KISS INSTITUTE FOR PRACTICAL ROBOTICS

CAD Basics for 3D Printing

Onshape

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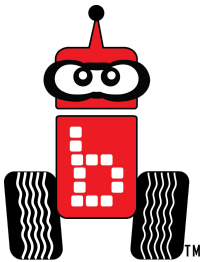


What is CAD?

- CAD stands for “computer-aided design”
- CAD programs are where you design and create your 3D models to be printed by your 3D printer
- There are many CAD programs available
  - Tinkercad, Onshape, FreeCAD, SketchUp, LibreCAD

This tutorial will give an overview of using Onshape tools

Setting up your OnShape Account

- Go to Onshapeservices.org
- Click on “Create a Student Account” in the top right
- Create an account
  - Submit all prompted information and activate account through email
- Login-in to your account
- Click “Create” in the upper-left corner, start a document, name it, and begin designing

You do not need to download anything onto the computer, Onshape will open in the browser

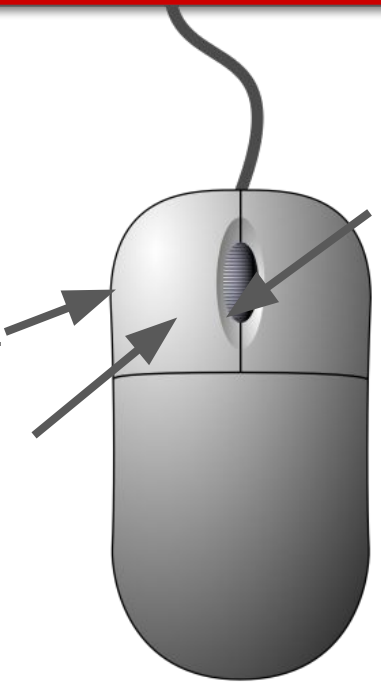
Navigating the Design Space

- Before starting your first design, become familiar with navigating the design space

Left button: Used to interact with the design (e.g. placing shapes, dragging, etc.)

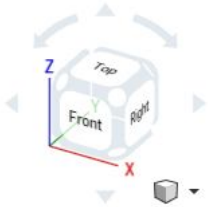
Scroll button: Used to move the whole screen left, right, up, or down when pressed. Zoom in and out by simply scrolling

To reset, right click in the white space and select “Isometric”



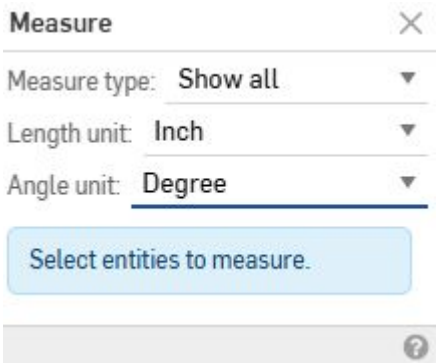
Right button: Used to change the angle of perspective

or toggle the perspective cube



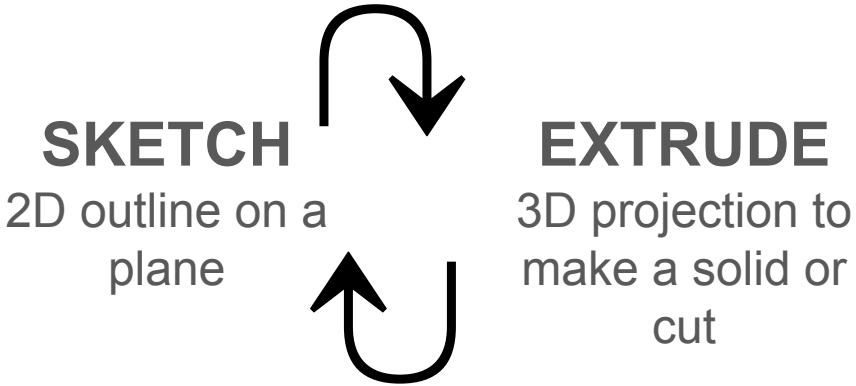
Changing Units

- By default, the units of measurement are usually in millimeters
- To change the units, select the measuring tape icon in the lower right corner
  - A menu will pop up where you may edit measurement settings



Sketching and Extruding

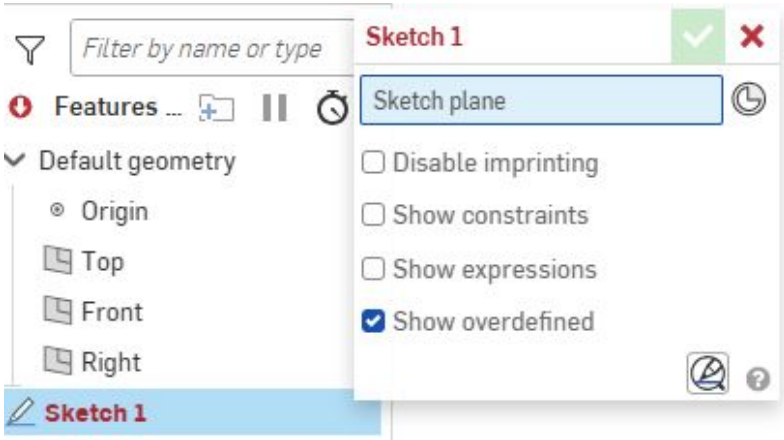
- There are many tools and features which can be confusing, but designing typically follows the two-step same cycle…



Sketching - Starting a Sketch

- Click the “Sketch” button in the top left

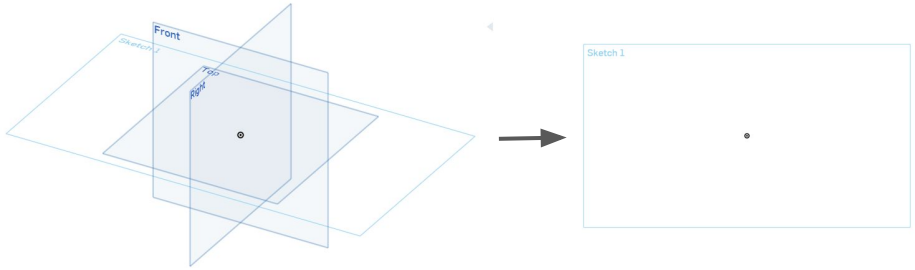
- A new sketch will be created in the left menu and a sketch menu will be opened
  - Select a sketch plane to begin sketching
  - Notice the top toolbar has changed to have sketching tools
  - Save or delete sketches with the checkmark or “X”

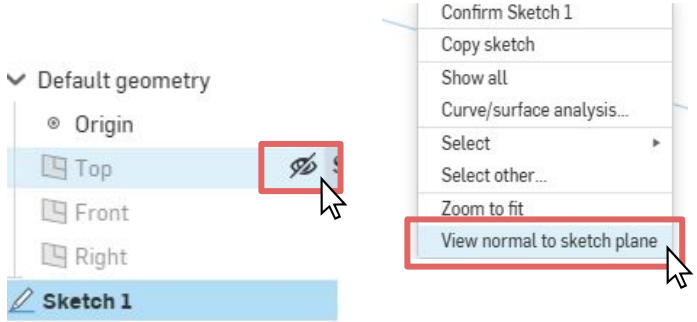


Sketching - 2D View

- We can declutter the screen to better sketch in 2D
  - Under “Default geometry” in the left menu, click the eye icons to hide the planes
  - Right-click in the design space and select “View normal to sketch plane” from the pop-up menu

Now you should be looking at a top-down view of your sketch space





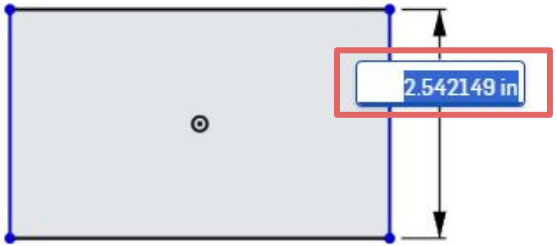
Sketching - Line and Shape Tools

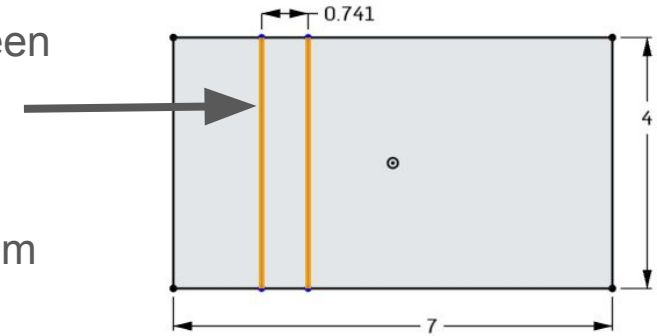
- When a sketch is open, the top toolbar houses various line and shape options
  - Create lines, rectangles, circles, arcs, inscribed polygons, splines, points, and text
  - Select the downwards arrow next to icons to open a dropdown menu with additional options

- Selected tools will highlight blue indicating that they are active
  - To quit a tool, deselect by clicking the icon again or hitting escape on your keyboard

Sketching - Setting Dimensions

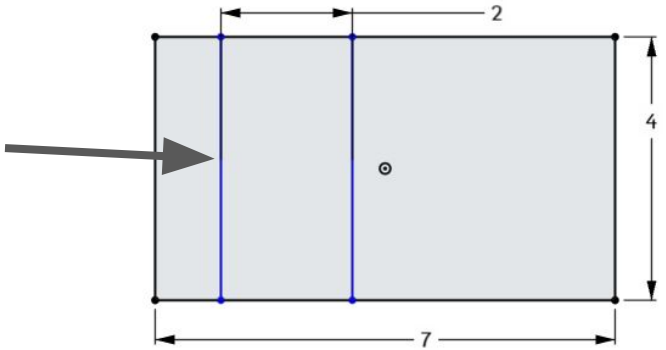
- Change sketch dimensions by selecting the dimension tool on the right side of the toolbar
  - Click on the an edge, line, or shape and place the dimension
  - Type the new dimension into the insert box and hit enter
- Set dimensions between lines and shapes by selecting both entities
- Double-click on set dimensions to edit them



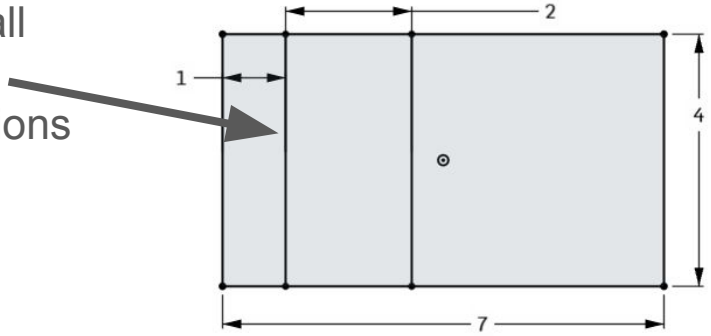


Sketching - Setting Dimensions Continued

- Unconstrained entities will be blue
  - These items can still be manipulated because not all dimensions have been set

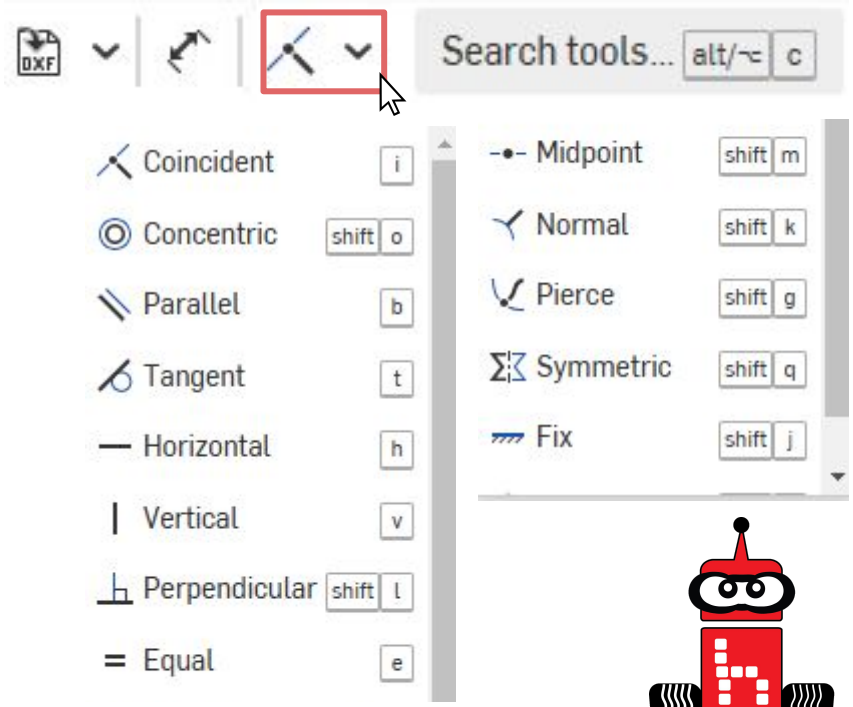


- Constrained entities are black
  - These entities cannot be freely moved because all dimensions are set
  - To edit, change dimensions themselves



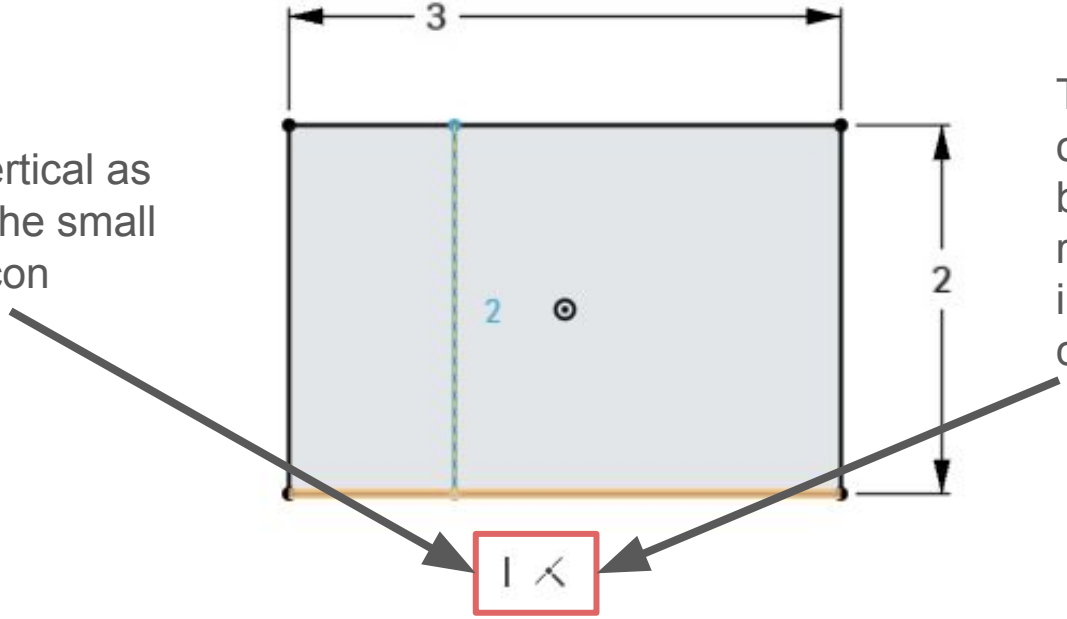
Sketching - Symbols

- The last button and dropdown menu in the top toolbar contains many important tools
- These options define an entity's relationship with other lines and entities
  - For example, the coincident line tells you that two entities share a location (they are touching)
- These symbols automatically show up in a sketch if they are true



Sketching - Symbols Example

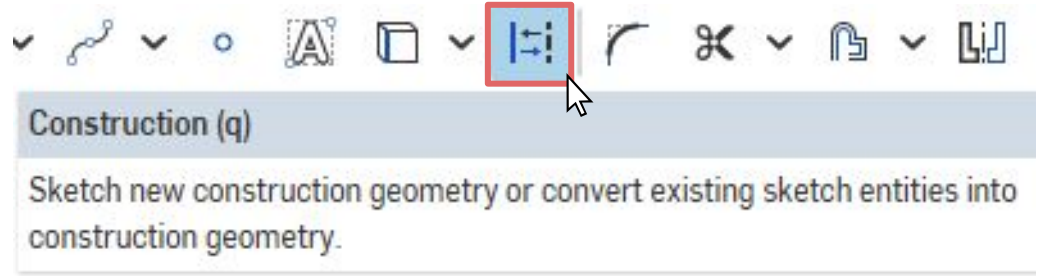
This line is vertical as indicated by the small vertical line icon

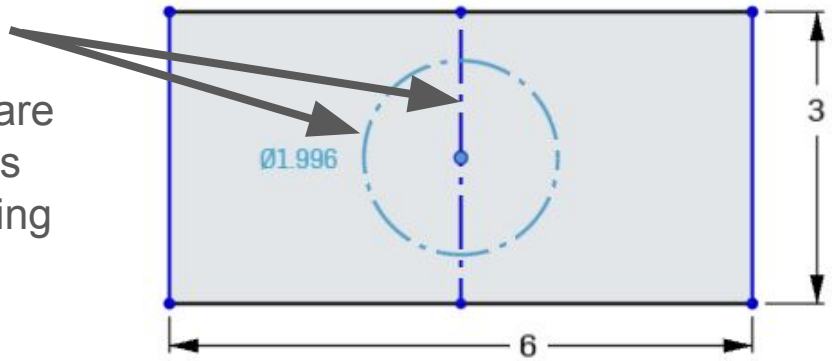


The line is also coincident with the bottom edge of the rectangle as indicated by the coincident icon

Sketching - Construction Tool

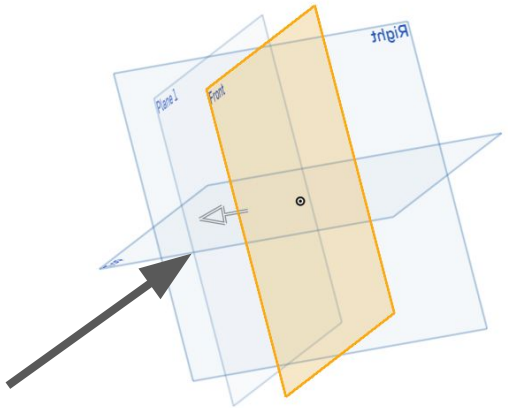
- Selecting the construction tool in the middle of the top toolbar while sketching creates lines that will be ignored during extrusion
  - Construction lines are dashed
  - Construction sketches are useful for things such as finding midpoints, aligning elements, creating intersections, etc…



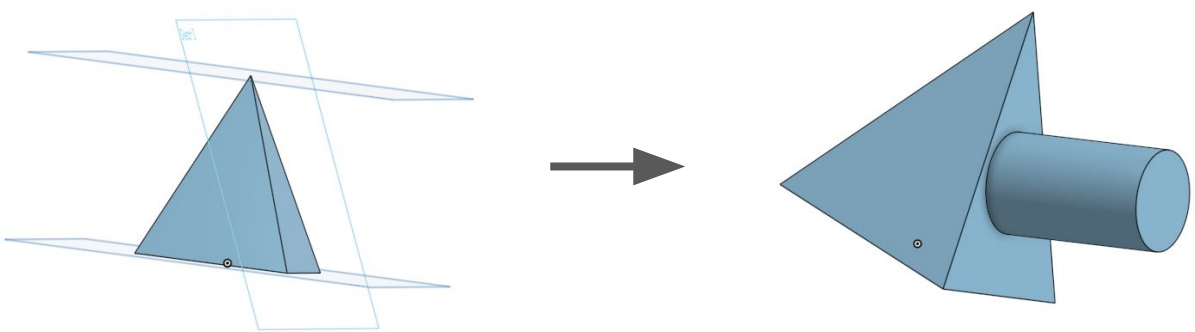


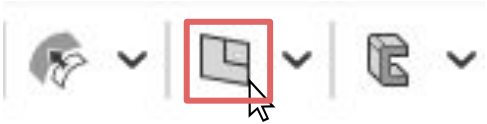
Sketching - Using Different Planes

- Use the plane tool to create new planes
  - This can be useful to create sketches on different planes other than the standard front, top, and right planes
- First, return to the 3D toolbar
  - Exit any sketch currently open
- Now, select the plane tool on the right side of the toolbar
  - Select one of the standard planes to create new plane parallel to the standard one
  - Offset the new plane by the desired amount and save



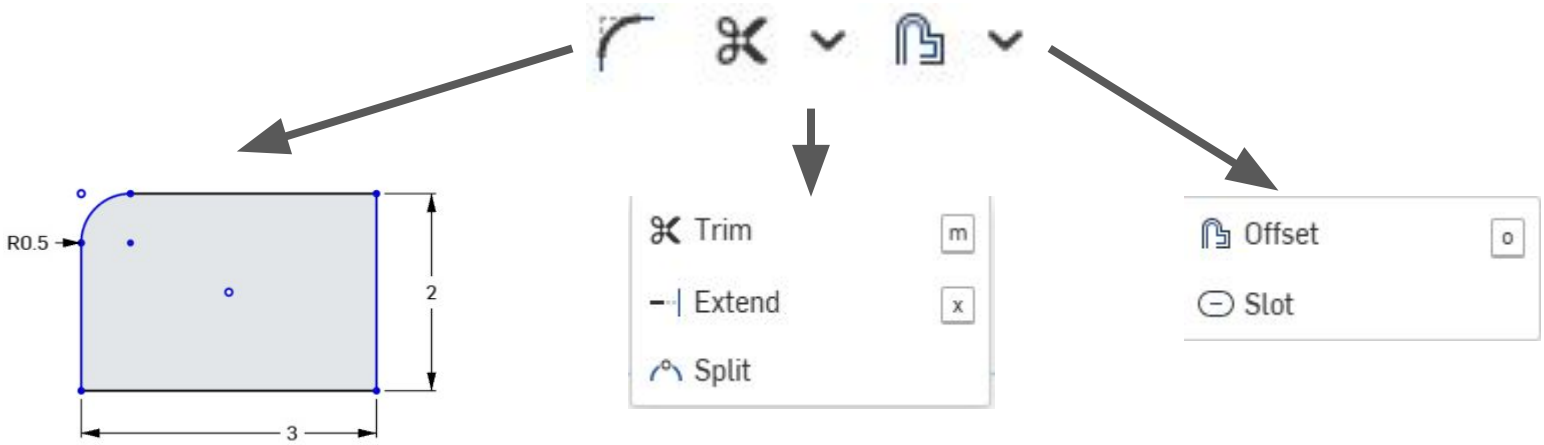
Sketching - Using Different Planes





- You may sketch directly on the surface of an extruded object
  - Simply create a new sketch and select the desired surface to sketch on instead of selecting a plane
- Example:
  - Square pyramid > “Sketch” > Select side of pyramid > Sketch circle > Extrude circle

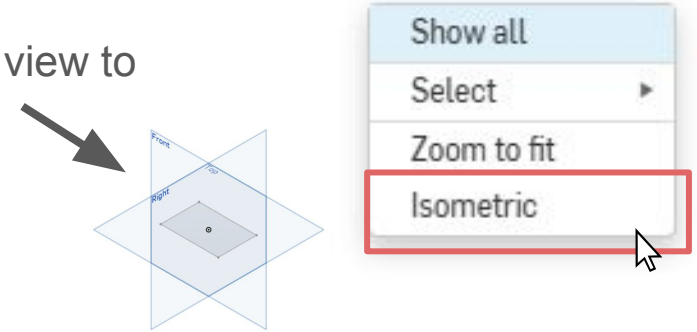
Sketching - Additional Tools

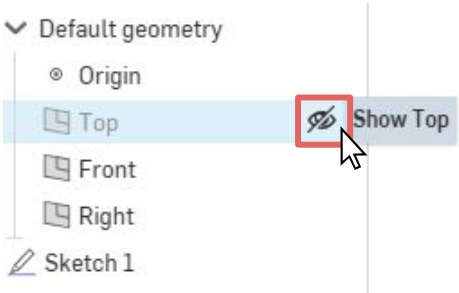


- Create curved corners
- Delete, extend, or split lines
- Offset lines or create slots

Extrusion - Return to 3D

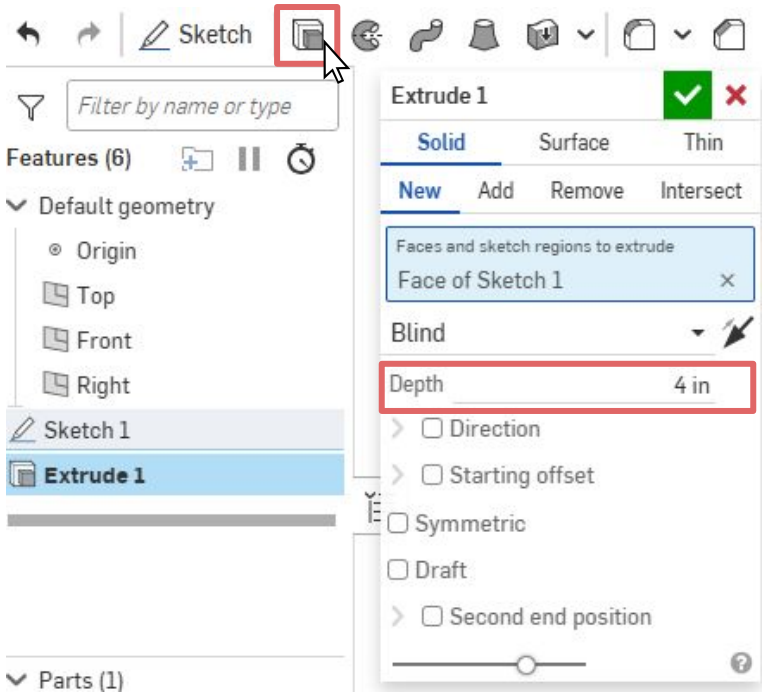
- Once a sketch is saved, you may return to a 3D view to extrude the sketch
  - Right click and select “Isometric”
  - Show the planes again by clicking on the eye icons of each plane under “Default geometry”
- Notice the top toolbar has changed to have 3D options





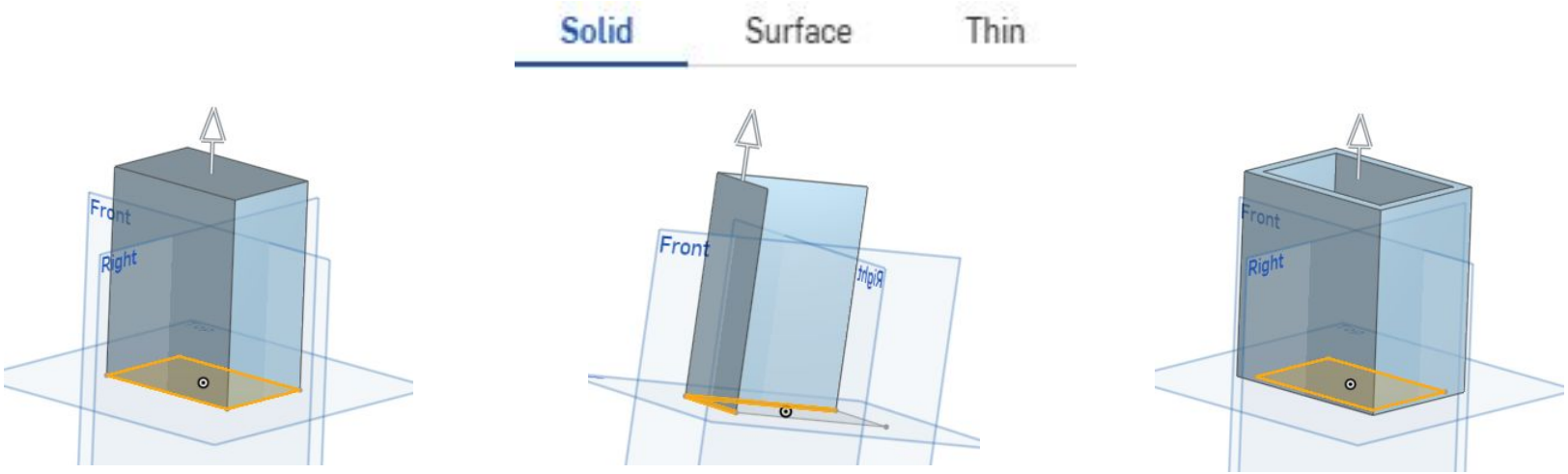
Extrusion - Starting an Extrusion

- Selecting the extrude button from the top toolbar creates a new extrusion in the left menu and opens an extrusion menu
- Select the sketch you wish to extrude



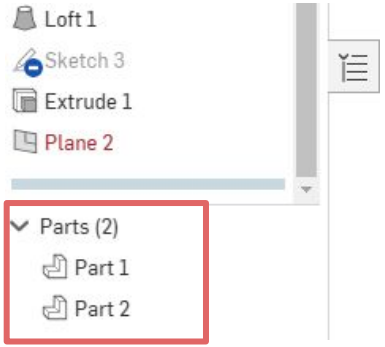
- For basic extrusions, simply set the depth to the desired amount

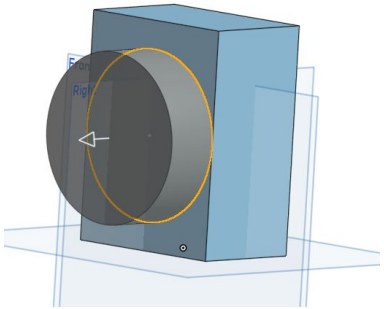
Extrusion - Extrusion Options

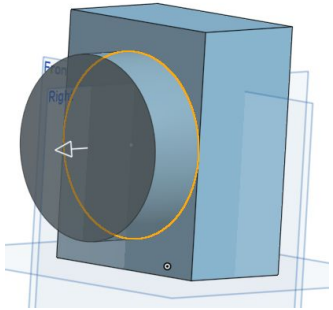


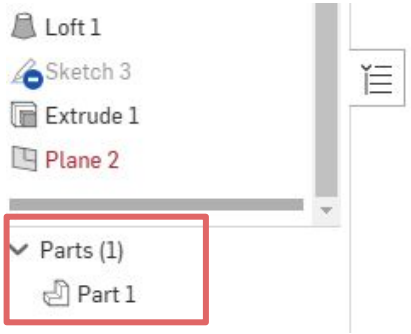
- **Solid:** Extrude the entire selected sketch as a simple solid
- **Surface:** Extrude only selected edges
- **Thin:** Extrude edges as thicker walls

Extrusion - Extrusion Options



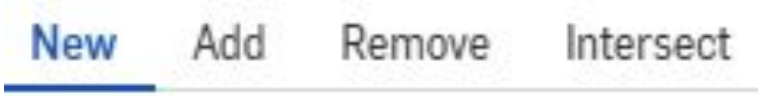




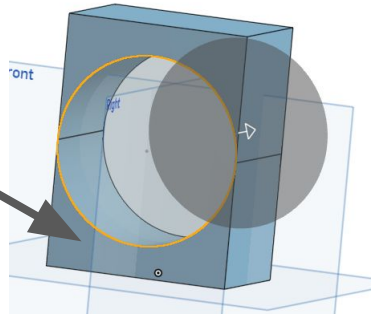


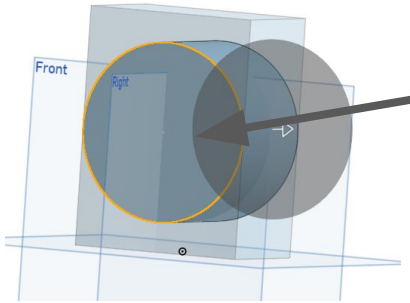
- **New:** Creates a new part, separate from the previous part, which can be accessed in the left menu
- **Add:** Creates a new piece/detail on a pre-existing part without creating a new one

Extrusion - Extrusion Options



Example: Remove extrusion creates a prism with a circular hole



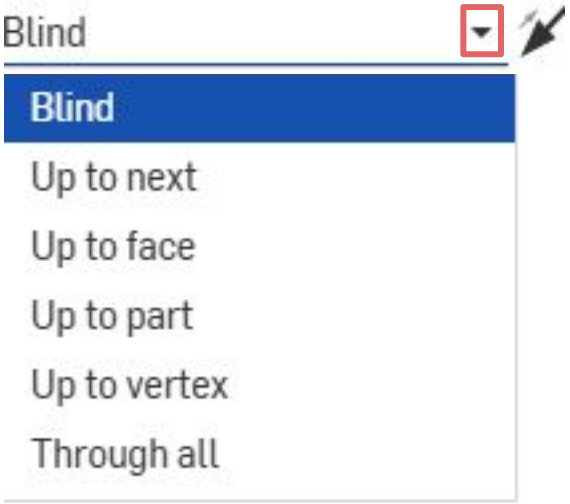


Example: Intersect extrusion creates a horizontal cylinder

- **Remove:** Removes all material that the sketch is extruded through
- **Intersect:** Leaves behind only where the 3D object and extruded sketch intersect

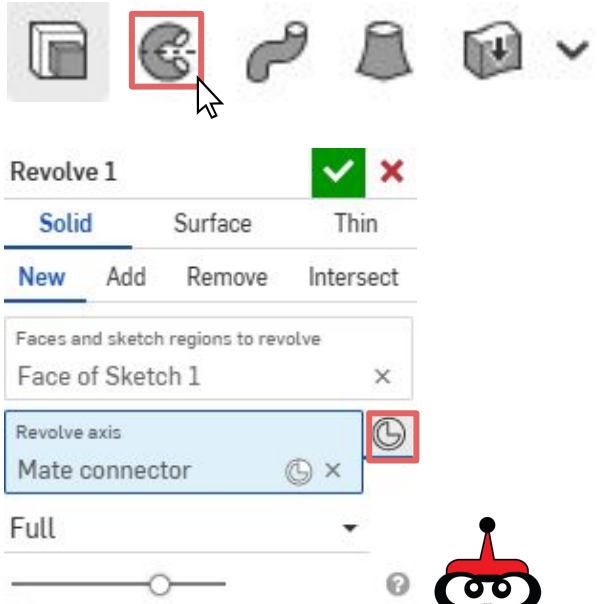
Extrusion - Extrusion Options

- On the extrusion pop-up menu, there will also be different extrusion end types to select from
- Access by clicking on the small arrow to open the drop-down menu
  - **Blind:** Extrude for a set distance in a selected direction
  - **Up to next:** Extrude to the next object encountered
  - **Up to face:** Extrude up to a face or plane
  - **Up to part:** Extrude to a specific part
  - **Up to vertex:** Extrude until in line with a selected vertex
  - **Through all:** Extrude entirely through a part

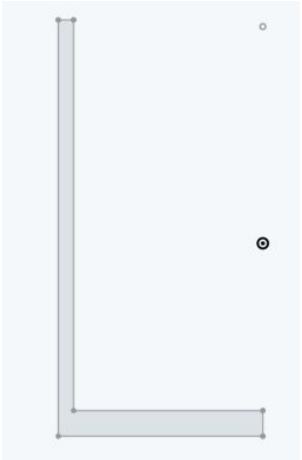


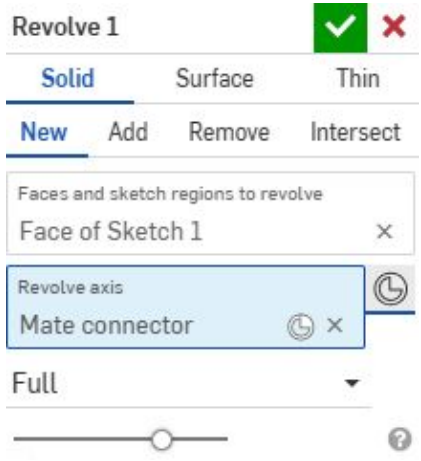
Additional Tools - Revolve

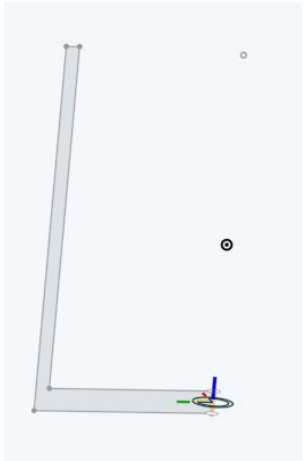
- Revolve sketches to create 3D objects
- First, create a sketch to be revolved
- Next, select the revolve button
- Once the revolve menu opens…
  - Select the sketch
  - Select “Mate connector”
  - Select the line or edge to revolve around
  - Save

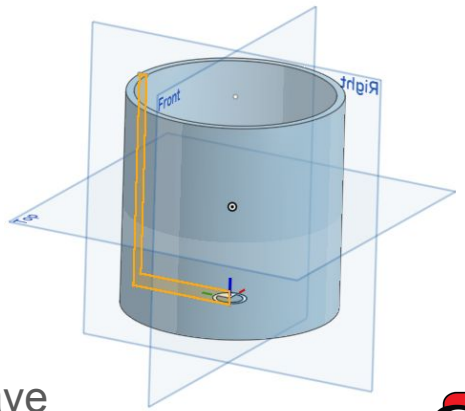


Additional Tools - Revolve Example



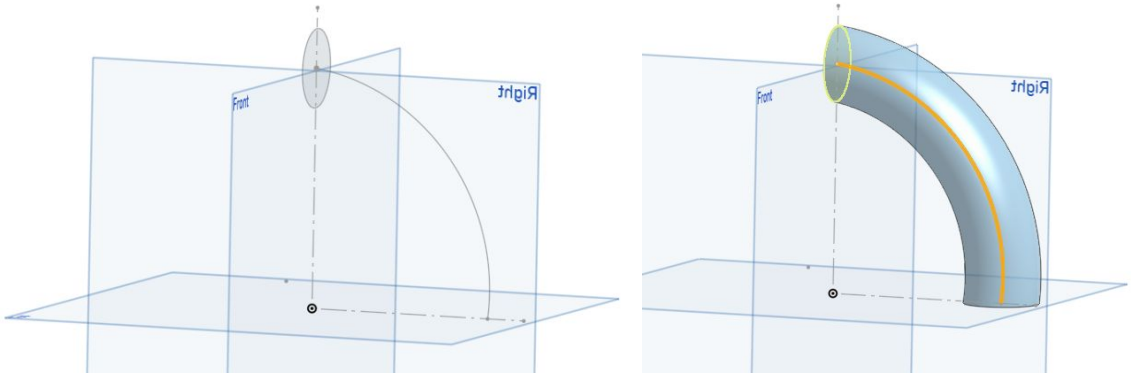


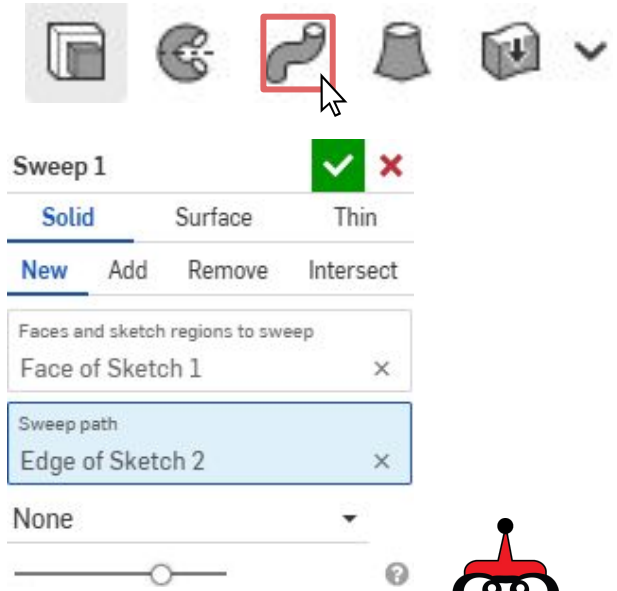




Sketch ⇨ Revolve ⇨ Mate connector ⇨ Select edge ⇨ Save

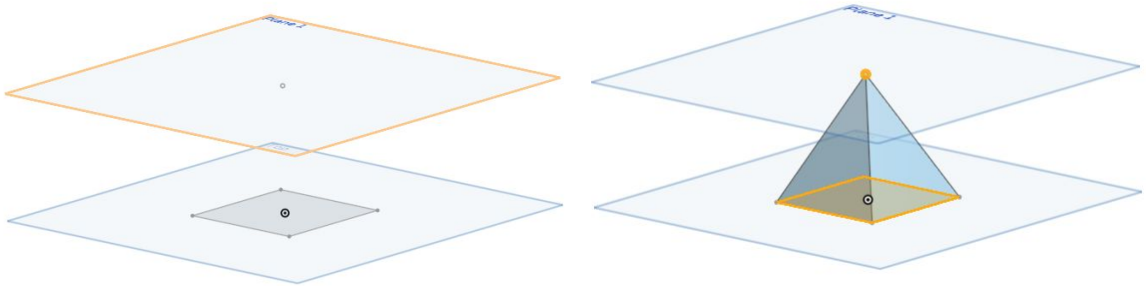
Additional Tools - Sweep

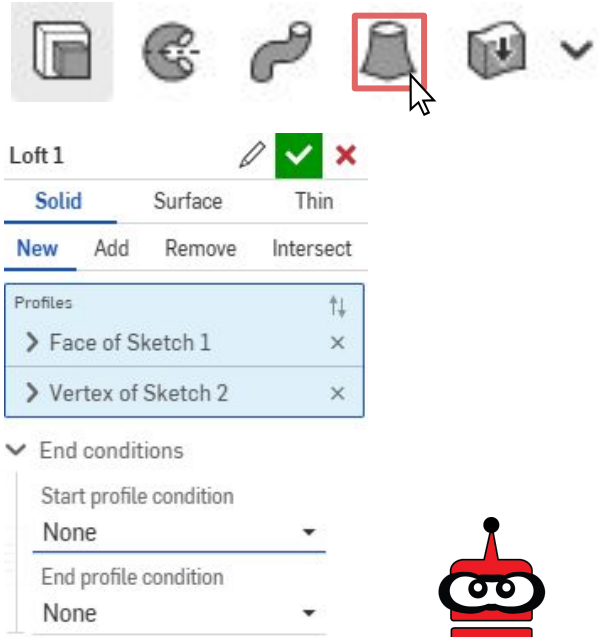




- To use the sweep tool, create two sketches
  - One sketch is the shape that will be swept along the other sketch, the sweep path
- Activate the sweep tool
  - Select the face sketch and the sweep path
  - Save

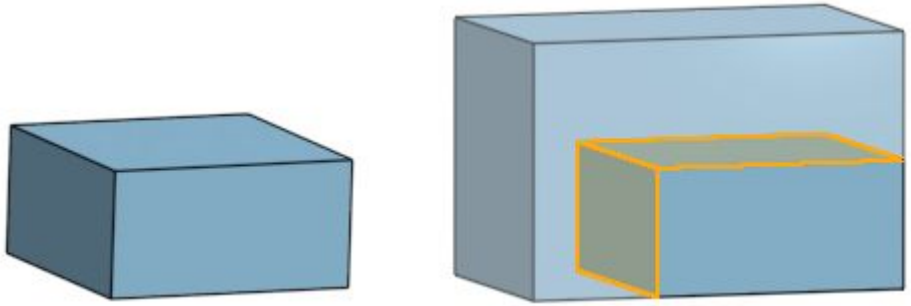
Additional Tools - Loft

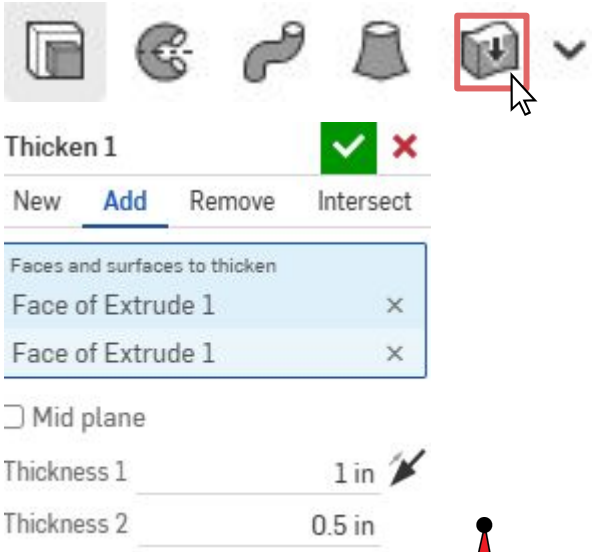




- Use the loft tool to create smooth surfaces extending from one sketch to another
  - Create two sketches
  - Activate the loft tool
  - Select the sketches in the order they are to be lofted in (1st sketch is lofted to the 2nd)
  - Save

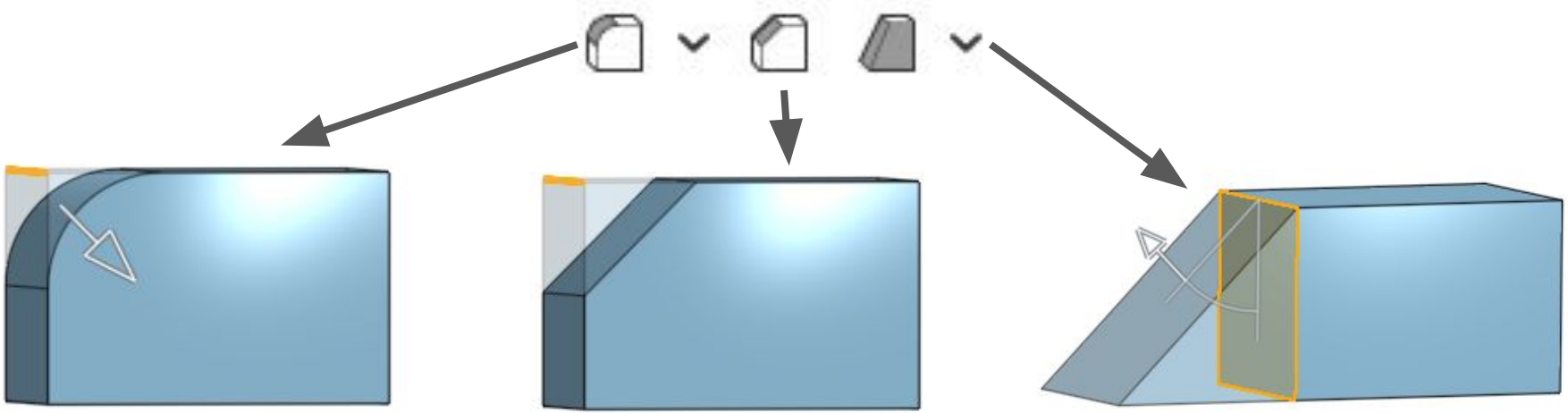
Additional Tools - Thicken





- Thicken selected faces of an object
  - Activate the thicken tool
  - Select the faces you wish to thicken
  - Set the thickness amount in the thicken menu
  - Save

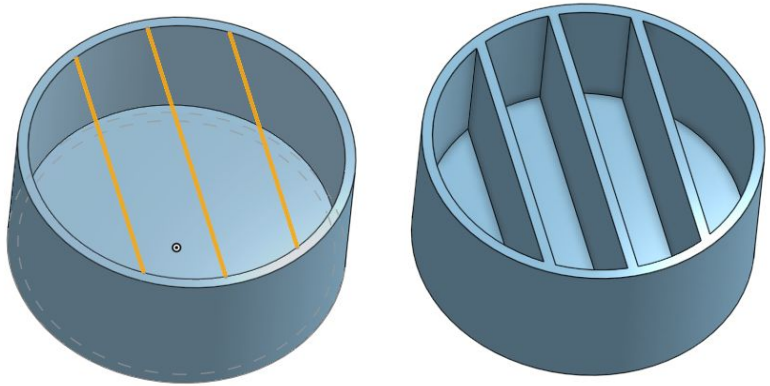
Additional Tools - Corners and Edges

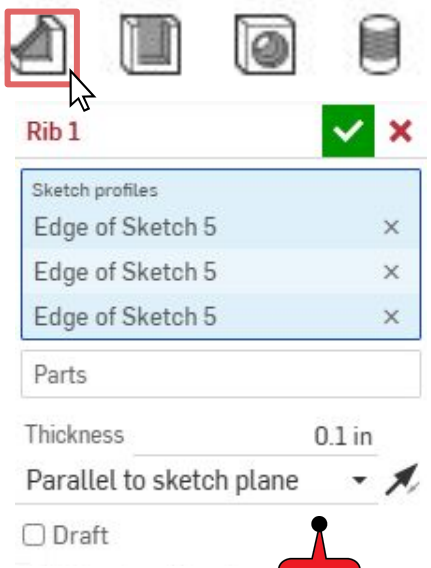


- **Fillet:** Create rounded edges on 3D objects
- **Chamfer:** Create a straight sloping edge on 3D objects
- **Draft:** Create tapered sides on 3D objects

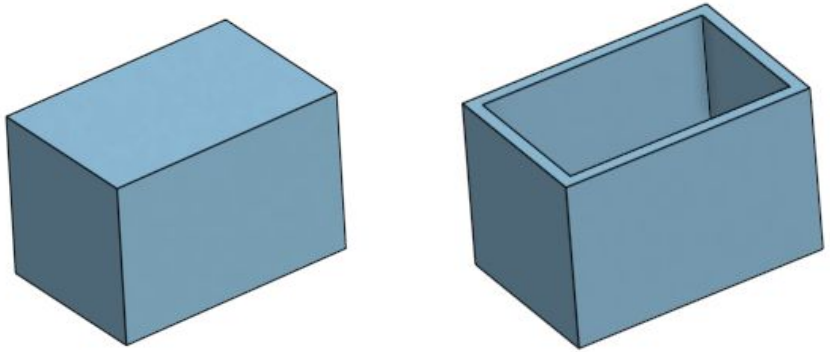
Additional Tools - Rib

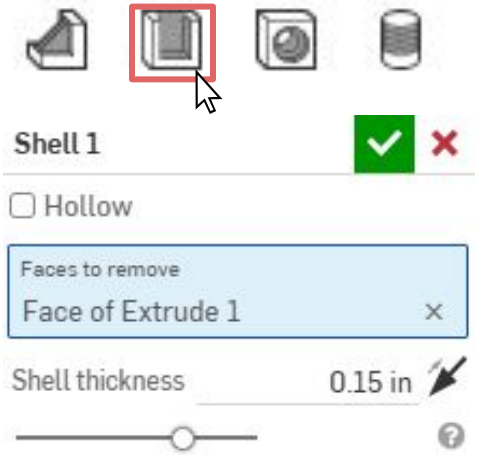
- Create ribbing by using the rib tool
  - Sketch rib lines across the part
  - Activate the rib tool
  - Select the sketch lines
  - Set rib thickness and toggle the rib direction
  - Save
- Note: A shell must first be created to add ribs. The ribs must also have set bounds defined by the shell. If there are no clear endpoints for the ribs, they will fail to generate





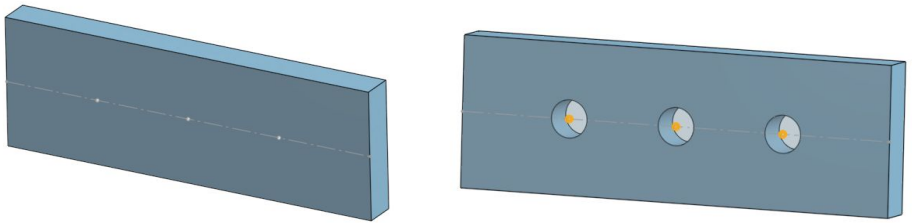
Additional Tools - Shell



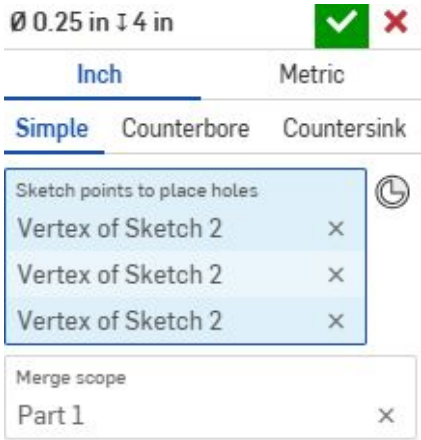


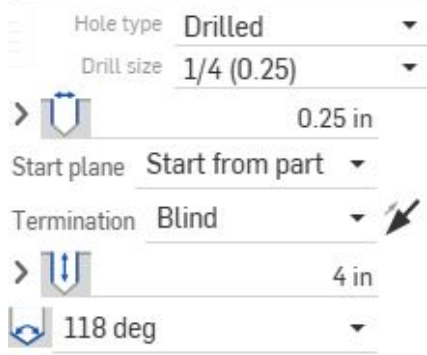
- Create hollow shells from 3D shapes by using the shell tool
  - Activate the shell tool
  - Select the face of the 3D object to be removed
  - Set the wall thickness in the shell menu
  - Save

Additional Tools - Hole



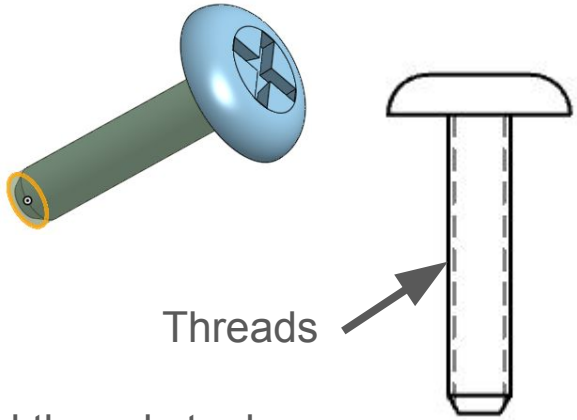
- Create holes in 3D objects using the hole tool
  - Create sketch points where holes are to go on a 3D part
  - Activate the hole tool
  - Select the sketch points
  - Toggle with hole settings (e.g. hole size, hole depth, end type)
  - Save

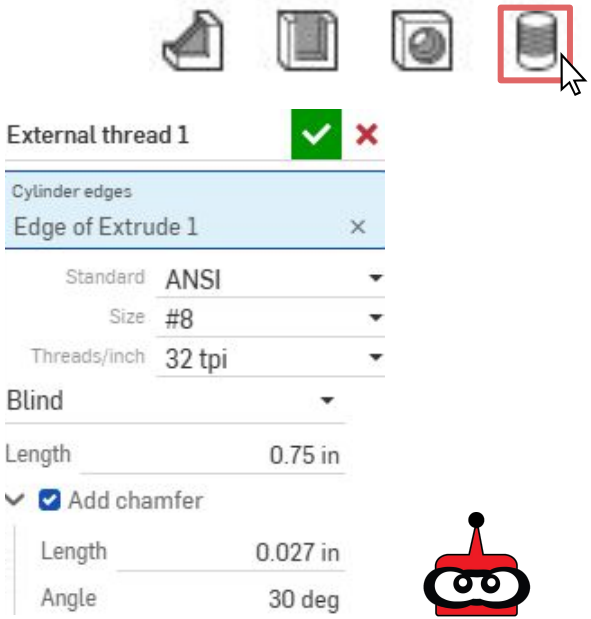




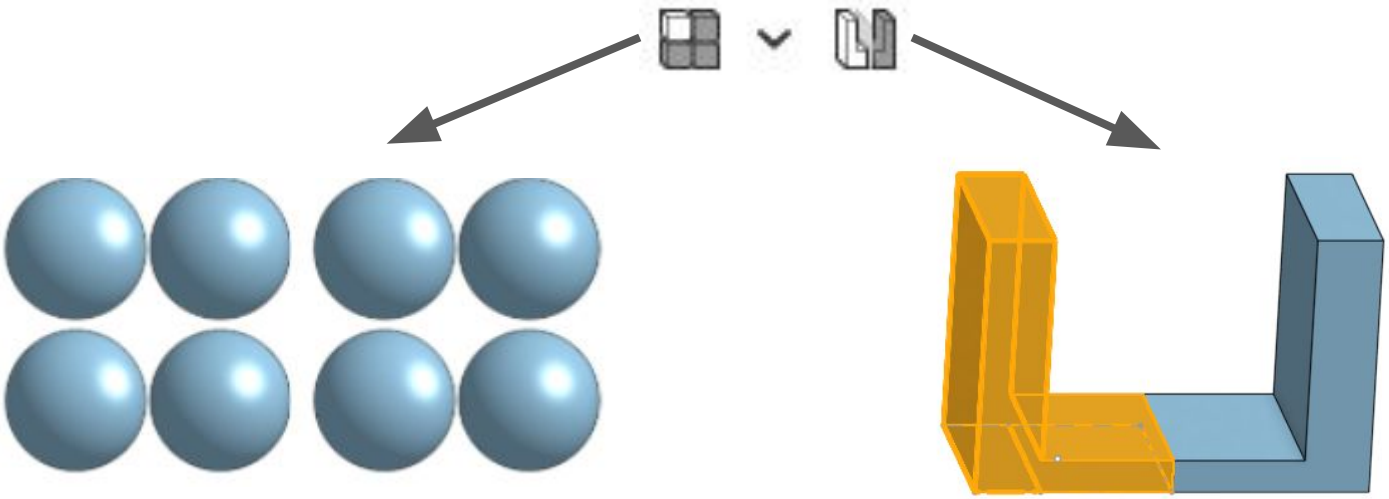
Additional Tools - External Thread

- Create threads using the external thread tool
- First, create a cylinder
- Next, to make the threads…
    - Activate the external threads tool
    - Select an edge of the cylinder
    - Make sure “ANSI” is selected
    - Set the length and save
- Note: External threads will not show up on the 3D part but they will appear in the design drawing



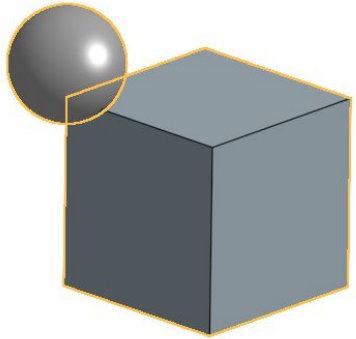


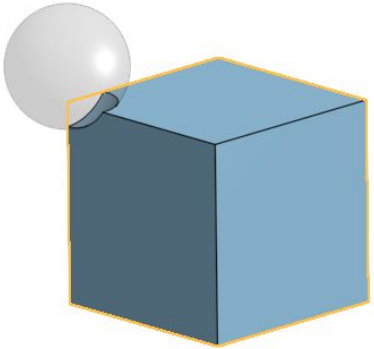
Additional Tools - Patterns and Mirroring

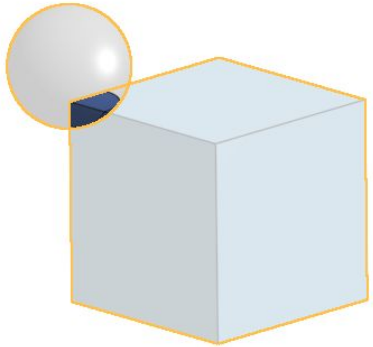


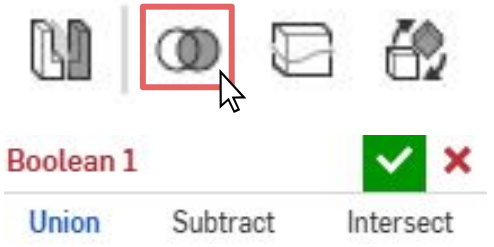
- **Patterns:** Create patterns with 3D objects
- **Mirror:** Mirror 3D objects

Additional Tools - Boolean



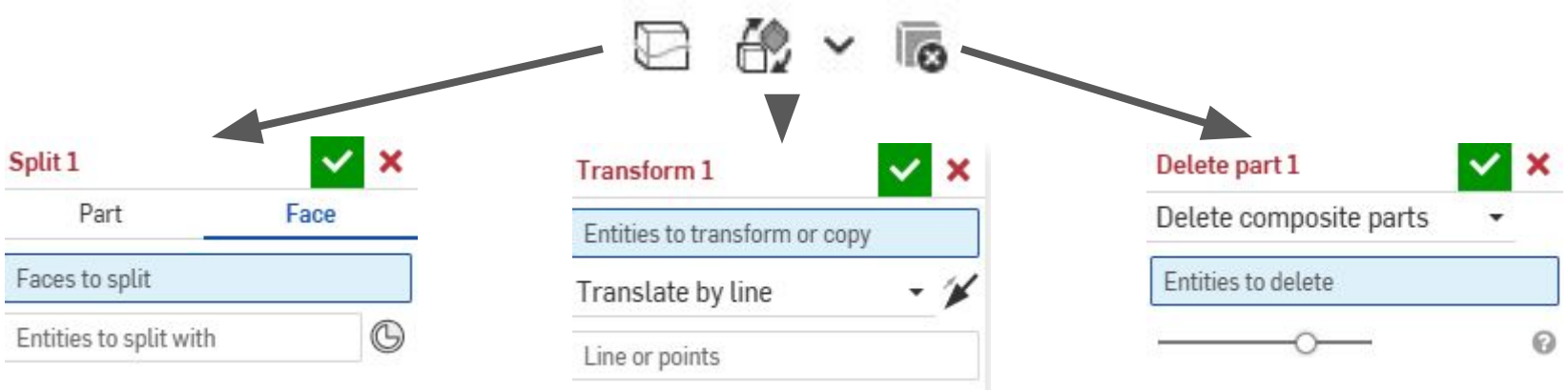






- **Union:** Combine separate parts into a single part
- **Subtract:** Cut away material at the overlap of two parts
- **Intersect:** Cut away all material except for where two parts overlap (opposite of subtract)

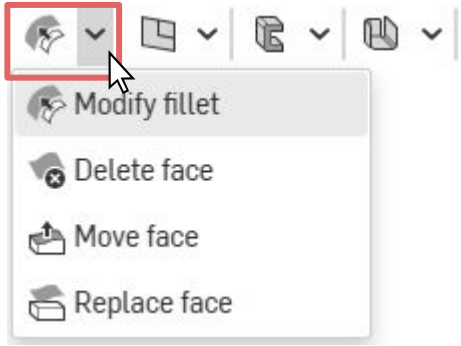
Additional Tools - Manipulating 3D Parts



- **Split:** Split parts into separate pieces
- **Transform:** Move 3D objects around the design space by translating them across edges and lines
- **Delete part:** Delete entire parts

Additional Tools - Editing 3D Parts

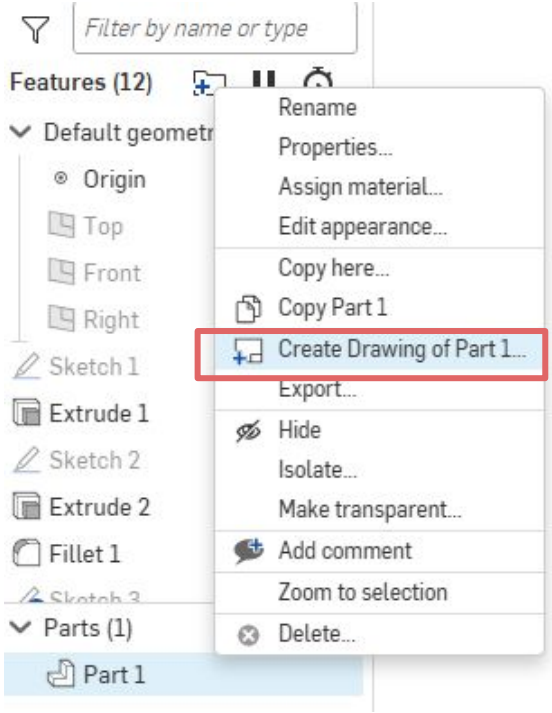
- Easily edit 3D parts
  - **Modify fillet:** Change the radius of pre-existing fillets or delete them
  - **Delete face:** Entirely delete a face. Either leave it open or select“Heal” or “Cap” to re-seal
  - **Move face:** Translate a face in the design space (similar to thicken)
  - **Replace face:** Replace a face with another one



Creating a Drawing

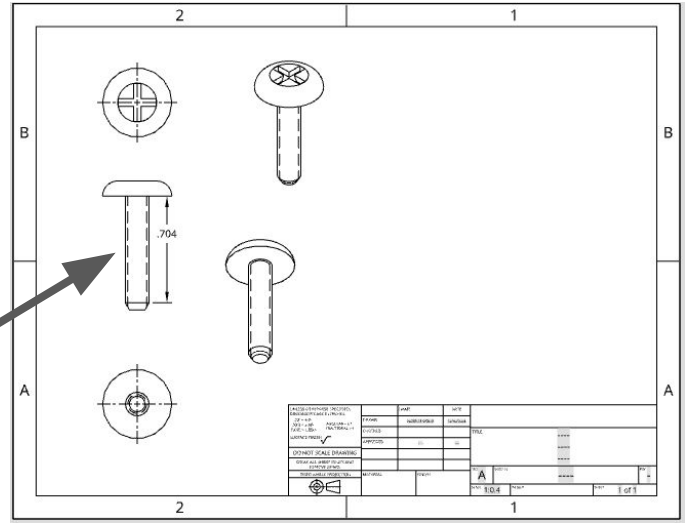
- 3D designs can be made into 2D drawings

- To create a drawing of a part…
  - Right-click on the part in the left menu
  - Select “Create Drawing of Part #”
  - From the window that pops-up, select the first template option and “No views”
  - Hit “OK” in the lower right corner



Creating a Drawing Continued

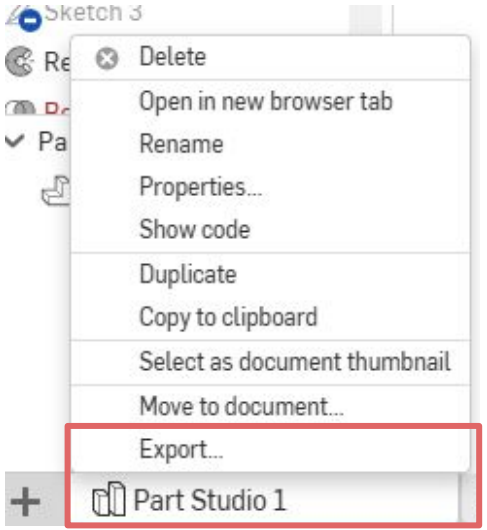
- A blank drawing will open
  - Create different views of the part by dragging and dropping in different directions from the parent view
  - Add dimensions using the dimension tool in the top toolbar
- Note: Parent view is the front on view that the other views are pulled from



- The drawing will save as a tab at the bottom of the document
- To export, right click on the drawing tab, select export, and save as a PDF

Exporting as.STL

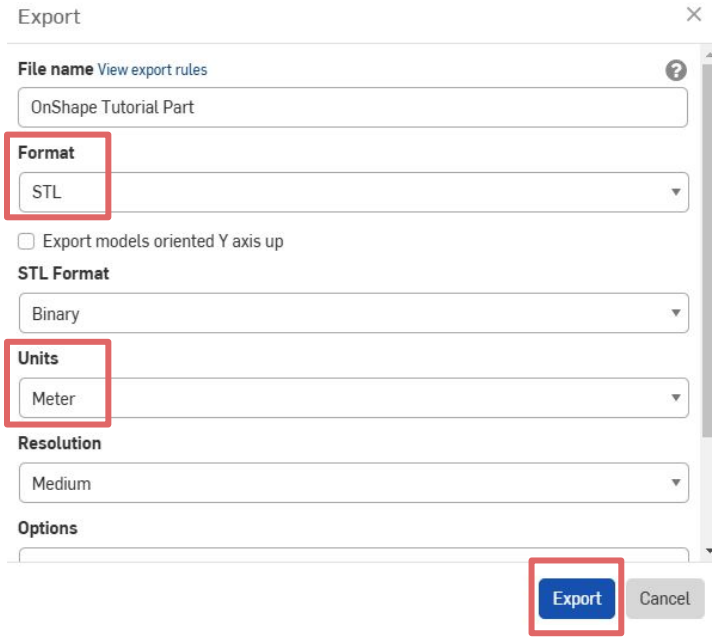
- To export a part for printing…
  - Right-click the part tab in the bottom left corner of the screen
  - From the menu, select “Export…”



Exporting as .STL Continued

- From the export window…
  - Name the file
  - Select “Format” and change it to “STL”
  - Set the exporting units to what your slicing program is in
  - Click “Export”

You have now downloaded your STL file for printing



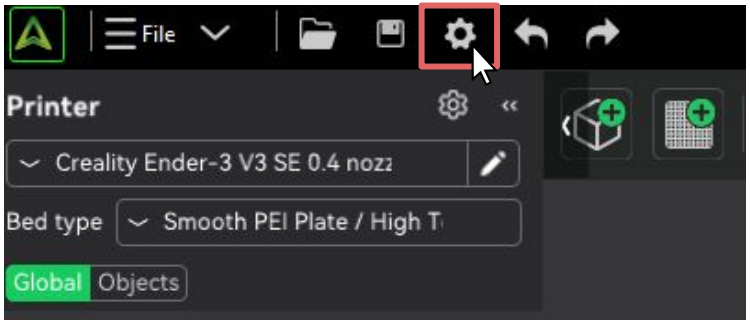
3D Printing - Slicing Software

- Download Creality Print at Creality.com
- Launch the program
  - Select “Creality Ender-3 V3 SE” as your printer from the menu that pops up when you open the program

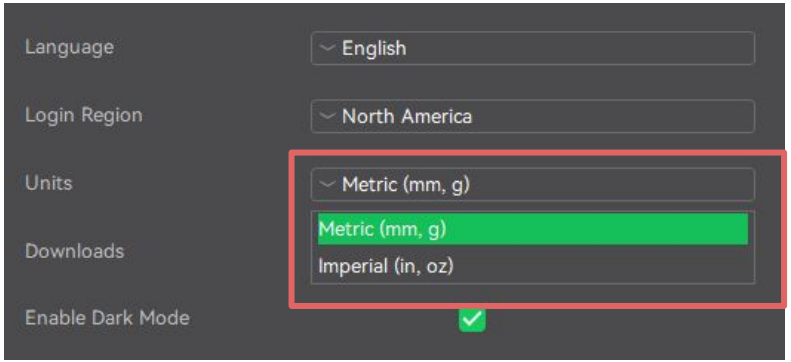
A workspace should generate that matches your printer’s plate

3D Printing - Checking Units

- Check the units in settings
  - Click on the settings icon in the upper left
- Open the “Units” dropdown menu to toggle between millimeters and inches



When exporting your design from CAD software, make sure the exporting units match the units of the slicing program (e.g. if Creality print is set to inches, export designs in inches)

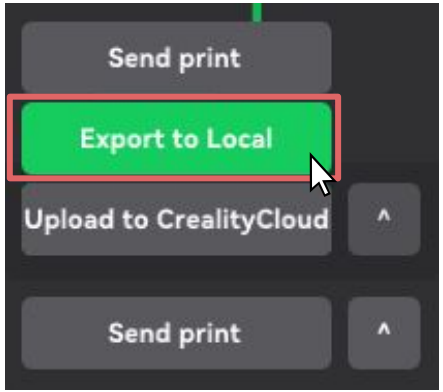


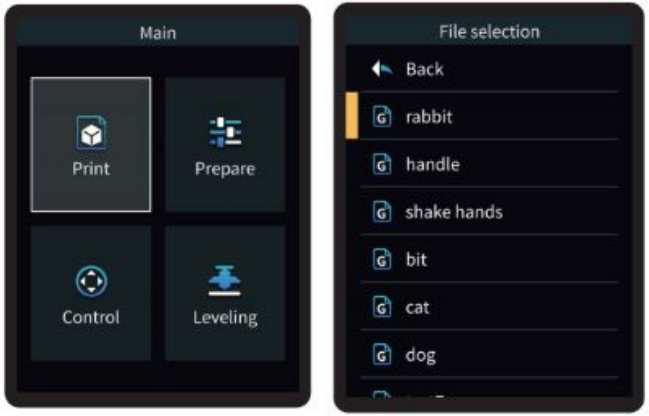
3D Printing - Importing Designs

- Select the add button in the upper left corner
  - Select your print from your files (.STL file)
  - Select “Slice plate”
- Double-check that dimensions are correct
- Insert a SD into the computer
- Select the upwards arrow next to “Send print”

3D Printing - Printing Designs

- Select “Export to Local” from the pop-up menu
  - Save to your SD
- Eject the SD from your computer and insert it into the 3D printer on the left-hand side
- Select “Print” and then choose the file to be printed





Your design is now being printed!