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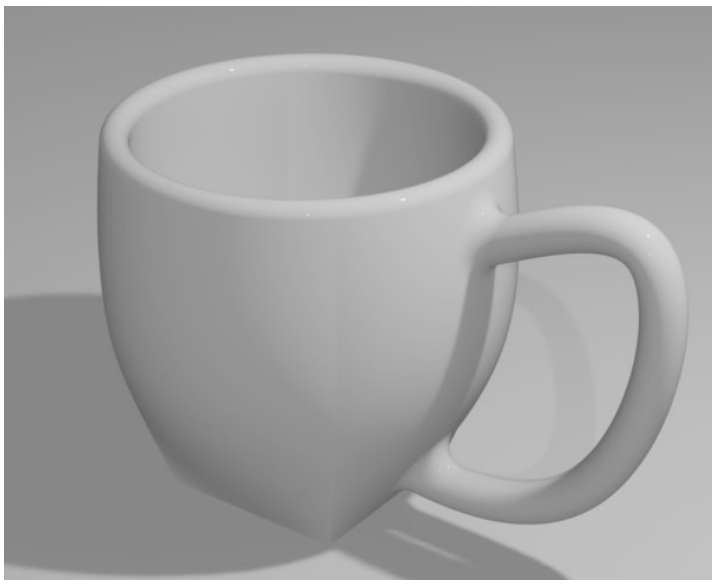
Play · Experience · Learn

3D Design and Printing

Chapter 3



Scope

- Creating a Part with Sweep
- Creating a Part with Loft

3.1 Sweep

Introduction to Sweep

In the previous lessons, we learned to create parts using extrude and revolve.

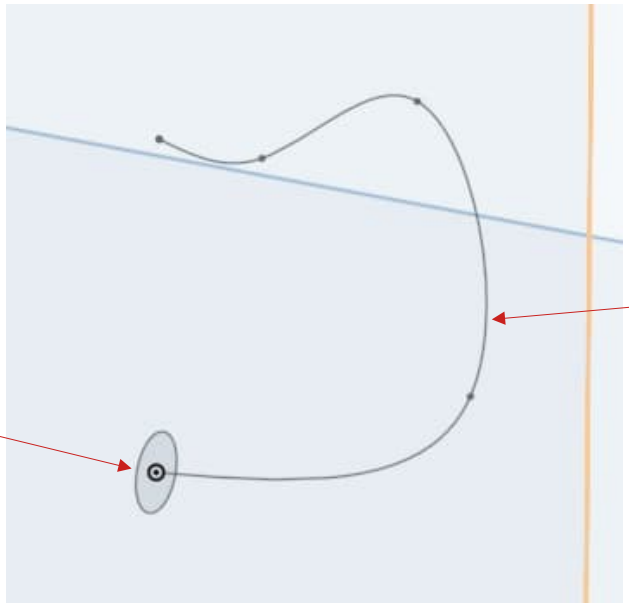
1. **Extrude.** Project the 2D shape in one direction.
2. **Revolve.** Project the 2D shape by rotating it about an axis.

The handle can't be easily made using either extrude or revolve.

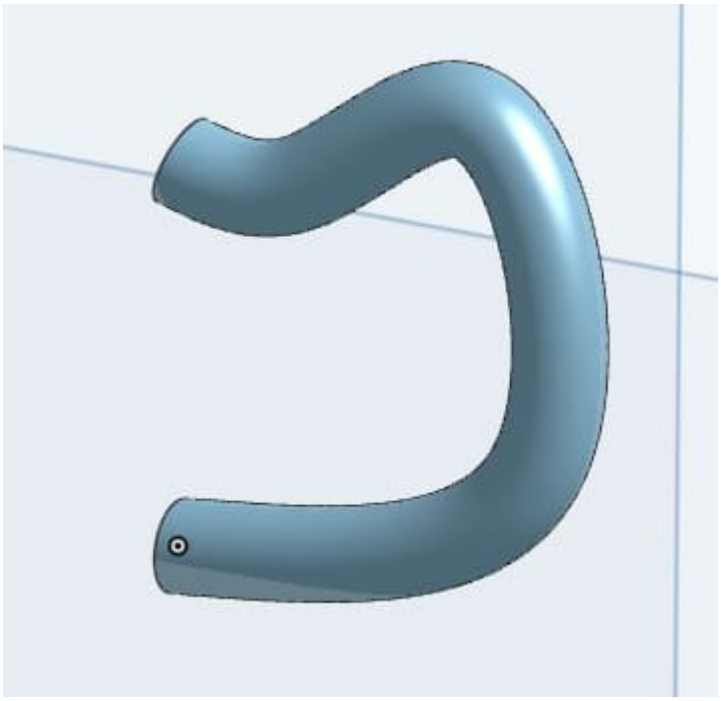
These two methods will cover the majority of your 3D design needs, but for shapes like the tea cup handle to the right, you'll need to use **Sweep**. Sweep is also good for modeling bent or twisted tubes. You won't need it as often as extrude and revolve, but when it's required, there's simply no replacements!

Some other shapes which are can be modelled with sweep: 1) Heater tubes, 2) Reusable straws

There is one main difference between extrude / revolve and sweep; and that is a sweep requires **two** sketches. The first sketch defines the cross-section, while the second sketch defines the path that is used to project the first sketch.



When the sweep is applied, Onshape will take the first sketch and move it along the second sketch. All the areas that it passes through will become a solid shape, like this…



Part Design Process

The part design process is similar, but we'll need to prepare 2 sketches.

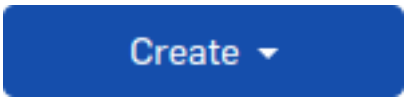
1. **Cross-Section: Surface**. Select a surface to draw on.
2. **Cross-Section: Sketch**. Draw a 2D sketch for the cross-section.
3. **Cross-Section: Dimension**. Set the dimensions for the sketch.

4. **Path: Surface**. Select another surface to draw on.
5. **Path: Sketch**. Draw a 2D sketch for the path.
6. **Path: Dimension**. Set the dimensions for the sketch.

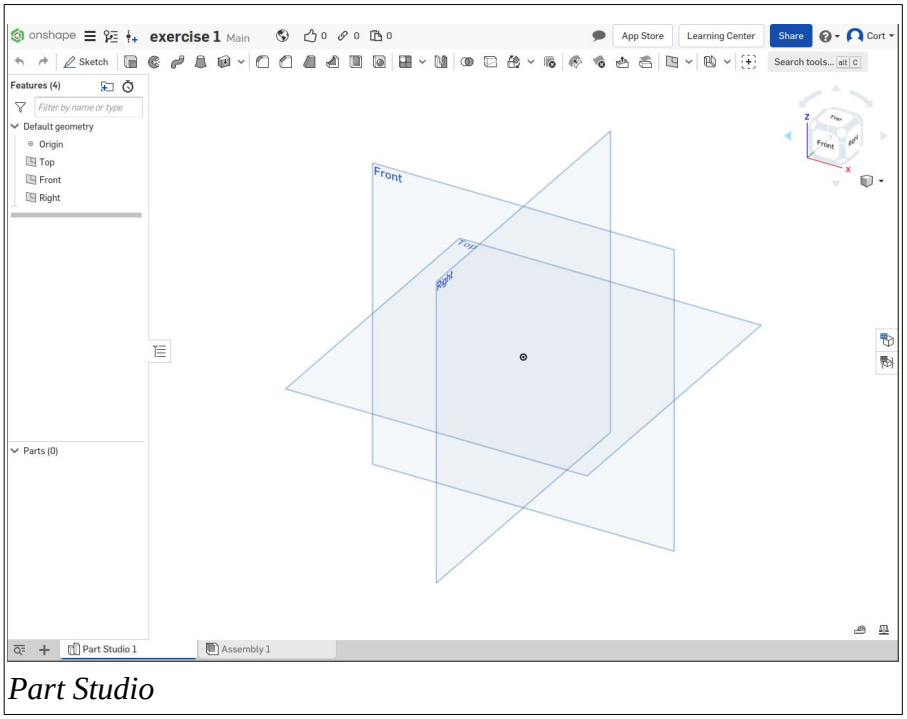
7. **Sweep**. Turn the 2D sketch into 3D.

New Design

From here on, we'll be using the Onshape software. Open the Onshape website and login. We'll be creating a new design. Go ahead and click...



.. select **"Document..."**, and give it a name (eg. "Sweep Exercise 1"). You should now be in the **Part Studio**, which looks like this…



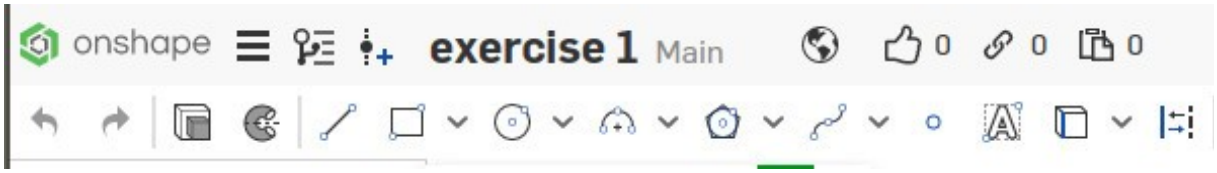
Part Studio

Selecting a Surface (Cross-Section)

As before, we'll need to start by selecting a surface to sketch on. There are 3 default planes available for us to select (Front, Right, Top). It doesn't matter much which one you choose. For this exercise, I'm going to choose Top, but you can pick whichever you like.

Click on the surface to select it (...it'll turn orange), then click...

You're now in the sketch mode, and the tool bar will change to show the sketch tools...

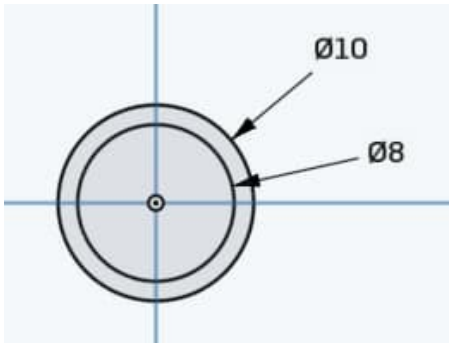


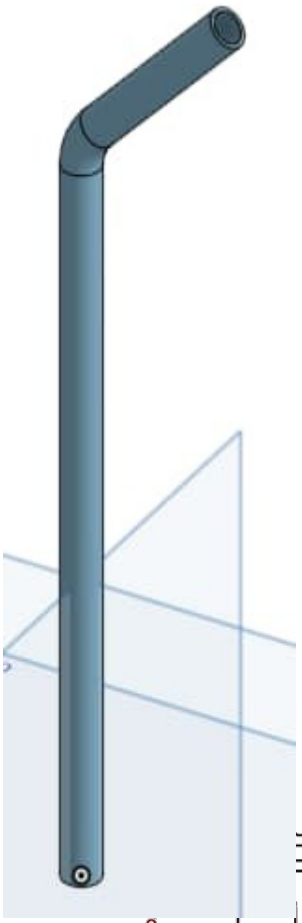
Sketch tools

Sketching (Cross-Section)

For this first exercise, we will be making a reusable straw. It is extremely simple, but it'll get you used to using sweep. Before you start, press **"n"** to rotate the view.

For the first sketch, we'll be drawing the cross-section of the straw. This is easy. The cross-section of a straw is basically just two concentric circles. Make sure that both circles are centered on the origin.





Dimensions (Cross-Section)

After you are done sketching, set the dimensions accordingly.

Click on the dimensions button... 

...then click on the drawing to set dimensions.

- Click on the outer circle to set the diameter to 10mm
- Click on the inner circle to set the diameter to 8mm

If you have done all of these correctly, there should be no blue lines or points remaining. The sketch is now “fully defined”.

If there are still some blue lines remaining, check for these common errors...

- Make sure both the circles are centered on the origin

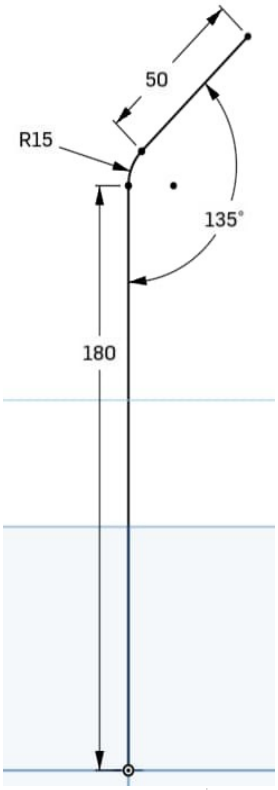
If everything is good, go ahead and click the green tick to complete the sketch.

Selecting a Surface (Path)

The second surface is meant for the path of the sweep. You should choose a surface that is perpendicular to the previous sketch. So if you have used the Top surface in the previous sketch, you should use either the Front or Right surface here. For this exercise, I’m going to choose Front.

Sketching (Path)

The second sketch will define the path of the sweep. Draw the shape on the right.



Important: The 2 straight lines are **tangent** to the arc in the center. This basically means that there isn’t a sharp angle at the point where they join. Such relationships are often implied in engineering drawing; watch out for them.

Recommended steps (...you don't have to follow these, but it's easier this way):

1. Start by drawing the 180mm vertical line. Make sure to start from the origin.
2. Use the "Tangent arc" to draw the arc (R15: radius is 15mm).
   - Place the first click on the end of the 180mm line. This will automatically set a tangent constraint with it.
   - Place the second click near where you see it in the picture.

3. Draw the 50mm line. Start from the end of the arc.

4. Select both the R15 arc and the 50mm line, then click the tangent button to set a tangent constraint. 

Dimensions (Path)

After you are done sketching, set the dimensions accordingly.

Click on the dimensions button... 

...then click on the drawing to set dimensions.

- Click on the vertical line and set the length to 180mm
- Click on the arc and set the radius to 15mm
- Click on the diagonal line and set the length to 50mm
- Click on the vertical line then the diagonal line to set the angle between them to 135 degrees

If you have done all of these correctly, there should be no blue lines or points remaining. The sketch is now "fully defined".

If there are still some blue lines remaining, check for these common errors...

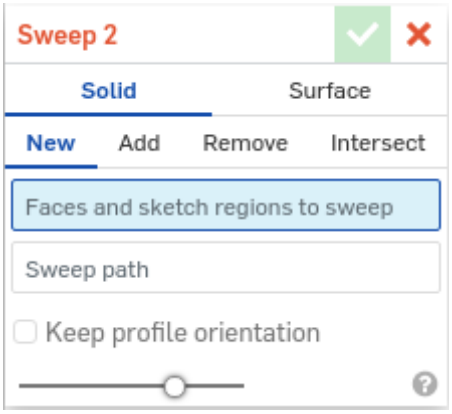
- Make sure the vertical line starts from the origin
- Make sure the arc is tangent to both the vertical and diagonal lines

If everything is good, go ahead and click the green tick to complete the sketch.

Sweep

To start a sweep, click on the **Sweep** button. 

You should now see the sweep dialog box.

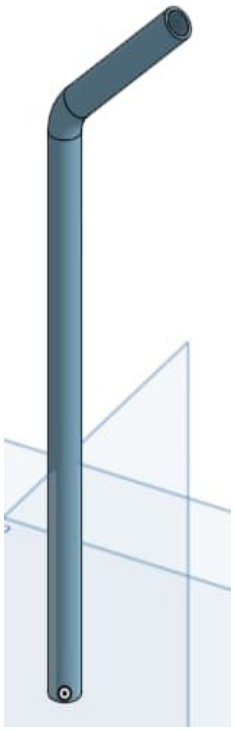


Click “Faces and sketch regions to sweep”, then select the first sketch.

Next, click “Sweep path”, then select the second sketch.

This will turn your 2D sketch into a 3D shape.

If everything looks correct, go ahead and click the green tick to complete the sweep. You should now have a shape that looks like this…

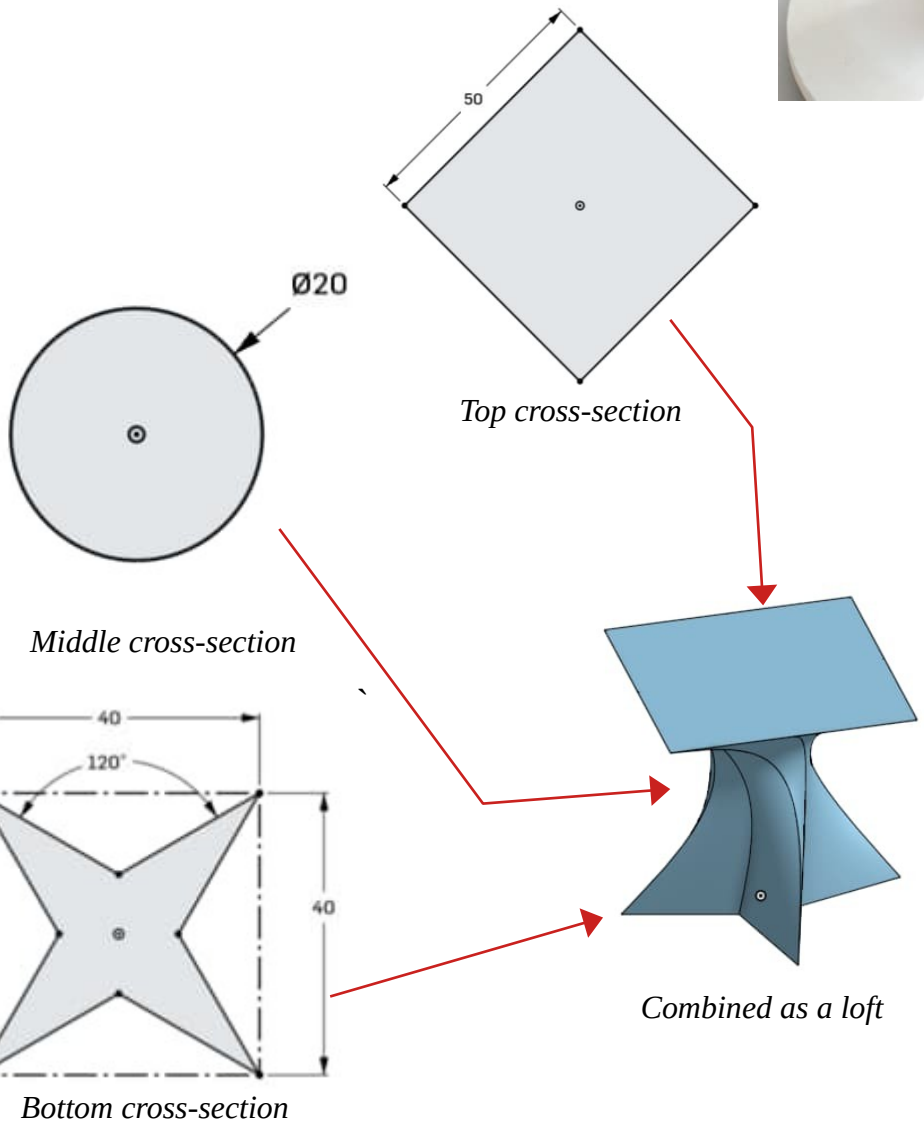


3.2 Loft

Introduction to Loft

One limitation of sweep, is that it can only handle shapes that have a constant cross-section. Shapes like the one on the right, can't be handled with a sweep, extrude, or revolve. For these, you'll need to use **Loft**.

A **Loft** allows you to provide multiple cross-section areas, and the 3D software will automatically generate a 3D shape that combines all of them.



Top cross-section

Middle cross-section

Combined as a loft

Bottom cross-section

Part Design Process

A loft will require multiple surfaces that are parallel to each other. Since none of the default surfaces (Top, Front, Right) are parallel to each other, we'll need to create some additional surfaces that we can use.

You will need at least two sketches for a loft, but depending on your design, you may use many more. For this exercise, I'll be using three sketches, but be aware that there are no limits; you can use as many as you want.

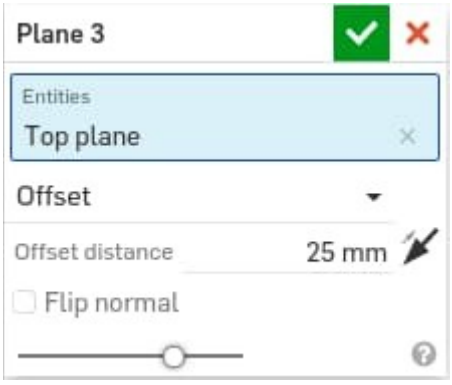
1. **Create Surfaces.** Create parallel surfaces to draw on.
2. **Surface**. Select a surface to draw on.
3. **Sketch**. Draw a 2D sketch for the cross-section.
4. **Dimension**. Set the dimensions for the sketch.
5. **Repeat.** Repeat the previous steps as many times as needed.
6. **Loft**. Turn the 2D sketches into 3D.

Create Surfaces

For this exercise, I'll need 3 parallel surfaces. I'll use the default Top surface as the first, and will need to create 2 more surfaces.

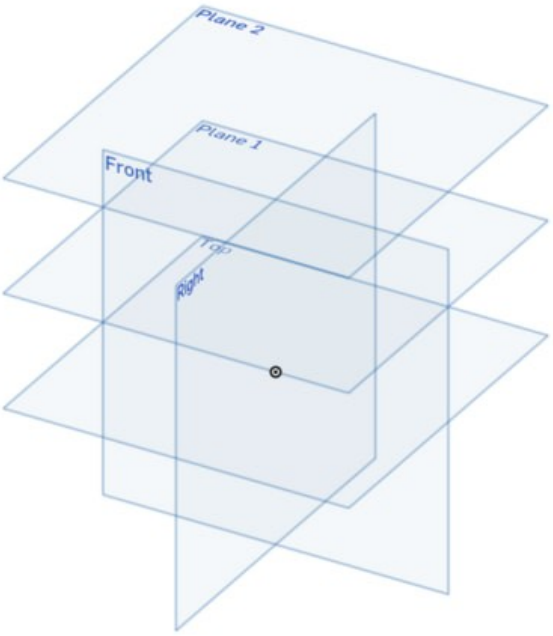
First, select the Top plane, then click the **Plane** button. 

You should now see the Plane dialog box.



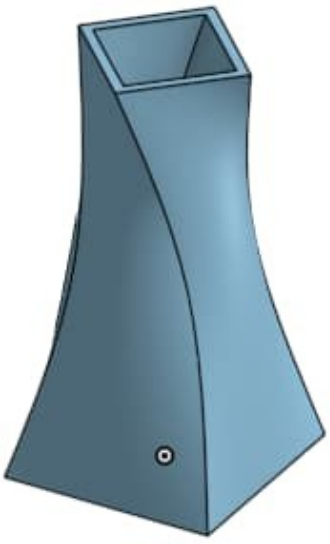
Change the offset distance to **50mm** then click the green tick to create your new plane.

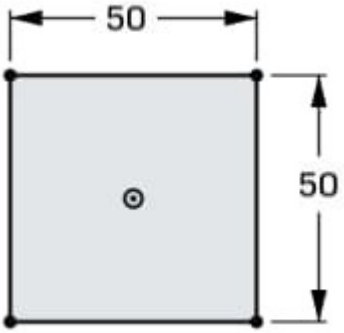
Now select the new plane, and click the Plane button again. Set the offset distance to 50mm then click the green tick. You should now have 3 surfaces looking like this…



Sketching and Dimensions

For this exercise, we will be making a vase. It'll require 3 sketches in total.

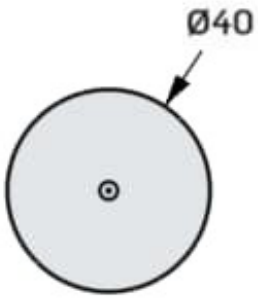




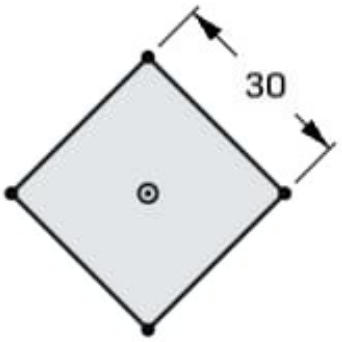
Bottom of vase. Draw this on the "Top" plane.

For the first sketch, use a center point rectangle and draw it centered on the origin.

The second sketch is just a simple circle. Draw it on Plane 1, and make sure it is centered on the origin.



Middle of vase. Draw this on Plane 1.



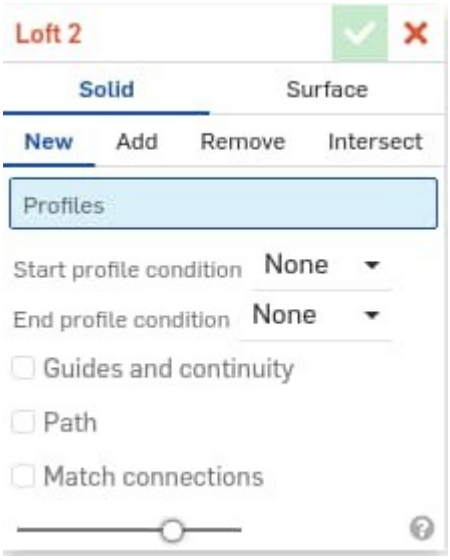
Top of vase. Draw it on Plane 2.

The last sketch should be drawn on Plane 2, and it should be drawn using lines. Make sure that the lines at each corner are perpendicular to each other using the perpendicular constraint button.

Loft

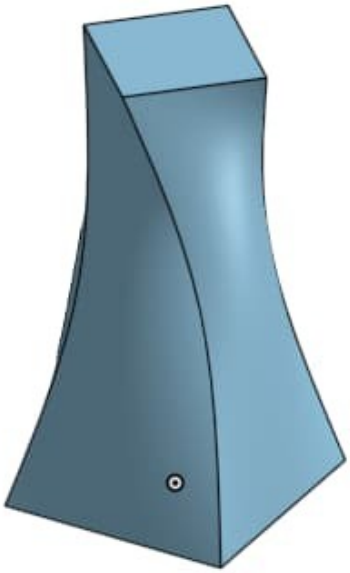
To create the loft, start by clicking on the loft button. 

You should now see the loft dialog box.



Click on “Profiles”, then select each of the sketch one by one.

If done correctly, you should now see this…



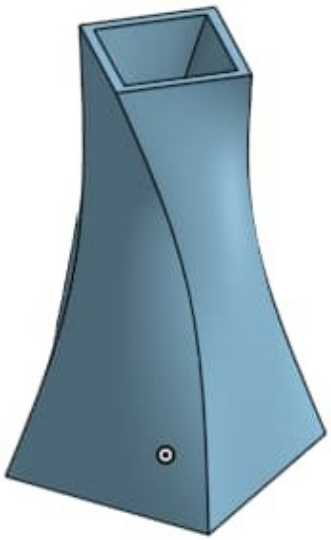
This isn’t much of a vase yet; there are no places to put your flowers! Fortunately, we can use the **Shell** feature to turn this solid shape into a hollow container.

Shell

The shell function “digs” out the insides of a solid object, turning it into a cup / bowl / vase shape.

Start by clicking on the top surface of your vase. then click the Shell button. 

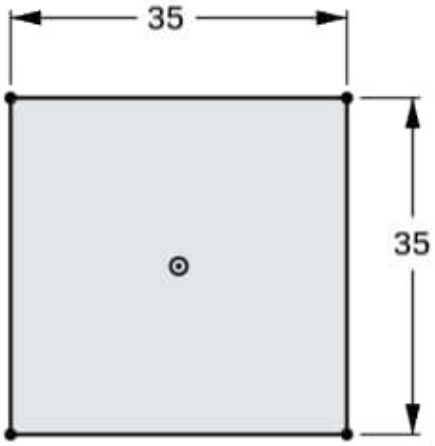
Feel free to adjust the “shell thickness” settings and see what it does. When you’re satisfied with the settings, click the green tick to complete the shell.



3.3 Exercise 6

For this exercise, we'll be creating a tea cup with a handle.

1) **Create -> Document...**
2) **Set name as "Exercise 6"**
3) **Add Surfaces**
    a) Select the "Top" plane
    b) Click the Plane button
    c) Set the offset to 30mm then click the green tick
    d) Select the new "Plane 1"
    e) Click the Plane button
    f) Set the offset to 40mm then click the green tick
4) **Select Surface**
    a) Select the "Top" plane
5) **Sketch**
    a) Click the Sketch button
    b) Press "n" to rotate the view
    c) Draw and dimension the sketch on the right



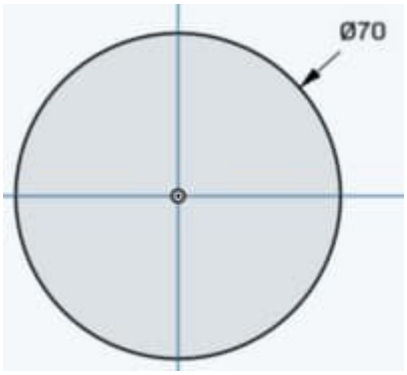
6) **Select Surface**
    a) Select "Plane 1"

7) Sketch

a) Click the Sketch button

b) Press “n” to rotate the view

c) Draw and dimension the sketch on the right



8) Select Surface

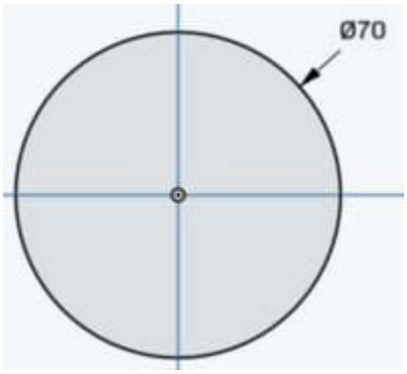
a) Select “Plane 2”

9) Sketch

a) Click the Sketch button

b) Press “n” to rotate the view

c) Draw and dimension the sketch on the right (Note: This is identical to the second sketch, but is on a different plane)



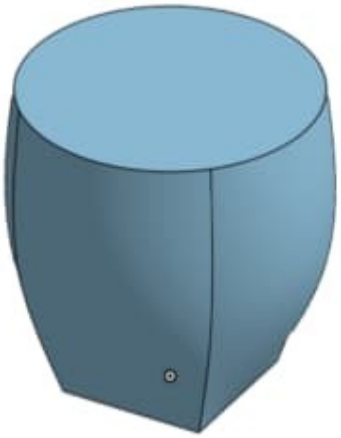
10) Loft

a) Click the Loft button

b) Click “Profiles”, then select the 3 sketches in this order: Sketch 1, Sketch 2, Sketch 3

c) Click the green tick to complete the loft. At this point, you should have the shape on the right

d) This completes the main body of your tea cup

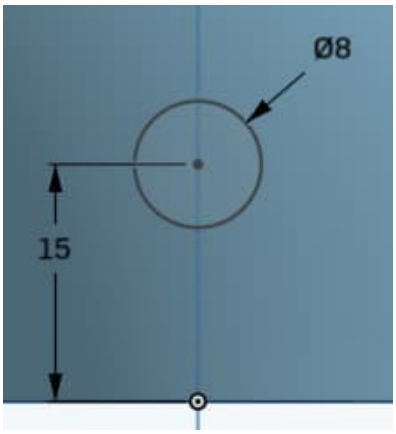


11) Tea cup handle

a) The following steps are for the tea cup handle

12) Select Surface

a) Select the “Right” plane



13) Sketch

a) Click the Sketch button

b) Press “n” to rotate the view

c) Draw and dimension the sketch on the right

14) Select Surface

a) Select the “Front” plane

15) Sketch

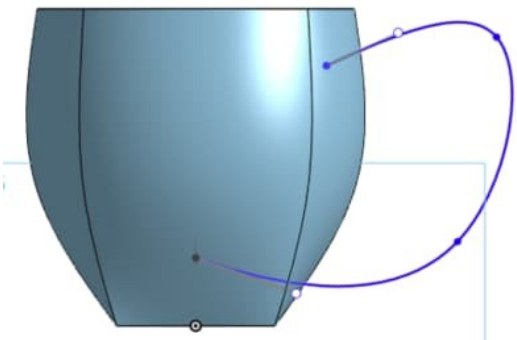
a) Click the Sketch button

b) Press “n” to rotate the view

c) Here we are mainly concerned with the appearance of the handle, and not so much the exact dimensions. In cases like this, it is useful to use the **spline** function.

d) Click the Spline button. 

e) Draw the shape on the right. Note that the line starts from the center of the circle in Sketch 4.



f) Don’t worry if your shape isn’t exactly the same as mine. Note also that there are no dimensions for this sketch.

16) Sweep

a) Click the sweep button

b) Click on “Faces and sketch regions to sweep”, then select sketch 4

c) Click on “Sweep path”, then select sketch 5

d) That completes the handle. Your cup should now look like this.

17) Shell

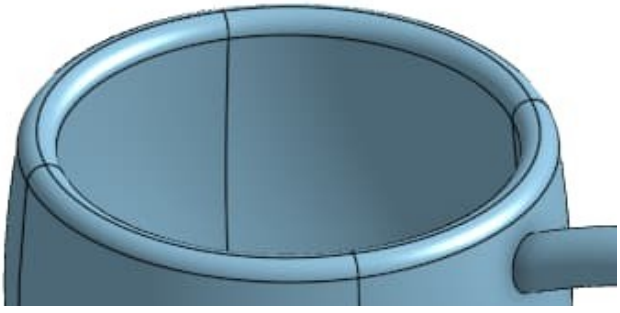
a) Select the top of the cup

b) Click the shell button

c) Set the thickness to 5mm, then click the green tick to complete the shell

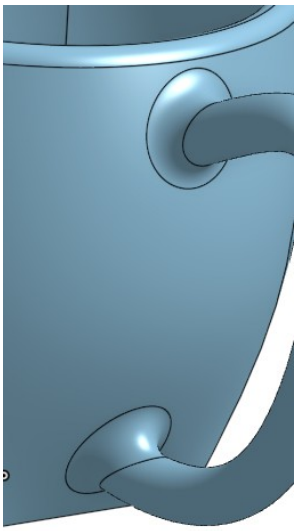
18) Fillet

a) Add a 2.5mm fillet to the top of the cup



19) Fillet

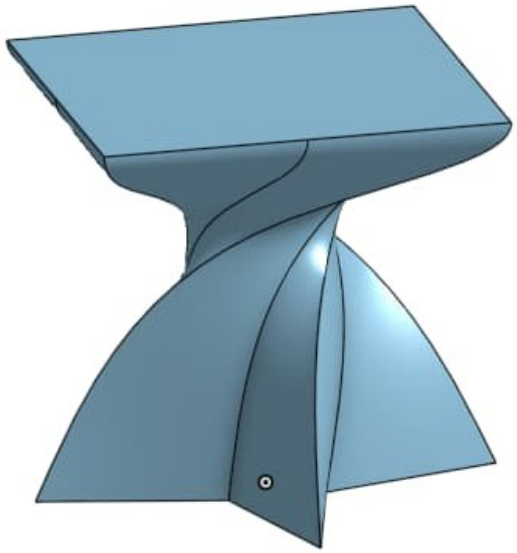
a) Add fillets to the point where the handle attaches to the cup. The fillet radius is up to you to decide.



3.4 Exercise 7

Design using Loft or Sweep

Create your own design using Loft or Sweep. I've provide a couple of my designs here to give you some ideas on what you can do.



Twisted Table

Vase

A table is a little too big to print on your 3D printer, but a vase is possible (...keep the height within 25cm), and you can print your design on your school's 3D printer.