Welcome to Code Crunch!

C12 | Crunch 3D: CAD & Modeling

[Tues 4-5PM] Weekly | Spring 2025

Attendance

Earn a Certificate of Completion by attending at least 70% of the workshops

CODECRUNCH.CS.FIU



Unit #2

Topic: 3D Part Modeling

C12 | Crunch 3D: CAD & Modeling

Key Dates and Structure
C12 - Crunch 3D: CAD & Modeling

- **Week 1: Introduction to OnShape, Sketching Fundamentals, & 2D Constraints**

  Learning what On-Shape is and how to properly sketch on OnShape.

- **Week 2: 3D Part Modeling**

  Learning how to use extrude, remove, fillet, chamfer, revolve, shell, and loft functions for 3D models.

- **Week 3: 3D Part Modeling Continued**

  Learning how to use draft, mirror, and sweep functions for 3D models.

- **Week 4: Assemblies & Mating Constraints**

  Learning how to create assemblies through mates and constraints.

- **Week 5: Final Project - Real-World Design Challenge**

  Applying all concepts to design a functional product.

- **Week 6: Final Project Showcase**

  Presenting your final project, sharing your experience, and receiving feedback.

3D Modeling Definitions

- **Profile** is a closed 2D shape/sketch.
- **Faces** are the different sizes of a 3d Model.
- **Mirror** is used to reflect a sketch entity across a line/axis or a 3D model across a plane. It is useful for creating symmetric parts.
- **Extrude** is a feature that converts a profile into 3D shapes.
  - **Base Extrude** is the first extrude done in the model.
  - **Boss Extrude** is any extrude done after the Base Extrude.
  - **Remove** (a.k.a. Extrude Cut) uses a sketch to remove a section from an extrusion.

3D Modeling Definitions

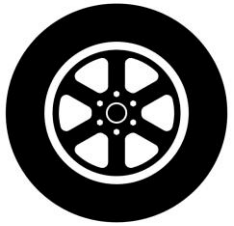
- **Fillet** is a modifier that rounds the edges or faces in a 2D sketch or 3D model.
- **Chamfer** is a modifier that created bevels on edges or faces in the model.
- **Revolve** is a type of extrude that creates a model around an axis.
- **Shell** is used to hollow out an extrusion. It leaves a consistent thickness for all the walls.
- **Loft** connects two or more profiles to form a 3D model or surface.

Session Overview

In this session we will be creating a link, a wheel, and a wine glass to practice the 3D modeling basics of extrusion, remove, fillet, chamfer, revolve, shell, and loft. For all models, we start off with a 2D sketch.



Link:
Stick-like members that connect to each other. They can be used to make a movement (linear or rotational).
To create a link, we will be using extrude, remove, fillet, and chamfer.



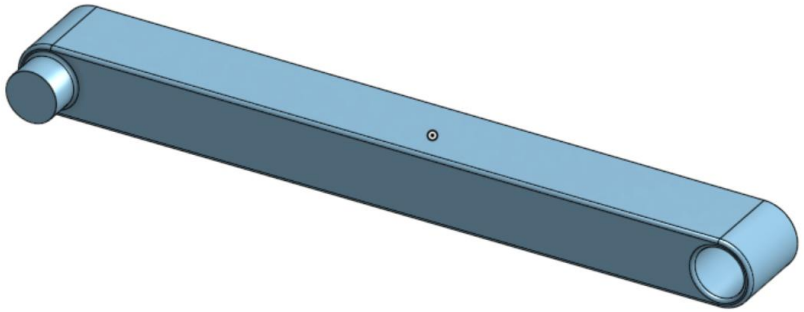
Wheel:
To create the tire for a wheel, we will be using revolve and shell.
This PowerPoint also demonstrates how to create a rim for the wheel using extrude.



Wine Glass:
To create a wine glass, we will be using extrude, loft and shell.

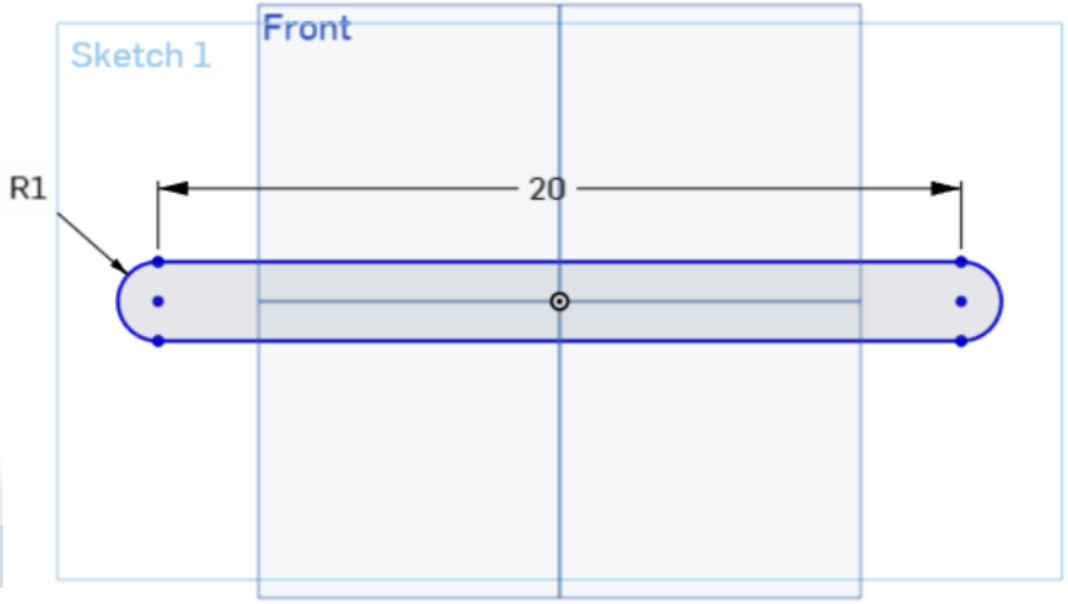
Time for the...

Link!

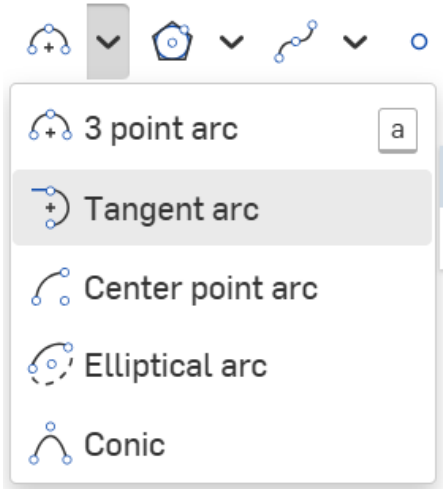


Link: Sketch (& Arcs)

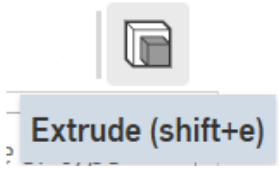
To start with the **link**, we are going to do a **sketch** on the **front plane**. The drawing consist of a **center rectangle** with the sides **trimmed**, and 2 **tangent arcs**. The width of the rectangle is 20 cm, and the radius of the arcs is 1 cm.



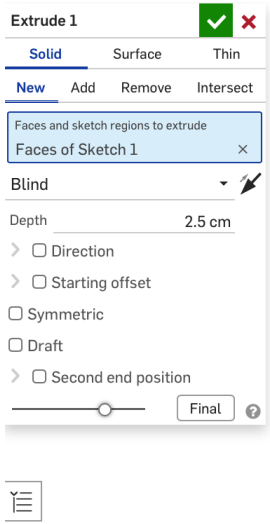
Note: Arcs allow you to make curves. A **tangent** arc allows for a smooth transition between the arc and another line/curve. Types of arcs:



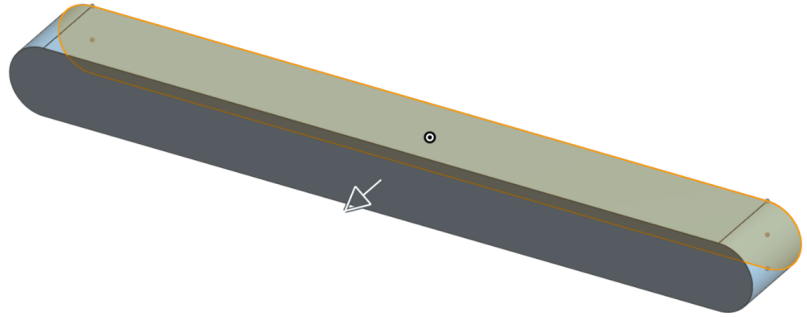
Link: Extrude Base



After the sketch is done, click the **green check mark** on the **sketch menu**. Then, select the sketch on the left side and click **extrude** on the top. On **depth**, write 2.5 cm, and click the **green check mark** to make the 2D sketch a 3D model.

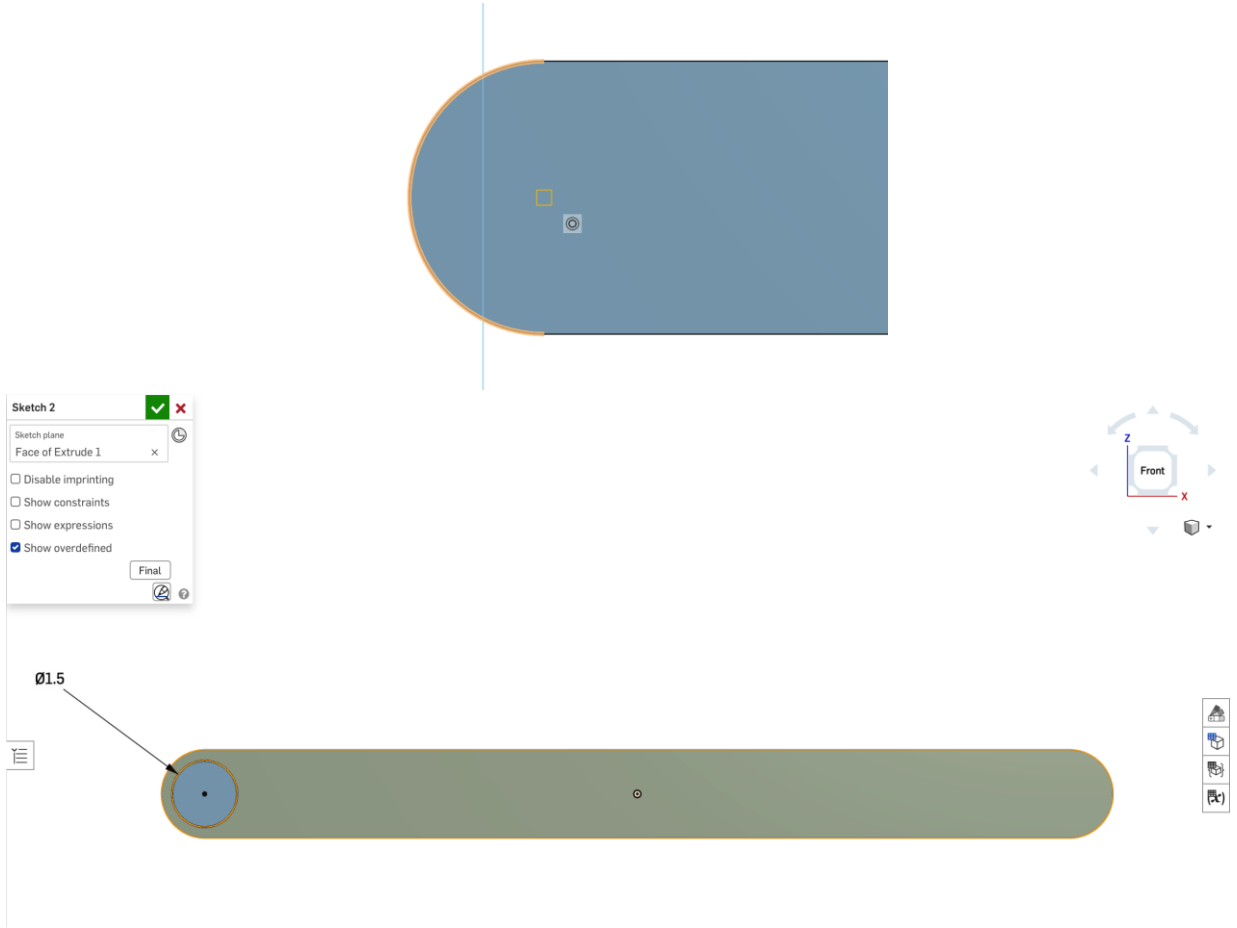


Note: when you finish the sketch, you can go directly to extrude by clicking the extrude feature at the top.



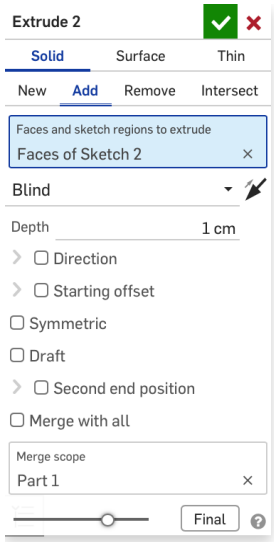
Link: Sketch on a Body

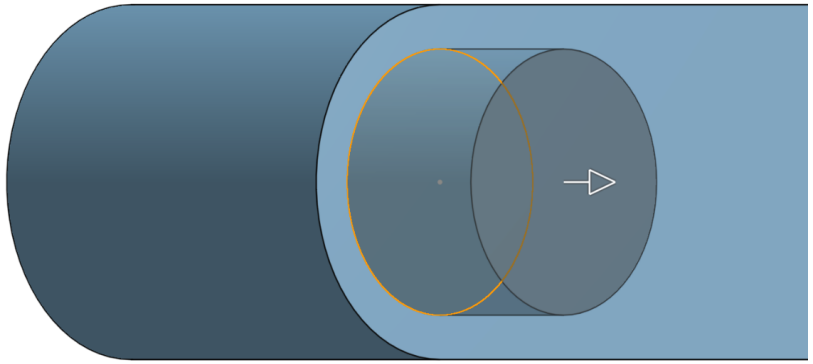
Just like in a plane, you can do **sketches on faces**. Select the **face** (in this case, the front one) and do a sketch. Draw a **center point circle** in the **center of the arc** (indicated by the top point of the arc) with a radius of 1.5 cm.



Link: Extrude Boss

Once the sketch is completed, click **extrude**, and change the **depth** to 1 cm, and click the **green check mark / Enter key**.

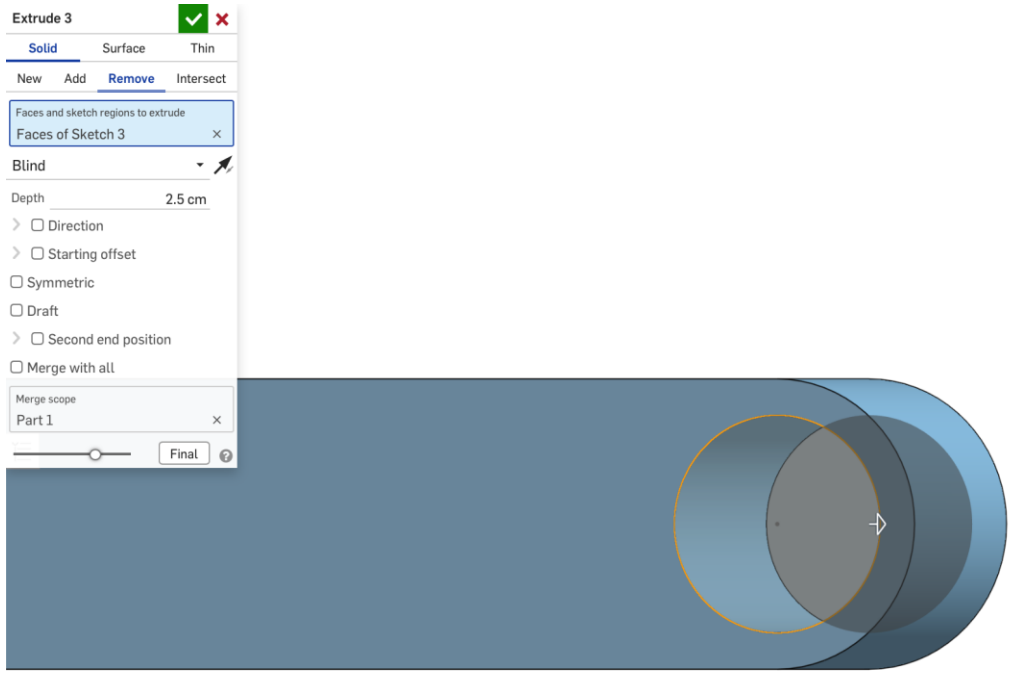




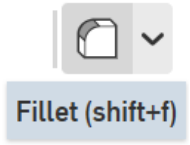
Link: Remove

Now, we do the same circle sketch on the other side of the bar. After we have the sketch, click **extrude**, change from "**add**" to "**remove**", and click enter.

Note: The remove feature went through the entire link because 2.5 cm is the **default** measurement. If a hole is not desired and rather an **indent**, the depth must be changed.



Link: Fillet

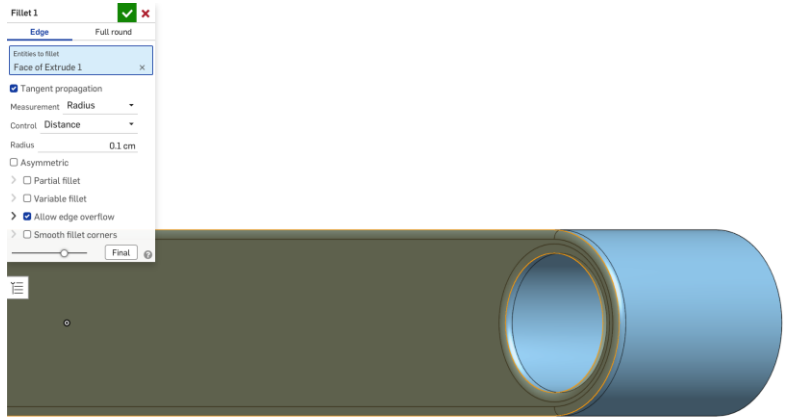


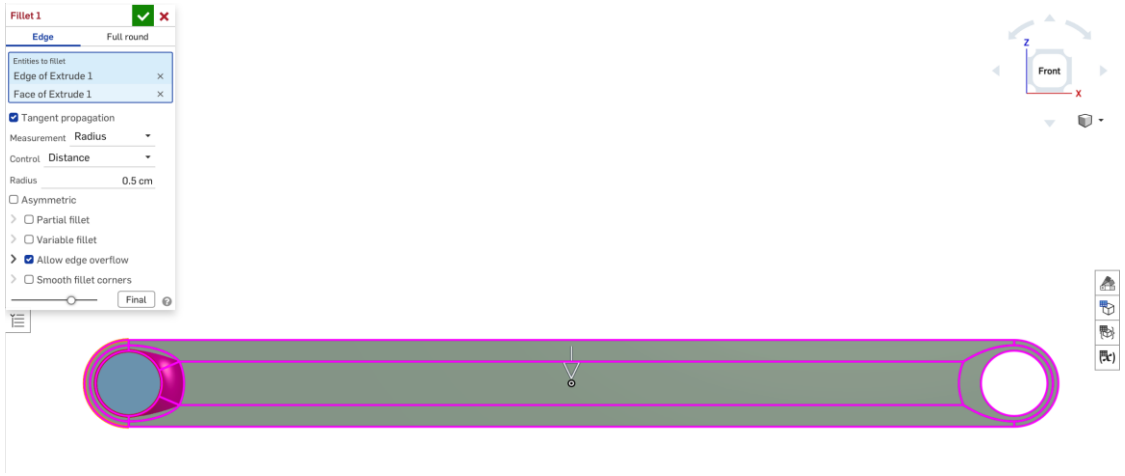
A **fillet** is used to rounds the edges or faces in 2D sketch or 3D model.

To do a **fillet** for the front face of the link:

1. Select the front **face** where you want to add it
2. Change the **radius** to 0.1 cm
3. Click the **green check mark**

Note: When the **fillet is too big**, OnShape will indicate it with **pink lines**. If any changes are necessary, select the **feature**, right click it, and select **edit**.

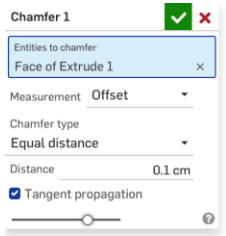




Link: Chamfer



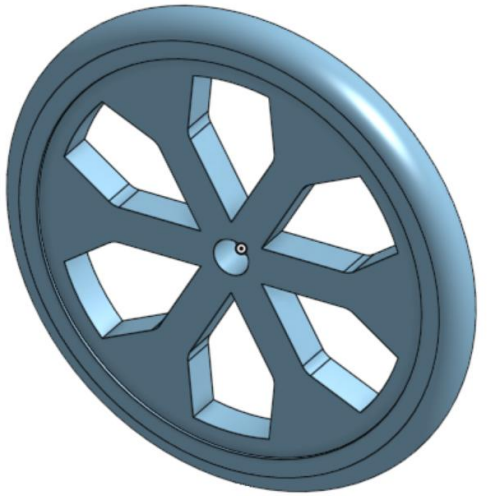
Chamfer is very similar to **fillet**. Select the back face of the bar and add a **chamfer** with a **distance** of 0.1 cm.





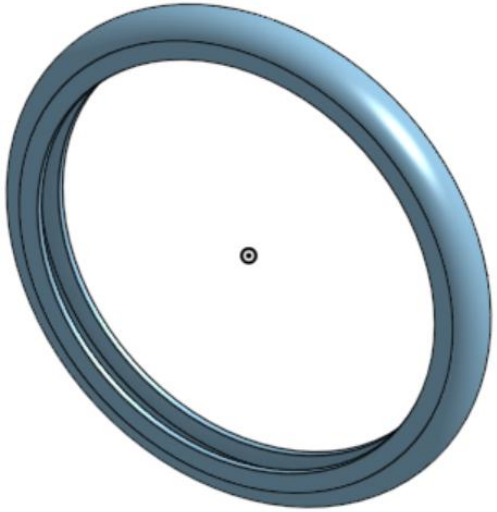
Time for the...

Wheel!



First...

Wheel Tire!

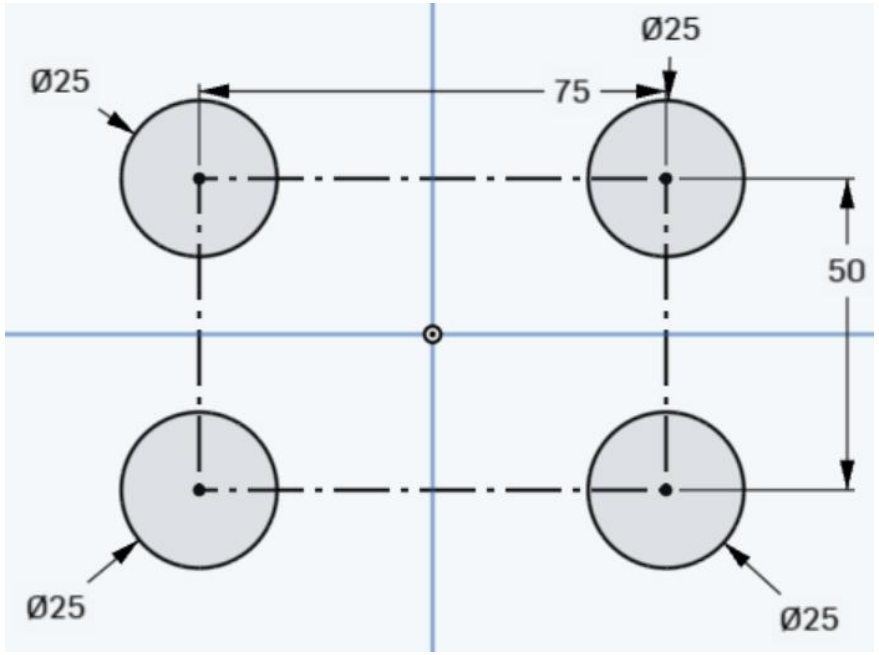


Sketching: Construction



Sketch entities can be made into **construction** lines/curves. **Construction** lines/curves are used as a **reference** or a **guide** for other components of a 2D sketch, 3D part, or assembly. They are **not** used to create features.

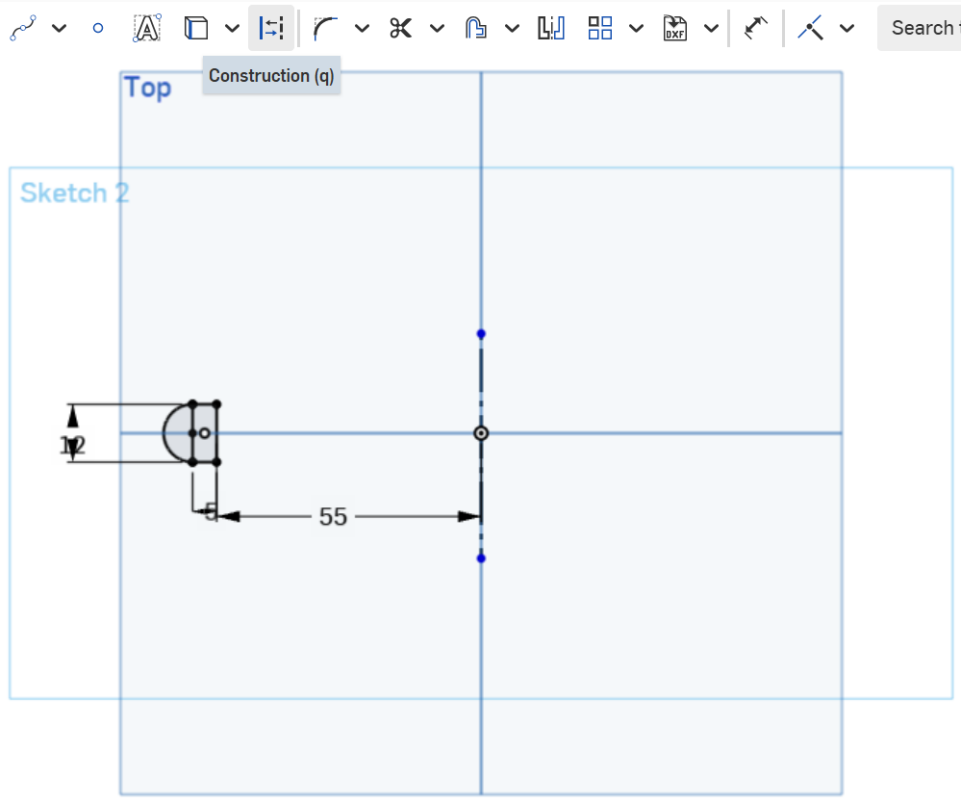
The image to the right displays a construction center point rectangle.



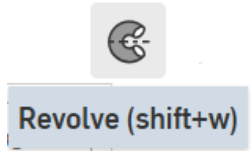
Wheel Tire: Sketch

To start with the **wheel**, we are going to do a **sketch** on the **top plane**. The drawing consist of a **center rectangle**, a **tangent arc**, and a **construction line**.

The **center** of the **center point rectangle** is made to align with the horizontal axis, and it is **displaced** 55 mm to the left of the origin. A **tangent arc** is added to its left side. The rectangle has a **width** of 5 mm and a **height** of 12 mm (this automatically makes the tangent arc have a radius of 6 mm). Lastly, a **construction line** is drawn vertically through the origin (its length does not matter).

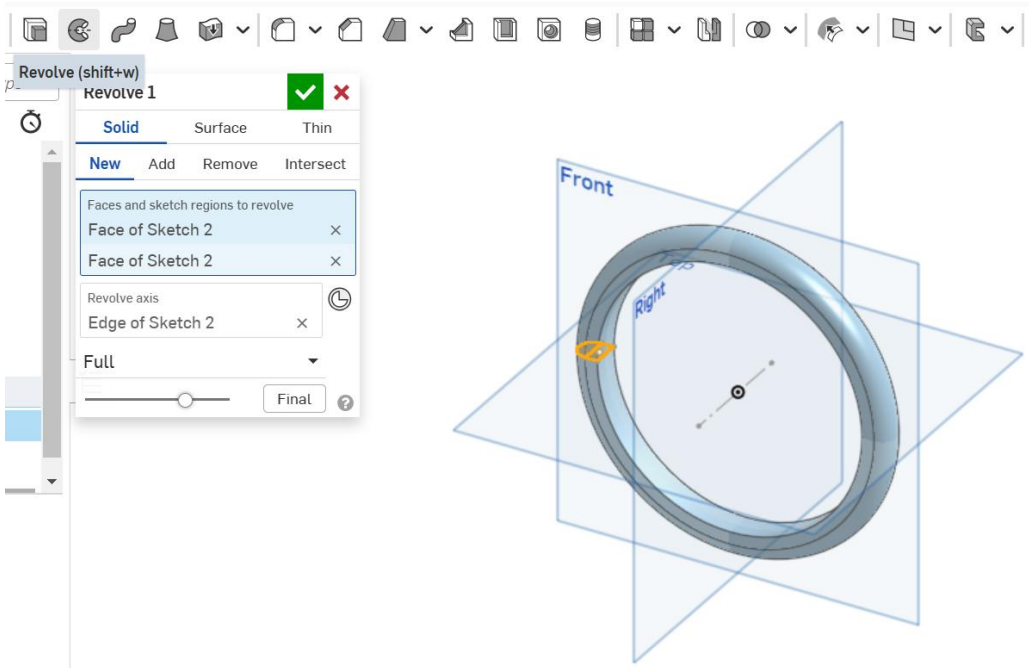


Wheel Tire: Revolve



Revolve allows you to rotate a 2D sketch about an axis of your choice.

For the wheel, we revolve our 2D sketch about the construction line.



Notes:

- Ensure that “**solid**” is selected for the revolve to make sure it is treated as a 3D part.
- As demonstrated, the construction line was used as a guide for the revolve feature, but it was not used as a component of our 3D model.

Wheel Tire: Shell

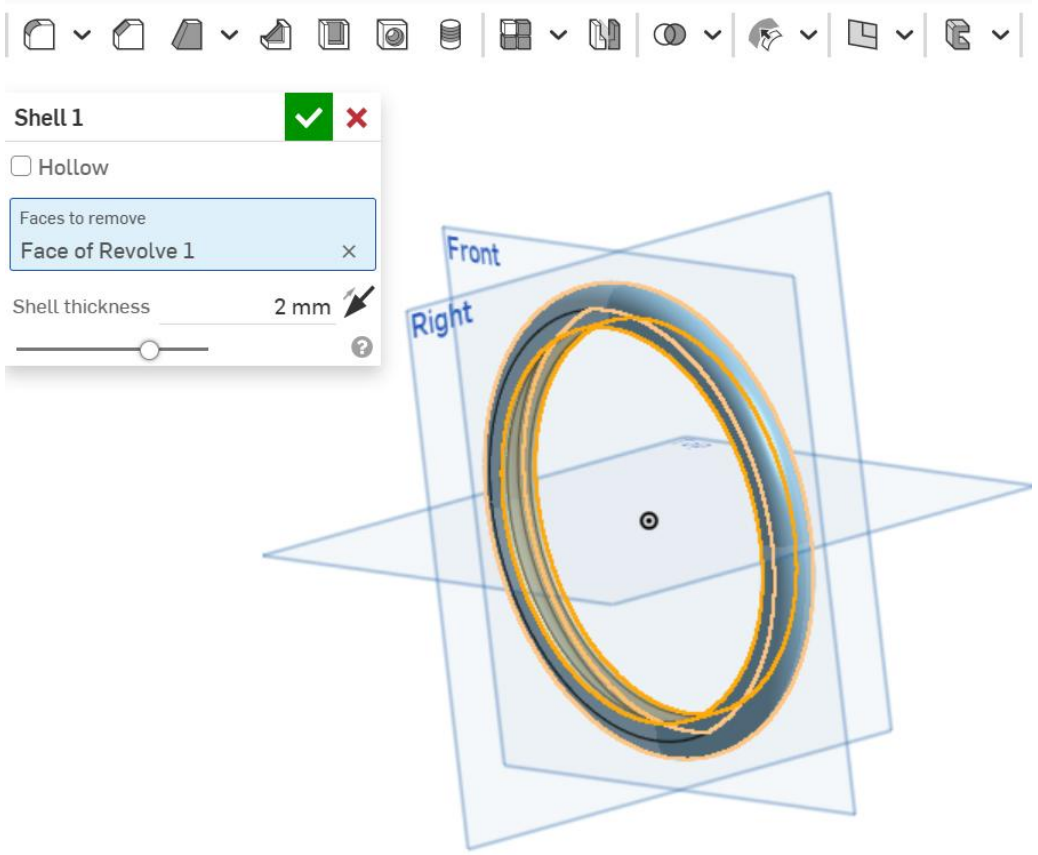


Shell is used to hollow out an extrusion with a consistent thickness for all the walls.

For the wheel, we used shell to hollow out the tire.

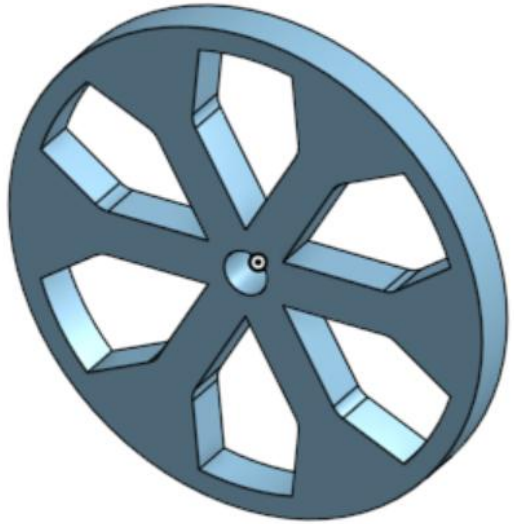
Steps:

1. Click **shell**
2. Click the **inside** cylinder of the wheel
3. Make **shell thickness** 2mm
4. Click the check mark to complete your tire



Extra: Then...

Wheel Rim!

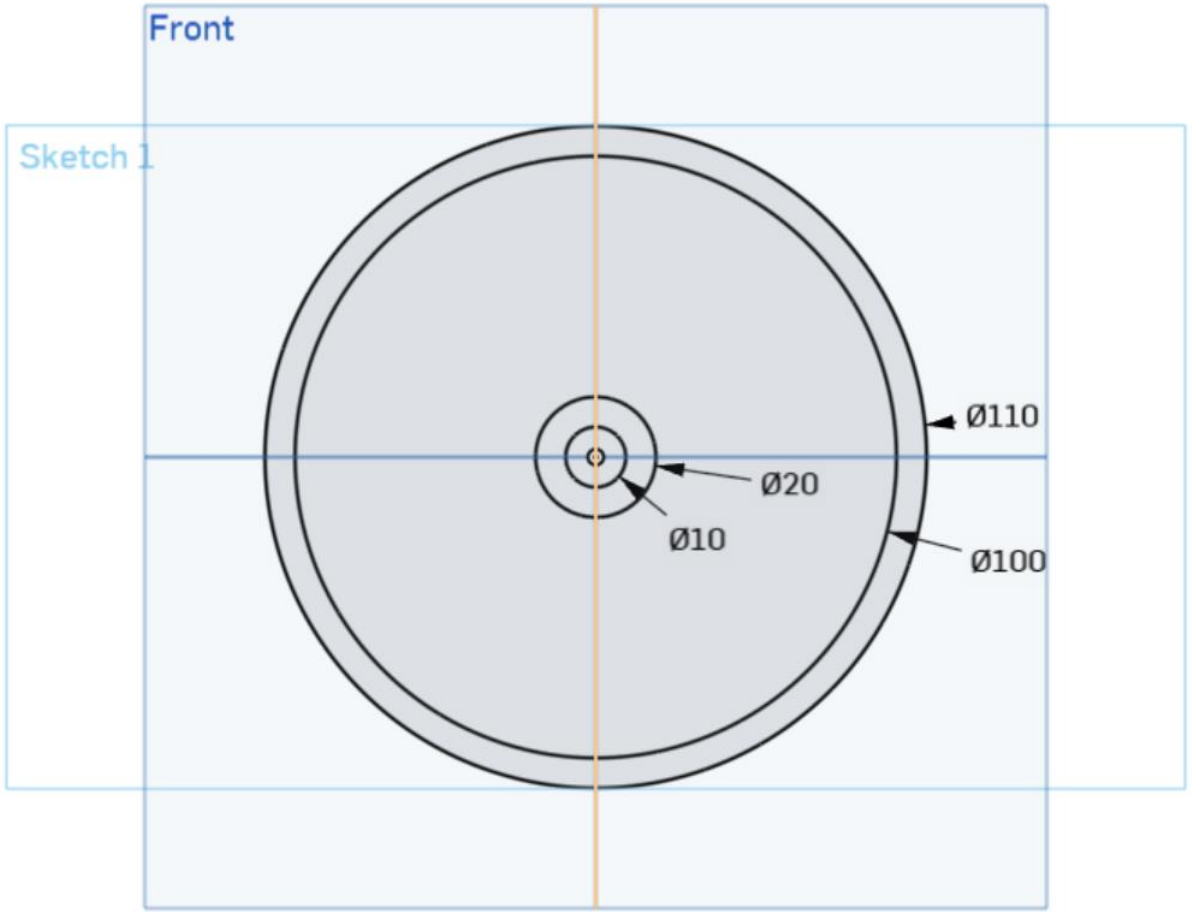


Wheel Rim: Sketch Pt. 1

The **sketch** on the **front plane** is the main feature for the rim of the wheel.

Step 1:
Create 4 **center point circles** with their centers at the origin. **Diameters**:

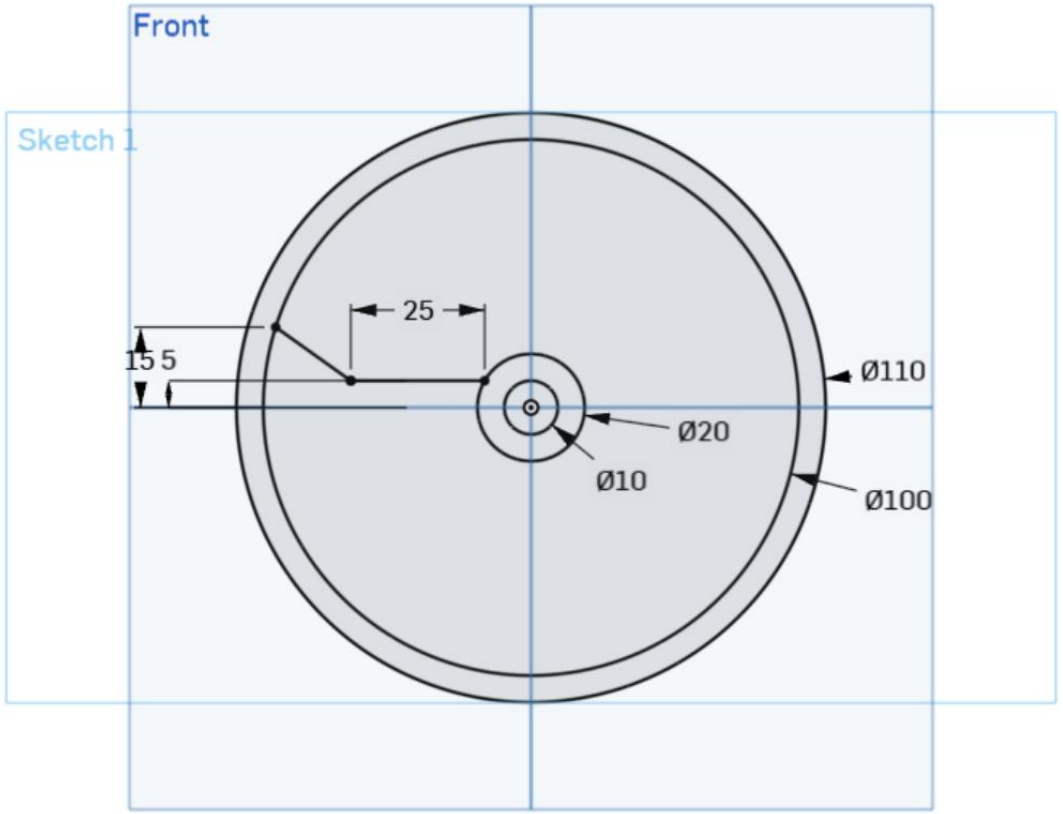
1. 110 mm
    - Note: 2x55 mm=110 mm. This is why the inner radius of the wheel tire is 55 mm.
2. 100 mm
3. 20 mm
4. 10 mm



Wheel Rim: Sketch Pt. 2

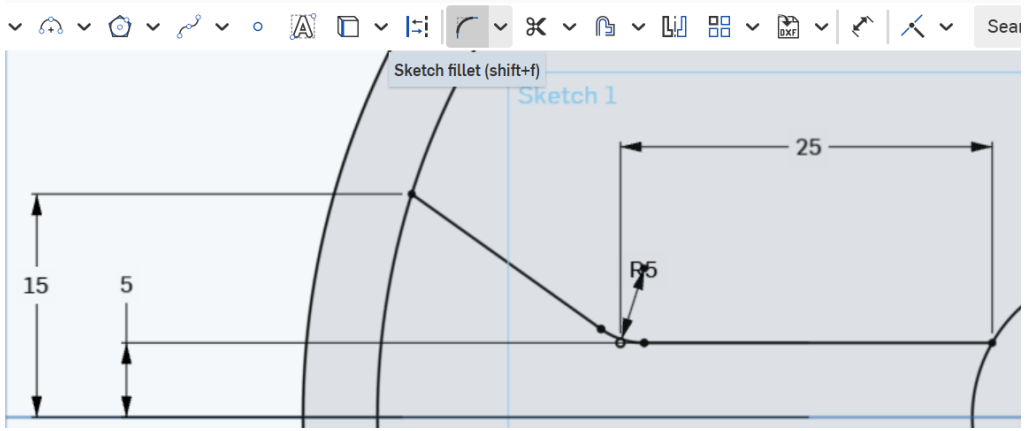
Step 2:

1. Create a **horizontal line** of length 25 mm that touches the 3rd circle (20 mm). Make it 5 mm away from the horizontal axis.
2. Make another **line** connecting the horizontal line to the 2nd circle (100 mm). Make the point at the end of this line that touches the 2nd circle be 15 mm away from the horizontal axis.



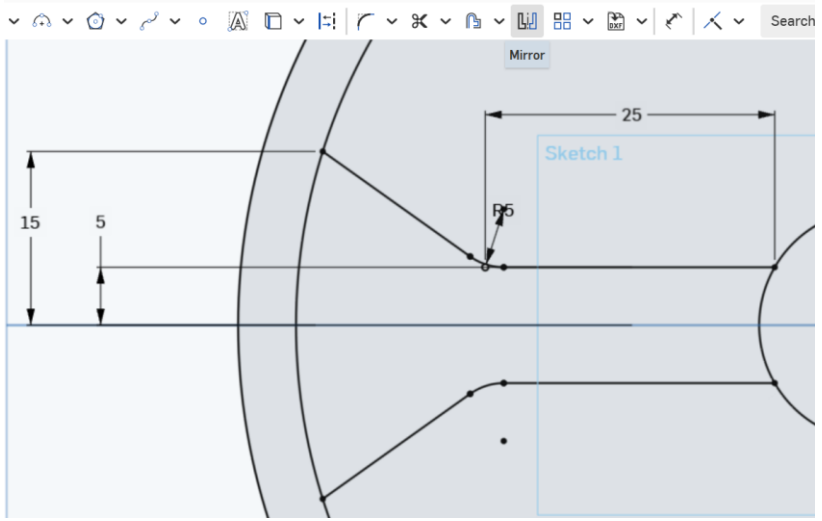
Wheel Rim: Sketch Pt. 3

Step 3:
Make a **sketch fillet** with a **radius** of 5 mm at the point where the 2 lines **intersect**.



Step 4:
Mirror the 2 lines and fillet about the horizontal axis.

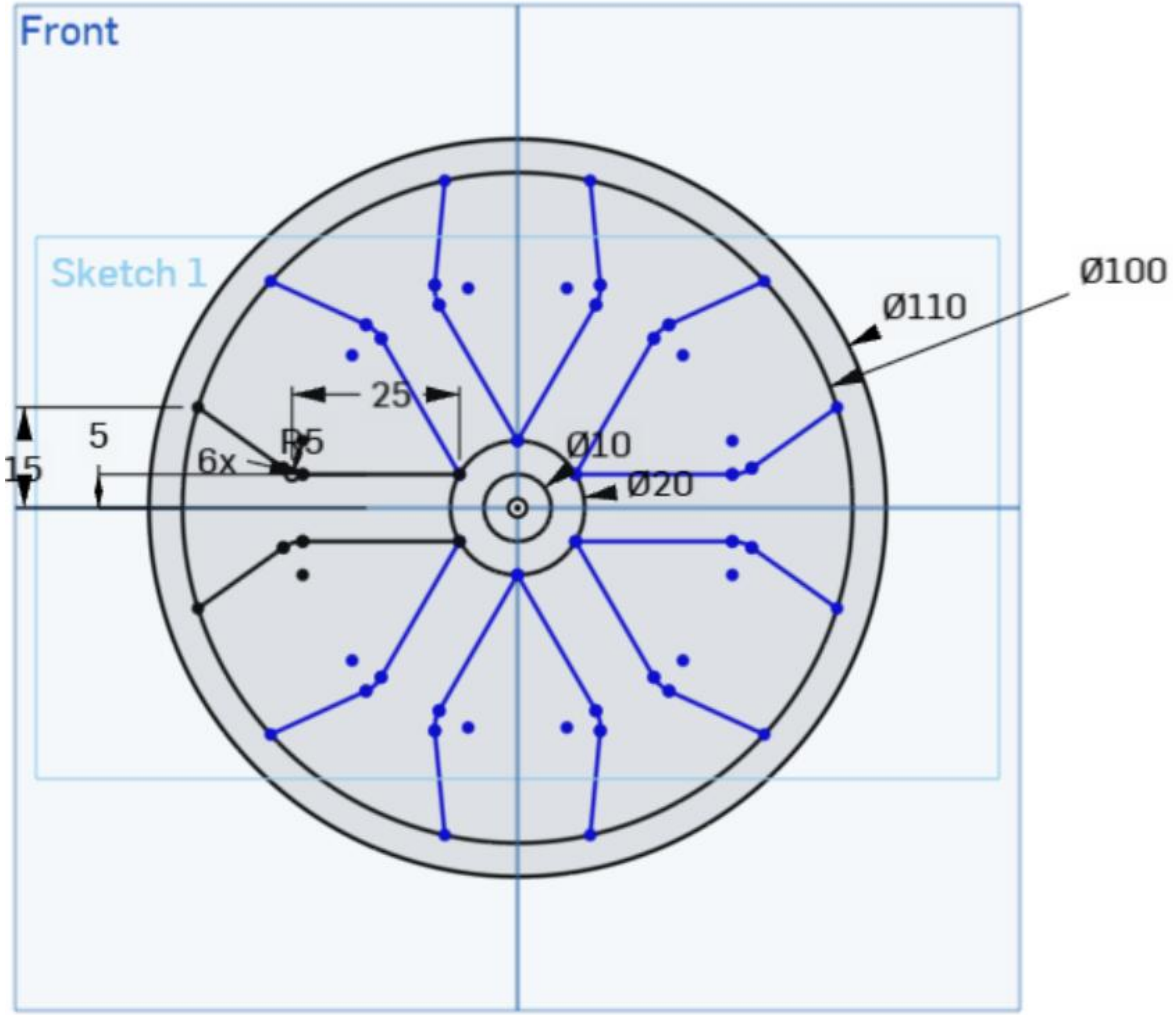
Note: **Mirror** is used to reflect sketch entities across a line/axis. In this case, select **mirror**, then the **mirror line** is the horizontal axis, and the 2 lines & fillet are the **entities to be mirrored**.



Wheel Rim: Sketch Pt. 4

Step 5:
Create a **circular pattern** of the 4 lines and 2 fillets in between the circles.
Make them repeat 6 times.

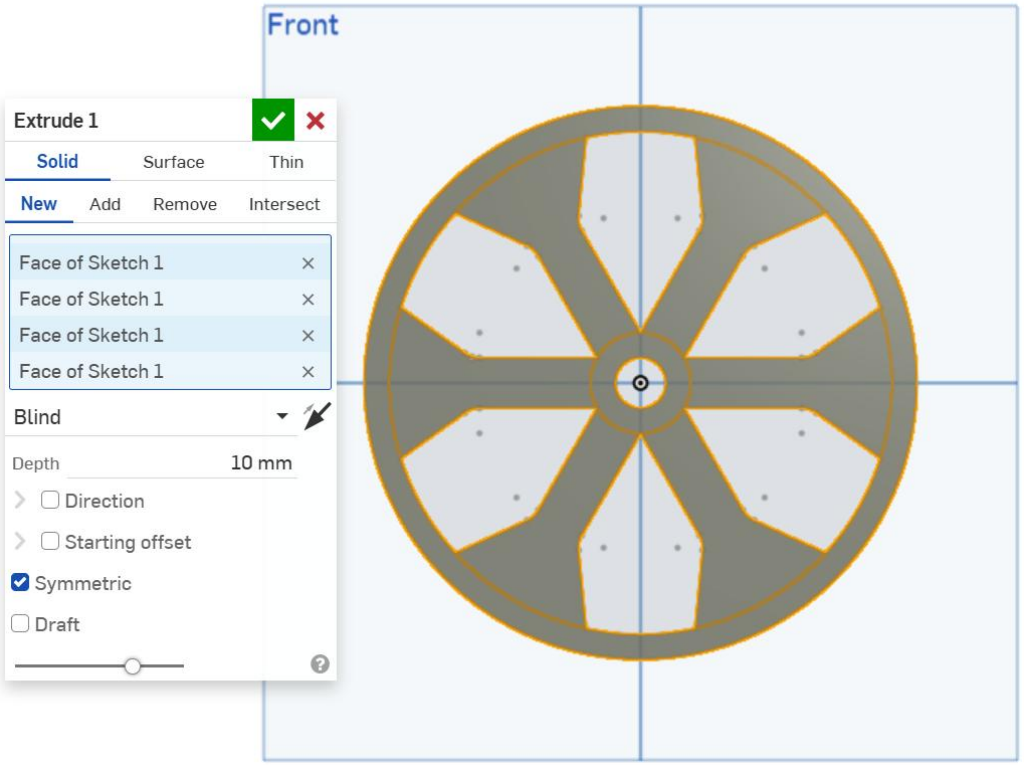
Step 6:
Click the green check mark to complete the sketch.

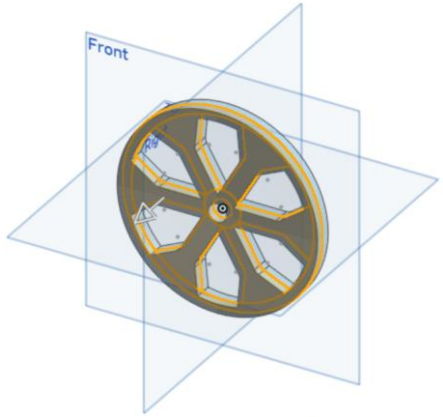


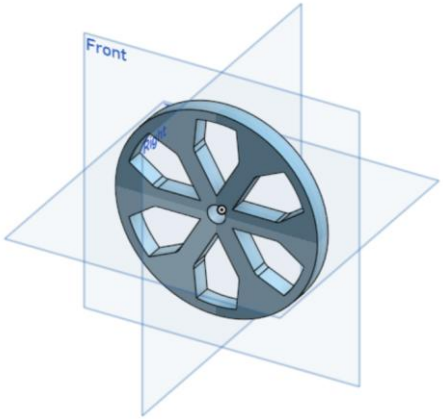
Wheel Rim: Extrude (using symmetric)

To make the rim a **3D model**, we are going to **extrude** the sketch and make the **depth** 10 mm.

Check **symmetric** to make the rim extrusion have an equal **depth**/thickness on both sides of the **front plane**.

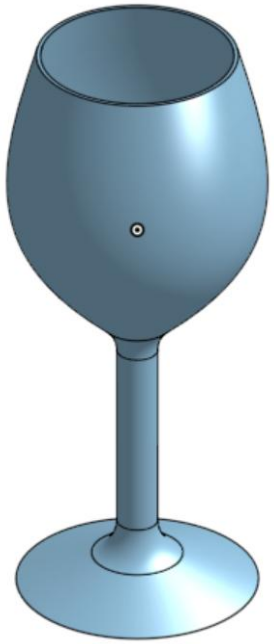






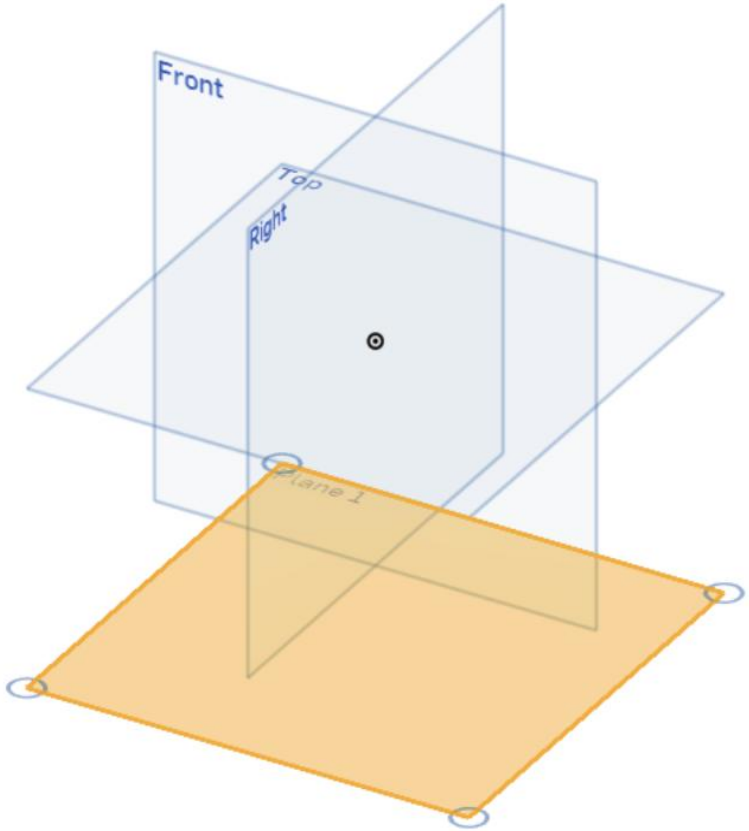
Time for the...

Wine Glass!



Offset Plane

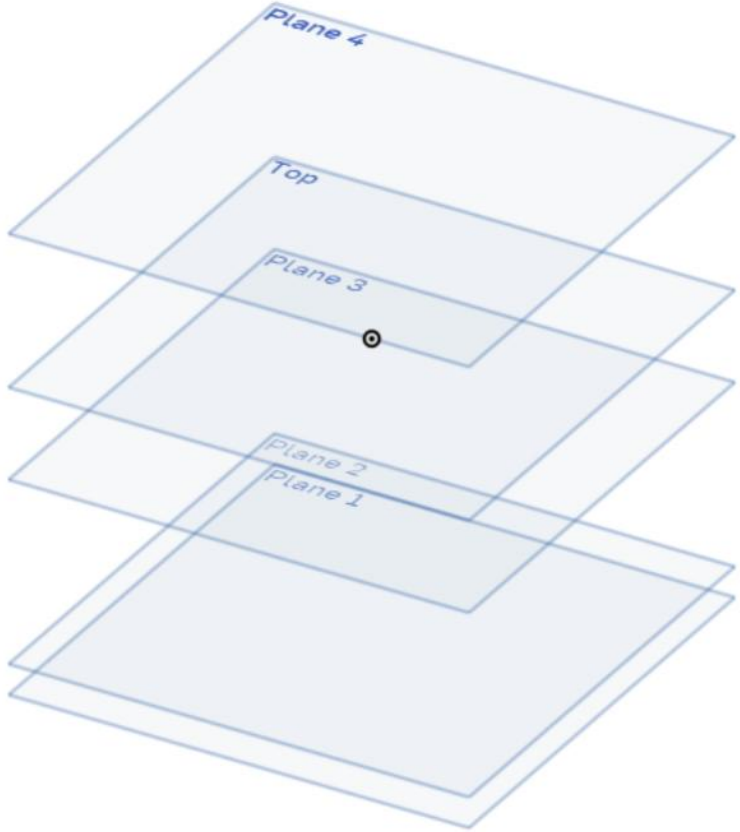
Before we can start with the **Wine Glass**, we need to learn about **offset planes**. **Offset planes** are copies of the **original planes** at a **distance**. To create an **offset plane**, click **“plane”**,  select the plane you want to make a copy of (front, top, or right), and choose an **offset distance** and the **direction**.



Wine Glass: Offset Plane

For the **wine glass**, we are going to need 4 **offset planes** of the **top plane**.

1. Plane 1: 10 cm downwards
2. Plane 2: 9 cm downwards
3. Plane 3: 3 cm downwards
4. Plane 4: 5 cm upwards
    - Note: The front and right plane were hidden for a better visualization.

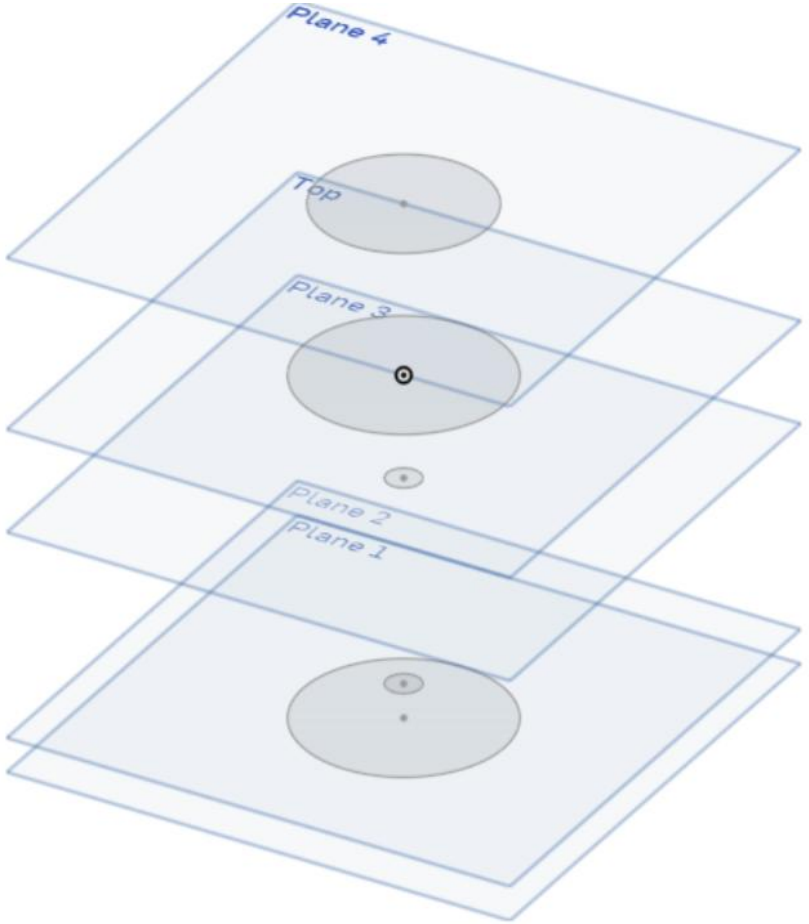


Note: To **hide** planes, place the clicker on top of it on the left, and click the eye next to it. For features, right click them and select suppress. To **unhide**, repeat the same steps.

Wine Glass: Sketch

For this 3D model, we are only going to **sketch circles**. Draw a circle with diameter of

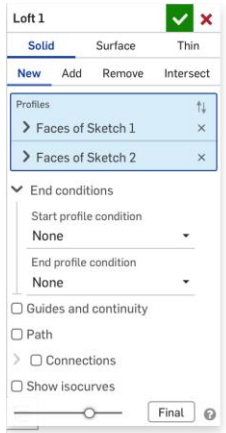
1. 6 cm for Plane 1
2. 1 cm for Planes 2
3. 1 cm for Plane 3
4. 6 cm for Top plane
5. 5 cm for Plane 4

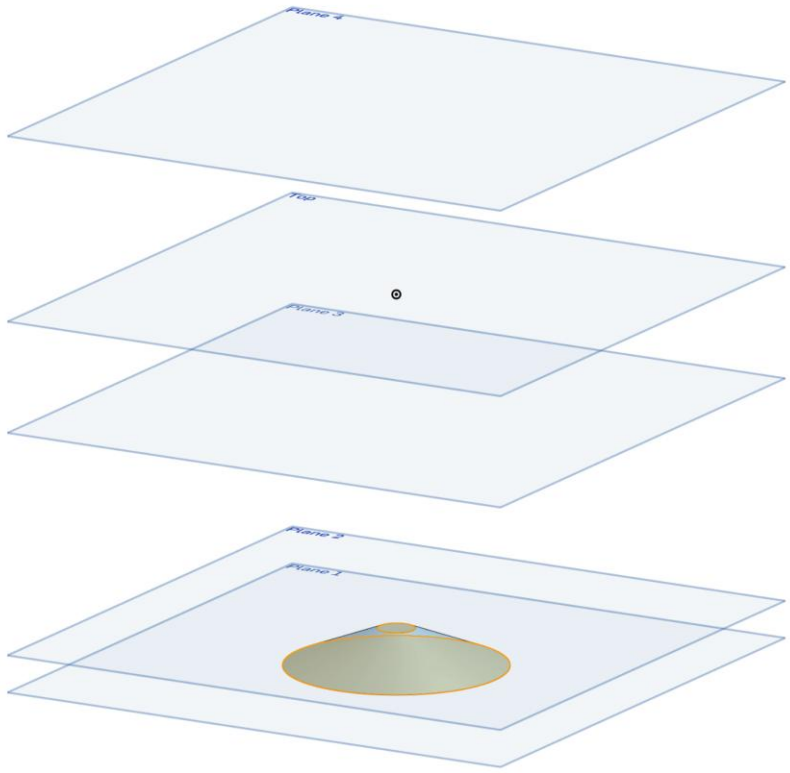


Wine Glass: Loft



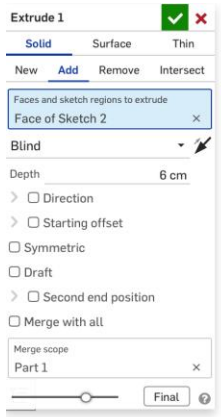
Select sketch 1 and 2, and then click on the **loft feature**. The loft feature will connect the two circles to create a 3D shape. Click the green check mark / Enter to finish.

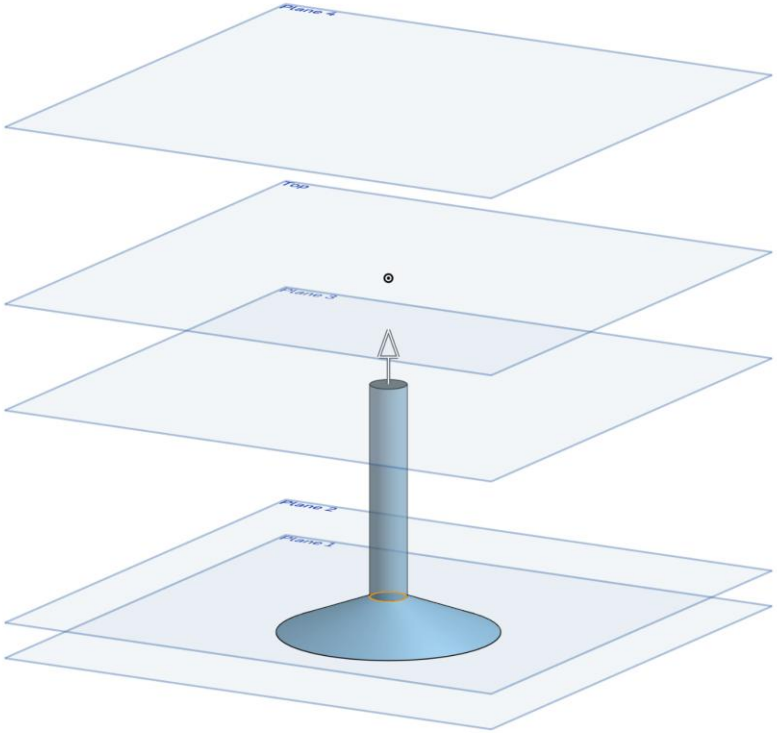




Wine Glass: Extrude

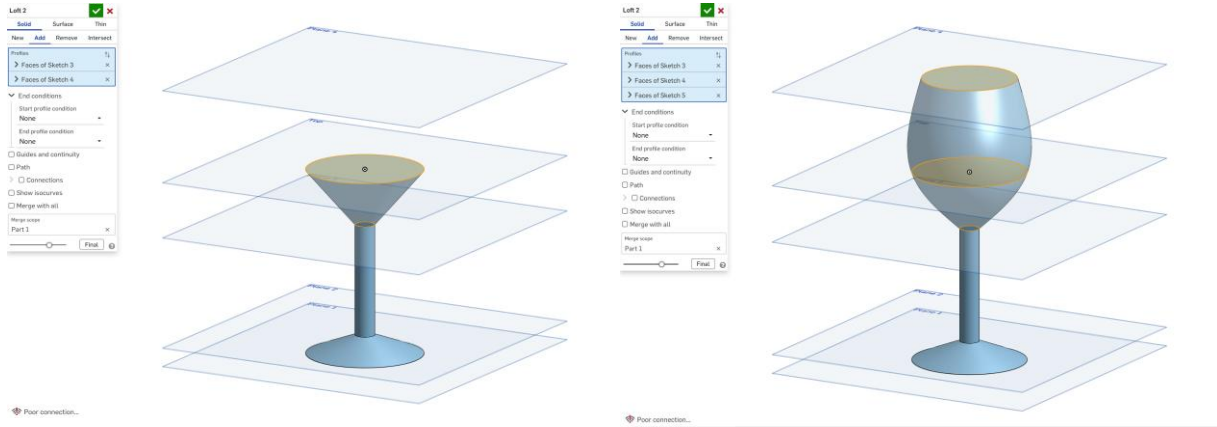
Click **extrude** and select the top (smaller) **face** of the **loft.** Change the **depth** to 6 cm so it reaches plane 3.





Wine Glass: More Loft

Click the **loft feature**, then select sketch 3 and 4 and hit the **enter key**. To edit the **loft**, you can **double click** it on the left side and add sketch 5.

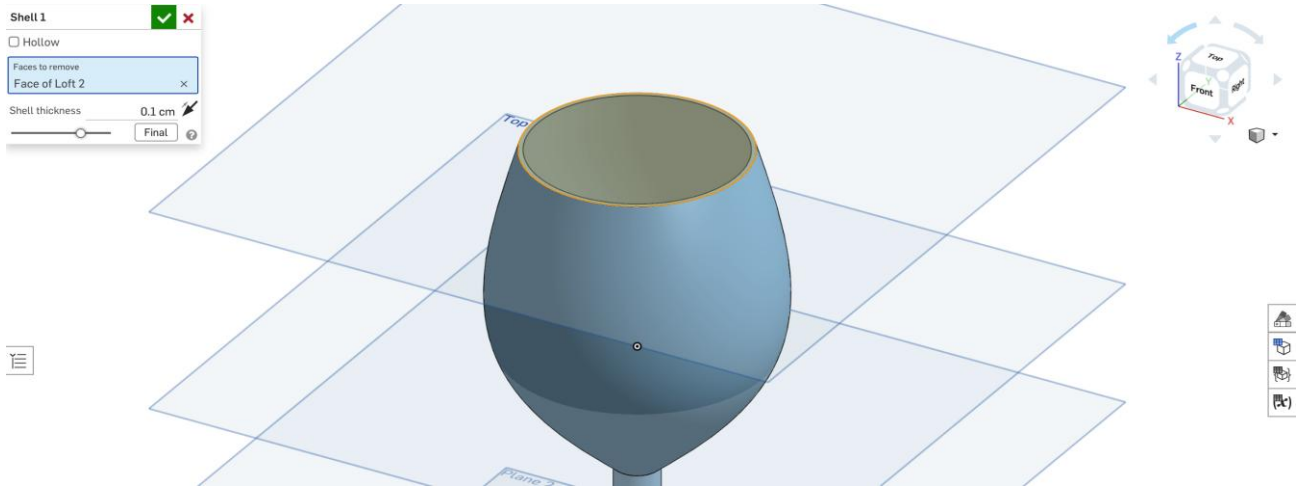


Note: You should click edit to make the loft one part. If you originally add the 3 sketches to make the loft, it is going to do the loft as a separate part.



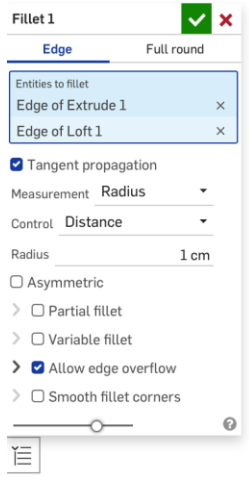
Wine Glass: Shell

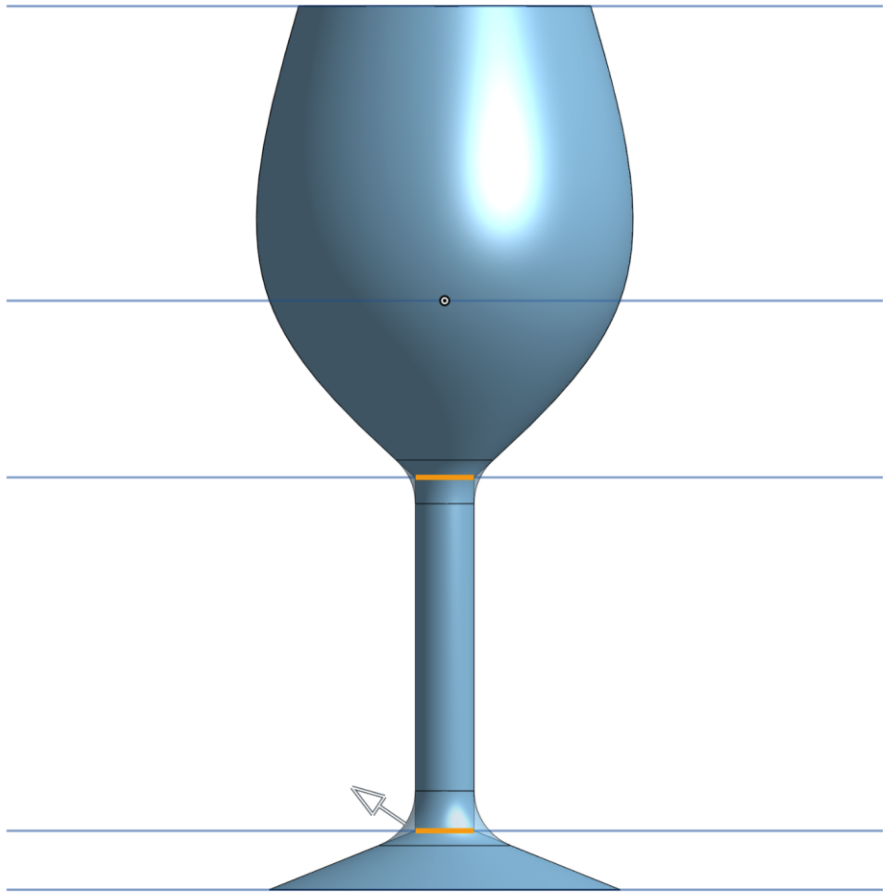
Click the **shell feature**, select the **top of the wine glass**, and set the **shell thickness** to 0.1 cm.



Wine Glass: Fillet

Click the **fillet feature**, select the **bottom and top circles of the handle**, and set the **radius** to 1 cm.





Q&A Session

Attendance

Join us for the upcoming weekly sessions this Spring 2025!
(Jan 6 - Apr 4, 2025)

Monday: C5 1PM
Tuesdays: C7 1PM / **C11** 3PM
Wednesdays: C5 12:30 PM / **C9** 1:30 PM (Varies)
C1 2:30PM / **C2** 3:30PM / **C13** 5:30PM
Thursdays: C6 1PM / **C8** 3PM
Friday: C9 2PM (Varies) / **C10** 3PM/ **C11** 4PM

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