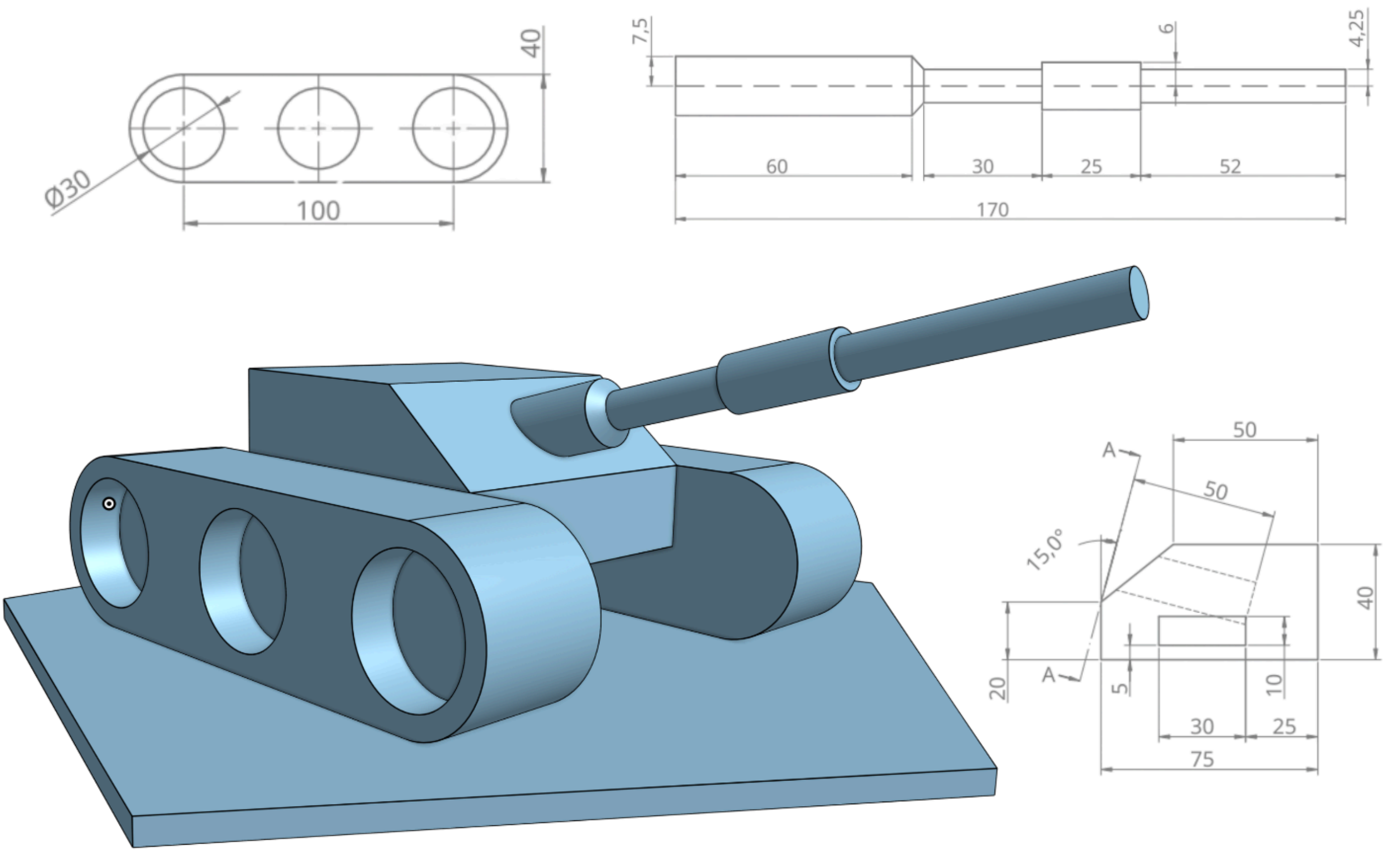
Introduction to Computer-Aided Design (CAD) with Onshape

by Billy Hau



Introduction to Computer-Aided Design (CAD) with Onshape

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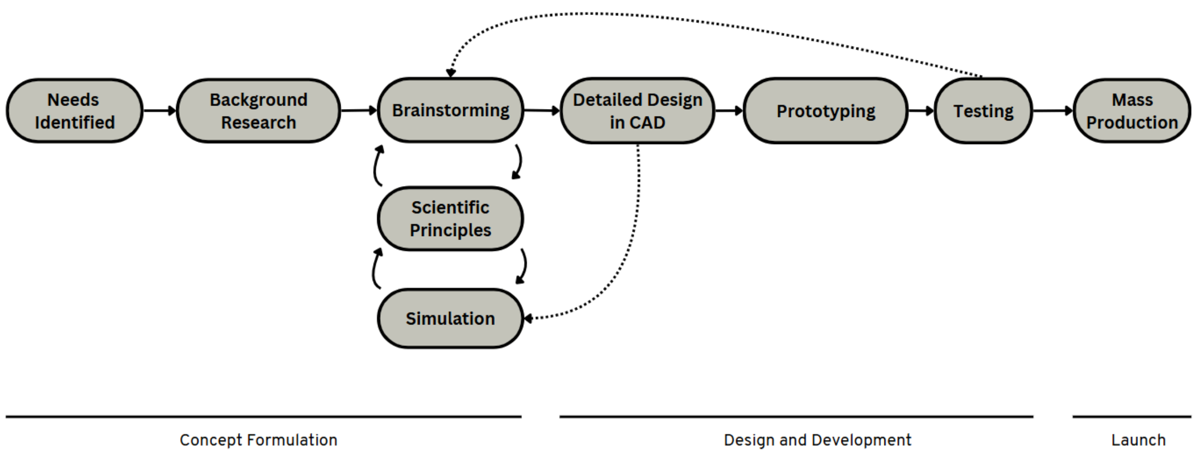
Computer-Aided Design

Computer-aided design (CAD) is the use of computers to help with the mechanical design process. CAD is an important skill to learn because it enables engineers to turn scientific principles into physical designs that can be manufactured to solve our everyday problems.

There is also a term called ECAD, which stands for electronic computer-aided design. ECAD are commonly used in electrical engineering to create printed circuit boards (PCB). CAD and ECAD are complementary skills serving different engineering disciplines.

Applications of CAD

Let's take a step back and look at the overall engineering design process. When a project team designs a product, they first start by identifying the needs and researching the relevant background information. Then they will brainstorm ideas to address the needs, often using science and simulation to validate their ideas. This is the concept formulation phase of the process. Next, engineers take the concept and make detailed designs in CAD. The resulting model can be used to control machines to produce the prototype in real life. The prototype will go through a series of tests and the data collected will be used to improve the design. This cycle repeats until the team accepts a final design, and it is sent for mass production.

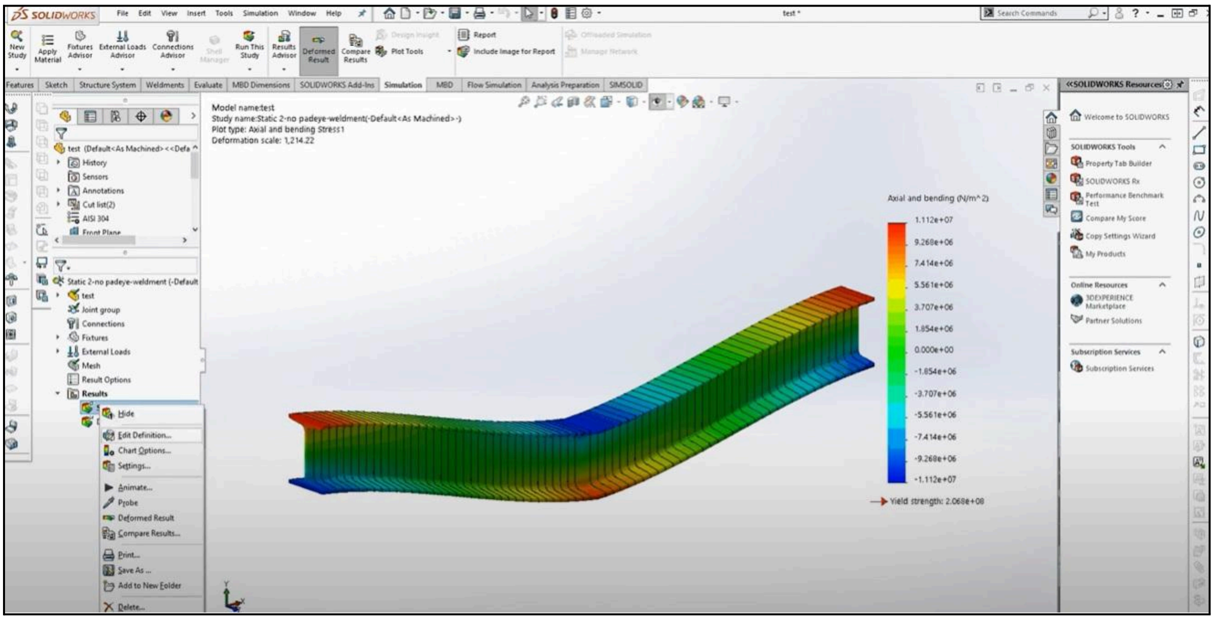


The Engineering Design Process

Aside from manufacturing, CAD models can also provide valuable information. For example, CAD software can calculate physical properties like mass, center of mass, and moment of inertia. These are important properties that can be used in physics simulation to see how the prototype would behave in real life. This is especially important in aerospace engineering, to calculate whether a rocket would be able to take off, for example.

Some advanced CAD software also has built-in thermal, stress, and motion analysis. Thermal analysis, for example, can be used to calculate whether a cooling system is enough to keep an engine from overheating. Stress analysis can be used to calculate how much weight a bridge can carry before collapsing. Motion analysis can be used to simulate a robotic arm's range of motion.

Outside of the realm of engineering, 3D models can also be used in video games, virtual reality experiences and computer animations. As you can see, the possibilities are endless.



PTC Onshape

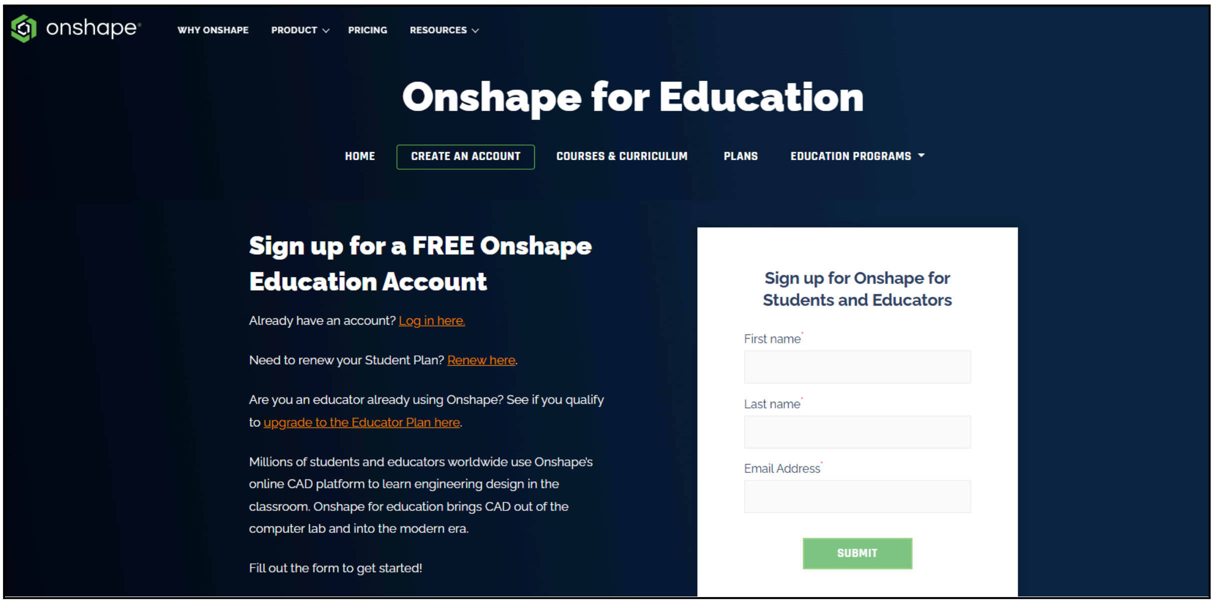
In this class, we will be learning CAD with a software called Onshape. Onshape is a user-friendly and free-to-use software made by PTC. It can be used online without any installation in school or at home. You can even use it on your phone if you want to! All you need is a stable internet connection and a supported web browser like Google Chrome.

There are actually many CAD software packages on the market. Onshape is just one of them that's more suitable for education. It is important to keep an open mind as you may encounter other CAD software in the future during your studies or career. As each has their pros and cons, you will need to choose the best option based on your use case. The fundamental modeling concepts that we will learn in this class will be the same. Therefore, learning a new software will not be difficult.

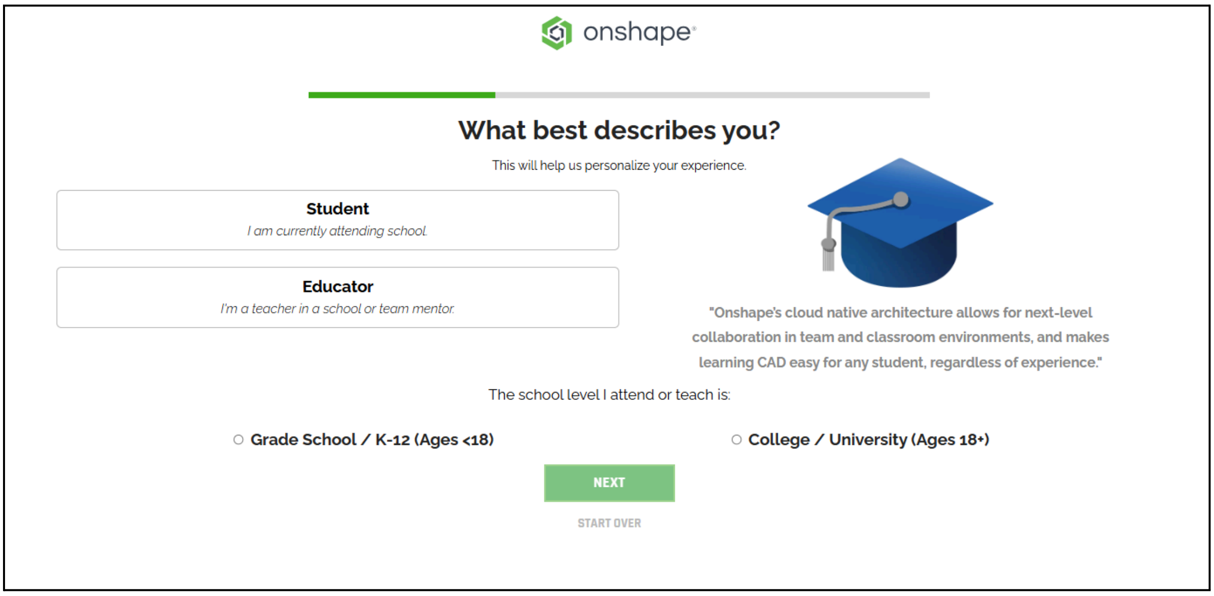
A list of popular CAD and related software can be found in the appendix.

Account Registration

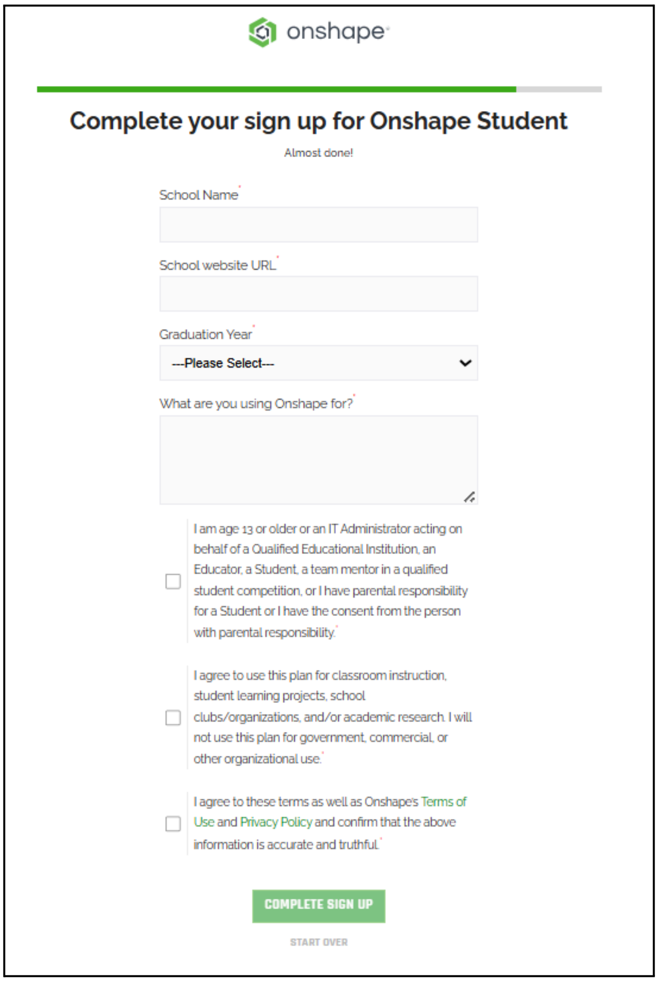
To get started, sign up for a free student account on the Onshape for Education website: https://www.Onshape.com/en/education/sign-up



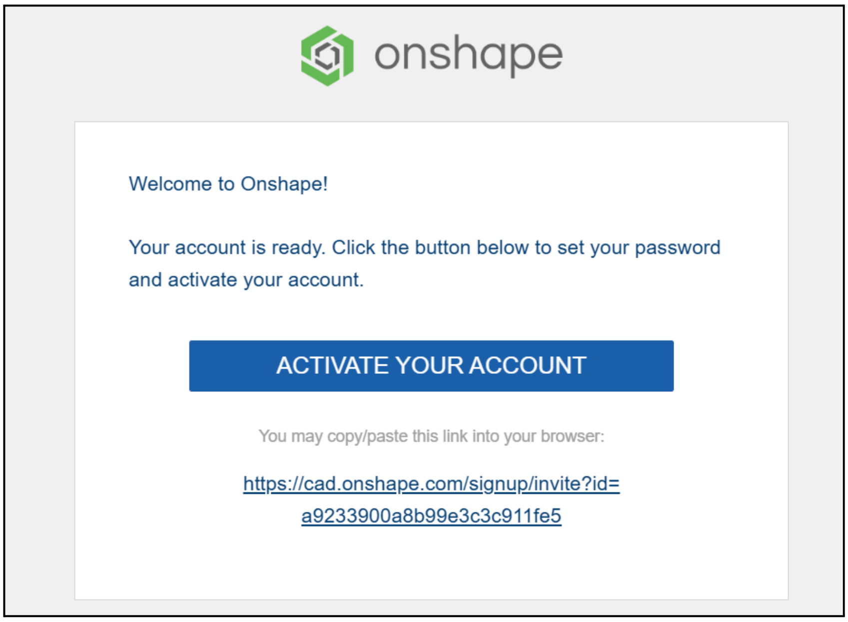
Select that you are a student in grade school, then “**Next**”.



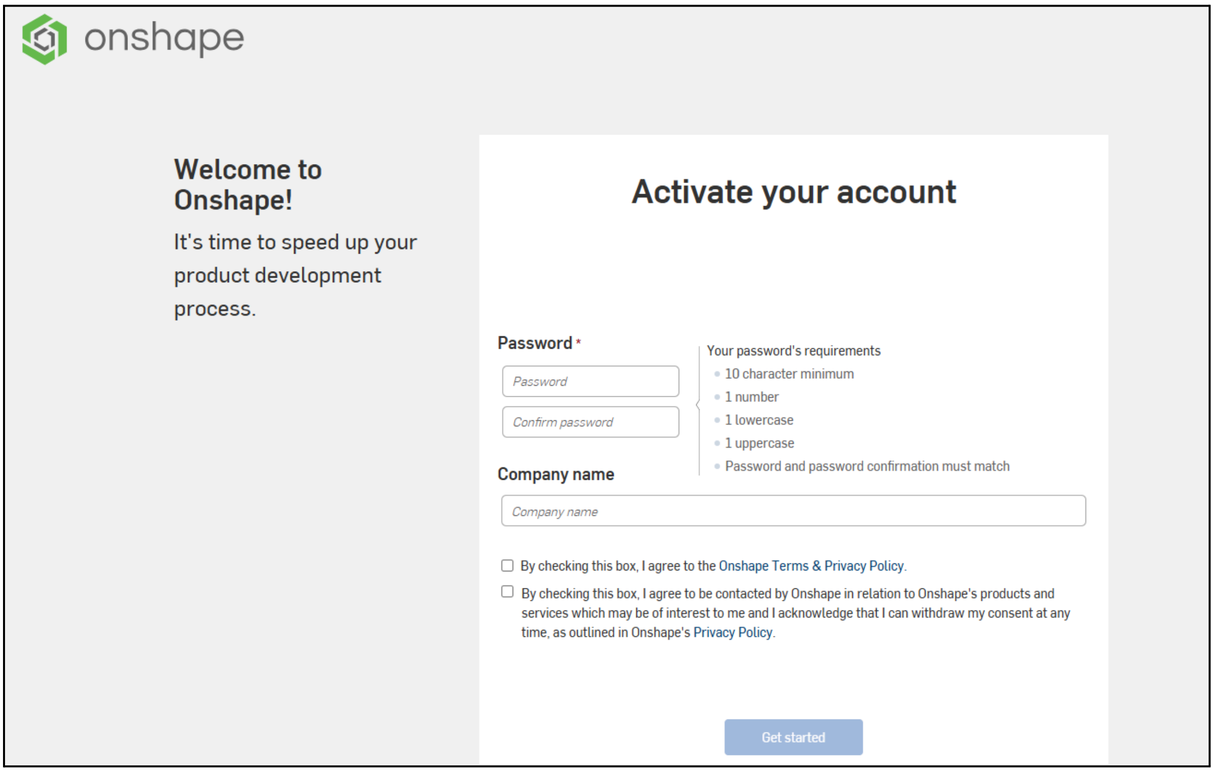
Complete the form with your school name, school website URL, graduation year, and specify you’re using Onshape to learn CAD. Check all boxes and click “**Complete Sign Up**”.



You should then receive an email from Onshape. Click “**Activate Your Account**”.

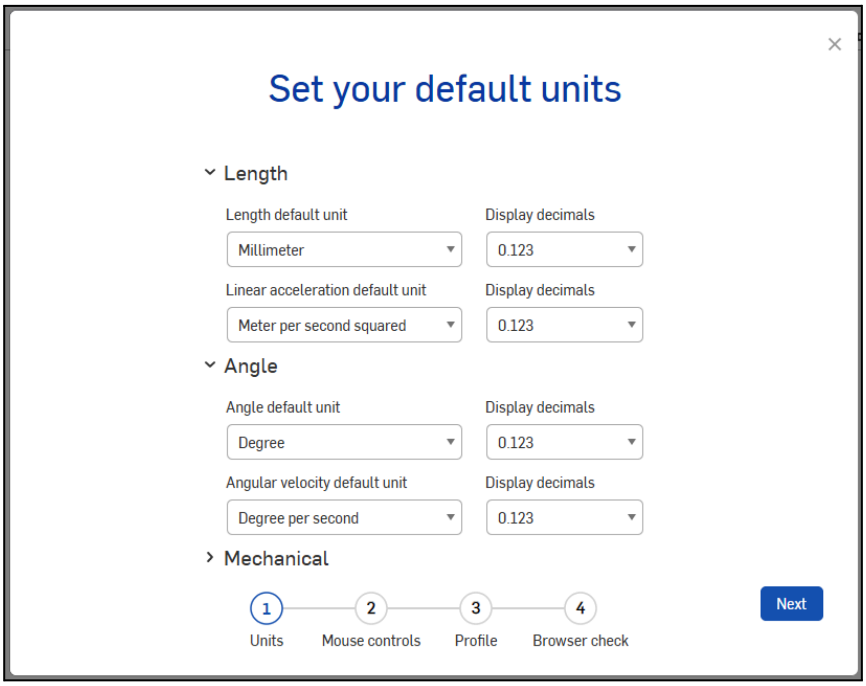


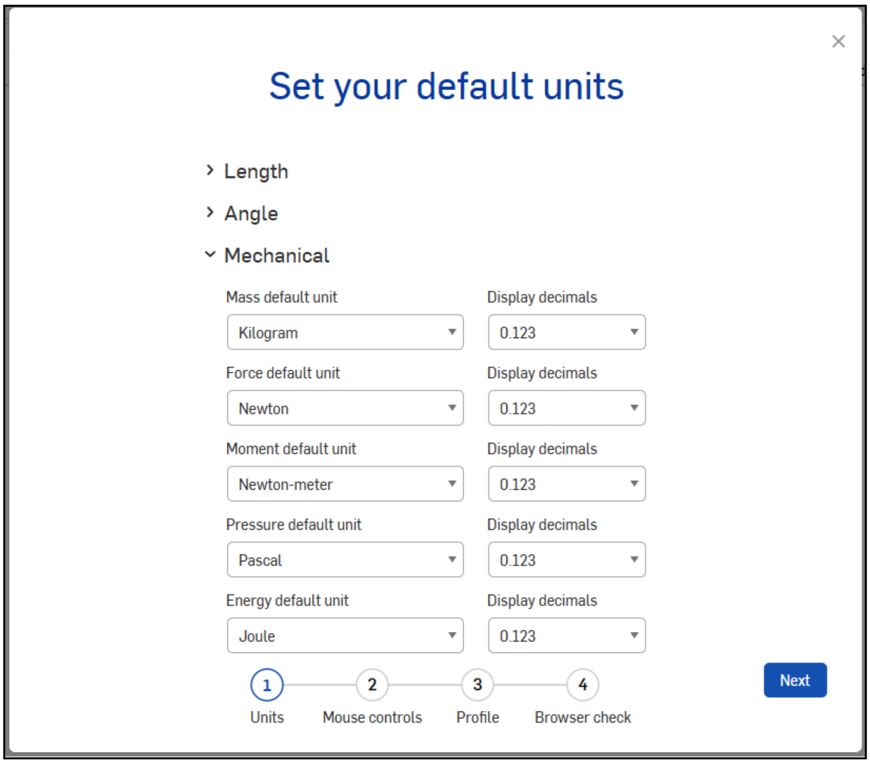
You will then be asked to choose a password, check both boxes, and click “**Get Started**”.



Default Units Configuration

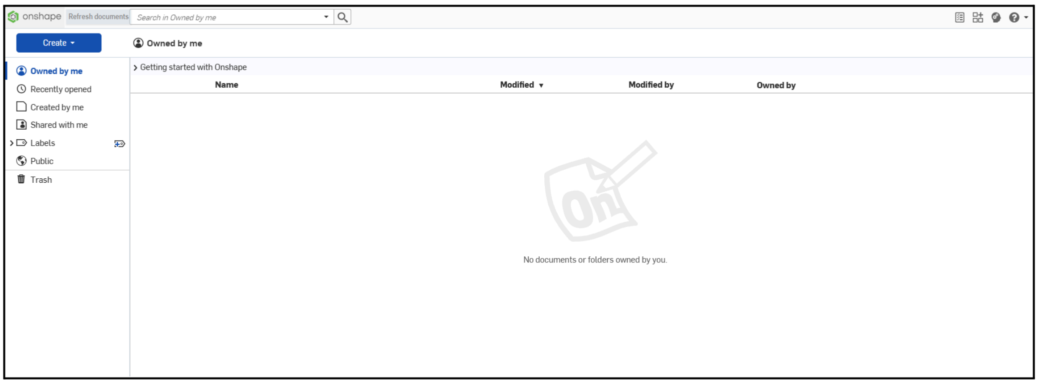
Finally, we will configure the default units to SI standards. Since we will only be designing small parts, let's set the length unit to **millimeters**. Remember to expand the "**Mechanical**" tab as well. Then click "**next**" till all the setup pages are completed.

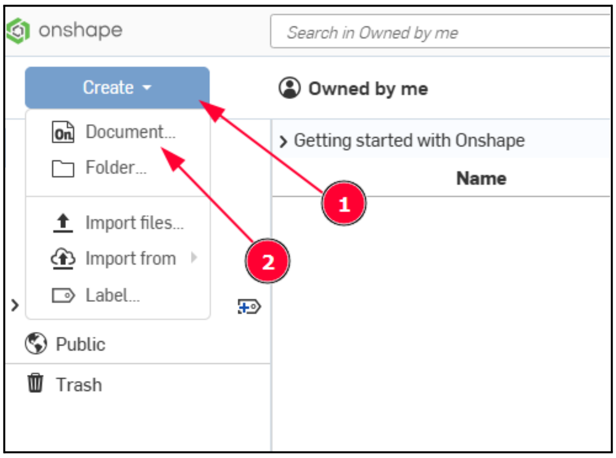


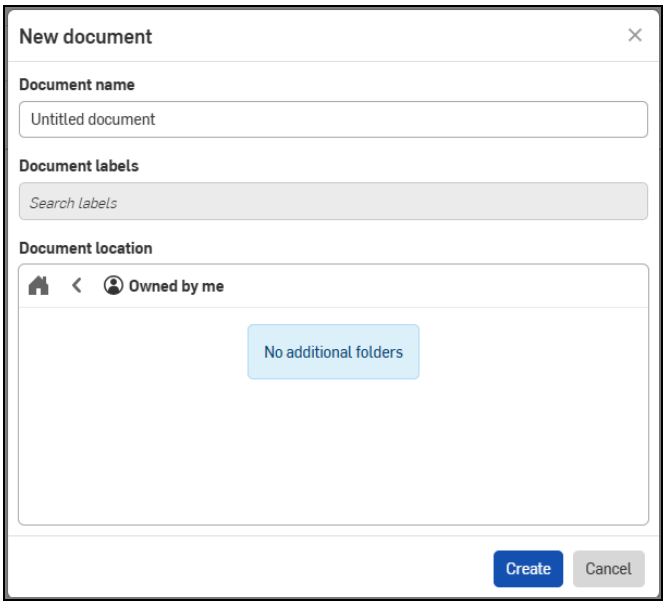


Creating a New Design

Welcome to your dashboard, this is where all your CAD designs will be located on the cloud. On the top left corner, click “**Create**” => “**Document**” to make a new design. Make sure to give your new design a name. You can also create folders to keep things organized.







Part Studio

After you open a new design document, you will arrive at the Part Studio. This is where we will model our parts for assembly later on.

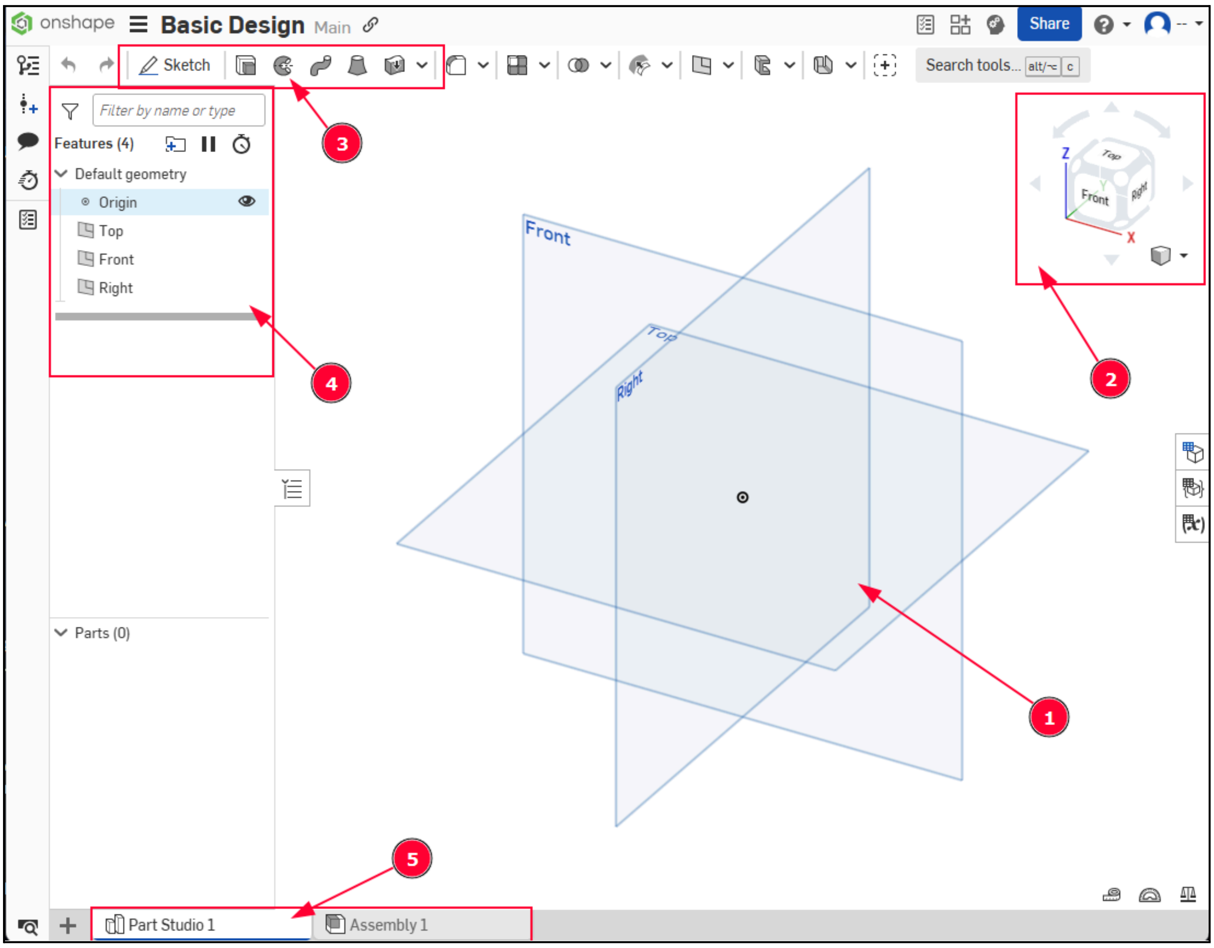
Onshape is a 3D CAD program, therefore, as you can see in **Balloon 1**, there are 3 planes representing the three dimensions.

In **Balloon 2**, there is a 3D gimbal to indicate the current orientation in 3D space. You can rotate the viewing angle by using the gimbal, or **right click** with the mouse. Holding the **middle mouse** button will pan the view around, and scrolling the middle **mouse wheel** will zoom in and out.

Balloon 3 is the toolbar where we will use to draw 2D sketches and convert them into 3D models using various operations.

Balloon 4 is a history log of all the operations performed. There is a powerful feature that lets you edit things in the timeline without starting all over again.

Balloon 5 lets you switch between different “apps”. For example, putting all the parts together in an assembly and creating technical engineering drawings.



2D Engineering Drafting

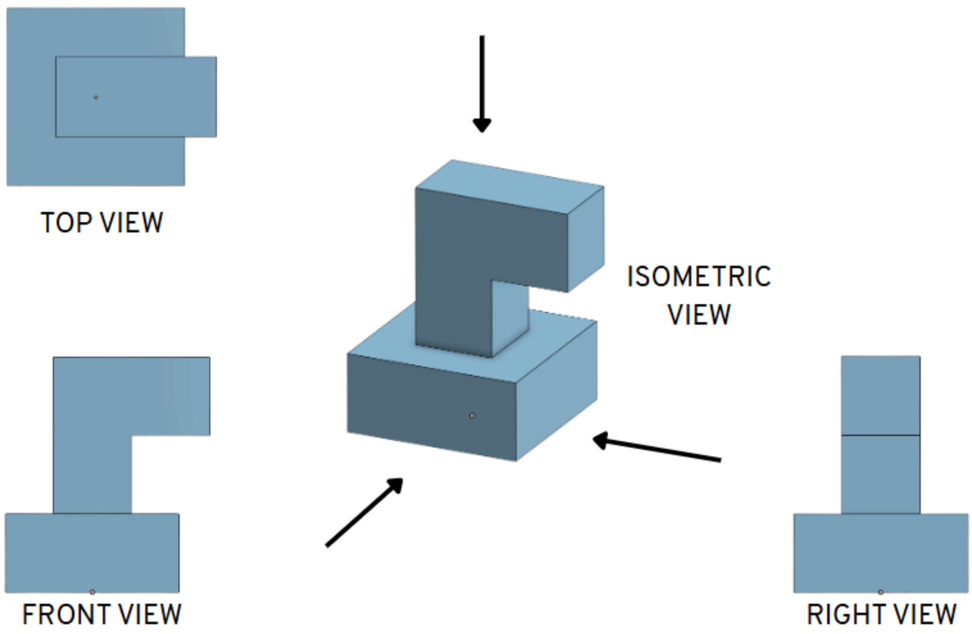
Orthographic Projections

Orthographic projection is the representation of a 3D object in 2D planes. For decades, multiview 2D orthographic drawings have been the standard way of communication between engineers and the technicians producing the parts. Before computers were widely available, the design concepts could only be represented in the limited 2D paper format.

Nowadays, technology enables us to easily share a 3D model via the cloud, where the recipients could easily interact with it on their phone. However, representing 3D objects in orthographic views remains the standard, since it can more clearly define part features and sizes. As you will see, it is also easier and more precise for us to model parts from 2D to 3D. It is difficult to draw in 3D space with a mouse on the 2D plane.

The engineering drawing below uses the standard 3-view format. The three views are front view, top view, and side view. Usually, three views are enough to represent the 3D object. However, additional views can be added if it is needed to convey more details. I regularly add an extra isometric view to help the reader envision the 3D object better.

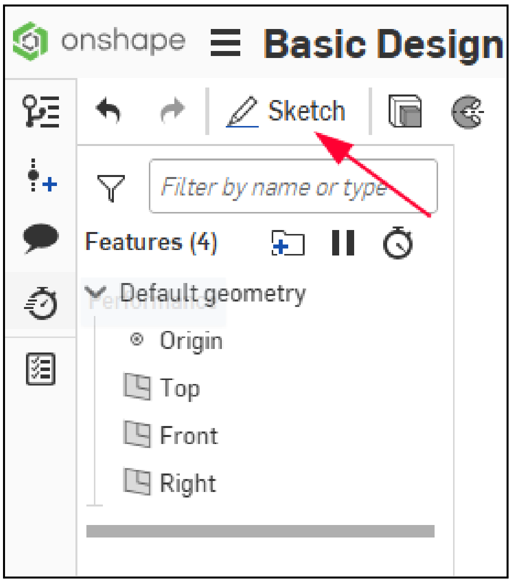
Take a look at the three views below, can you reconstruct the 3D model in your mind?

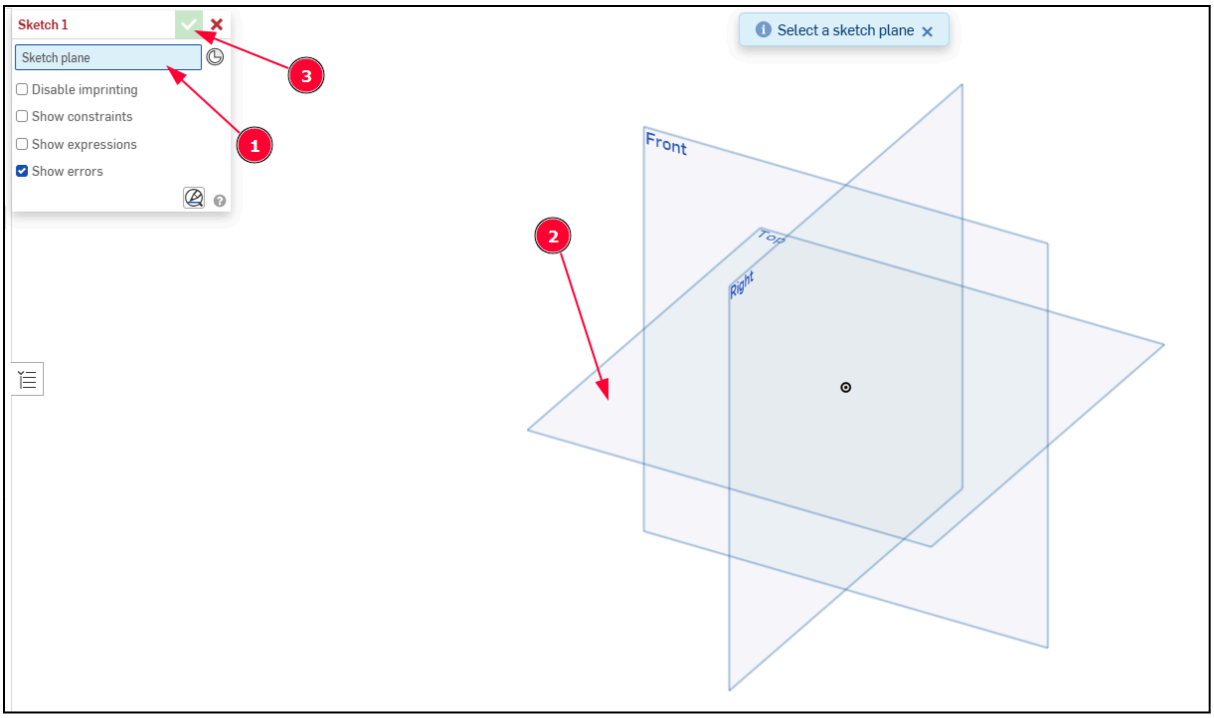


Creating 2D Sketches

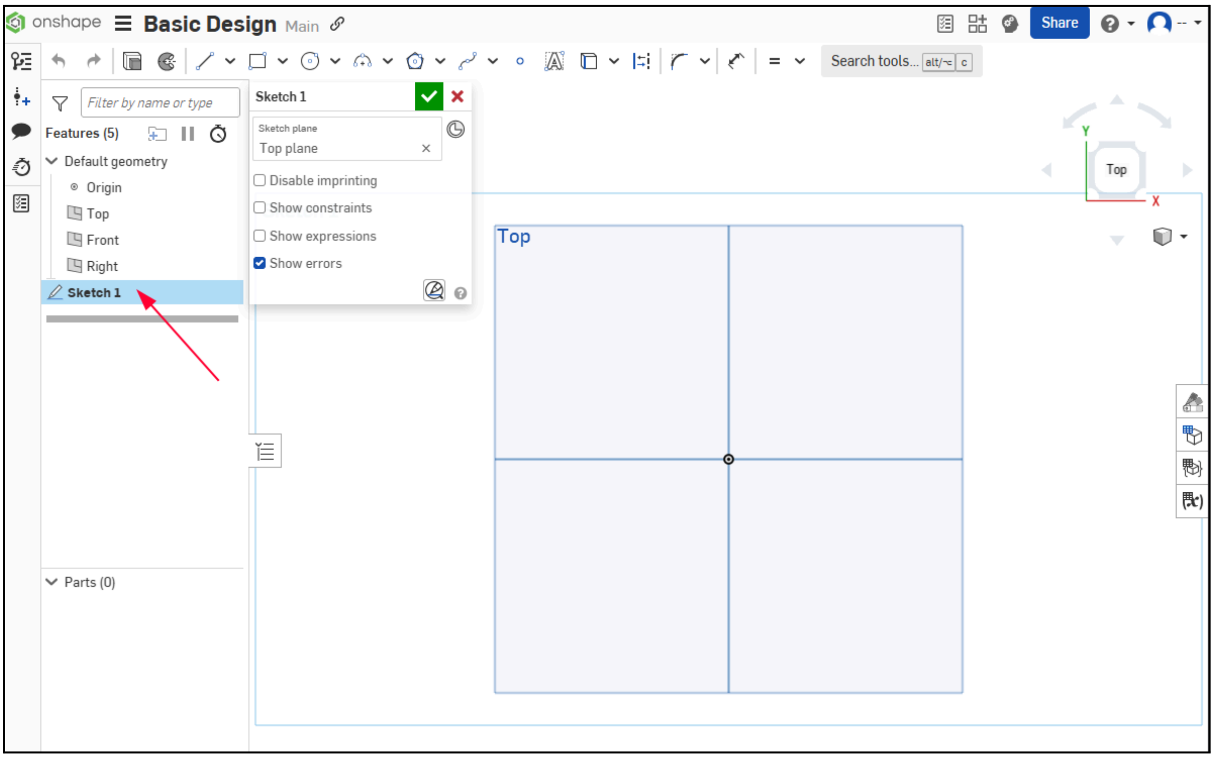
Let's try recreating a 3D object. First start by creating a 2D sketch by clicking "**Sketch**" on the toolbar. Then, select the **top plane** as the sketch plane. We can actually create sketches on any plane, including new planes that we created ourselves.

Press the **"N"** on the key to orient the view to face the sketch plane. You can also do that manually by clicking "**Top**" on the 3D gimbal.

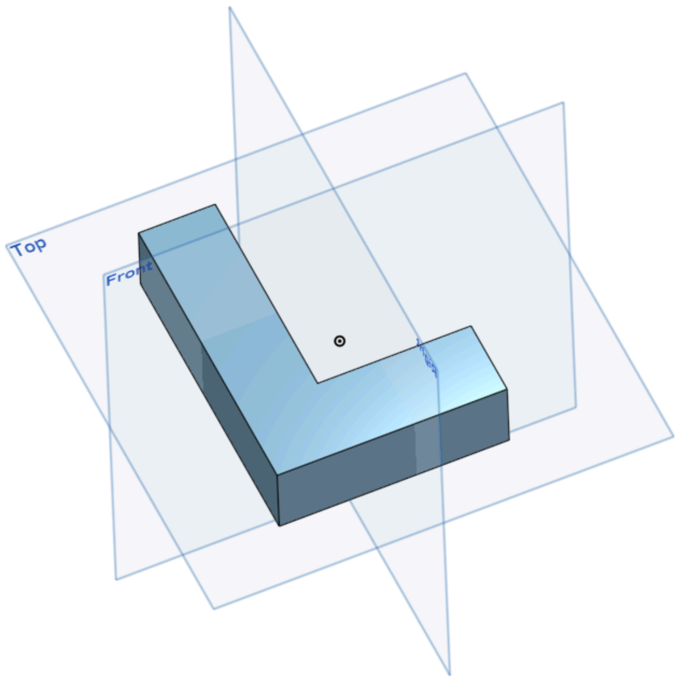




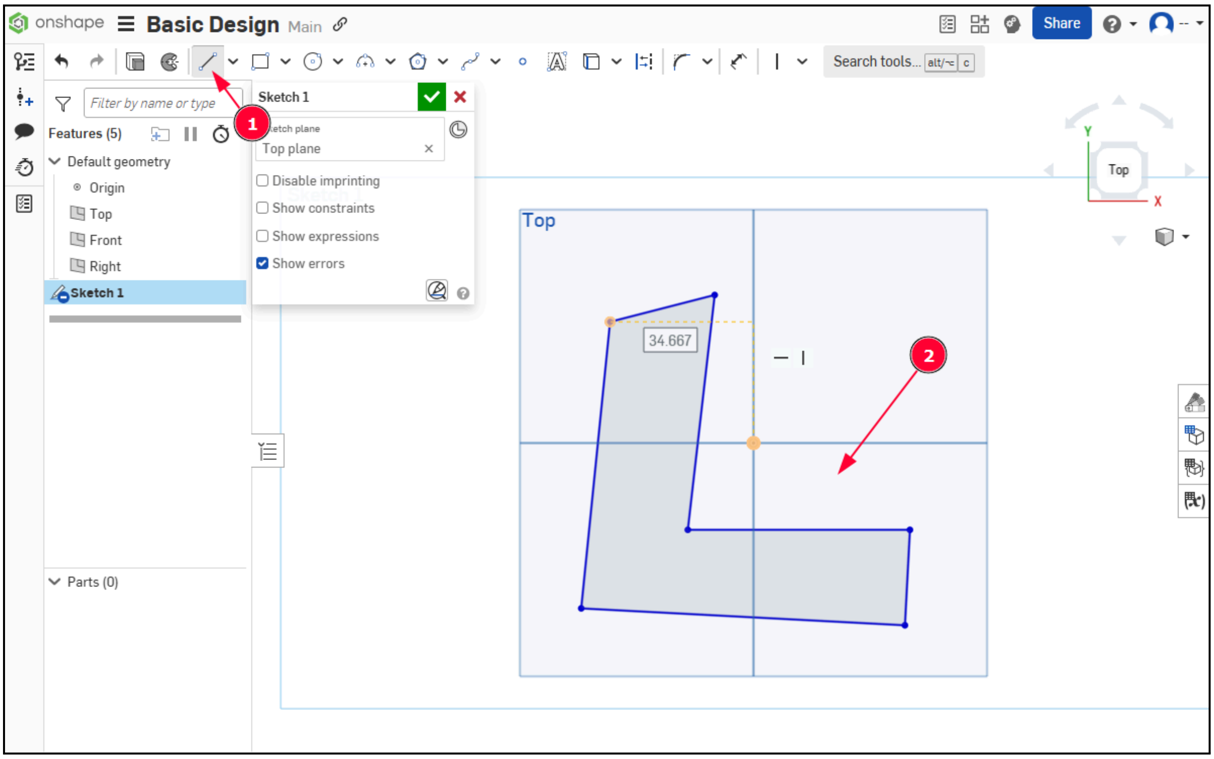
Notice that a new item called **Sketch 1** is added to the timeline. If you ever need to come back and edit this sketch, you can **right click** and select **edit**.



For practice, we will recreate this L-shaped part. If you have to convey the most information about this part, would you use the front, top or side view? Can you imagine what it would look like after the orthographic projection?

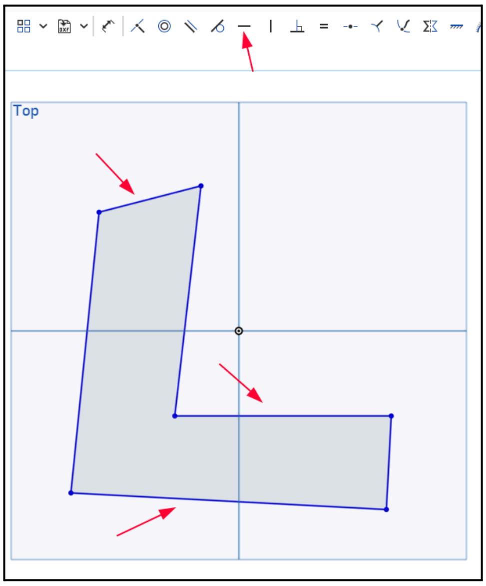


From the top view, select the **line tool** from the toolbar, then roughly draw the overall shape of the part. In this case, we will draw a L. Onshape has built-in snapping tools to help you draw straight horizontal and vertical lines. If you messed up like I did, no worries, we will set some constraints to properly define the shape. Press **"ESC"** to deselect the line tool.



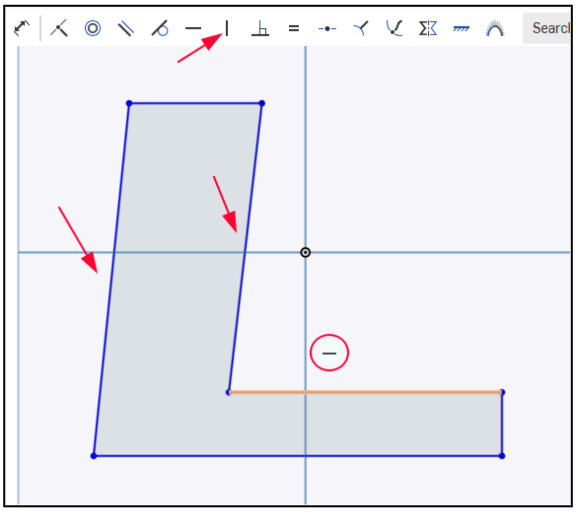
Geometric Constraints

Looking at the sketch below, the L looks quite crooked. Let's start by constraining the horizontal lines of the L with **horizontal constraints**.

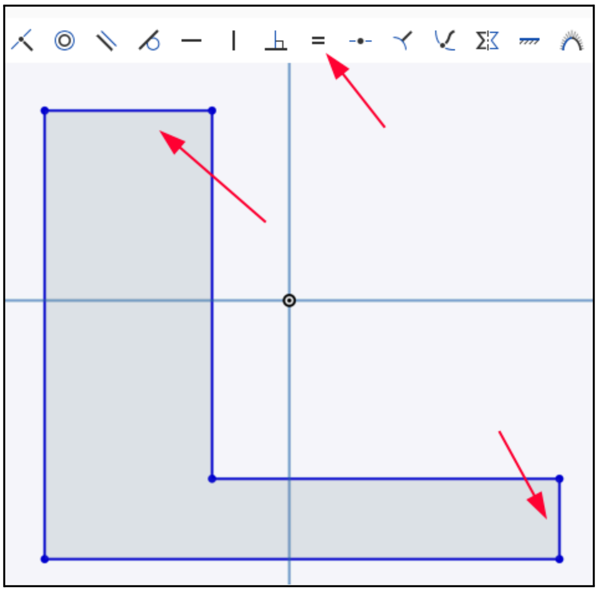


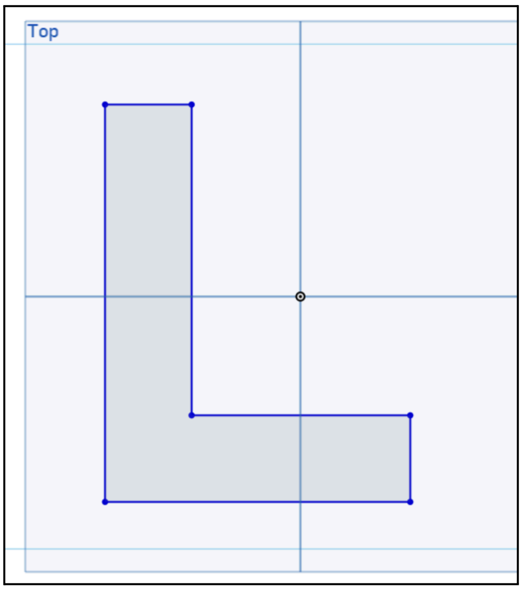
If you hover your mouse over the horizontal lines, you will now see that a horizontal constraint symbol appears. This indicates that the line can only be horizontal from now on. You can click on the constraint symbol and delete it if you want to remove the constraint.

Next, **constrain the vertical lines**.



Finally, let's constrain the width of the L as equals. Hold **shift** and select the two thickness lines, then set the **equal constraints**. After that, you should be able to drag the lines in the sketch, the sketch will look somewhat like a L regardless.

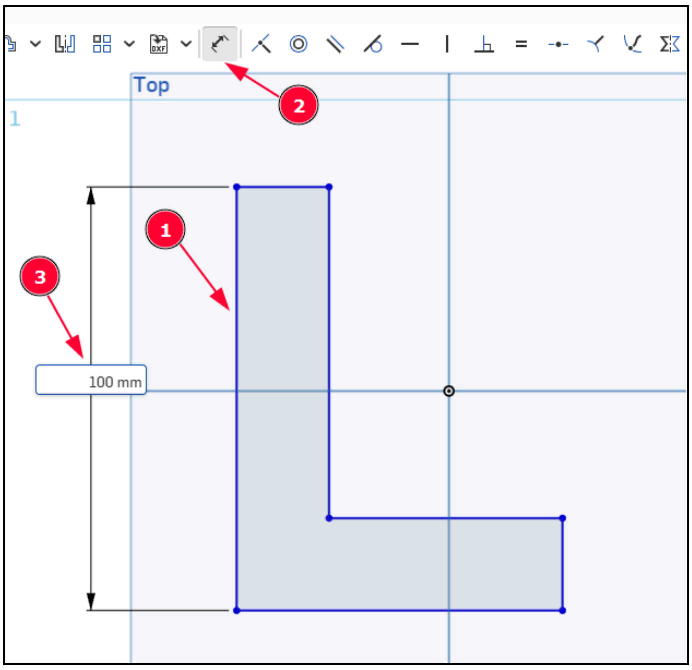




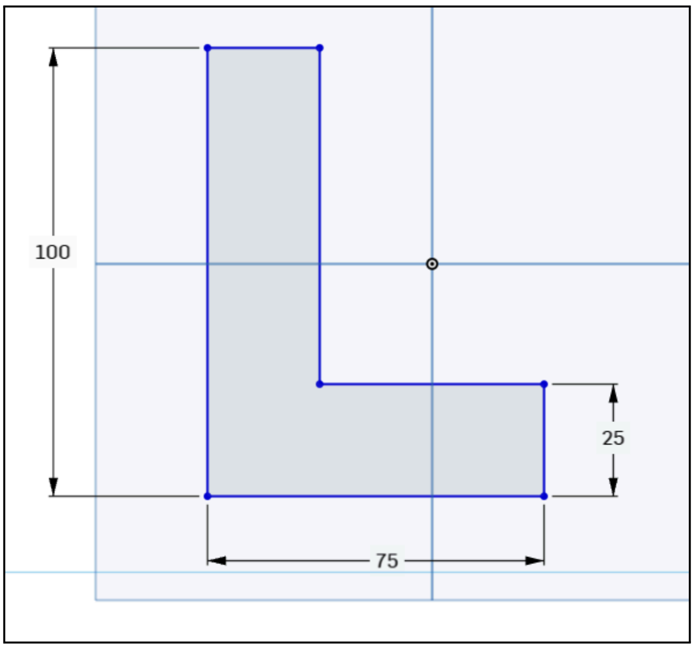
Dimensions

Right now, the sketch looks like an L, but we have not defined the size of it. We can do that using the **dimension tool**.

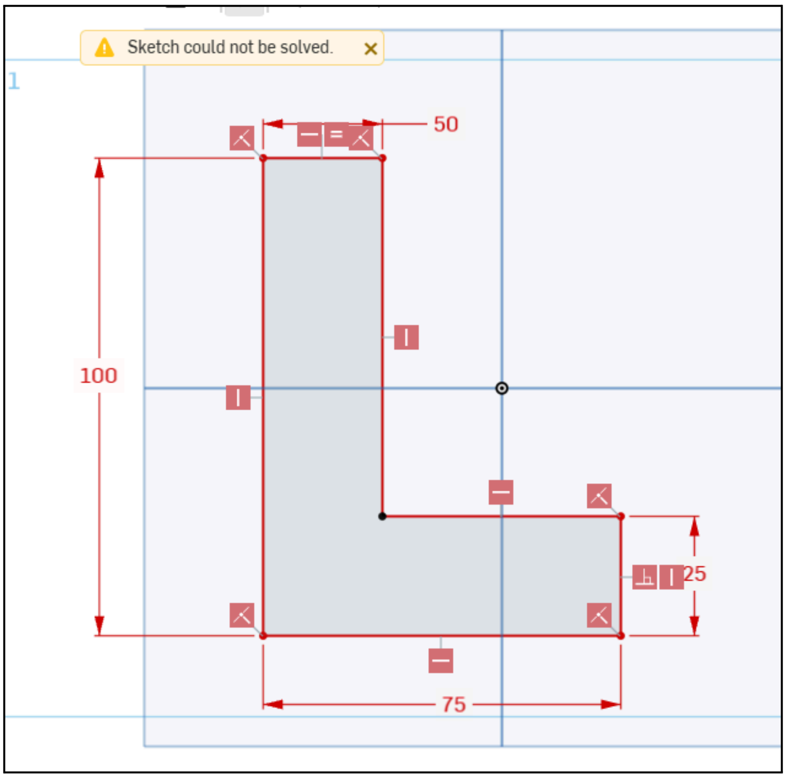
Click on the vertical line and **dimension** the height of the L as 100 mm.



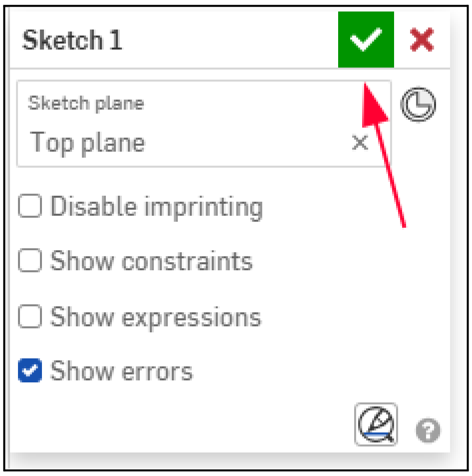
Finally, **dimension** the width as 75 mm and the thickness as 25 mm. There's no need to define the dimension for the other lines, as they are all constrained already.

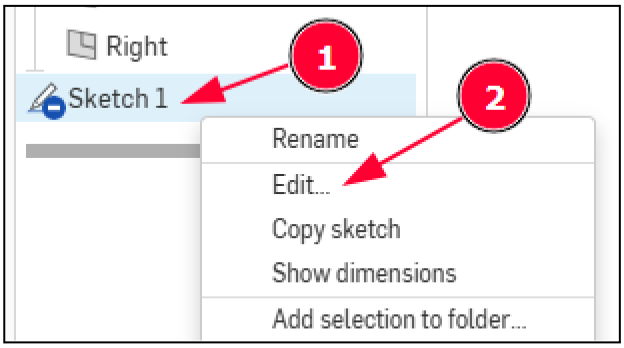


Let's say that we try to define the top thickness as 50 mm, that would over-define the sketch. We had already defined the bottom right thickness as 25 mm, and we previously defined the top thickness with equal length as the bottom right thickness. Defining the top thickness as 50 mm would contradict the previous constraints. We call this condition over-constrained, and the sketch could not be solved. Hence, Onshape shows the sketch as red to indicate an error. You can resolve this by removing the violating constraint or dimension.



Click on the check mark to finish the sketch. Remember that you can always **right click** and select **Edit** to edit the sketch.





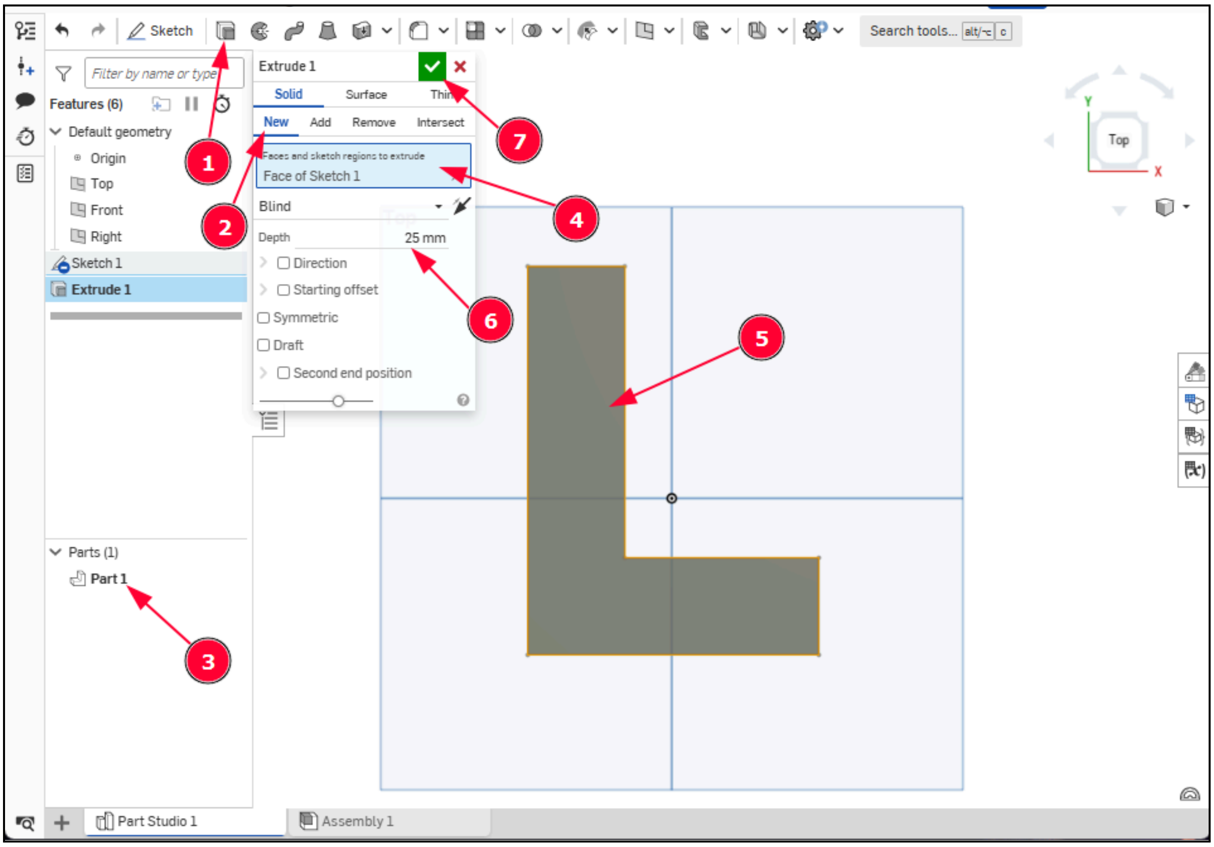
Fundamental 3D Modeling Operations

There are four fundamental 3D modeling operations in CAD: extrude, revolve, sweep, and loft. In a way, you can associate the extrude operation with the milling machine, and the revolve operation with the lathe. Those are the machines that you will use to create parts modeled using those operations. Sweep and loft are more advanced modeling operations that can create a smooth and curvy surface. However, they require more configuration and multiple sketches to get the desired results. Therefore, it will be a good idea to learn them at a later time.

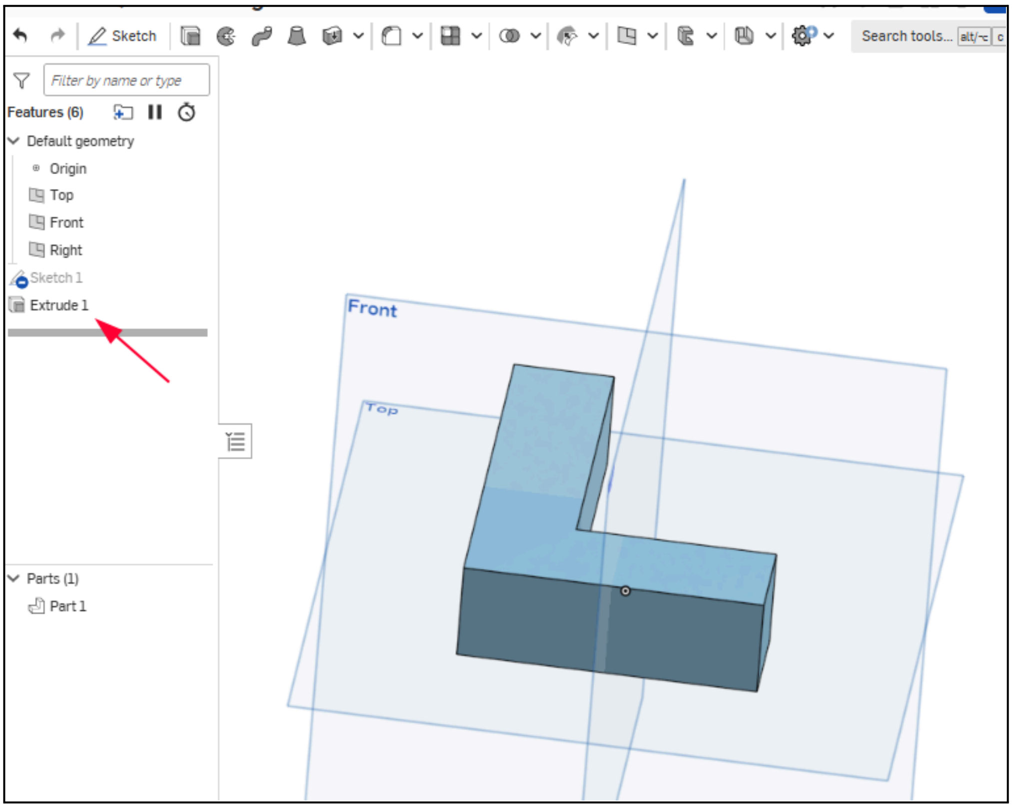
Extrude (New / Add / Cut)

The **extrude** operation simply takes a 2D sketch and gives it a height. Take our L-shaped sketch for example, if we apply the extrude operation, it will look like a 3D L geometry.

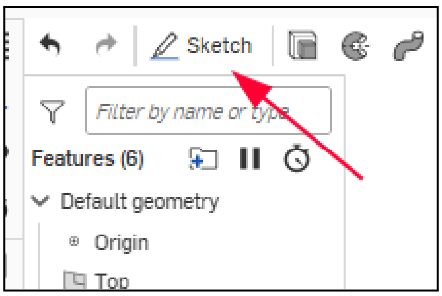
Let's give that a try, click on the **extrude** icon in **Balloon 1**. The extrude window will pop up. Next, notice that **New** is selected in **Balloon 2**, this will create a new part called **Part 1**, as indicated in the parts window in **Balloon 3**. You can always rename the part later. Next, we will need to select a face or sketch to extrude, click on the box in **Balloon 4**, and then select the L-shape contour in **Balloon 5**. It is important to remember that only closed geometry can be extruded. Next, give it a height of 25 mm in **Balloon 6**, and click the check mark.

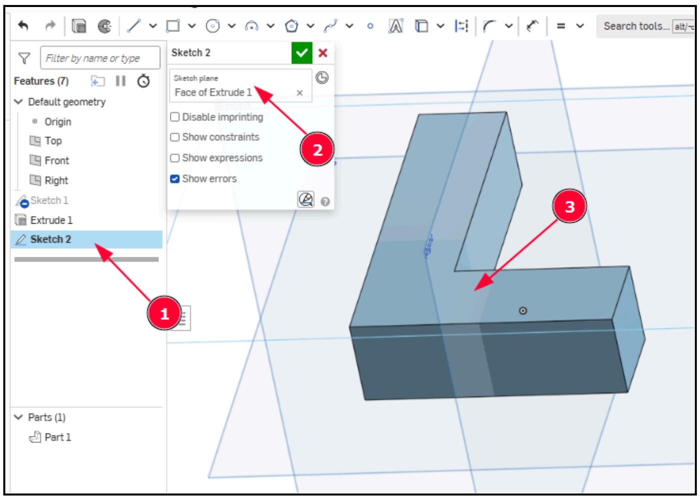


And there we have it, a 3D L model. Notice how the extrude operation is added to the timeline as **Extrude 1**. You can always right click and edit the settings if you like.

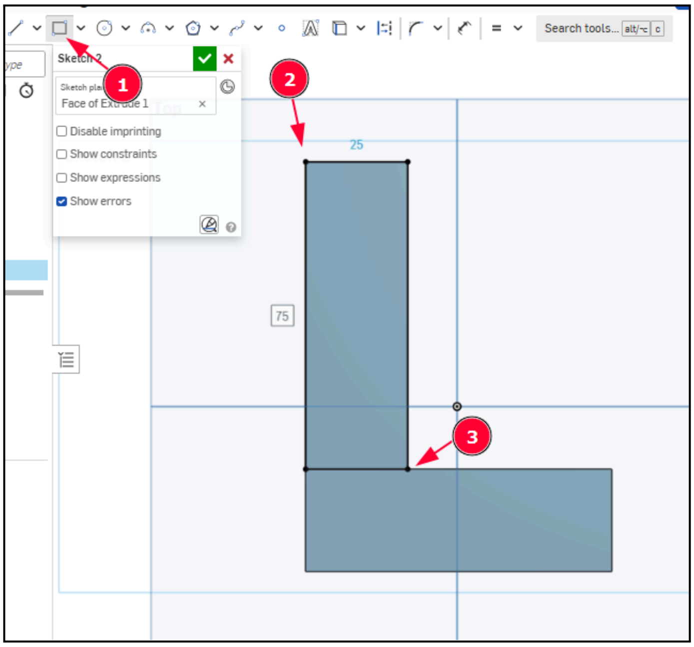


Now let's try to **add** another extrusion on the model. First, create a new sketch by clicking on the **Sketch** button. Notice how a new sketch called **Sketch 2** is added. Next, click on the box in the sketch window and then select the top face of the L geometry to sketch on that face. Press **"N"** on the keyboard to orient ourselves toward the sketching plane.

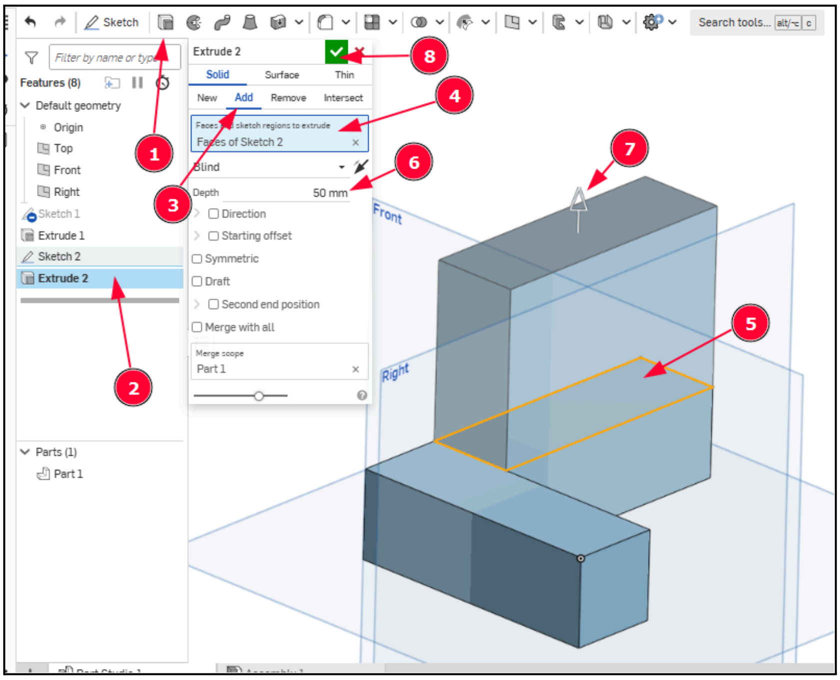




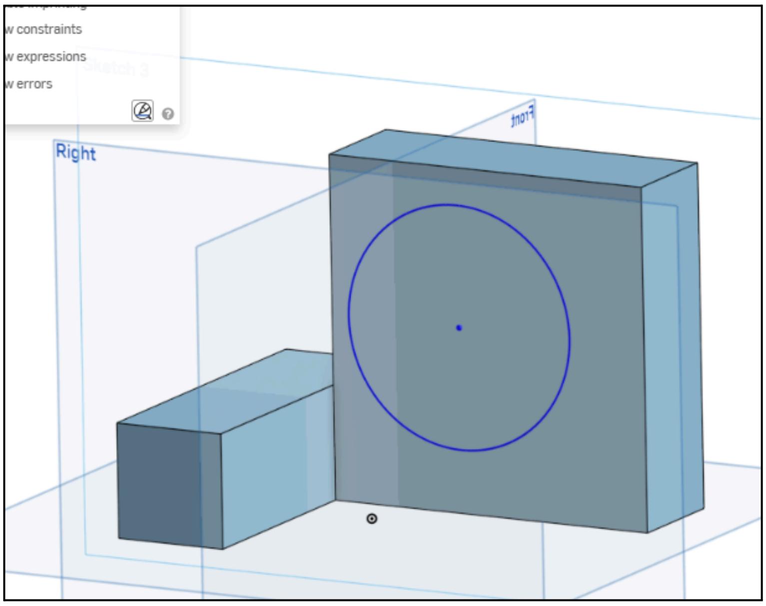
Next, we will create a rectangle using 2 points. If you hover your mouse over an existing geometry, Onshape should automatically snap to the nearest line or vertex. Select the top left corner of the L in **Balloon 2** and then the inner corner on the right in **Balloon 3**. You can sketch a rectangle like the one shown below.



Use the **extrude** tool, notice that a new **Extrude 2** operation is added to the timeline. Select **Add** from the Extrude tool window in **Balloon 3**, this indicates that we are **adding** to an existing part. Select the rectangle that we just sketched in **Balloon 5** to extrude it. Enter 50 mm as the height in **Balloon 6**. Alternatively, you can also drag on the extrude arrow in **Balloon 7**. But it is more accurate to enter it manually. You can rotate your view to see better. Finally, click the check mark to complete.

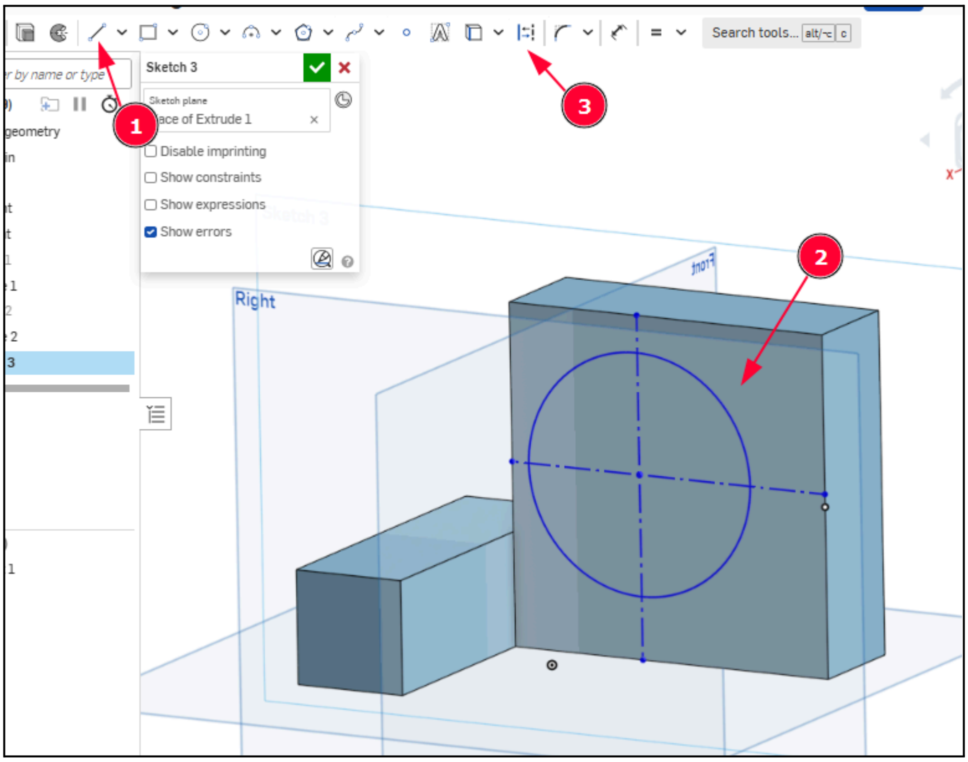


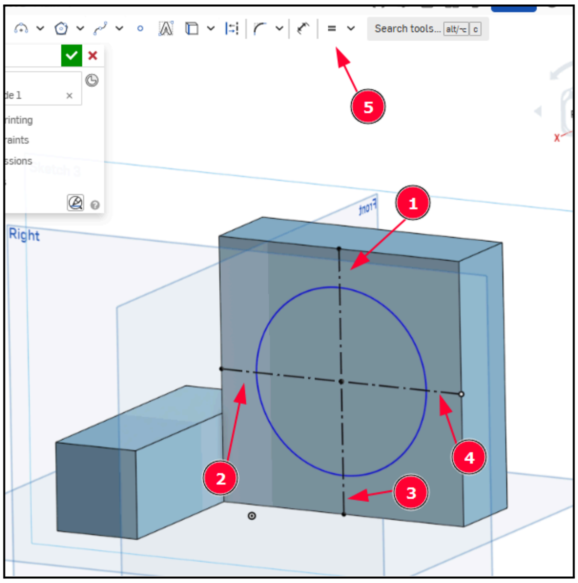
Finally, let's try the **extrude cut** (remove) operation to make a cut in the existing geometry. We will cut a circle in the wall we have just created. Again, we will start by creating a 2D sketch on the vertical wall face, then draw a circle.

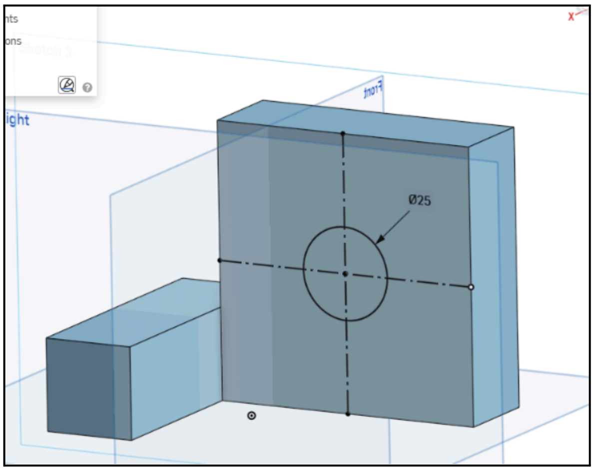


We will then draw four straight lines spreading vertically or horizontally from the center of the circle to the edge of the face. Now it is very important to make sure that they are straight, constrained vertical or horizontal. Then, select all of them and click the **construction line** button to mark them as helpers.

With all 4 lines selected, constrain them as **equals** to center the circle.

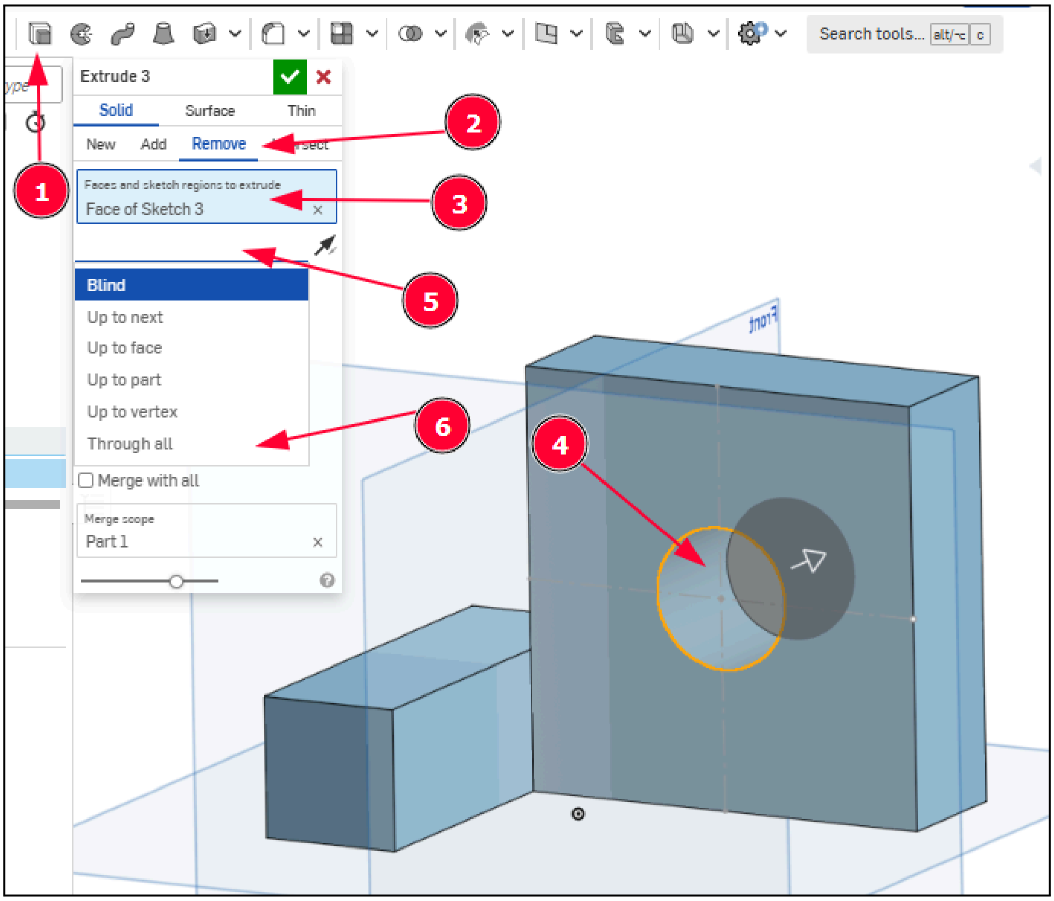






Use the dimension tool and click on the circle to **set its diameter as 25 mm**. Click the check mark to close the sketch.

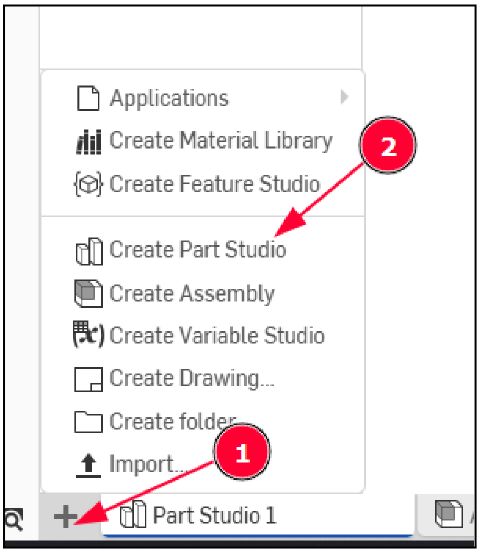
Next, use the **extrude tool** again and select **Remove** in **Balloon 2**. This will let us remove or cut materials from the part. Select the entire circle, then in select **Through all** from the menu that said Blind in **Balloon 5**. This lwill let us cut through the entire part as opposed to a fixed length. And now we have a part with a circle cut out.

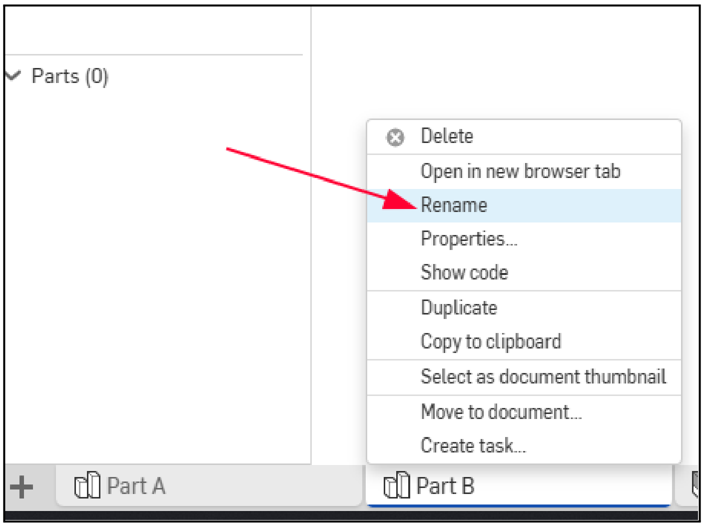


Revolve

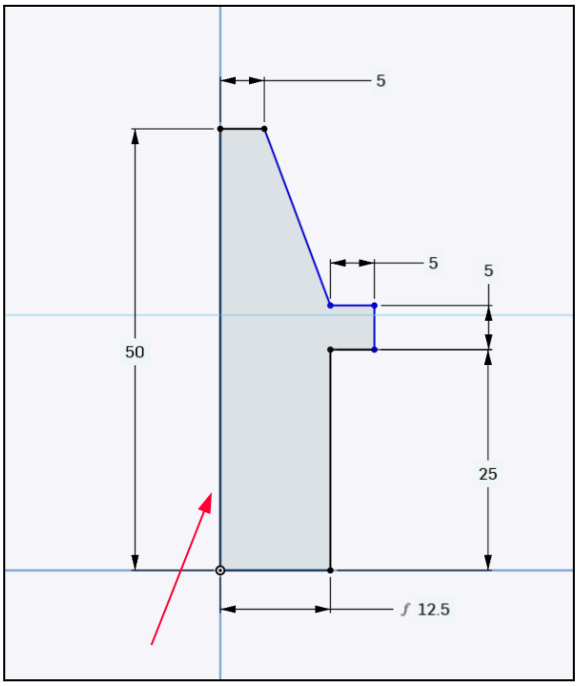
The **revolve** operation takes a 2D profile and revolves it around an axis to create a 3D object. Let's create a new part that can be inserted into the hole that we created earlier.

On the "App" manager tab, create a new Part Studio. We could have created all the parts within the previous Part Studio, but keeping them apart might be a bit cleaner. It might also be a good idea to rename them to Part A and Part B.

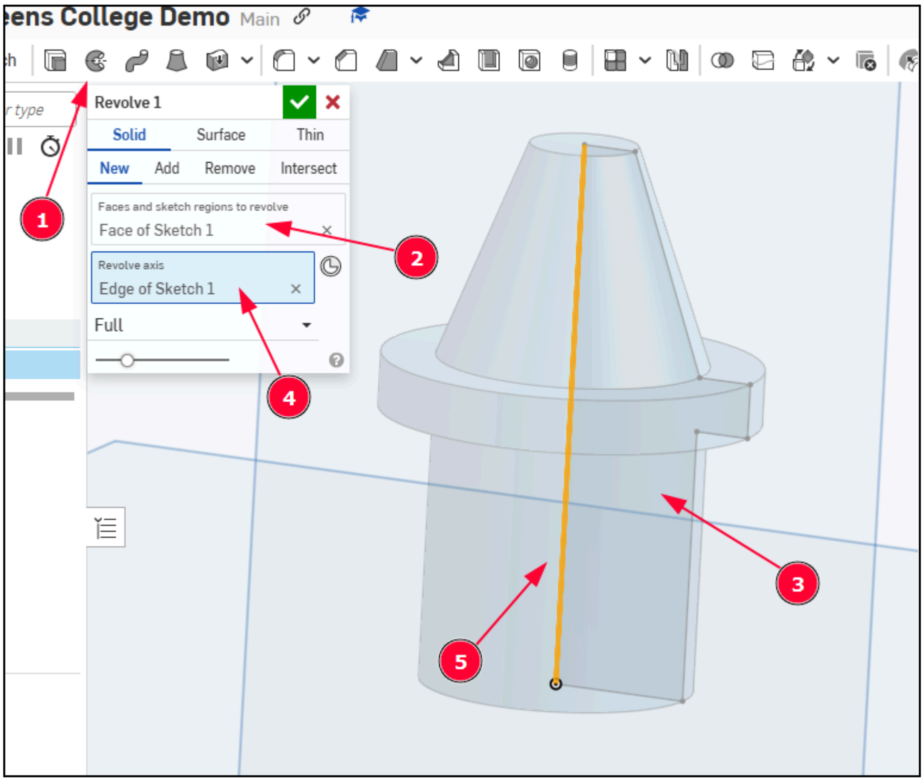


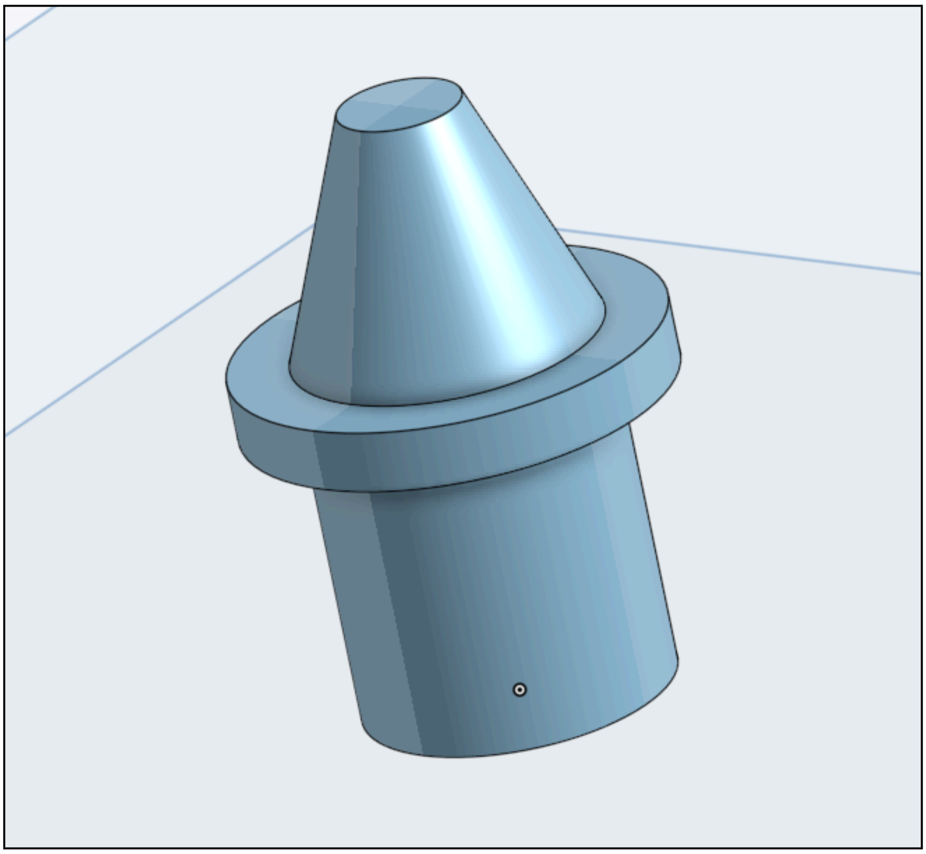


Create a **new sketch** on the **Right** plane and draw the following shape. It is good practice to place the rotating axis along the vertical axis line in the center. That way, you can see what the rotating profile looks like. Click check to close the sketch.



Select the **revolve** tool. The revolve tool window should open. Select the first box in **Balloon 2**, then select the **2D profile** we sketched earlier. Next, select the second box in **Balloon 4**, and then select the rotating axis in **Balloon 5**. Click check to apply the revolve operation.



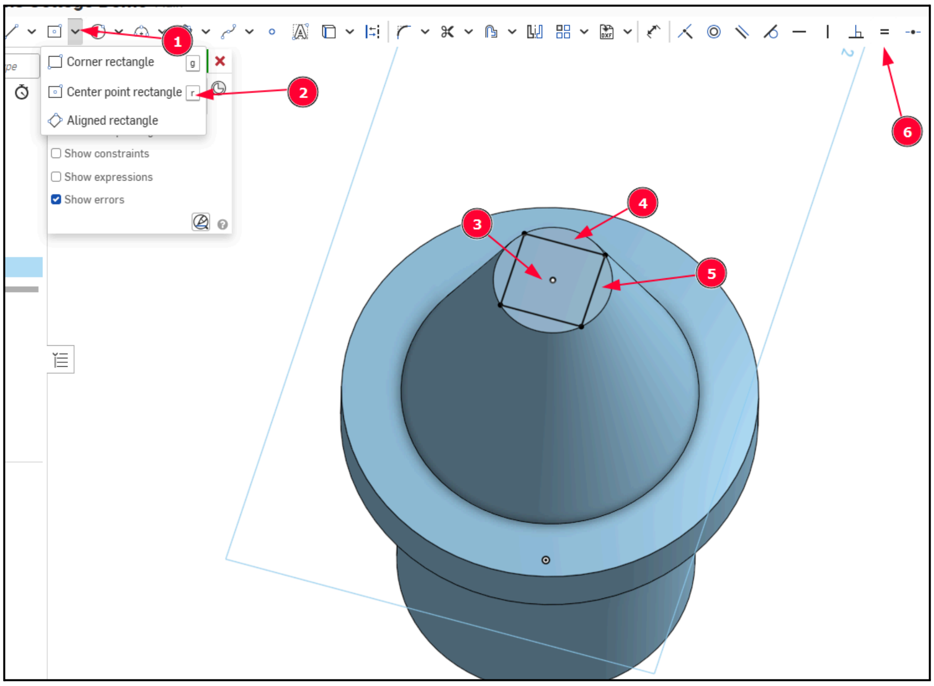


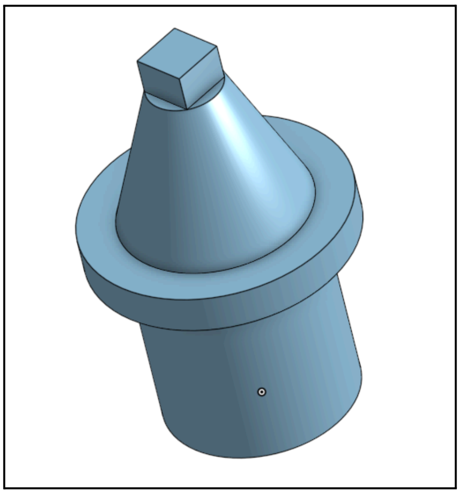
Assembly

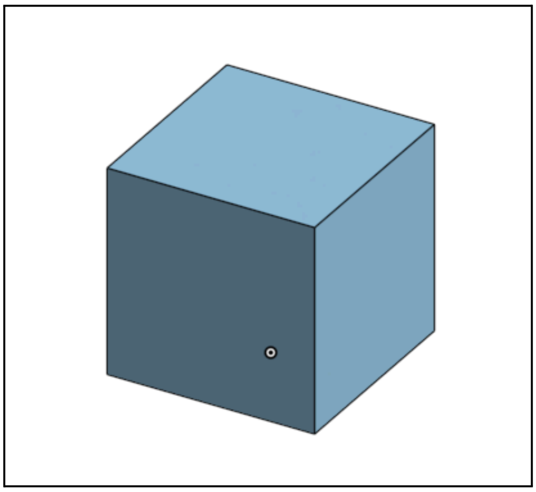
Engineering projects, like airplanes, are made of thousands to millions of parts. Making sure all of them fit together is a challenge. With the help of CAD software, we can create a virtual assembly to test fit all of them before production. To create an assembly, we need to use mates to attach different parts together. There are many types of mates in assembly and each software does it a little differently.

To learn Onshape's assembly functionality, let's modify Part B so that it has a square extrude on top. We will also create a new Part C that's a 25 x 25 mm cube.

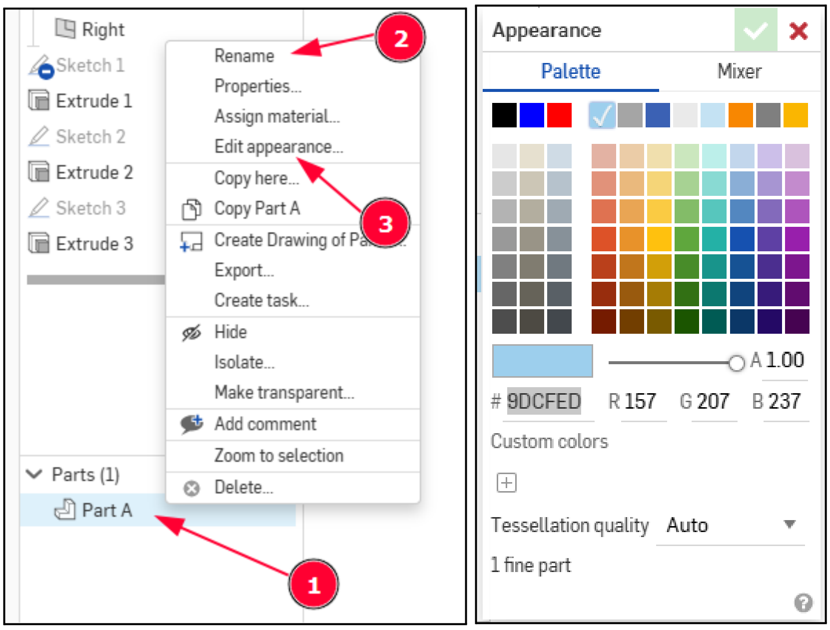
For Part B, create a sketch on the top face, and select the **center point rectangle** tool. Snap to the center of the face and then snap the other point to the circle's edge. Next, select the two adjacent lines on the rectangle and constrain them as equals. This will force the sketch to a square. **Extrude** it by 5 mm.





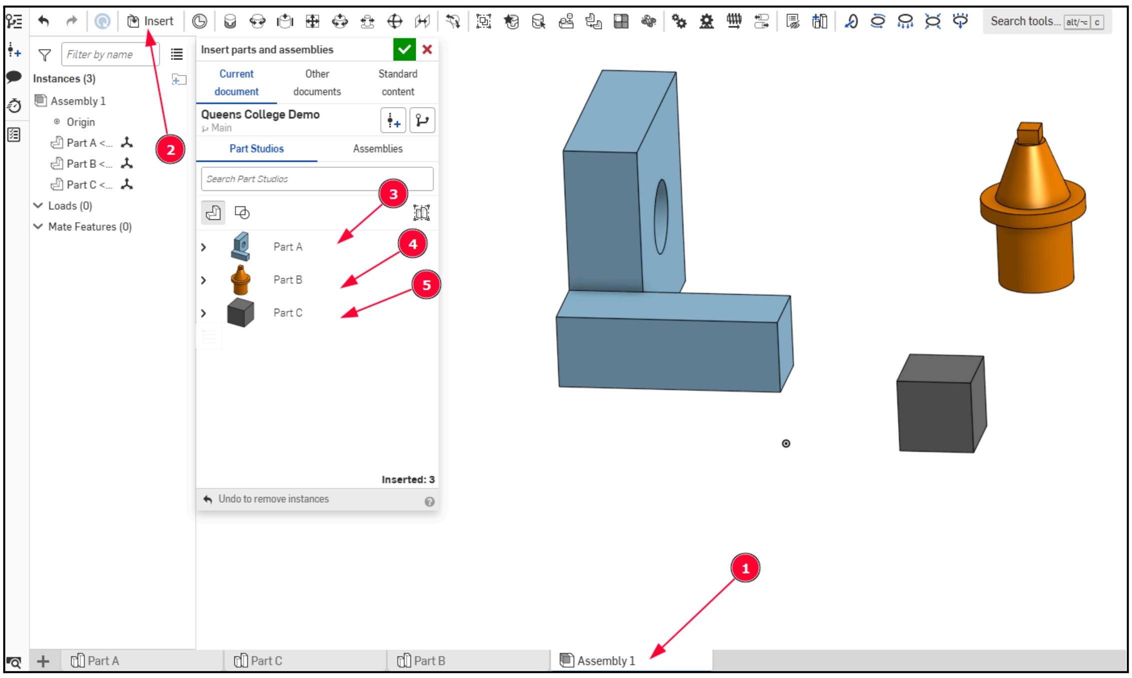


Before we start with assembling the parts, let's rename each of them and set their appearance by choosing a different color.

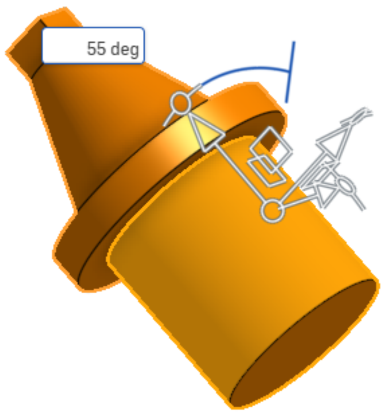


Fastened Mate

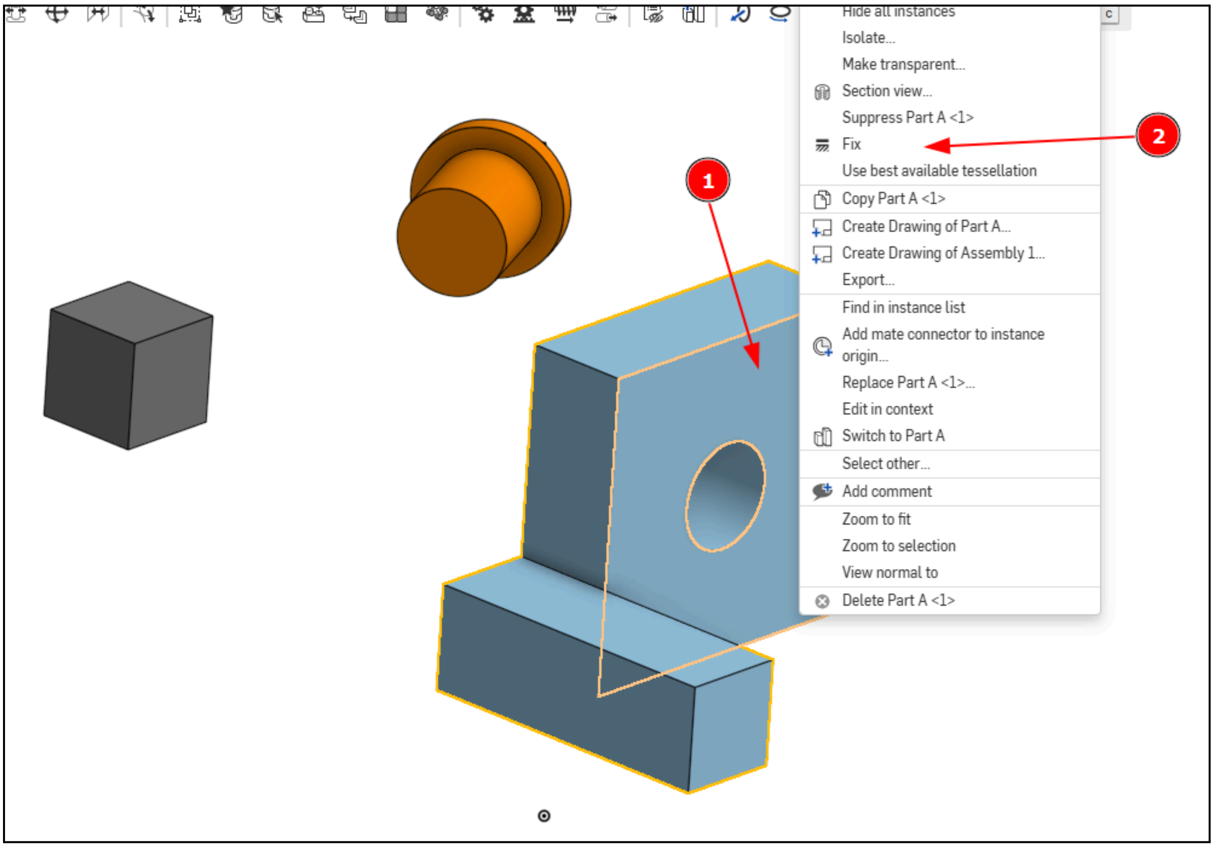
The **fastened mate** is a mate that fixes a part to another without allowing any movement. This is a good mate to use for non-moving parts. Switch to the **Assembly** tab, **insert** all the parts into the workspace. You can move the parts around by left click and drag.



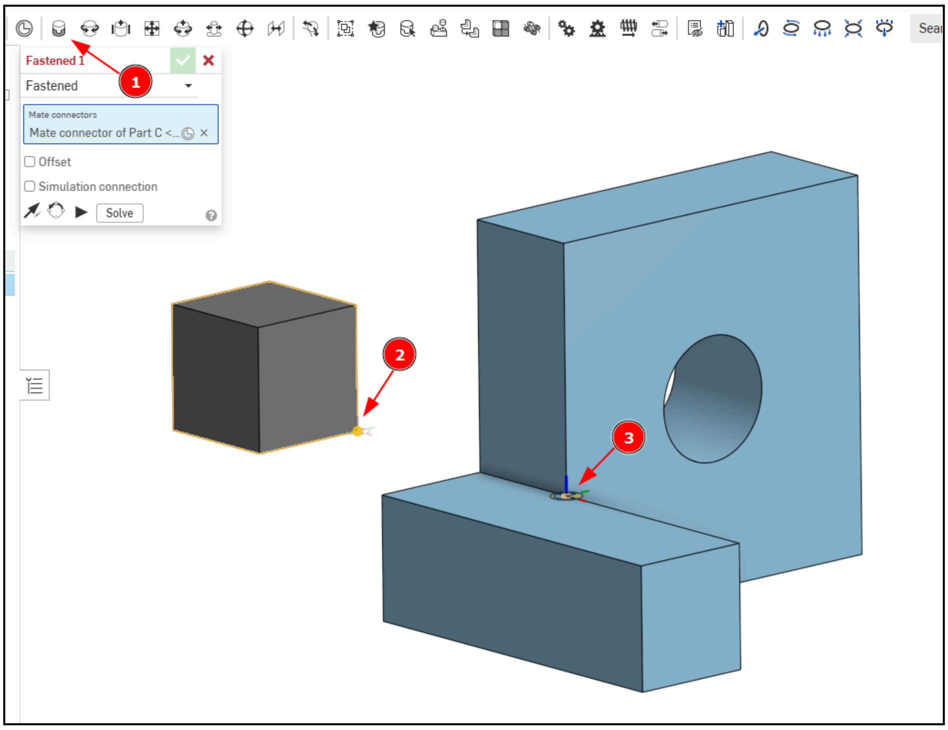
You can also left click on the part to bring up the 3-axis translation and rotation gizmo. The inner arrow is for controlling translation and the outside arc is for rotation.



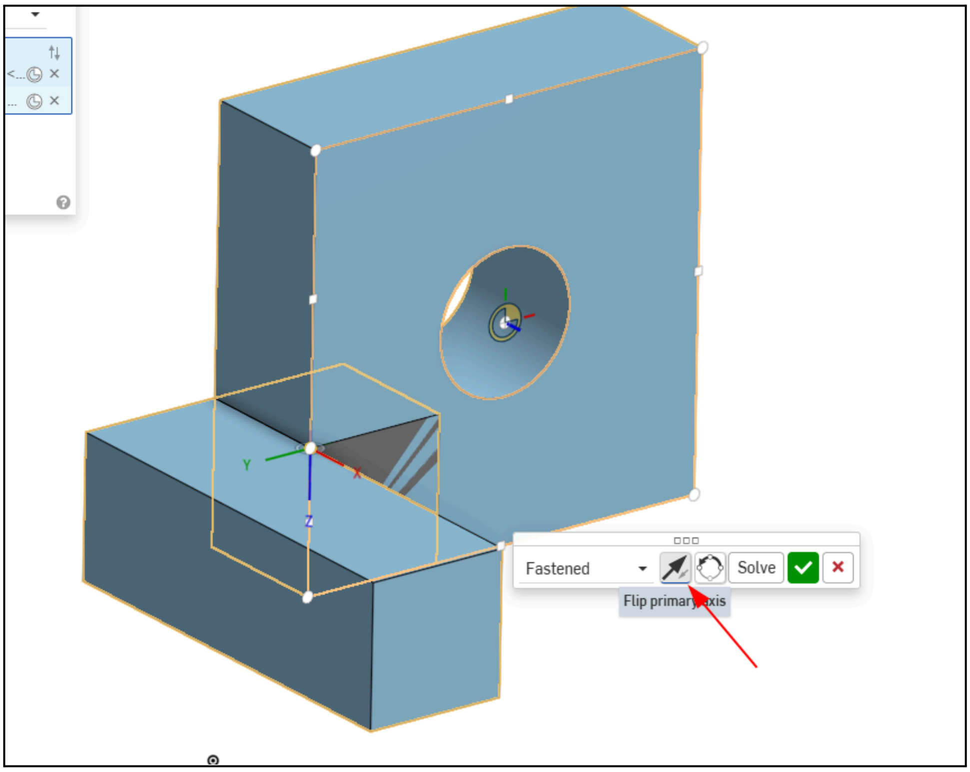
Next, we will need to pick a part and fix it in space so the assembly has a fixed reference point. **Right click** on Part A and click **Fix** to fix it in place.



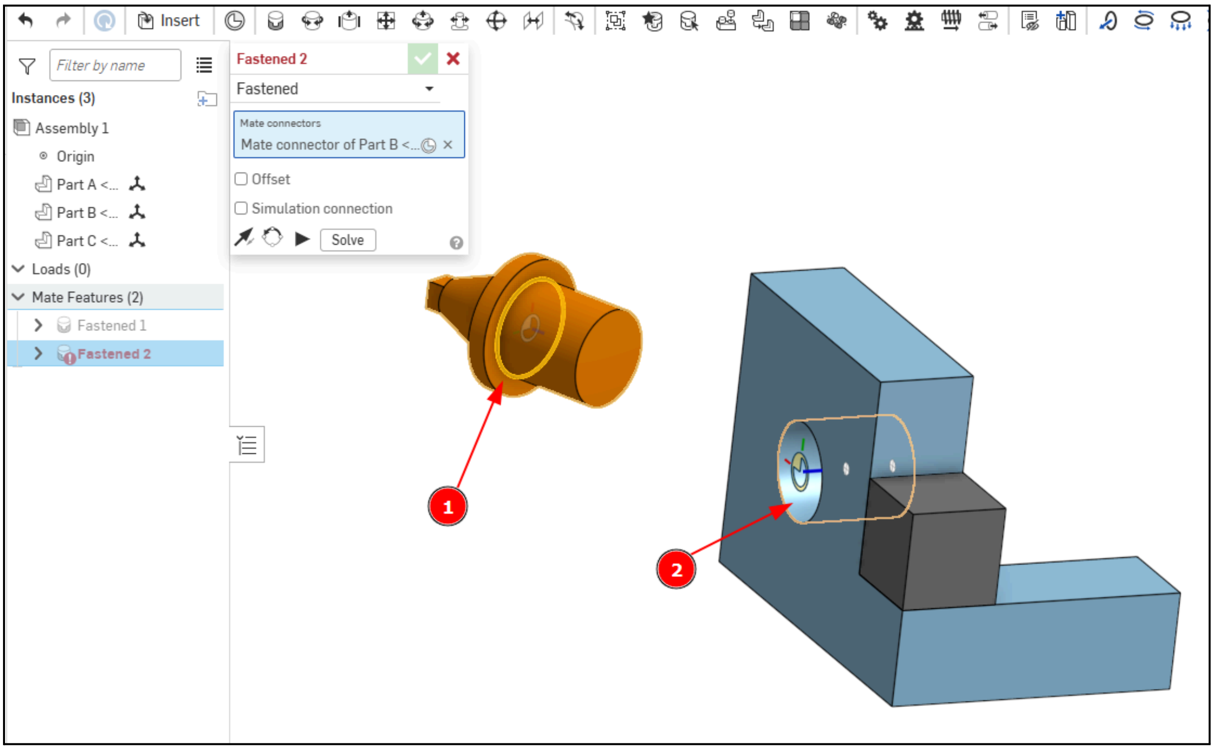
Click the **fastened mate** icon, then pick the **vertices** on **Part A** and **Part C** where we would like them to snap together. Notice that a circular gizmo will appear when you select the right location. You are also able to mate edges and faces as well.



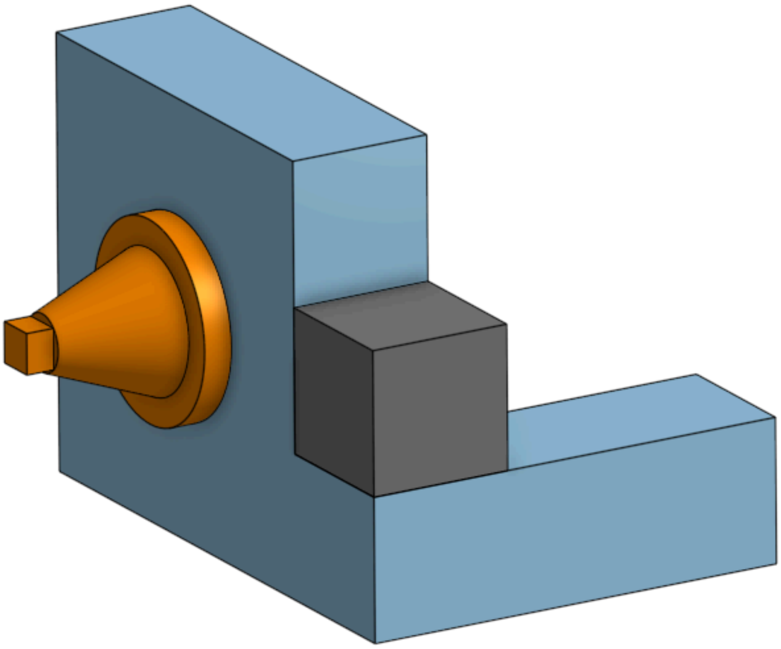
Sometimes, after the mate is applied, the part might be mated the wrong direction. We can flip that around by clicking the **flip primary axis** button. Click check to finish the mate.



Next, we will **mate** Part B's **circular edge** to Part A's **circular edge** together.



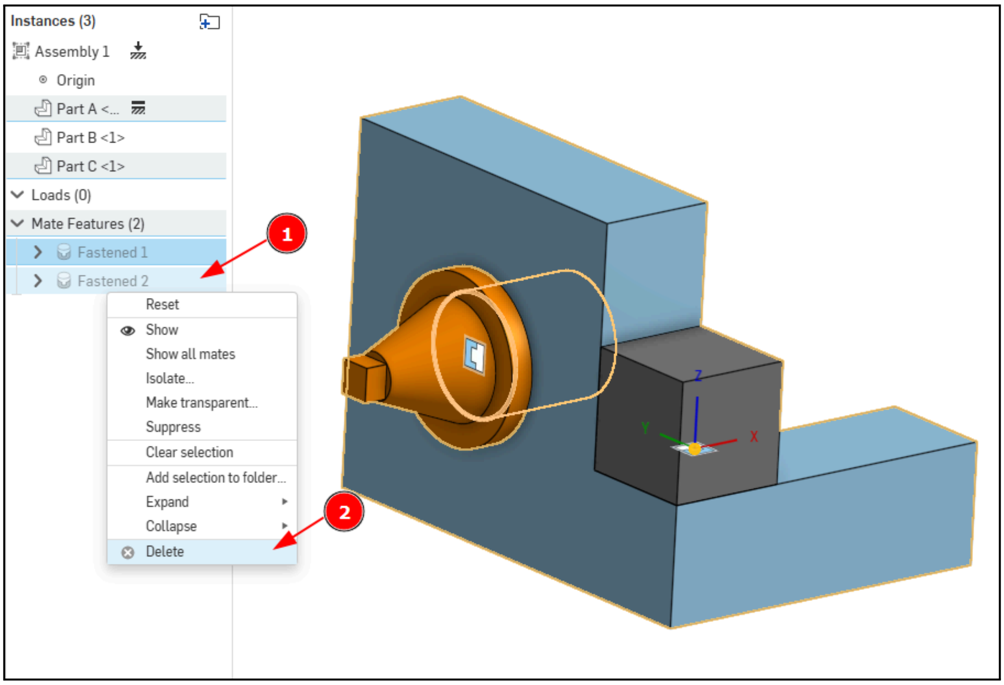
After the mate, everything should be fixed. You can try clicking and dragging on Part B and Part C, but no movement will be possible.



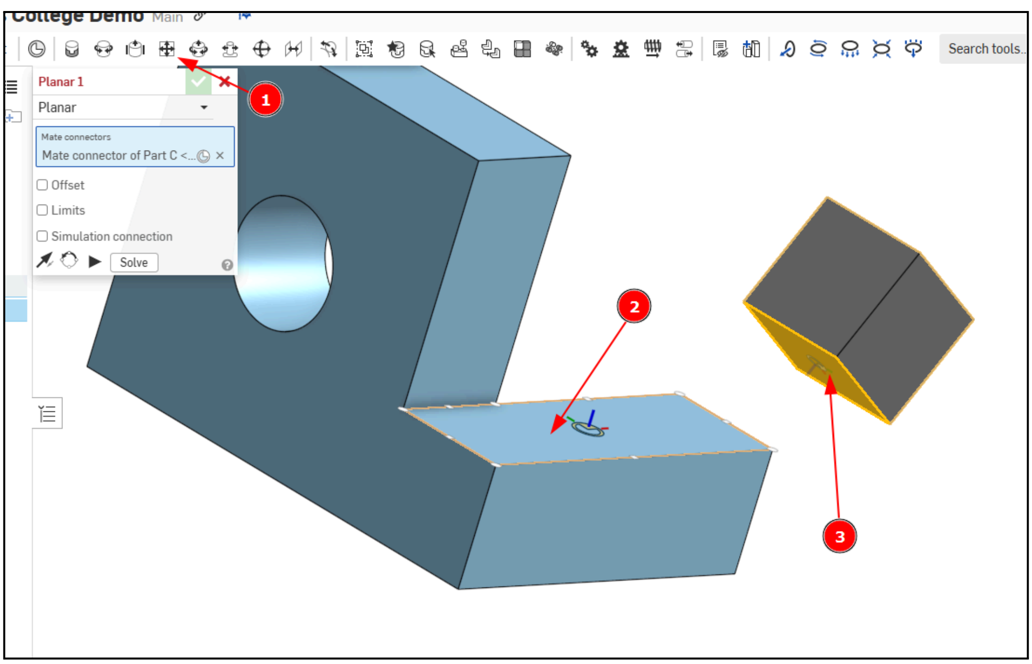
Planar Mate

The fastened mate is good for fixing parts together. However, what if we want to allow sliding movement? There are plenty of movement mates in Onshape. The **planar mate** is one that locks a part's surface onto another's.

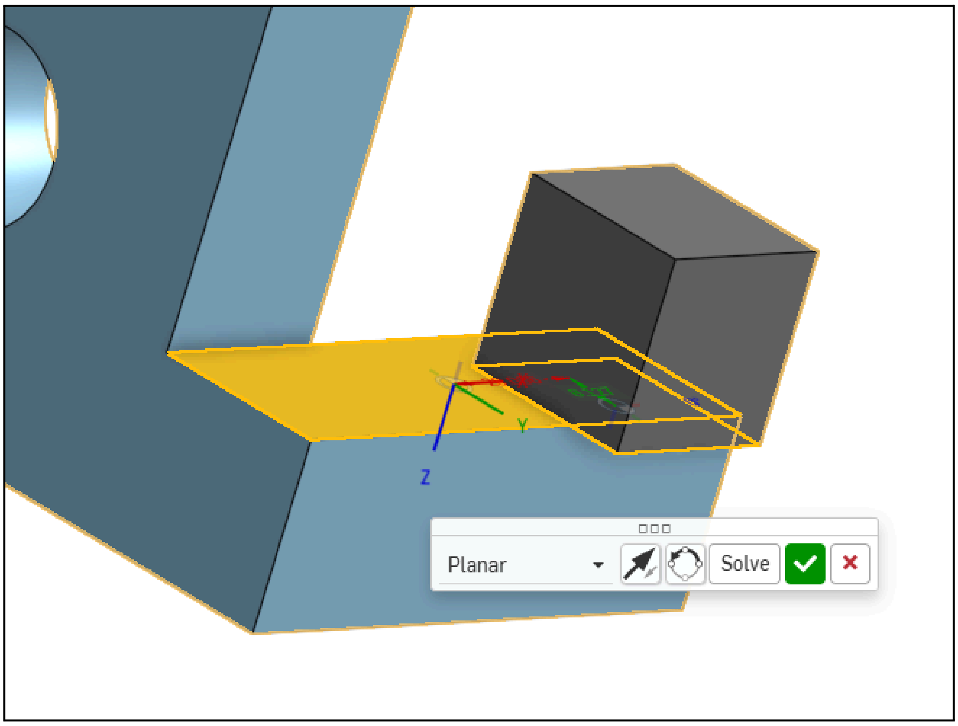
Start by removing the two mates that we added earlier by **right clicking** on them in the timeline and select **delete**.



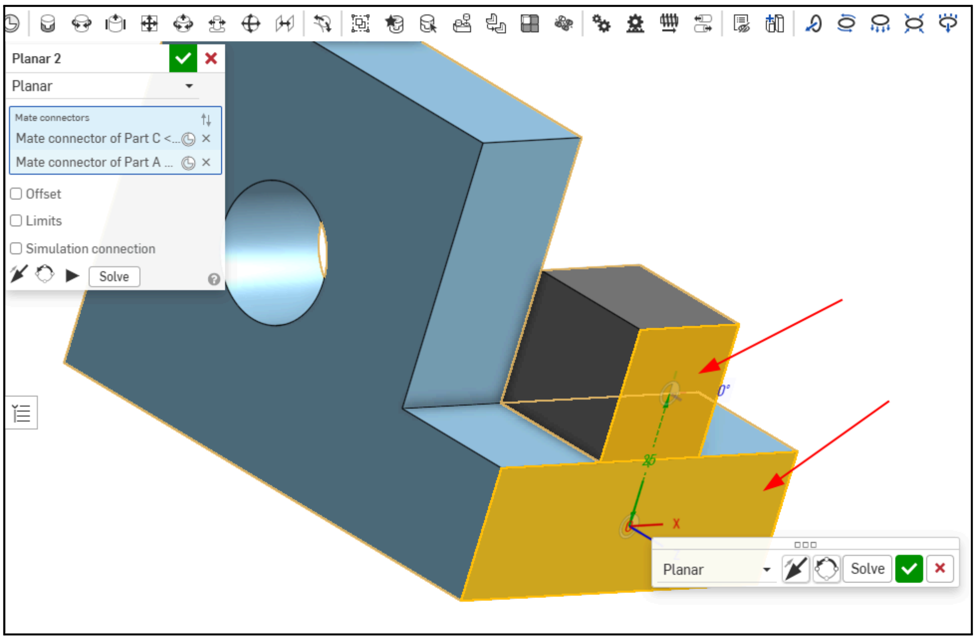
We will mate Part C onto Part A such that Part C can slide on the surface. Select the **planar mate** tool and then mate the two faces together.



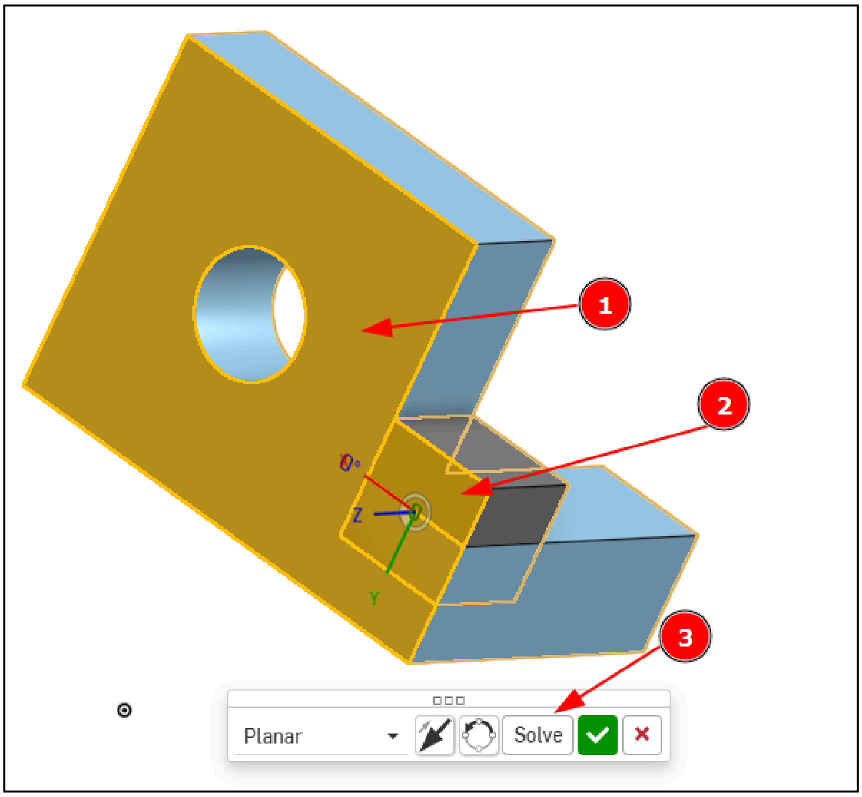
As you can see, the cube can now slide freely on Part A's arm. However, what if we want to limit the sliding motion to just the x-axis? We can add another planar mate.



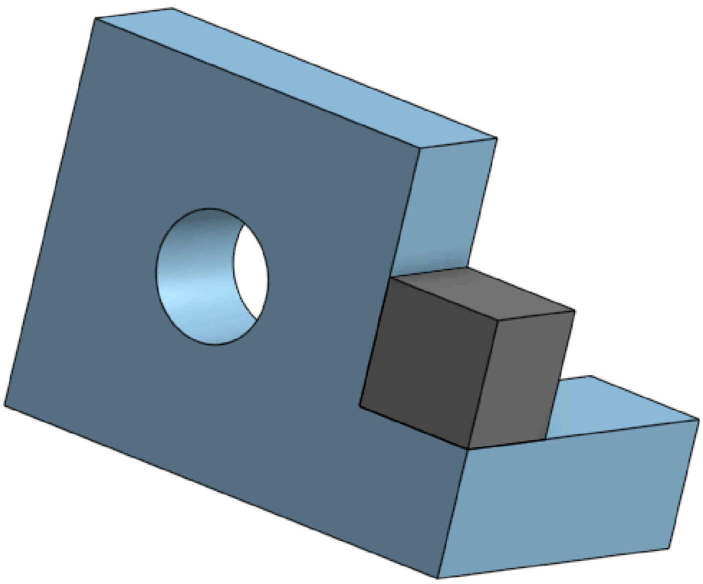
Next, add another planar mate on the vertical face of the Part A's arm and the cube's. This limits the cube's motion on the x-axis.



If we **add a third planar mate**, we can actually lock the cube in place just like before using the fastened mate. However, sometimes, the assembly will look like it satisfied the new mate but violated the old one, like shown below. You will need to click the solve button for Onshape to solve the geometry with all the mates.

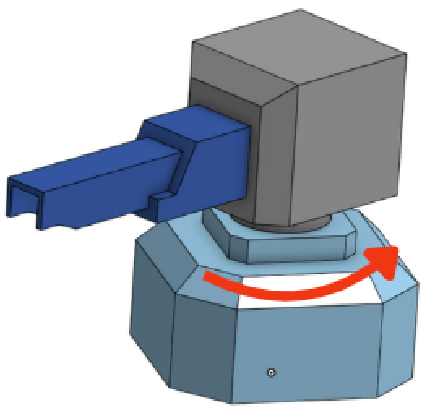


After that, the cube will be fully mated to Part A just like before without allowing any movement.

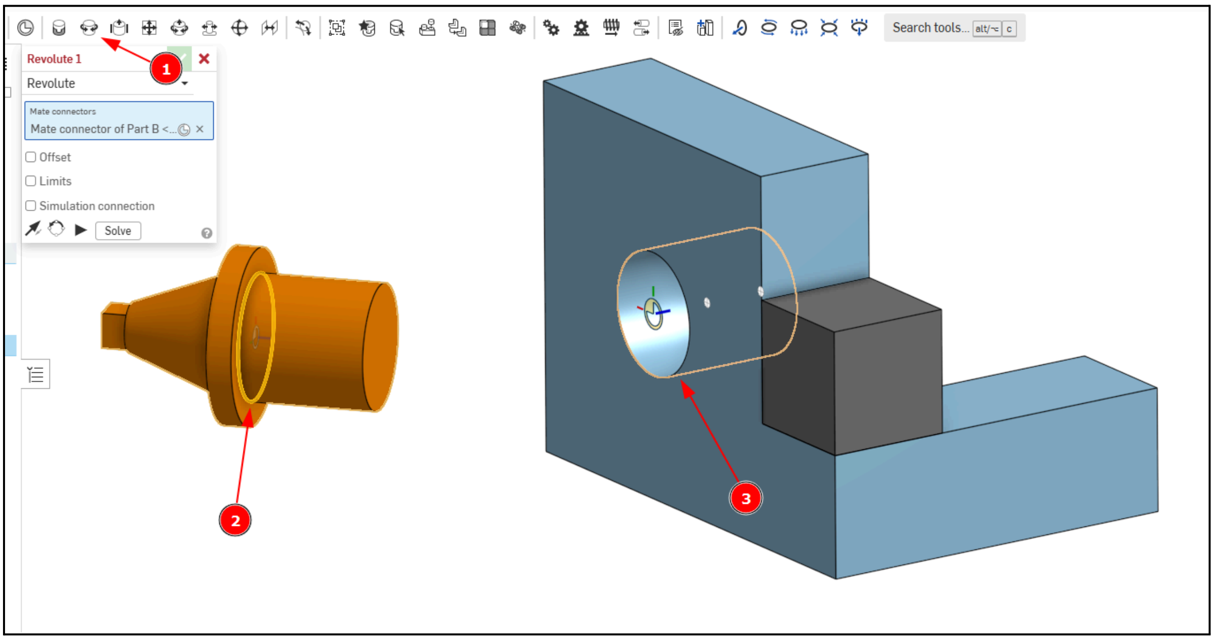


Revolute Mate

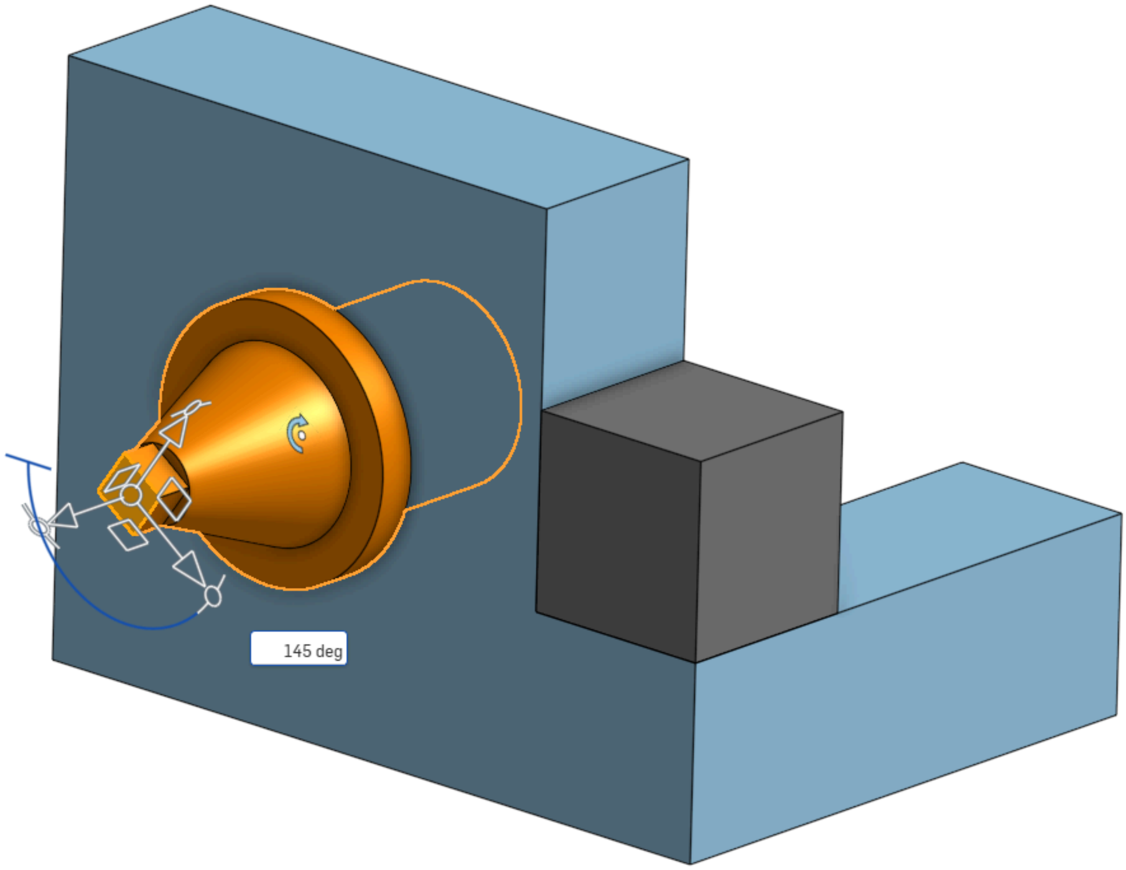
The revolute mate is for circular parts to mate with each other and allow for rotational movement. This is useful for situations like mounting a tank's turret to the body or a propeller to an engine.



Click on the **revolute mate** tool icon, and select the **circular edge** on Part C and the **hole** in Part A. This will estable the revolute mate.



After adding the revolute mate, you can rotate Part C along its rotational axis.



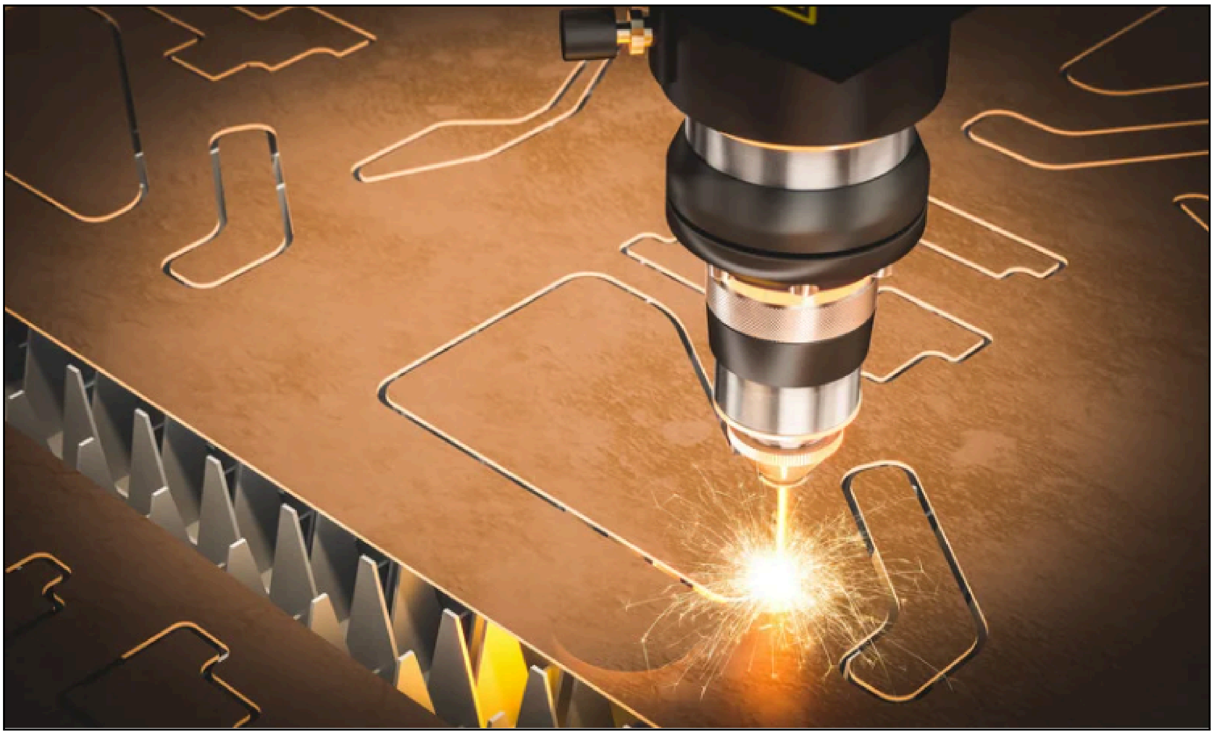
Design for Manufacturing

Now that we have learned the basics of CAD modeling and assembly, we have to discuss designing for manufacturability. We can is a super impressive design if we can't make it?

Manufacturability is limited by the machines we have access to. Oftentimes, makerspace will have laser cutters and 3D printers available. These machines are great for "quick" rapid prototyping. However, they come with some limitations that we have to keep in mind.

Laser Cutting

The laser cutter uses a powerful laser to make cuts and engrave in stock materials on a 2D plane. Laser cutting is relatively fast and inexpensive. Laser cutting service in mainland China cost around 45 RMB for designs that fit inside a 880 x 450 x 3 mm basswood sheet. Other materials like acrylics and metal are possible too.



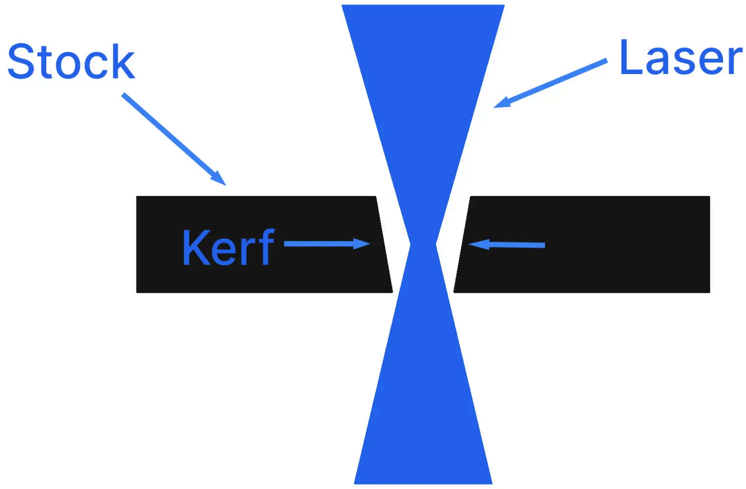
As mentioned before, laser cutters operate on a 2D plane. Therefore, they can only be used to create 2D parts. This is a severe limitation if you are designing for something in 3D. You will have to piece together multiple 2D parts like a puzzle to make it work.

The thickness of the stock materials also poses an issue. As laser cutters use heat to burn away materials, thick stock will take a long time to cut. If the laser is not powerful enough, or that the material is too thick, it might not be possible to cut through at all. The heat created from the burning operation can also cause the stock material to catch on fire.

When we design a square that is 10 by 10 mm, we would expect it to come out 10 x 10 mm after laser cutting. However, a cut is never as clean as we would like. During the cutting process, material is burnt away by the laser beam. The material removed in the cutting process is called the kerf. The kerf for laser cutter is around 0.08 to 1 mm depending on how focused the laser beam is and how fast the laser travels. The ticker the stock material, the more burn time is needed. Therefore, a 10 x 10 square might come out more like a 9 x 9 square, with 0.5 removed from each side of the path. You can also have burn marks around the material, though there are ways to mitigate it like using masking tape.

If you need an exact dimension for something like a press-fit joint, this could pose a problem. You will have to compensate by designing the square larger to a dimension like 11 x 11. The thickness of the stock material can also contribute to fitting issues. A 3 mm thick basswood sheet may vary between 2.6 – 2.8 mm due to material inconsistencies depending on supplier and environmental humidity level. This is a hard problem to address since it's less consistent.

It should also be noted that since a laser is a form of light, it does not do well with reflective materials. Things like glass and metal are often hard to cut and require special setup.



Credit: https://www.fabworks.com/blog/laser-cutting-kerf-explained

Waterjet

Waterjet is like a sister machine to laser cutting. They are very similar in how they work, except that waterjet uses water with abrasive particles like sand instead of a laser beam. The process does not create heat, but the material will get wet.

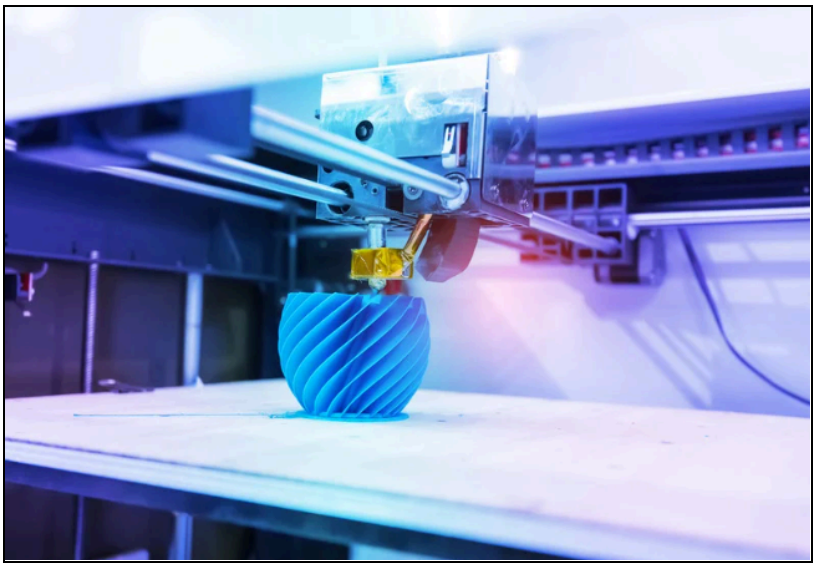
Waterjets are usually used to cut thicker materials, like metal. The edges they create are often smooth and cleaner. However, since a pressurized water beam is not as focused as a laser beam, the kerf is often wider at around 0.76 to 1.02 mm.

3D Printing

Laser cutting, waterjet, along with milling and lathe, are known as subtractive manufacturing. They work by removing materials. 3D printing is different, it is a form of additive manufacturing. 3D printing is relatively new to the manufacturing industry only becoming mainstream in the mid-2000s. In comparison, other computer controlled mills and lathe have been around since the 1960s.

3D printers work by having a printhead traveling along a 2D plane. The printer heats and extrudes plastic filament layer by layer to build up a 3D object. In recent years, it is even possible to use metal and even concrete as the printing material. But for most makerspace, you will only be able to print in ABS or PLA plastic.

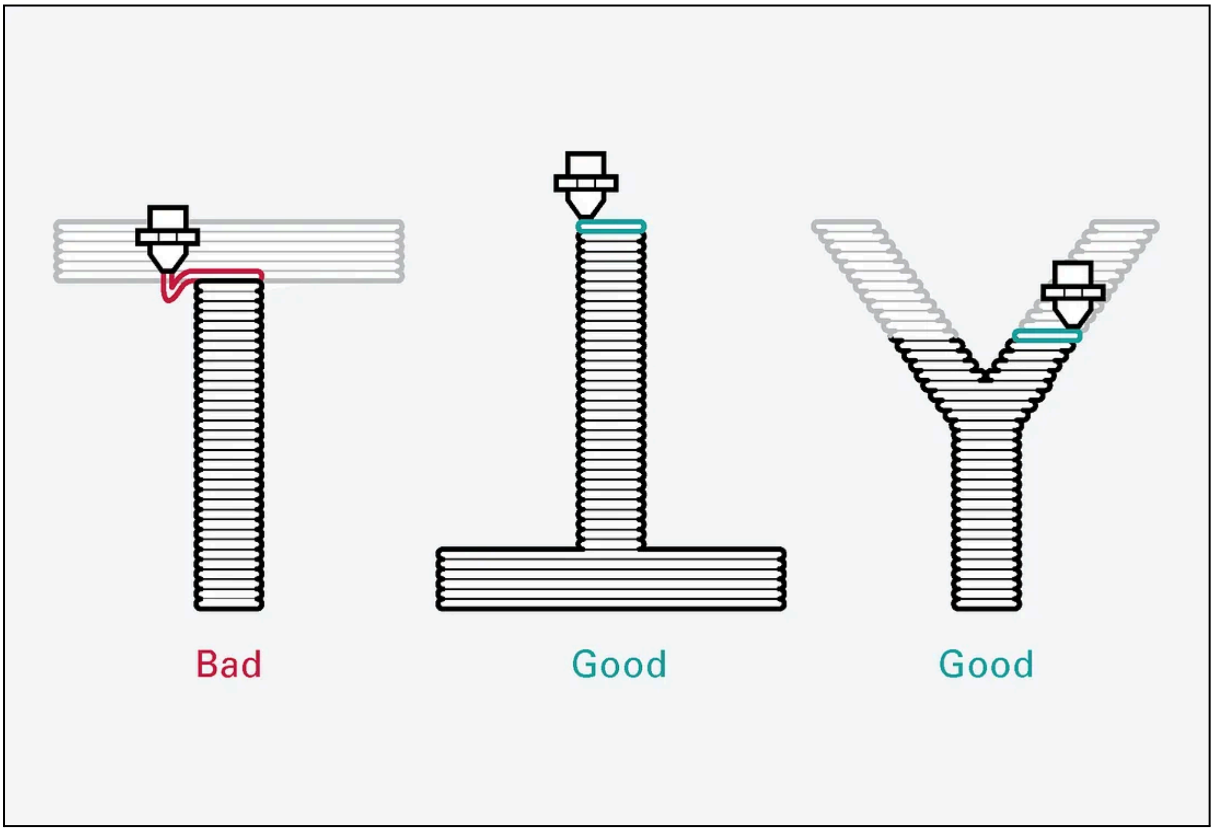
3D printers are great, you can take a CAD model and the printer will print it out in 3D. Though that do come with some major caveats.



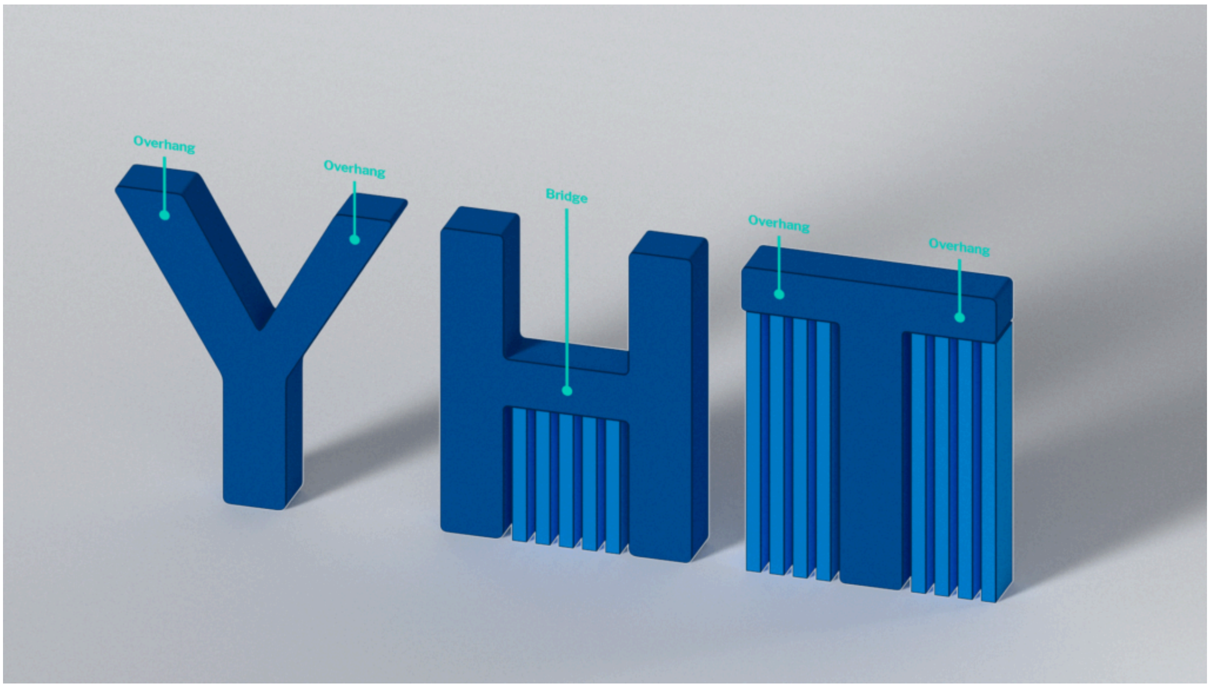
Since 3D printers add layers from bottom up, models with overhang features are problematic. Take a look at the images below. In the first model, the top beam of the T-shaped model is problematic because nothing is supporting the overhang. As the printer extrudes filaments, the plastic will sink due to gravity. The result is a spaghetti mess. Similarly, features like holes are affected by this issue as well.

Designers can attempt to resolve this by reorienting the model so that the overhang is eliminated by it resting on the print bed. Alternatively, designers can add columns as support for the printed filaments, though that will need to be removed later. It is best to keep this limitation in mind during the design process, as adding support doesn't always work and the clean up is quite time consuming and messy.

3D printed models are often weak and brittle. The model tends to have rough surface and inaccurate dimension due to the nozzle’s extrusion diameter. The printing process itself is also quite time consuming. It takes around an hour for a small simple object, and could take hours or days for something bigger.



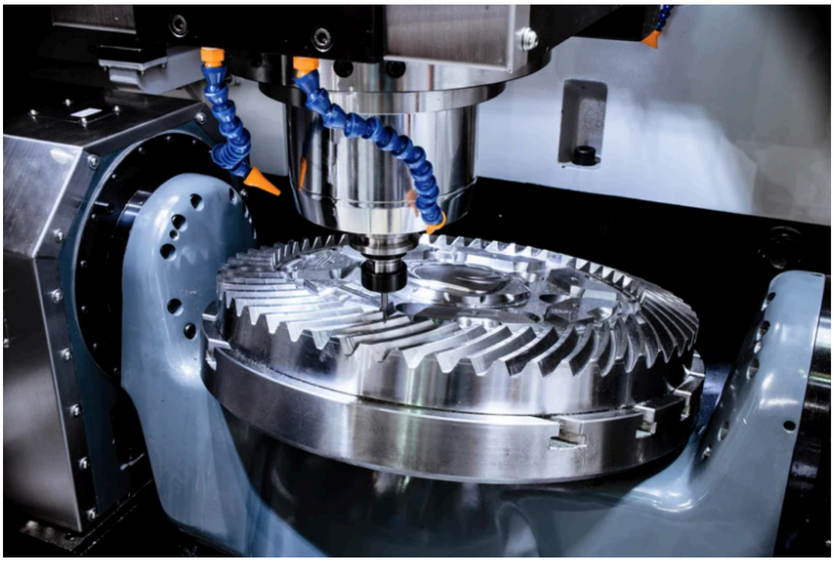
Credit: https://medium.com/@ahrengantri/overhangs-overview-e1c5b6c9563b



Credit: https://sybridge.com/support-structures-why-they-matter-and-how-to-design-for-them/

CNC Milling

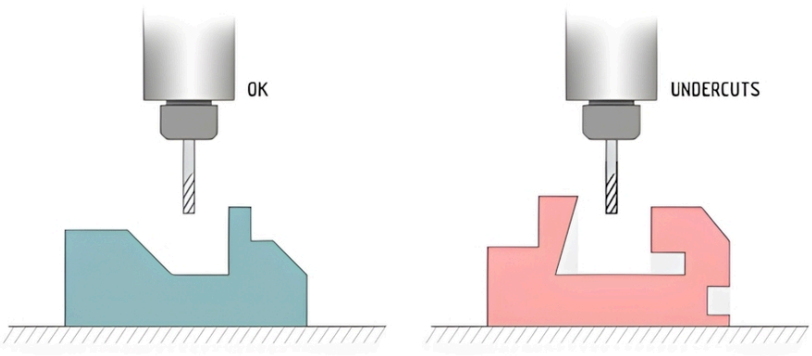
Since the 1800s, workers have been using milling machines to produce parts. The mill has a rotating cutting tool, called a mill bit, that move around to remove materials from the stock. Usually the stock material is a solid block of metal, wood, or plastic. Workers had to use gears to move the stock and cutting tool around manually to manufacture the part according to drawing specification. Nowadays, we have computer numerical control (CNC) milling machines that produce parts automatically from computer generated toolpaths from CAD models.



Since a mill works by physically removing material using an end mill, the operation is often fast, much faster than 3D printing. The resulting product is also quite strong and smooth. The kerf is determined by the end mill's diameter. The computer-aided manufacturing (CAM) software will take this into account when generating the toolpath.

Traditionally, mills can only cut from one side, therefore, overhangs are also problematic similar to 3D printing. The machine simply can't cut something underneath, and adding support is not an option. This is one big limitation. If the undercut is required, it might be possible to achieve that by rotating the part, or the cutting tool, to perform a separate cutting operation from another angle. Most of the time, this has to be done manually. More advanced milling machines are able to do this automatically, but they are expensive.

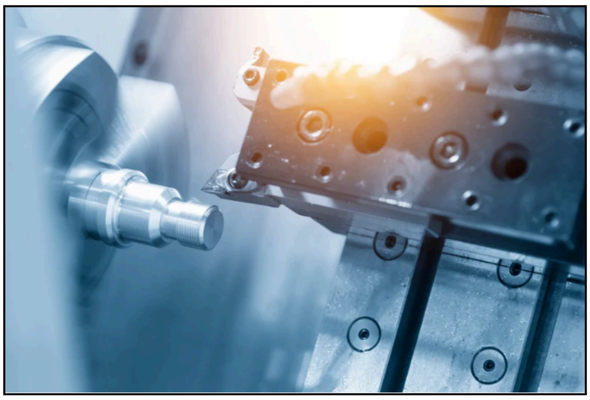
Another thing to keep in mind is that the part's feature is limited by the end mill's diameter. If the cutting feature is supposed to be small, the machine will need to use a smaller end mill. But this poses a risk since the tool can break. In addition, it will take a long time to cut bigger features, therefore a tool swap is required.



Credit: https://at-machining.com/undercut-machining/

CNC Lathe

The CNC lathe is designed to produce parts that are symmetric along its rotating axis. In a way, models designed from the revolve operation can be produced using a lathe. Similar to a CNC milling machine, lathe work by removing materials using a cutting tool. One major difference is that the stock material is rotating to create a symmetric part.



File Export for Rapid Prototyping

The most common rapid prototyping machines in makerspace are the laser cutter and the 3D printer. Here, we will learn how to export our CAD model for production.

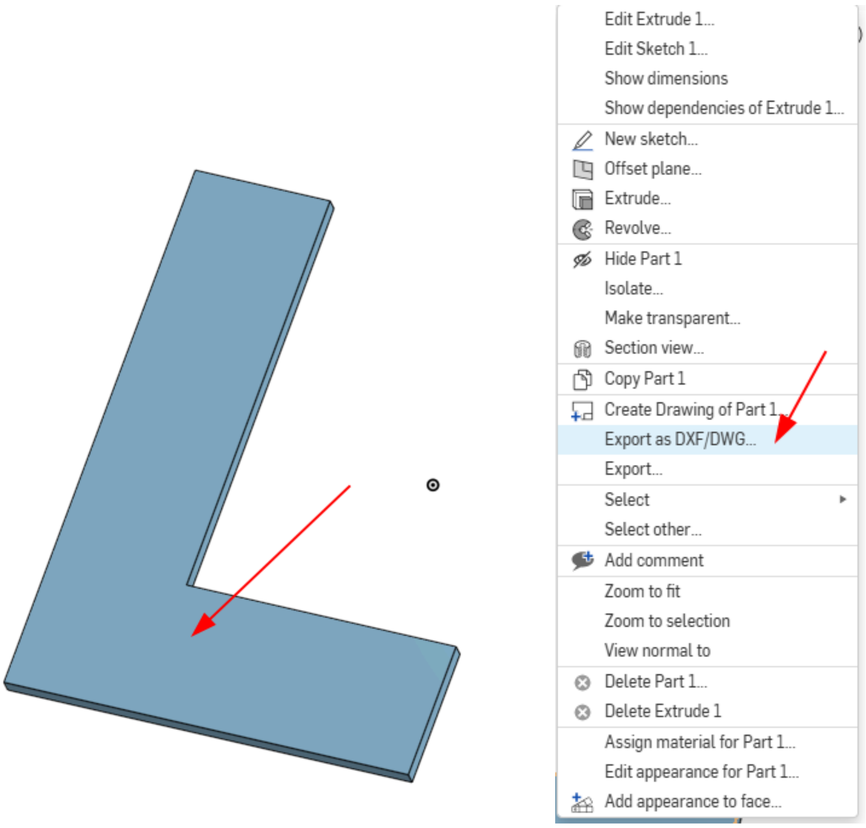
Laser Cutter (DXF)

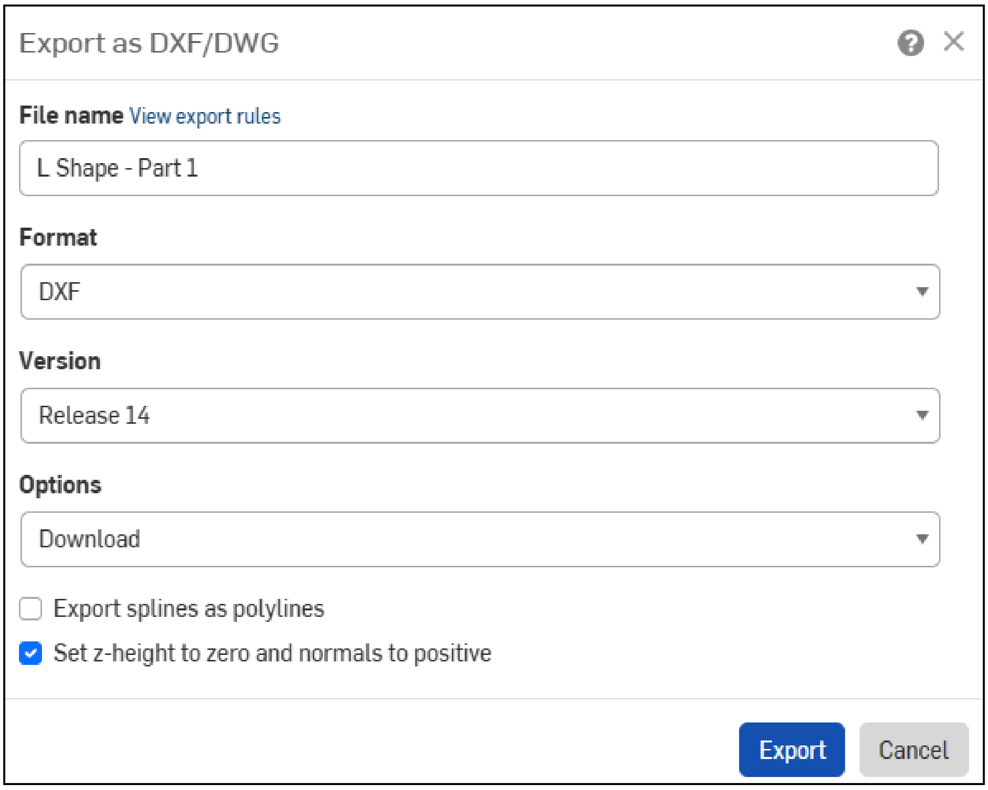
Inspecting the models that we had created thus far, you will realize that none of them are suitable for laser cutting. Part A and Part B both contain 3D features, hence, it is not possible to produce with laser cutting. The L-shaped geometry and the Part C cube are both way too thick for laser cutting.

We will have to **modify** the **L-shaped geometry** so that the **thickness** is around **3 mm**.

Right click on the face on which you will perform the laser cut and select **export as DXF / DWG**. Select the DXF format. This will export a 2D path file for the laser to cut or engrave.

Usually, you will have to post-process the DXF file in a vector editing software, like Inkscape, Illustrator or CorelDRAW. Mark all the paths as hairline width, and then change the cutting lines and engraving lines to different colors. You will then have to communicate with the vendors and let them know the color designation of each operation. You can refer to Appendix III for reference.




Export as DXF/DWG
File name View export rules
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Download
Export splines as polylines
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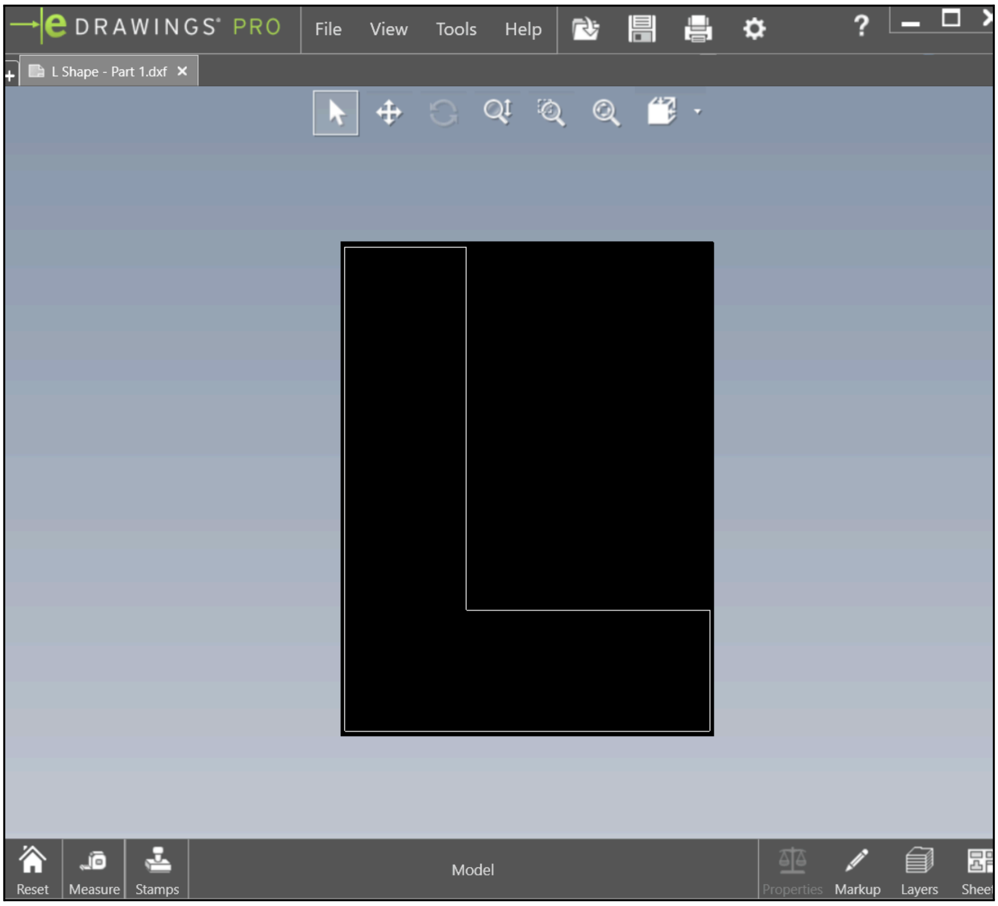
DRAWINGS PRO
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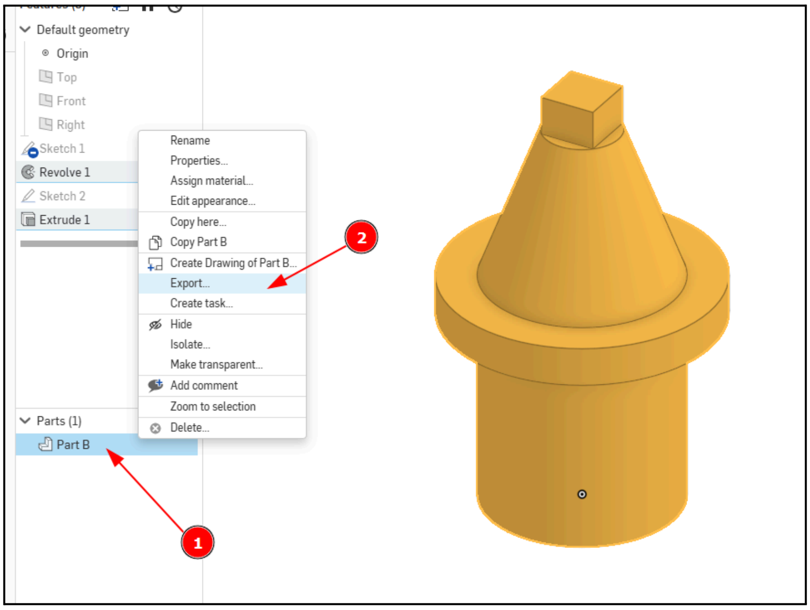
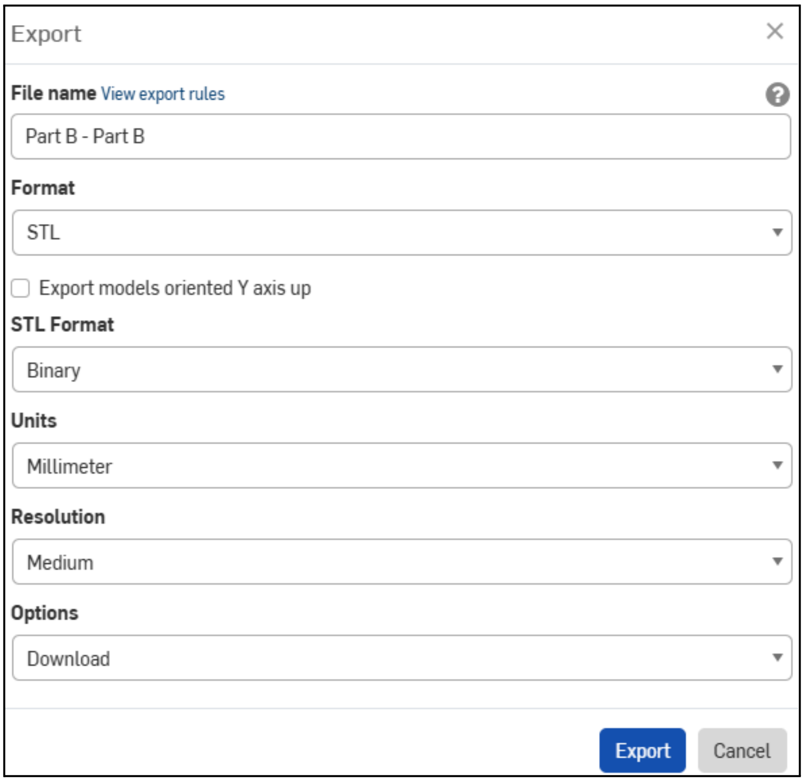
Markup
Layers

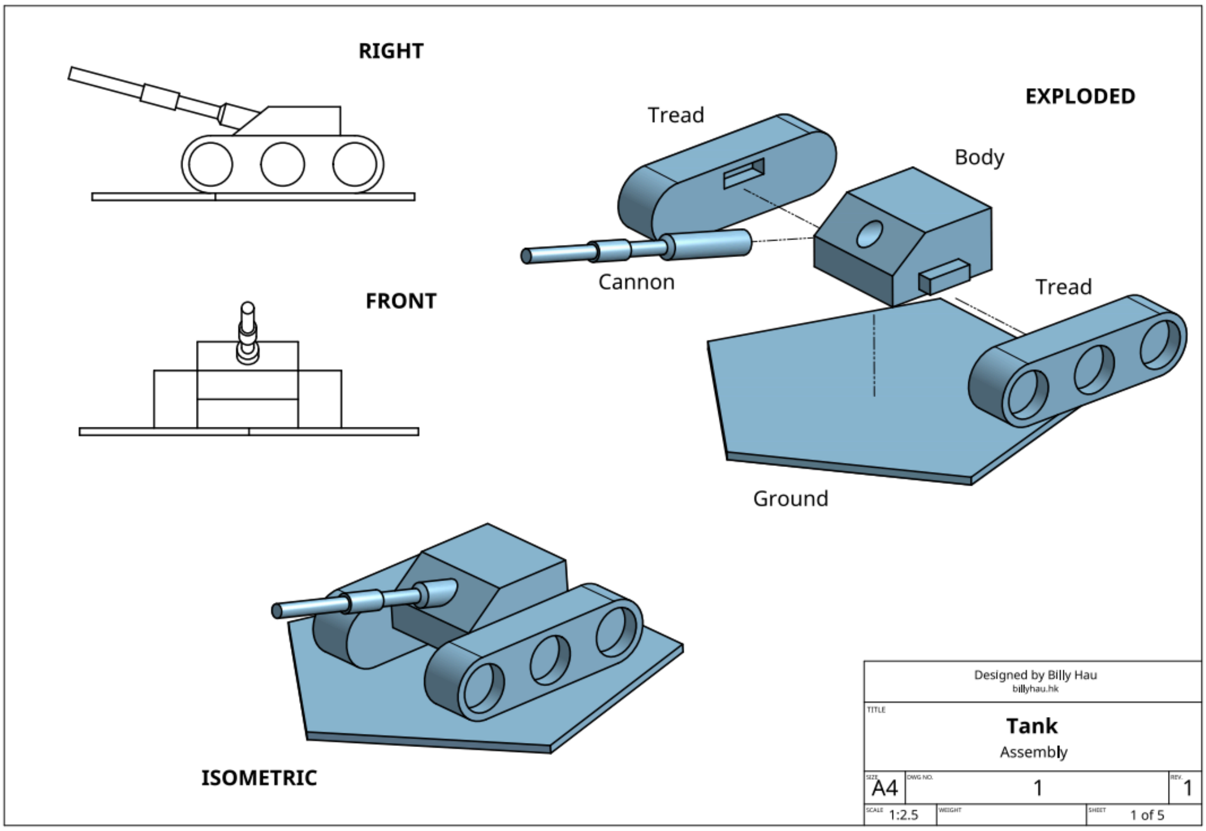
3D Printer (STL)

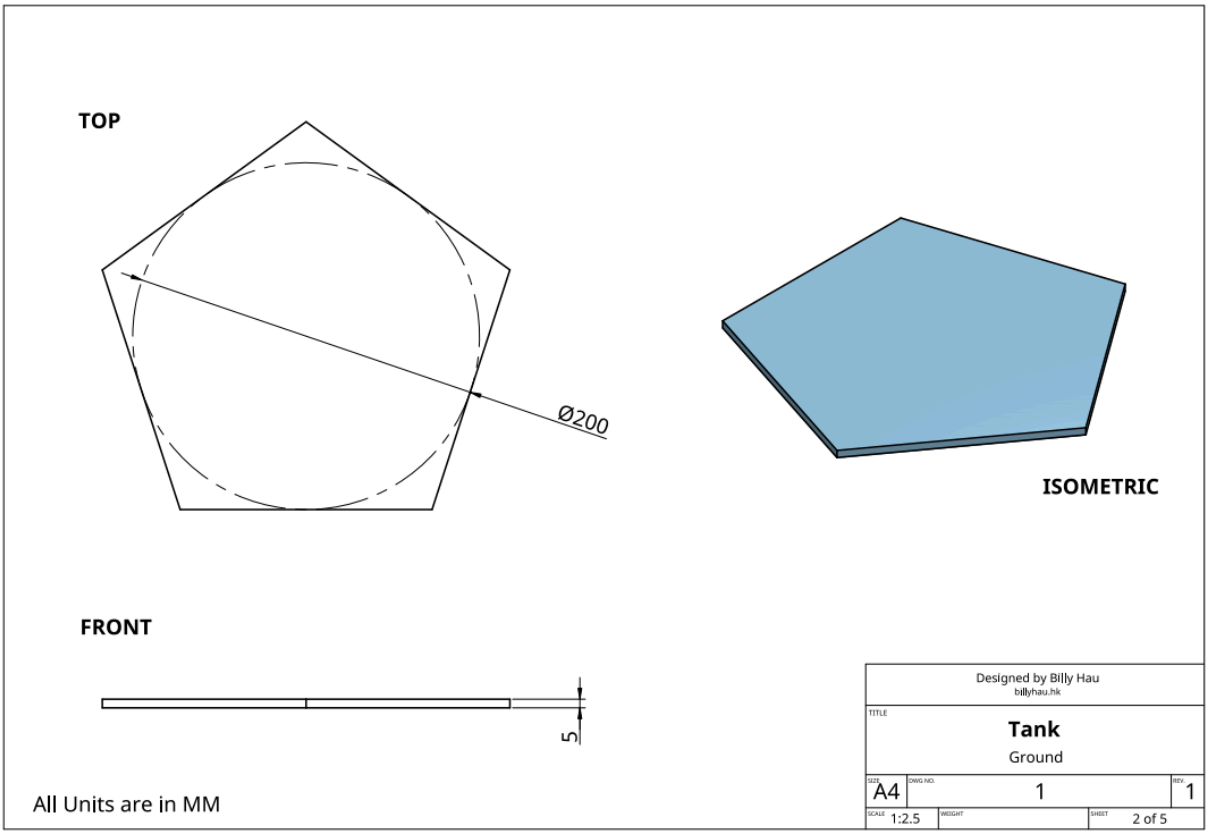
To export for 3D printing, open the desired part. From the parts menu, **right click** on the part, then select **export**. Make sure that the format is **STL** and the unit is **Millimeter**. You will then bring the STL file into the 3D printer's software. The software will then slice the model into layers and generate the filament extrusion path to print the model.

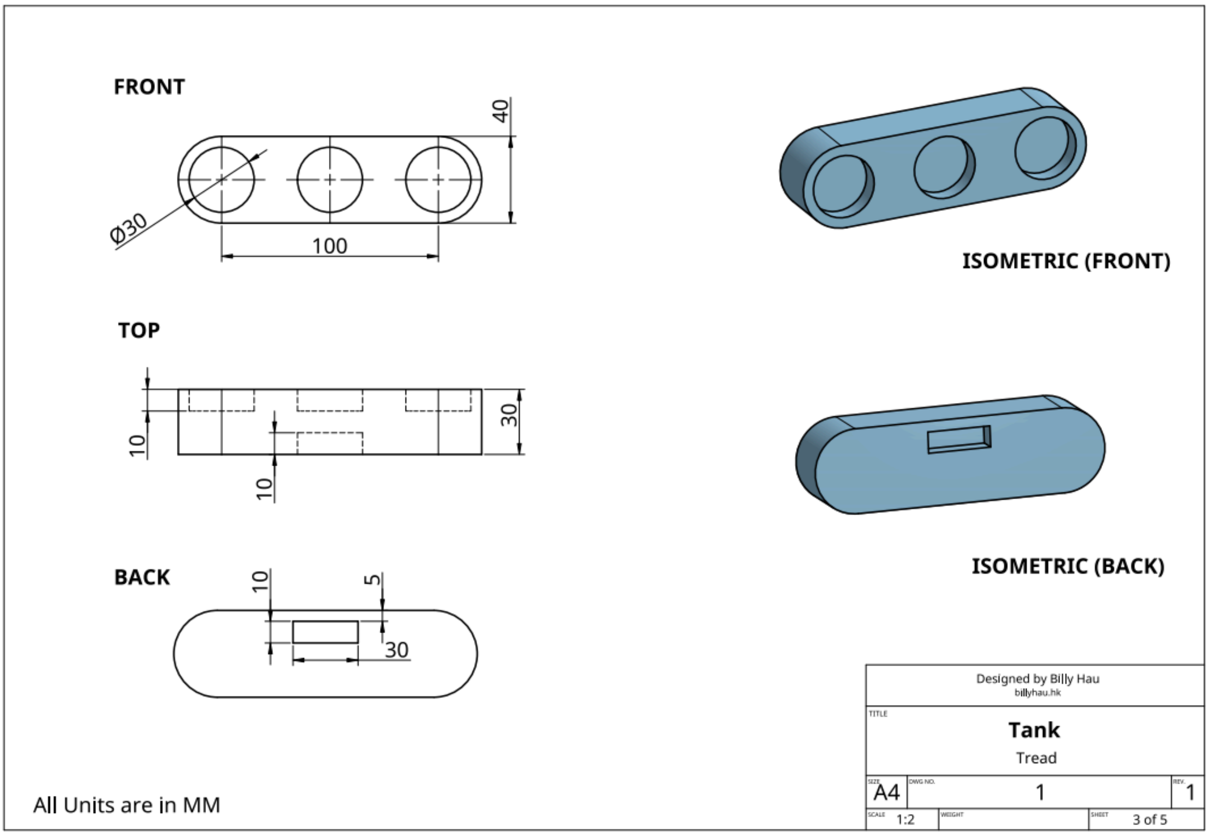




Project 1 - Tank






FRONT
40
Ø30
100
TOP
10
30
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BACK
10
5
30
ISOMETRIC (FRONT)
ISOMETRIC (BACK)
All Units are in MM
Designed by Billy Hau
billyhau.hk
TITLE
Tank
Tread
SIZE
A4
DWG NO.
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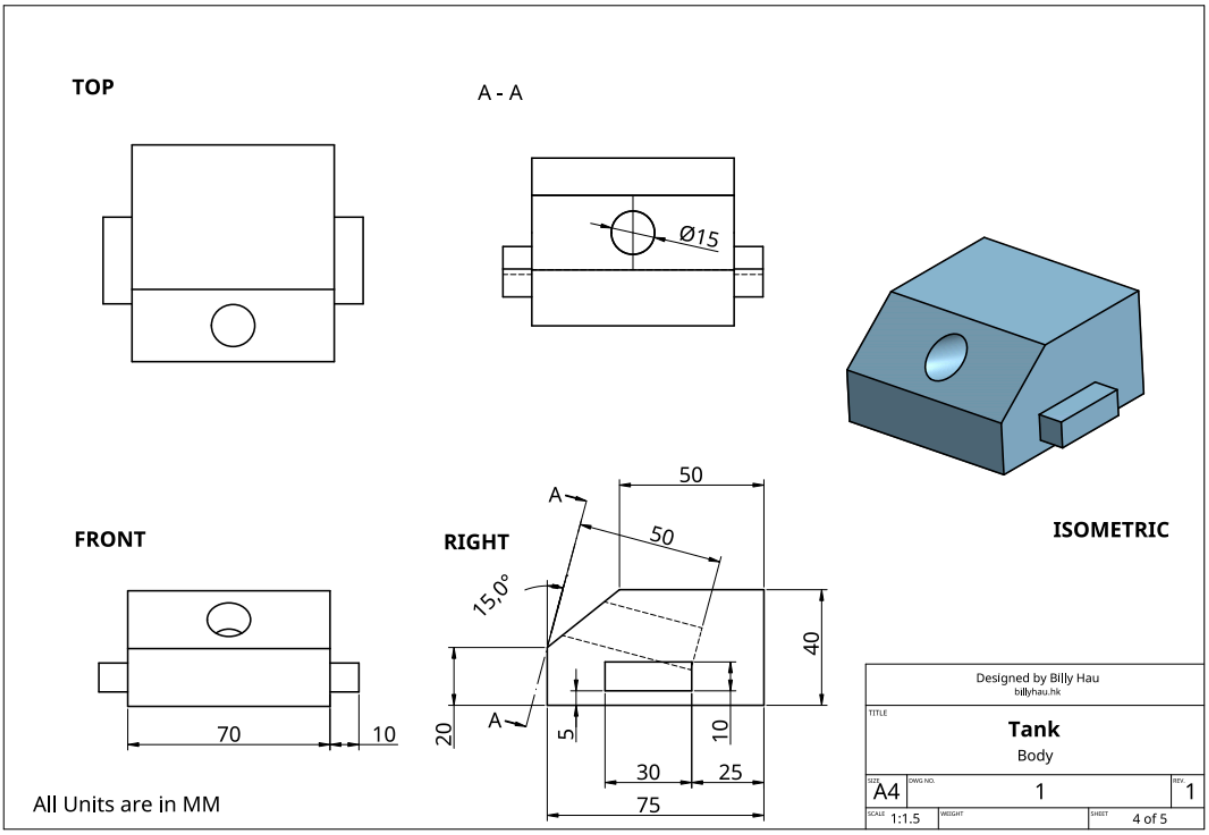
SCALE
1:2
WEIGHT
SHEET
3 of 5


TOP
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Ø15
ISOMETRIC
FRONT
70
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RIGHT
A
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50
15,0°
40
20
5
10
30
25
75
All Units are in MM
Designed by Billy Hau
billyhau.hk
TITLE
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Body
SIZE
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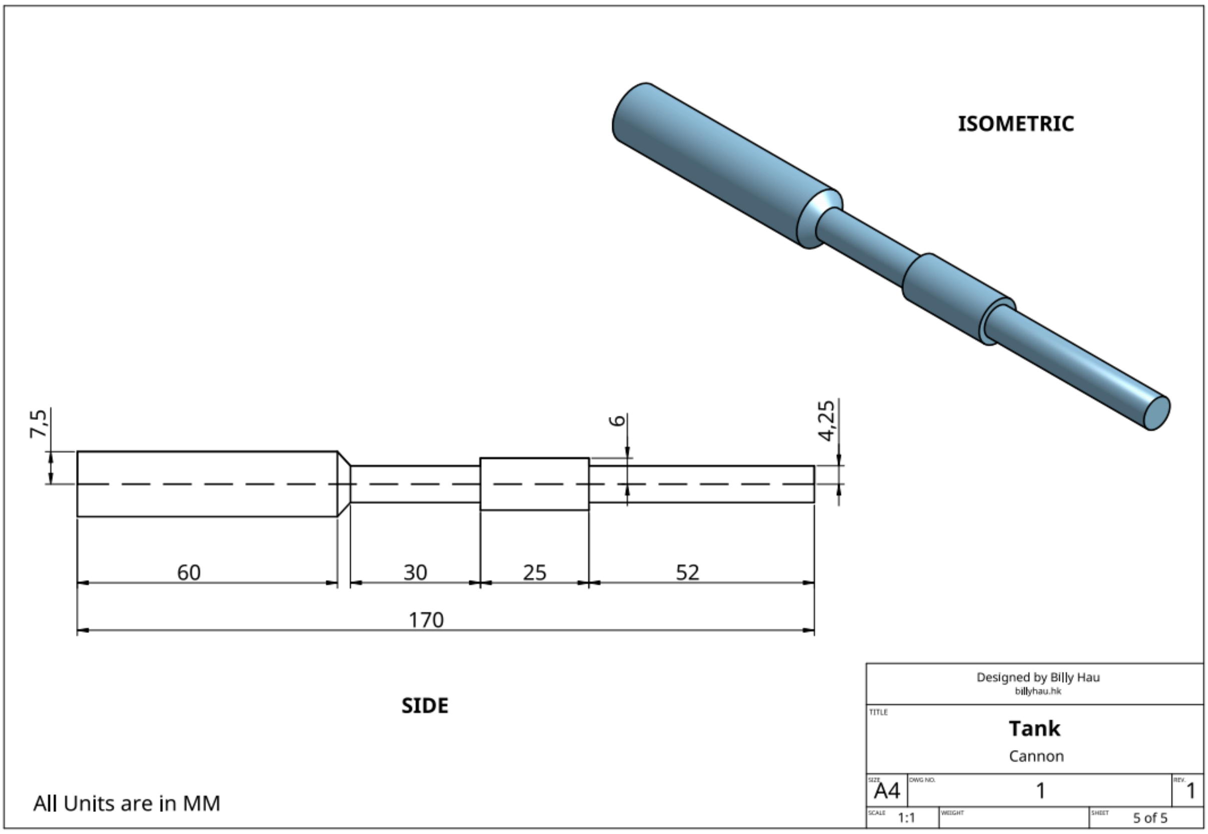
DWG NO.
1
REV.
1
SCALE
1:1.5
WEIGHT
SHEET
4 of 5


ISOMETRIC
7,5
6
4,25
60
30
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SIDE
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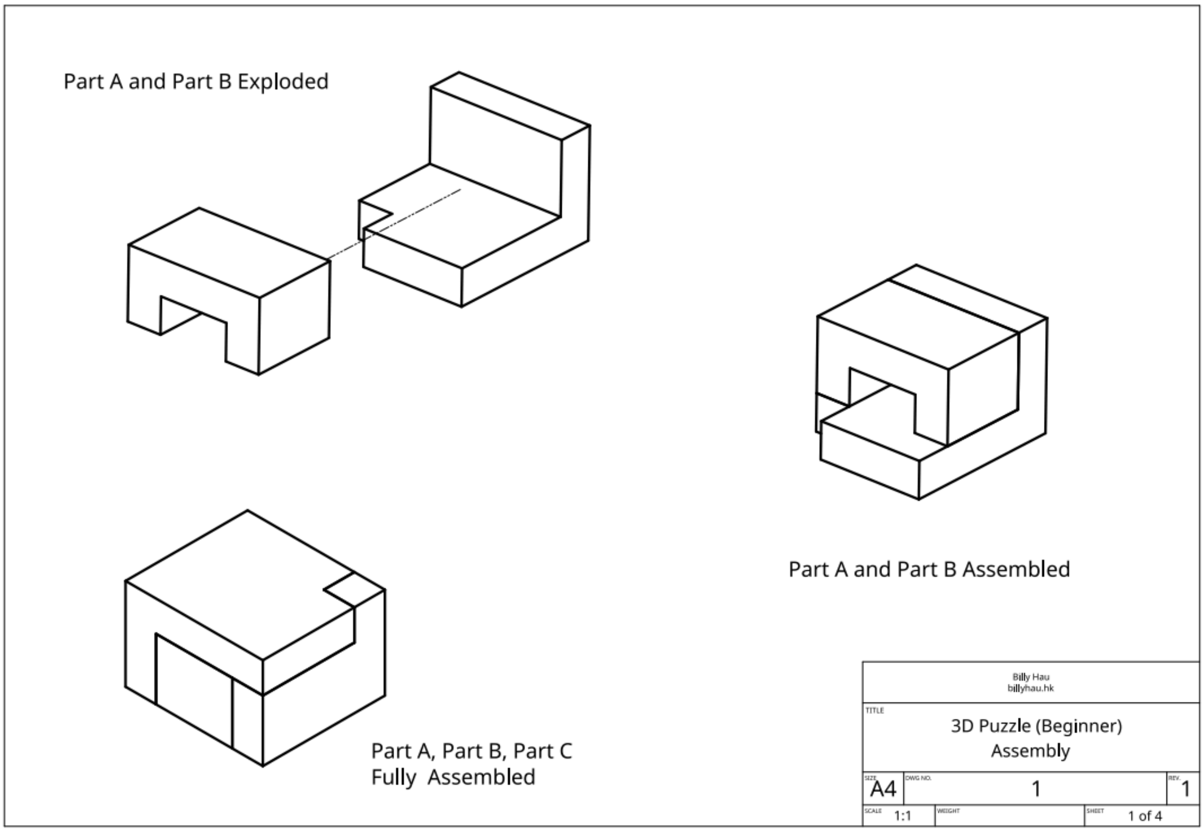
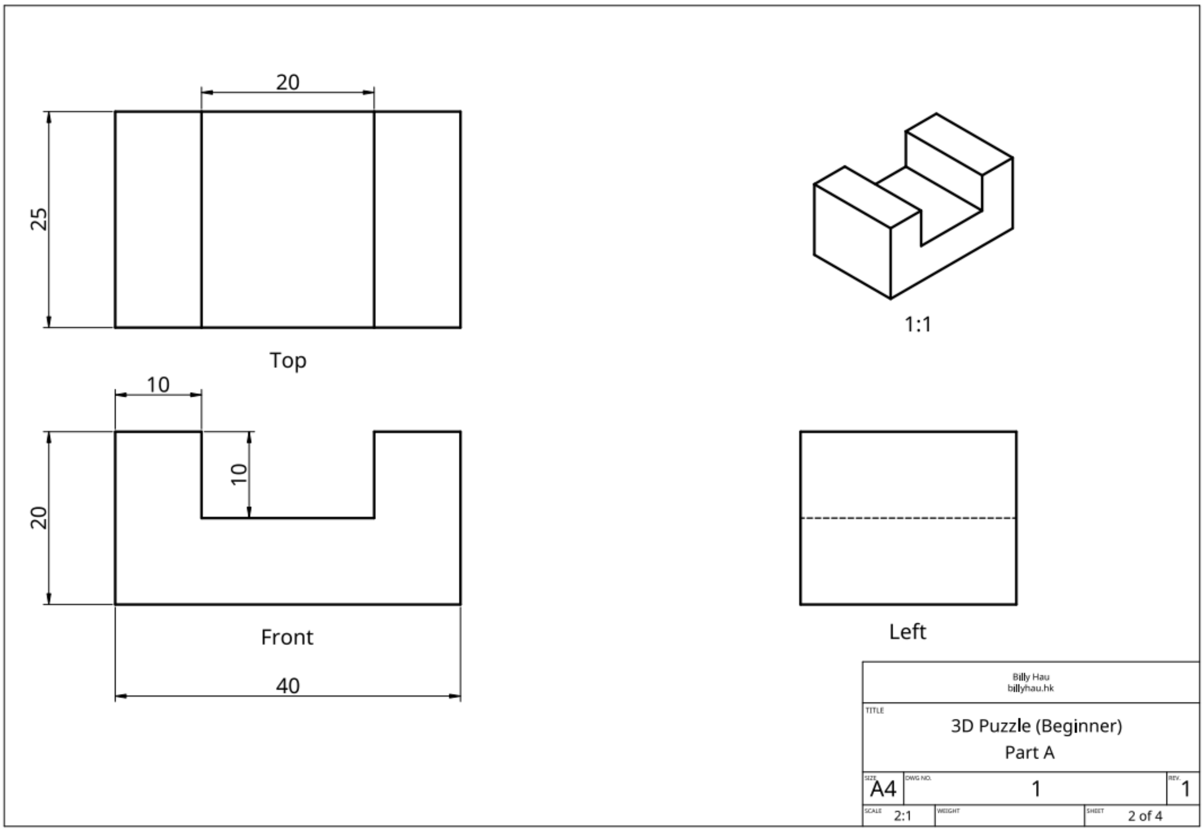
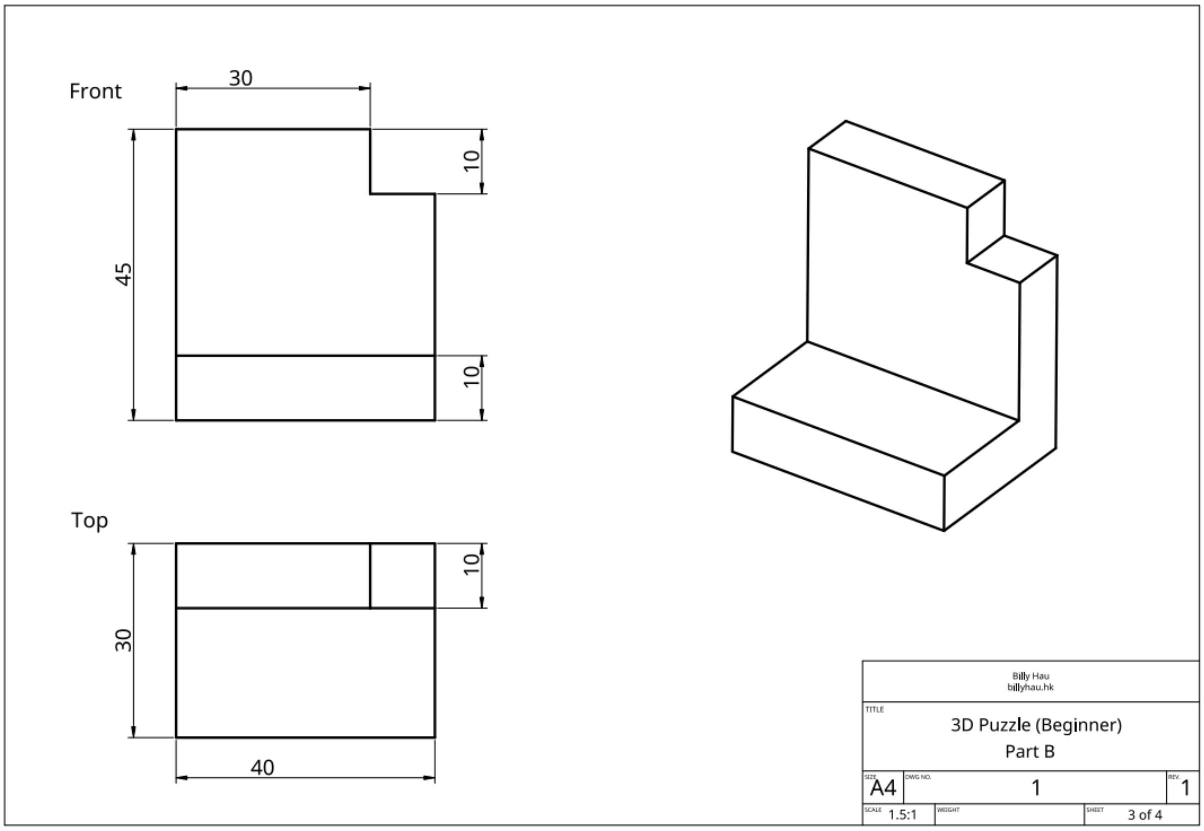
Designed by Billy Hau
billyhau.hk
TITLE
Tank
Cannon
SIZE
A4
DWG NO.
1
REV.
1
SCALE
1:1
WEIGHT
SHEET
5 of 5

Project 2 - 3D Puzzle







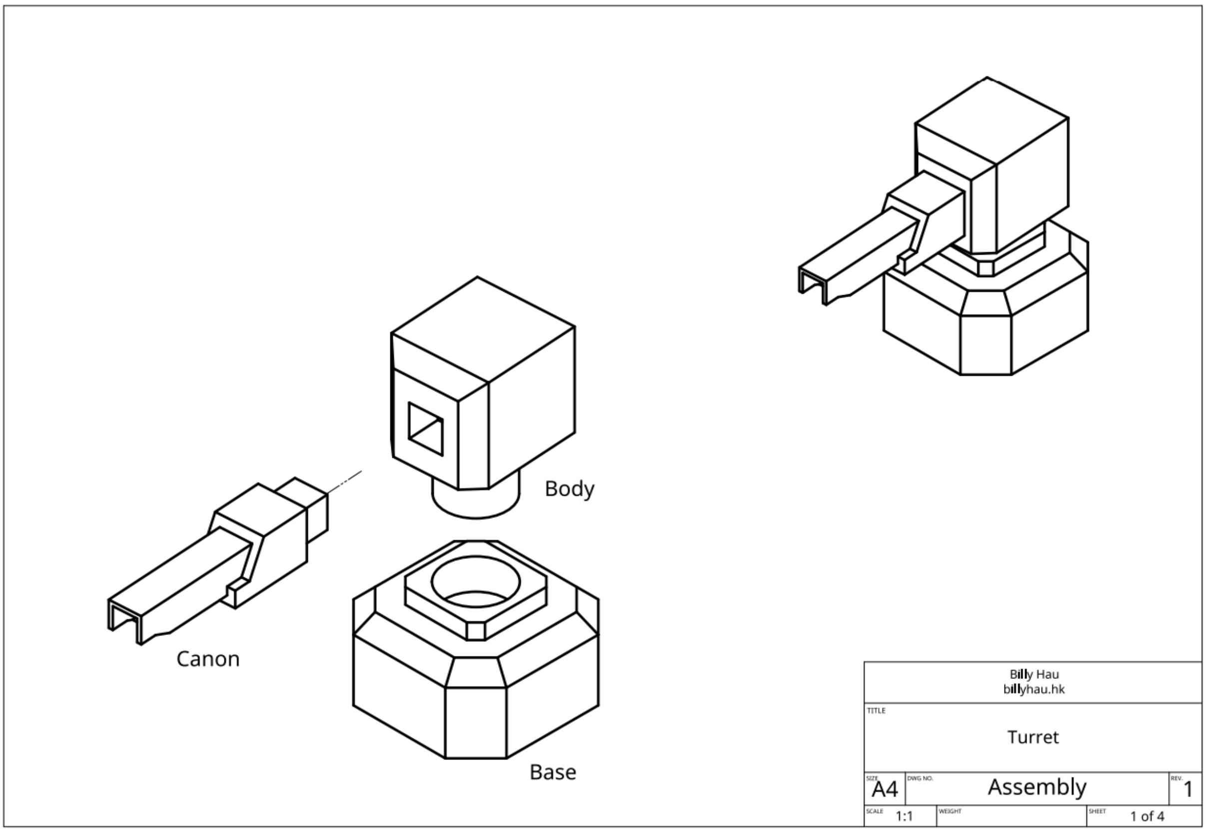
For this exercise, only drawings for Part A and Part B are given. You will have to imagine what Part C looks like and model it in CAD. Create an assembly so that all three parts form a perfect block together.

Additional Learning Resources

Thank you for attending this course. It was a pleasure to introduce you to the world of 3D modeling! To keep going on your CAD journey, you will need to continue learning and practicing. Here are some resources that you can use.

The Onshape Learning Center is a good place to start:
https://www.Onshape.com/en/education/courses-curriculum

Furthermore, you can find more CAD project blueprints and other STEM related learning materials on my website:
https://billyhau.hk



Appendix

Appendix I - CAD Software Packages

Software	Developer	Type	Target Audience	Impression
LibreCAD	Open Source	2D	Hobbyist	Free, but a bit hard to use. Suitable for verifying drawings.
AutoCAD	Autodesk	2D	Professional	Free education license, industry standard for 2D drafting and being used heavily in architecture. It is also used for viewing 2D engineering drawings for manufacturing.
Tinkercad	Autodesk	3D	Student	Free and good for learning the basics. Ability simulate electronics is cool too, but they are not really integrated.
SketchUp	Trimble / Google	3D	Student / Hobbyist	Free and good for quick 3D sketches to convey conversational ideas.
FreeCAD	Open Source	3D	Hobbyist	Not recommended, free but not user friendly and can be unstable.
SolidWorks	Dassault Systèmes	3D	Hobbyist / Professional	Industry standard for small and medium-sized engineering projects. Widely used in university programs and also the industry for some quick engineering design, CubeSats and mobile robots for example. Integrated motion analysis, thermal and material stress / deformation simulation packages. Quite user-friendly, yet feels more formal like other enterprise level software. Student and Maker license: ~$50 USD. A professional license is quite expensive, starting at ~$2800 with the base standard version.
Fusion	Autodesk	3D	Hobbyist / Professional	Free for students and non-commercial use. Integrated manufacturing tool-chain like CNC toolpath gcode generation. Modeling flow is quite free like Tinkercad, which encourages liberal design rather than precision. This could be good or bad depending on the use case. The software is semi web-based, but quite slow and buggy at times.

Onshape	PTC	3D	Hobbyist / Professional	Free for education and non-commercial use. Modeling flow feels more traditional, like SolidWorks, and puts emphasis on precision. Completely web-based and run smoothly. Generally user-friendly, with the exception of mating in assembly and setting geometric constraints. They are not as intuitive as others. Missing advanced simulation and manufacturing features like others. But this is probably the best option available to students, since it's free and has virtually no setup.
Inventor	Autodesk	3D	Enterprise	The "new kid on the block" in terms of enterprise level CAD software. While others are like 40 to 50 years old, Inventor is around 25 years old. Therefore, it is not widely used since large enterprises are mostly already committed to other software while there are better options for smaller engineering companies.
NX	Siemens	3D	Enterprise	"Old school" industry standard CAD software that has a long legacy and is being used in large enterprise engineering projects. Very not user-friendly and unintuitive. Most engineers use it because their companies have used it for a long time and they are being forced to use it.
CATIA	Dassault Systèmes	3D	Enterprise	Industry standard and mostly used in the automotive and aerospace engineering sector. It is the "big brother" of SolidWorks, but the emphasis is on large engineering systems with lots of complex parts rather than user friendliness.
Creo | Pro/E	PTC	3D	Enterprise	Industry standard, used in large engineering enterprises and agencies like NASA. But it's very hardware demanding and runs quite slowly if your computer is not top-tier. There's a running joke that it requires a supercomputer to open a model. Unintuitive and not user-friendly, but not as bad as NX. It's built for designing large engineering projects with lots of subsystem parts, like rockets or airplanes.

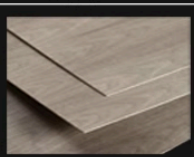
Appendix II - Related Software

Software	**Developer**	**Notes**
Inkscape	Open Source	To process exported 2D design for laser cutting.
Illustrator	Adobe	To process exported 2D design for laser cutting.
Mastercam	CNC Software	Computer-aided manufacturing (CAM) software to process 3D design and generate toolpath for computer numerical control (CNC) manufacturing machines.
Blender	Open Source	3D modeling and animation software that is geared toward video game development. Though models can be used for 3D printing as well.
Rhinoceros 3D	TLM, Inc	3D modeling tool used in architecture and product design. It is quite flexible and offers the precision of CAD, yet, it flows freely like other media-focused modeling tools. Its strength lies in the ability to procedurally generate complex geometry via programming.
3DS MAX	Autodesk	3D modeling and animation software geared toward video game development. However 3DS MAX is also used in architecture and product development for its rendering engine to create realistic images and animation for the proposed design.
Maya	Autodesk	3D modeling and animation software with a focus in animated film.
3D Printing Software	Depends on Vendor	To process exported 3D design and generate toolpath for 3D printer to manufacture the prototype.
Unity	Unity Technologies	Game engine that can also be used to create simulation, virtual reality and interactive experience with your design.

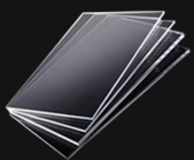
Appendix III - Laser Cutting Service Reference

The laser cutting price list and design specification are taken from 大师傅模型店 on Taobao. The RMB pricing is taken in June 2025 and is inclusive of material and cutting/engraving. Shipping is extra. There are many vendors on the platform, so feel free to shop around. https://item.taobao.com/item.htm?id=539734021637

椴木层板	厚度(mm)	880x450	880x880	1180x880
 模型常用板、工艺板、三合板	1.5	45	90	\
	2	45	90	\
	3	45	90	140
	4	55	110	150
	5	65	130	160
	6	85	170	\
	8	110	220	\
	10	150	300	\
	15	详询客服		

黑胡桃层板	厚度(mm)	380x580	780x580	1180X780
 黑色椴木板层板	2	55	110	220
	3	65	130	260

奥松板	厚度(mm)	380x580	780x580	1180x780
 密度板、中纤板	2	40	80	160
	2.5	45	90	180
	4.5	60	120	240

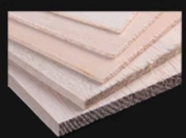
亚克力	厚度(mm)	380x580	780x580	1180x780
 透明玻璃板、PMMA、进口材质	1(透明、白色)	45	90	详询客服
	2(透明、白色、黑色、磨砂、绿色、黄色、蓝色、红	65	130	
	3（透明、白色）	75	150	
	5（透明、白色）	100	200	

雪弗板	厚度(mm)	380x580	780x580	1180x780
 PVC、发泡板、安迪板	2（白色）	40	80	详询客服
	3（白色）	50	100	
	5（白色、黑色）	70	140	
	8（白色、黑色）	90	180	

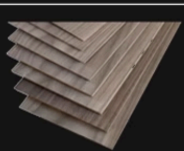
卡纸	厚度(mm)	400x580	800x580	1160x780
 颜色：黑色、白色、灰色、牛卡	1	30	60	90
	2	40	80	160
	3	50	100	200

瓦楞纸	厚度(mm)	760x760	1180x880	\
 AB坑加强纸	3	28	\	\
	5	46	76	\

桐木实木片	厚度(mm)	880x90
 实木航模材料	1	20
	2	
	3	
	4	25
	5	

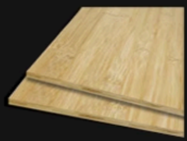
轻木实木片	厚度(mm)	880x90
 巴沙木、航模材料	1	35
	2	
	3	
	4	45
	5	

沙比利实木片	厚度(mm)	480x90
 刚果进口实木片	1	40
	2	45
	3	45

黑胡桃实木片	厚度(mm)		480x90	
 北美黑胡桃实木	1		40	
	2		45	
	3		45	

松木实木	厚度(mm)	380x580	780x580	880x90
 进口松木实木	8	70	140	\
	12	120	240	\
	2	\	\	20
	3	\	\	25
	4	\	\	30
	5	\	\	40

橡木实木	厚度(mm)	380x580	780x580	
 橡胶木	8	70	140	

竹子版	厚度(mm)	480x480	480x880	
 楠竹板	5	80	160	
代加工各类布艺、卡纸				

如何做好图纸

1 图形的宽度大于 2MM
2 图形的间距大于 2MM
3 另开一个文件放置切割图
4 删除重线（命令'OV'+ 回车）
5 设置三个图层（切割、雕刻、断口）
6 白色线切透，红色线雕刻，绿色线断口
7 切割线间距 2MM，雕刻线 1MM 以上
8 断口线间距 0.7MM，数量 2-5 个
9 图块、图案要炸开（命令 X），
文字炸开（命令 TEXP）
10 排版尺寸 880X450(880X900 算两个版面）
11 切割尺寸为 1:1，即图纸多大，真实尺寸就多大
12 导出版本 DXF-2007

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Published on Summer 2025

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