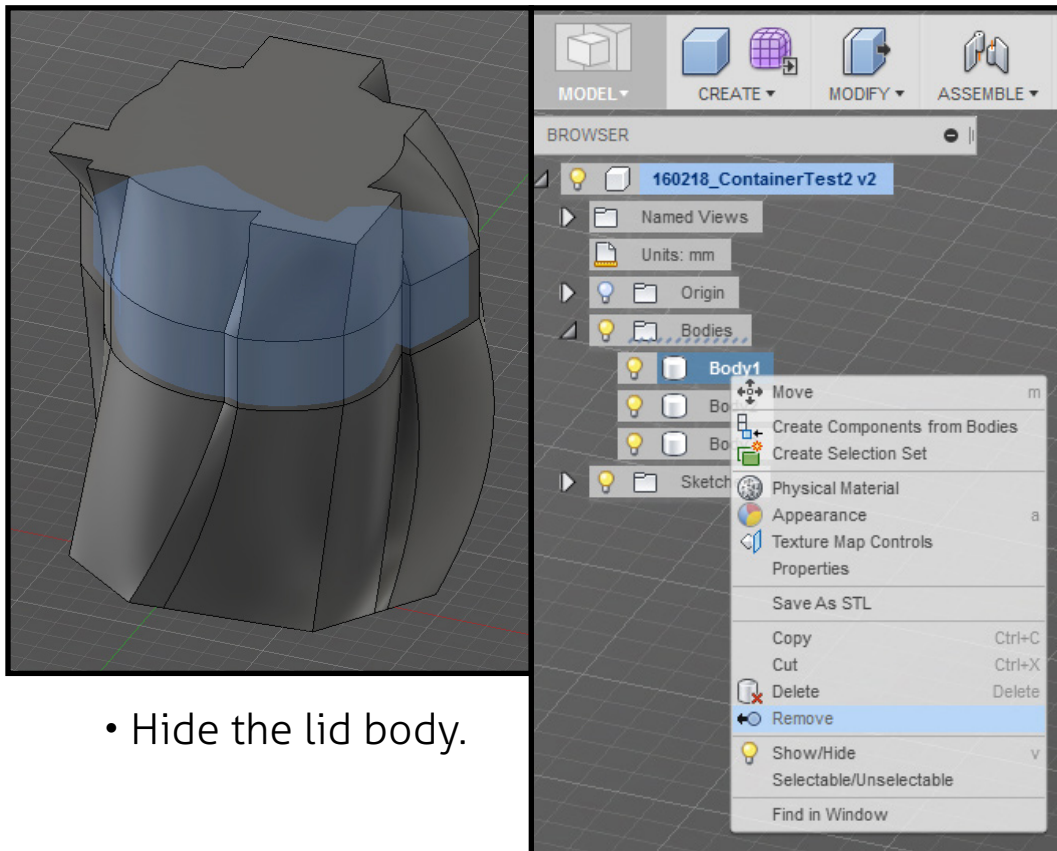


### STEP 3: CUT CONTAINER BOTTOM LIP FROM LID

Use the "Split Body" tool to cut bottom container lip from lid of container.



- In the tool bar at the top, click "Modify ▼" to expand the menu, and click "Split Body". Select the lid body first, in the dialog box on the right click gray box with "Select" near "Splitting Tool", click container bottom body to select, and click okay or press enter/return on the keyboard.



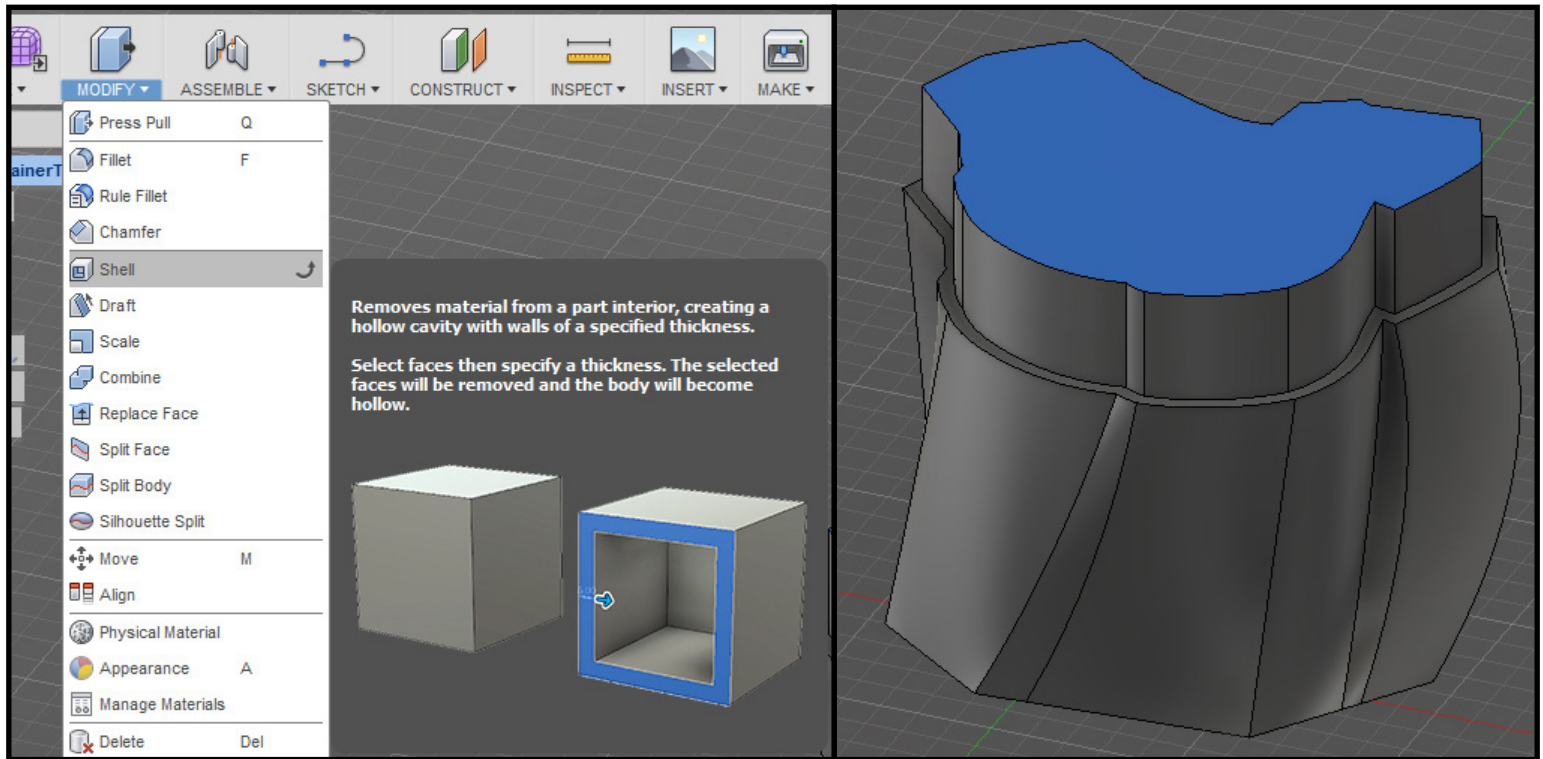
Remove the cut piece from the project. To do this, in the browser tree on the left in the "Bodies" section, hover over the body that only highlights inside the container, right click or option + click, and select "Remove" in the pop-up menu to remove.

- Hide the lid body.

## STEP 9: HOLLOW OUT CONTAINER BOTTOM

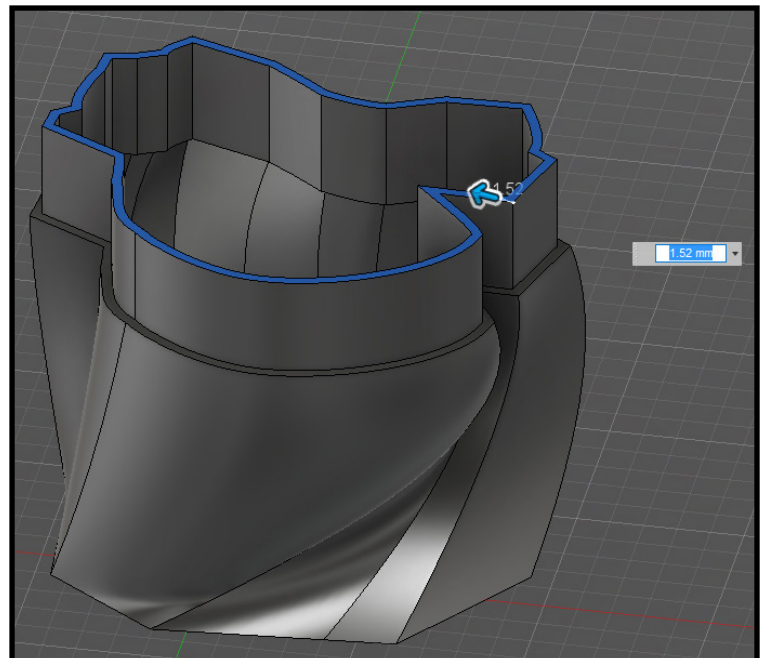
Use the Shell Tool to hollow out the inside of the container bottom.

- To do this, in the tool bar at the top, click "Modify ▼" to expand the menu, and click "Shell". Hover over the top face of the container bottom and click to select (it will turn blue with arrow). Set shell value between 1.00mm - 2.00mm. Start with "2.00mm" and if shell tool fails, continue to minus "0.10mm" until shell succeeds. Some designs do not work with 2.00mm shell and this is why a ranged is suggested. Below 1.00mm shell is too thin for printing.



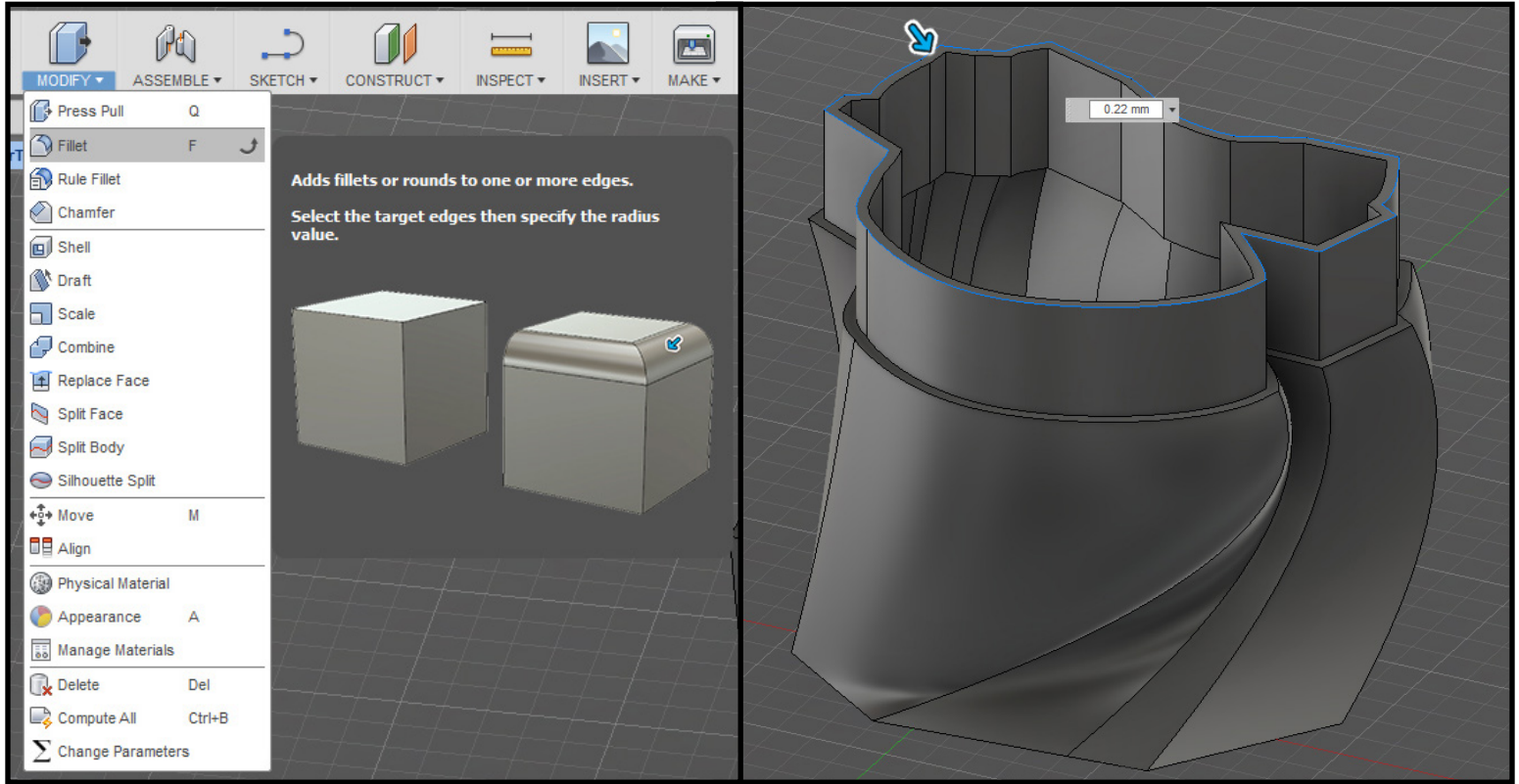
Note: Shell tool failing is often due to curves in design that create intersecting faces from using the shell tool. To fix the shape, edit sharp corners or tight curves in sketch profiles and attempt shell again.

Existing sketches can be edited by double clicking the action in the timeline to activate or double clicking the sketch title in the browser tree on the left.





- After shelling the container bottom it is helpful to round off the sharp outside edge on the container bottom for a smooth fit with lid. To do this, go to the tool bar at the top, click "Modify ▾" to expand the menu, and click "Fillet" (or press "F" on the keyboard). Click and drag arrow to desired radius. Hold CTRL/CMD, click on additional outside top edges to add to selection, and let go of CTRL/CMD to update Fillet.

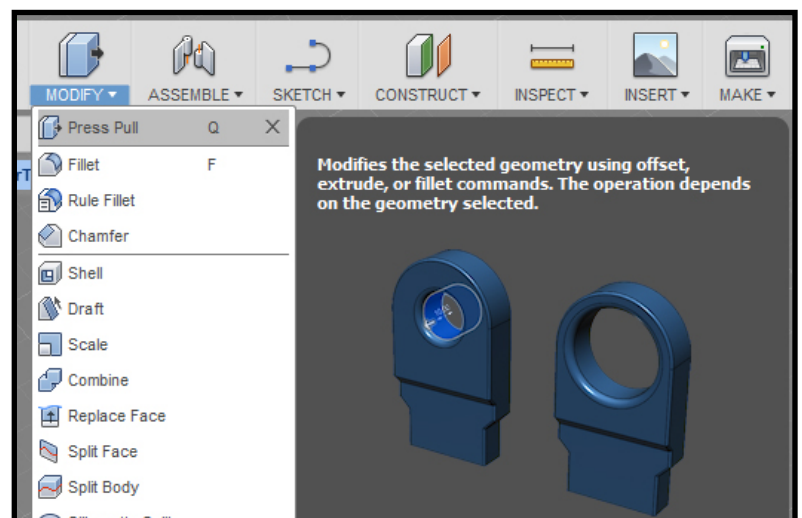


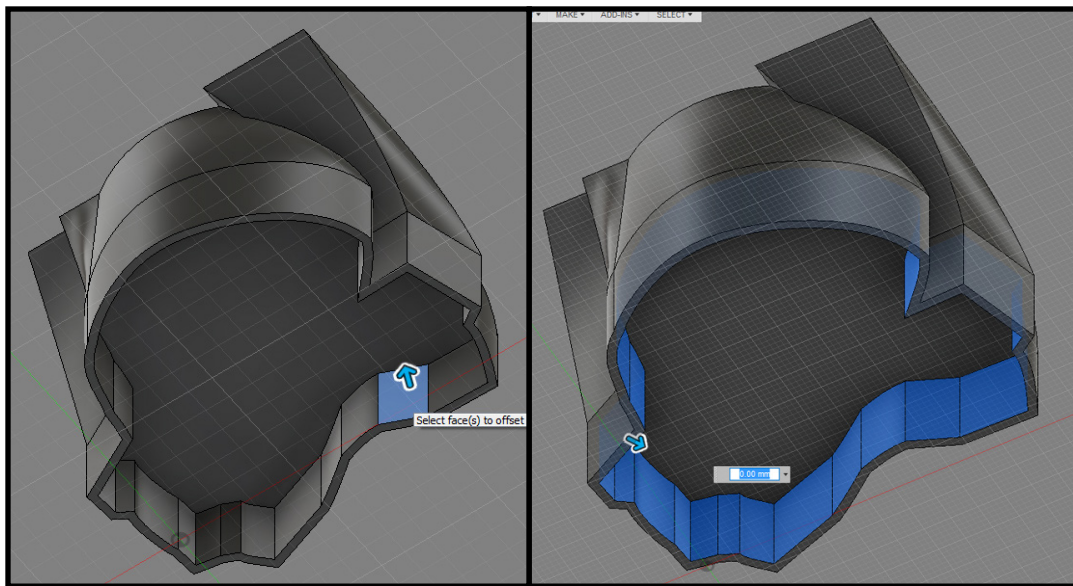
Note: Sometimes only very small radius is acceptable depending on angle of faces touching each other. Also use Fillet tool on sharp outside edges of container bottom lip if desired.

- Hide container bottom body.
- Unhide container lid body.

### **!IMPORTANT! STEP 10: ADD FITTING TOLERANCE TO LID**

In order for the lid to fit on container bottom after printing a fitting tolerance must be added.





• To do this, select an inside face of lid lip, go to the tool bar at the top, click "Modify ▼" to expand the menu, and click "Press/Pull" (or press "Q" on the keyboard). Continue selecting all the faces on the inside of only the lip.

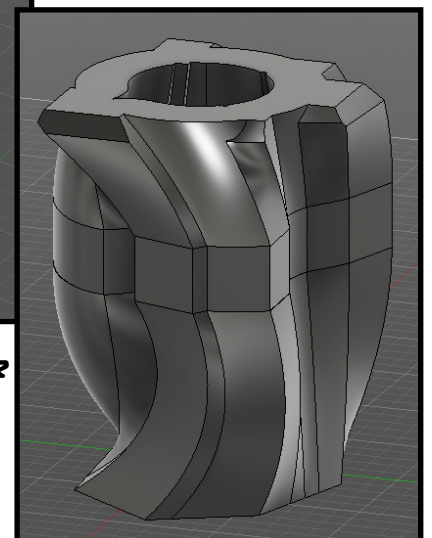
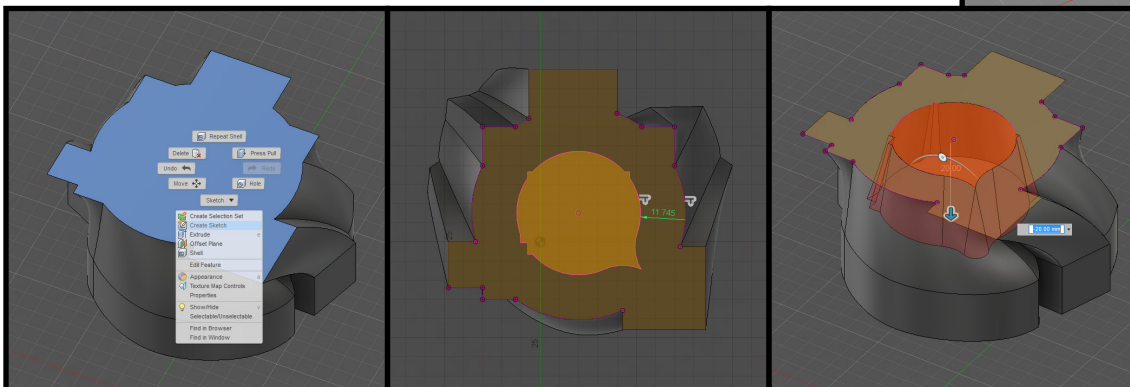
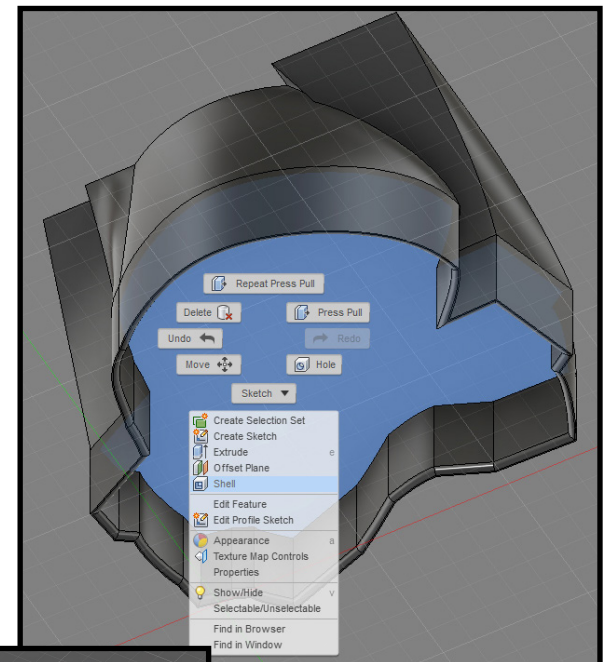
Enter the value of "-0.40mm" and click okay or press enter/return to accept.

Also use Fillet tool on sharp inside edges of container lid lip if desired. This will result in a smoother fit.

## STEP 11: HOLLOW OUT CONTAINER LID

Use the Shell Tool to hollow out the inside of the container lid. To do this, select bottom face inside lid lip, select shell from modify menu, and set value to "2.00mm".

Depending on design, this edit may not work, and will result in a very thick lid. Get creative with offsetting the lid top face edges and extrude at an angle into the lid. This is a good fix to cut back the amount of material needed for the lid.



THE CUSTOM CONTAINER IS TECHNICALLY NOW FINISHED AND READY TO EXPORT FOR PRINTING. IT IS POSSIBLE TO CONTINUE ADDING TO THE DESIGN BY ADDING INSET TEXT ON THE LID OR EVEN A HANDLE ON THE SIDE OF CONTAINER BOTTOM. KEEP IN MIND THE LID HAS TO HAVE A FLAT TOP SINCE IT WILL BE FLIPPED AND PRINTED ON ITS TOP