3D DESIGN COURSE

Hansei Technology

By : Gionea Larisa

1. Introduction
    a) What is a CAD? —> page 3
    b) File types —> page 3
2. Design
    a) CAD Software
        - Onshape —> page 4
        - Fusion 360 —> page 15
3. Printing
    a) 3D printers —> page 16
        - Cartesian printers
        - Delta printers
        - Core XY printers
    b) Operation mode —> page 17
    c) Calibration —> page 17
    d) Slicing
        - Slicing Software—> page 19
        - Types of filaments + temperatures —> page 20
        - Tips —> page 20
        - Settings in slicer —> page 21
            a) Layer height
            b) Layer width
            c) Number of walls (wall count)
            d) Infill and pattern
            e) Supports

Introduction

a) What is a CAD?

Through design, people's ideas become reality. CAD (Computer Aided-Design) software allows building the desired 3D model from scratch, using functions that are simple to understand and apply. This is the basis of your future object that will come to life only after printing.

b) File types

3D design software exports files of various types. The most used are STL, STEP, but there is also OBJ, PVZ format, etc.

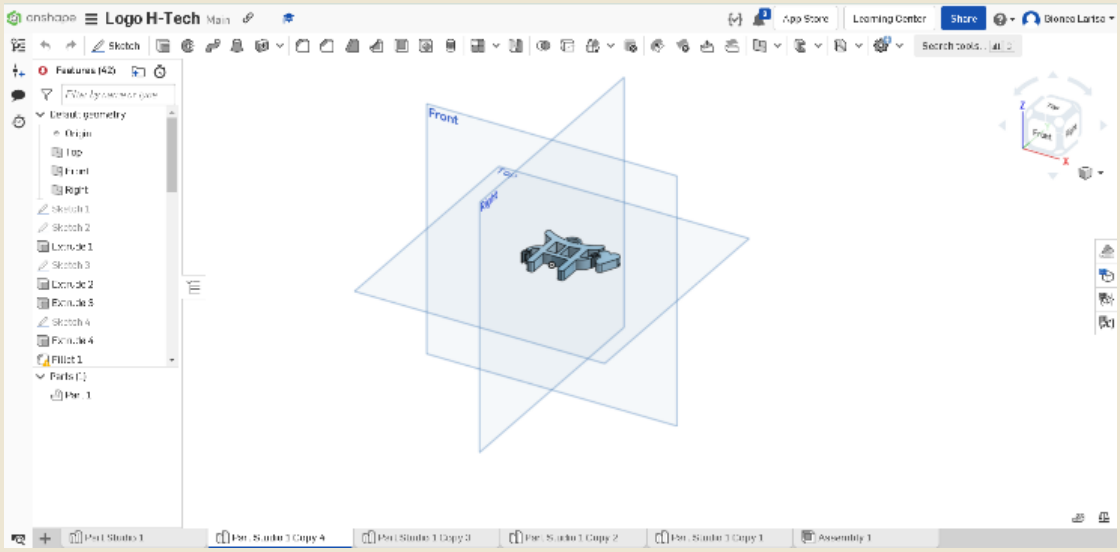
When downloading a CAD from the internet, it will often be saved in the form of an archive (zip file) that must later be unzipped to access the desired files (STLs, STEPs, etc.).

Design

a) CAD Software

- **Onshape**

This was the program we used most in the previous season, PowerPlay. Its interface has suggestive images and the possibility of navigating through all the commands made available. Also, its complexity made us realize that it is a suitable program for creating our parts, which were to be assembled on the robot.



Student account creation:

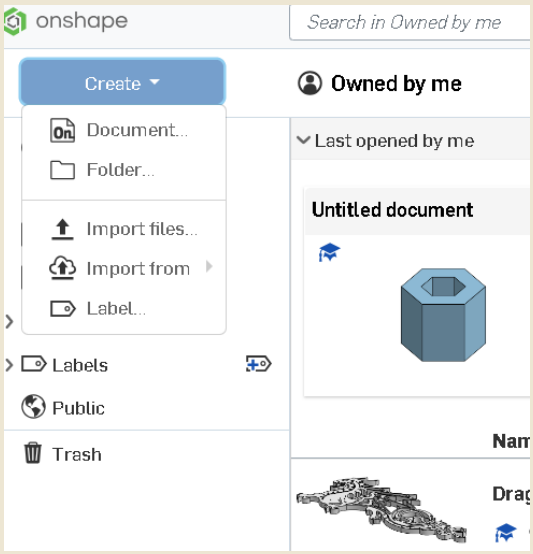
- Unlike the rest of the programs, Onshape is a Software in the Browser. This means that to use it you just need to create an account.

1. Access the following link: Onshape for Education

2. Follow the steps on the site by filling in the fields

Create a new document

1. Click the *Create button* in the top right
2. In the window that has opened, select *Document*
3. Type in the name of the document then click *Create*



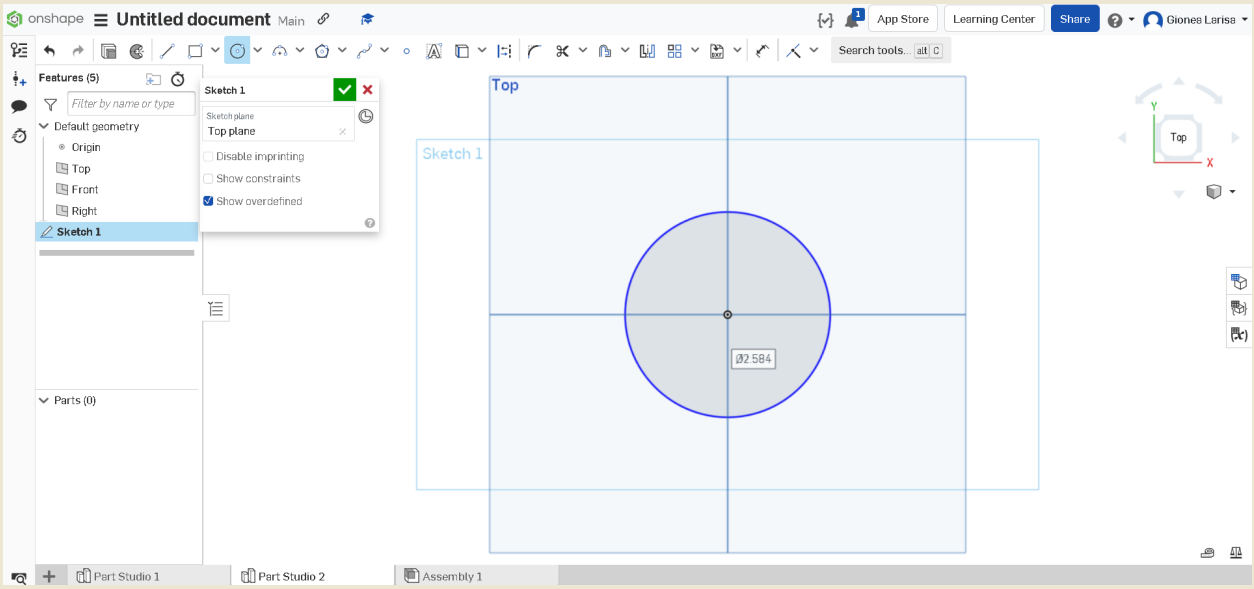
Part Studio

Sketch

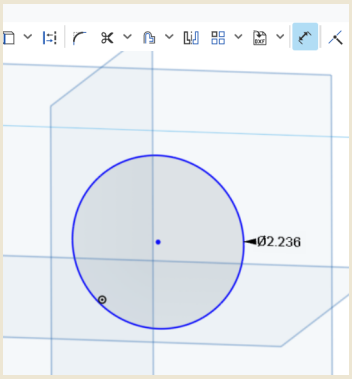
Now that you've created your first document, we'll create your first 3D objects. But to do that, both in Onshape and in the other programs, we must first draw them 2D. This function is called *Sketch*.

At the top of the screen is the toolbar, the first of which is *Sketch* (right next to the Undo/Redo buttons).

The first step to using it is to select one of the existing plans (top, front, right) where you will create your object. In *the Sketch* function there are many shapes that you can "draw", such as circles, straight lines, curved lines, semicircles, polygons, etc.

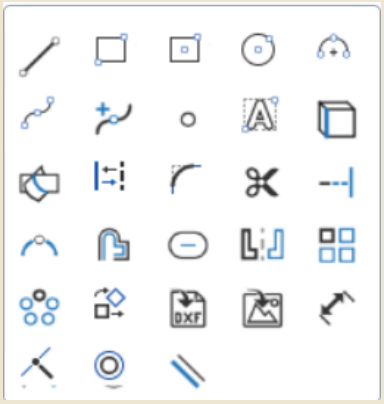


Dimension



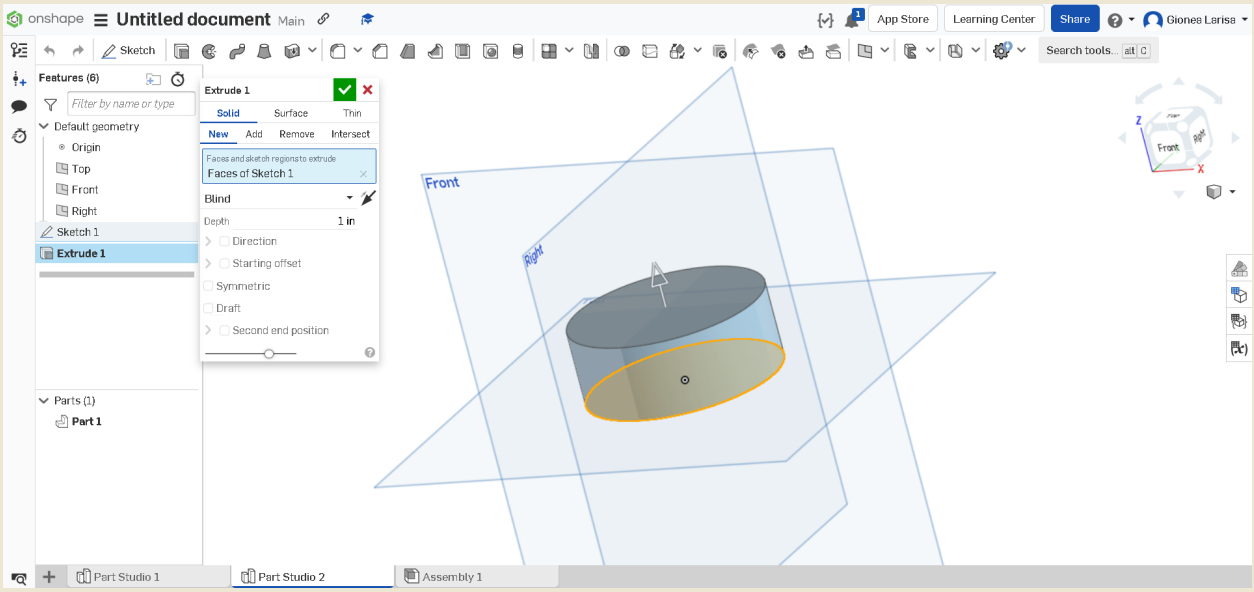
To measure all lines, shapes, and angles in a sketch, Dimension is used.

Tools in the *Sketch function*:



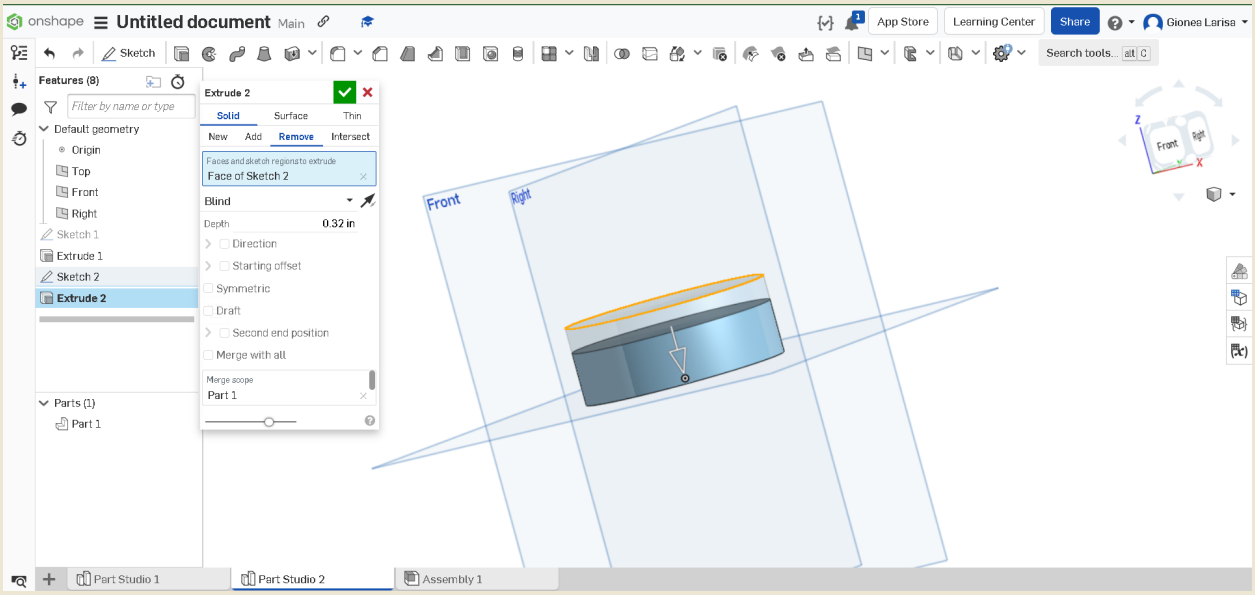
Extrudes

Through these sketches, you can then create a 3D object using the *Extrude* function that gives the 2D shape a height, thickness that can be modified to your liking.



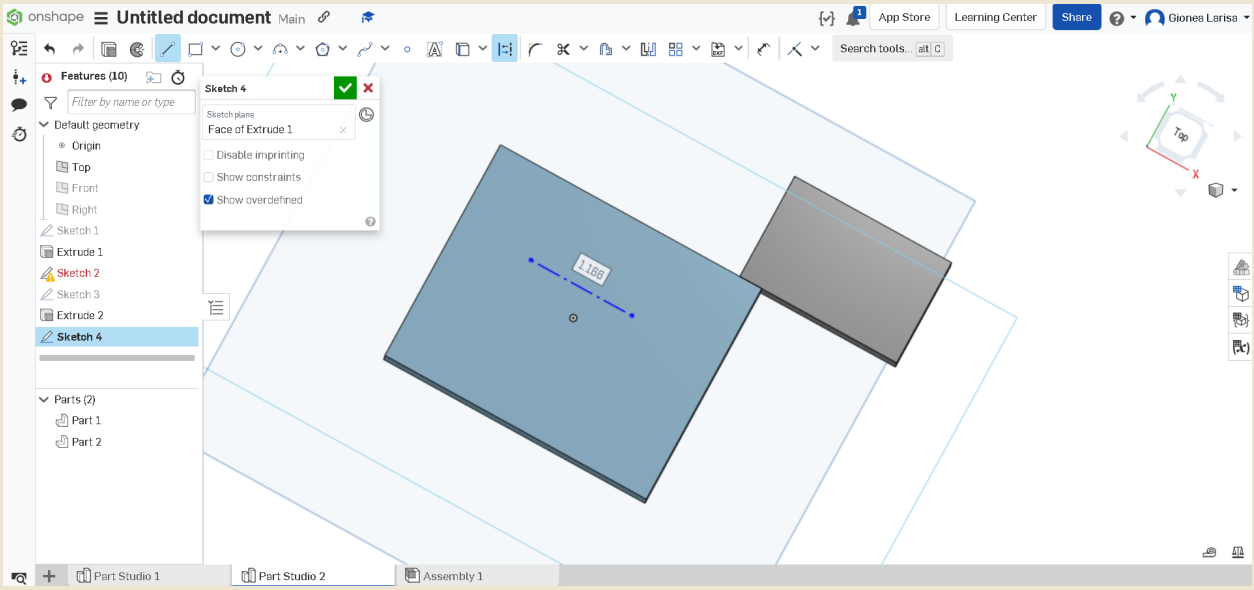
Remove

With *the Extrude*, you can remove parts by selecting the Remove option.



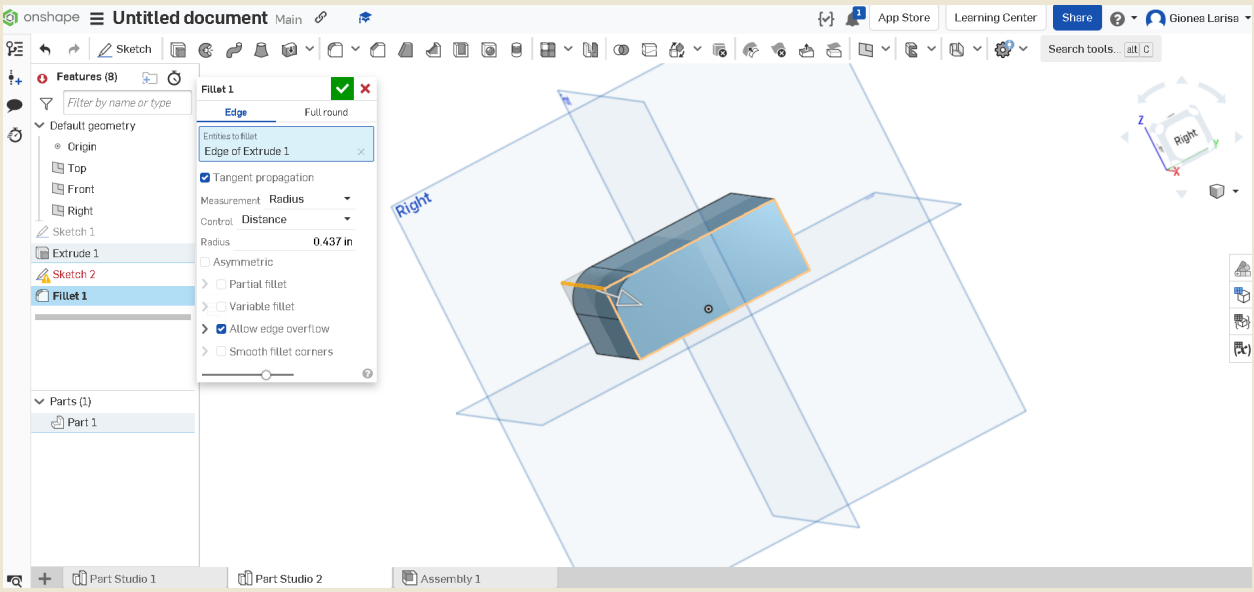
Construction

The *Construction* option makes dotted shapes within sketches. These are guidelines that will not be seen outside of the sketch in which they were created.

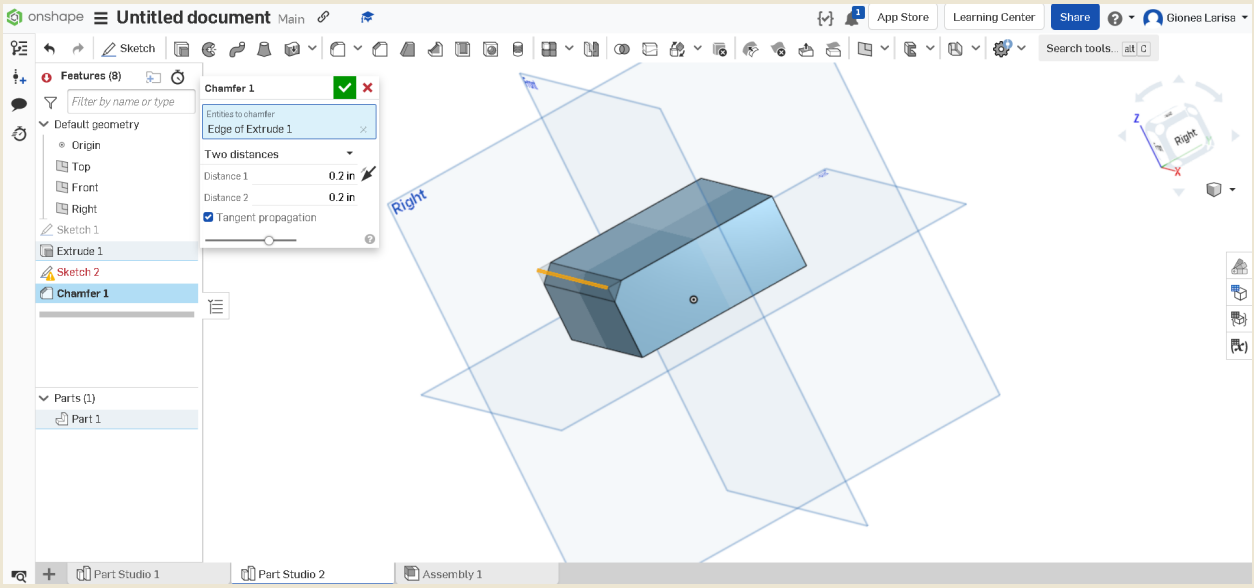


To drill down into objects, you have a wide range of functions at your fingertips, such as:

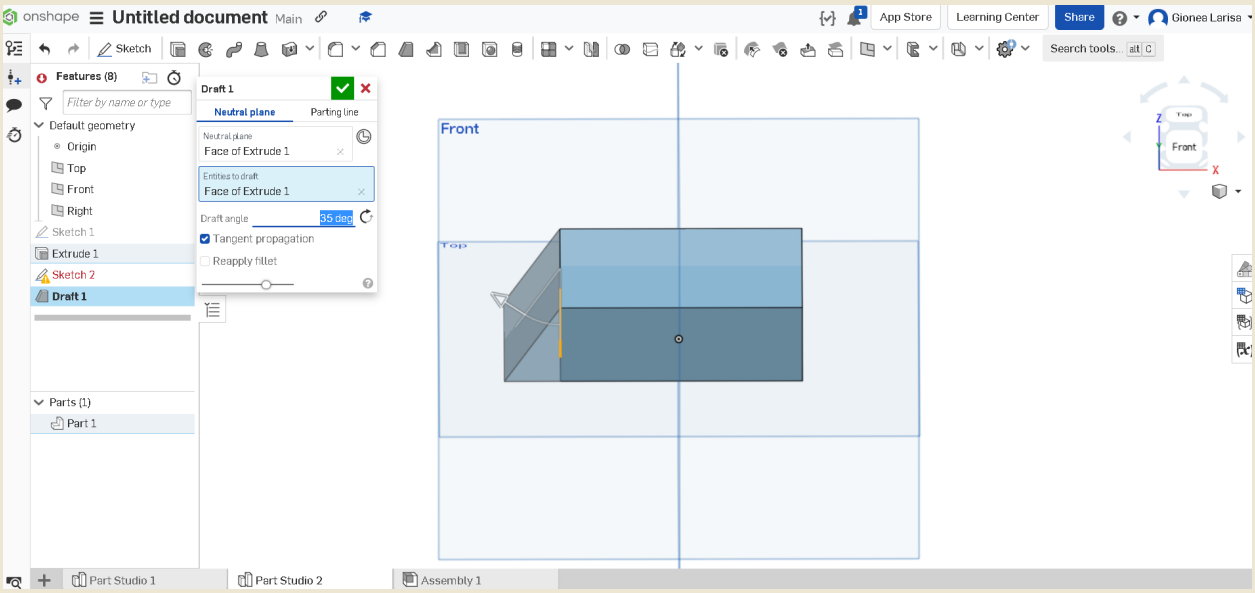
a) Fillet- rounding



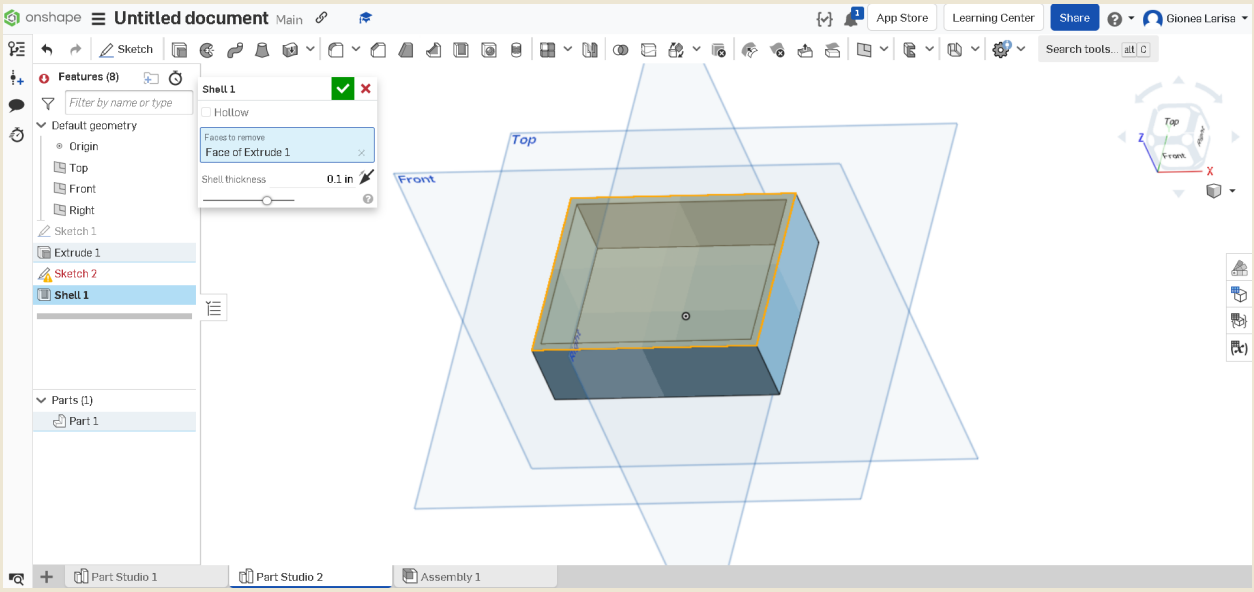
b) Chamfer- smoothing of cols



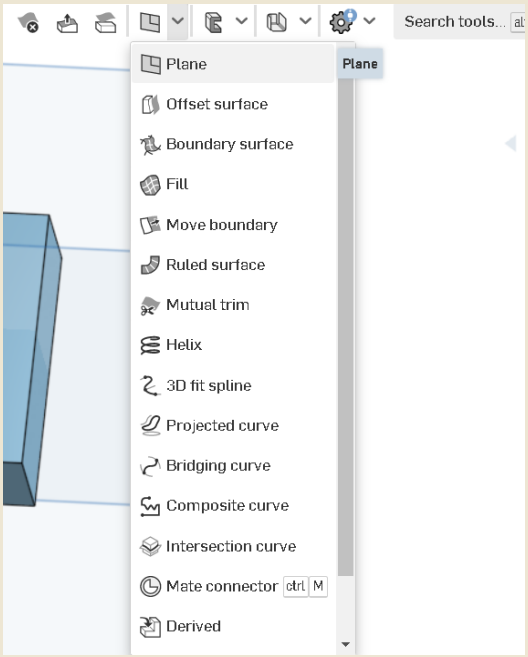
c) Draft- trapezoid creation



d) Shell- drilling



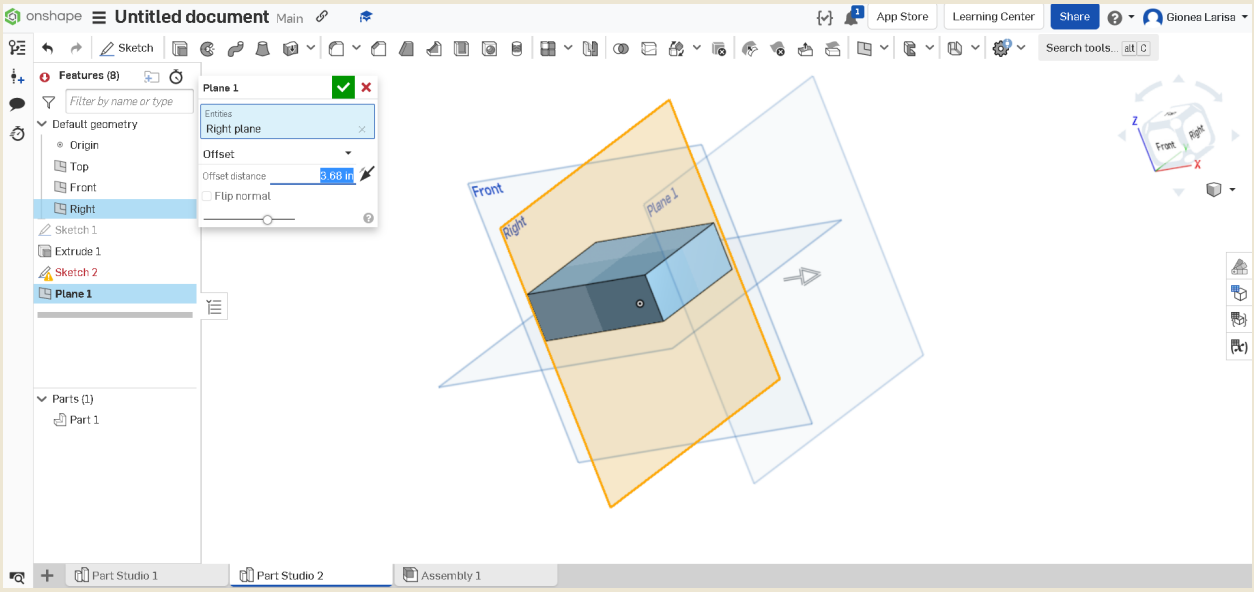
Plane



Also, in Onshape, you are offered 3 standard planes: top, front and right, but through the *Plane* function, you can create a plane where you need it to exist.

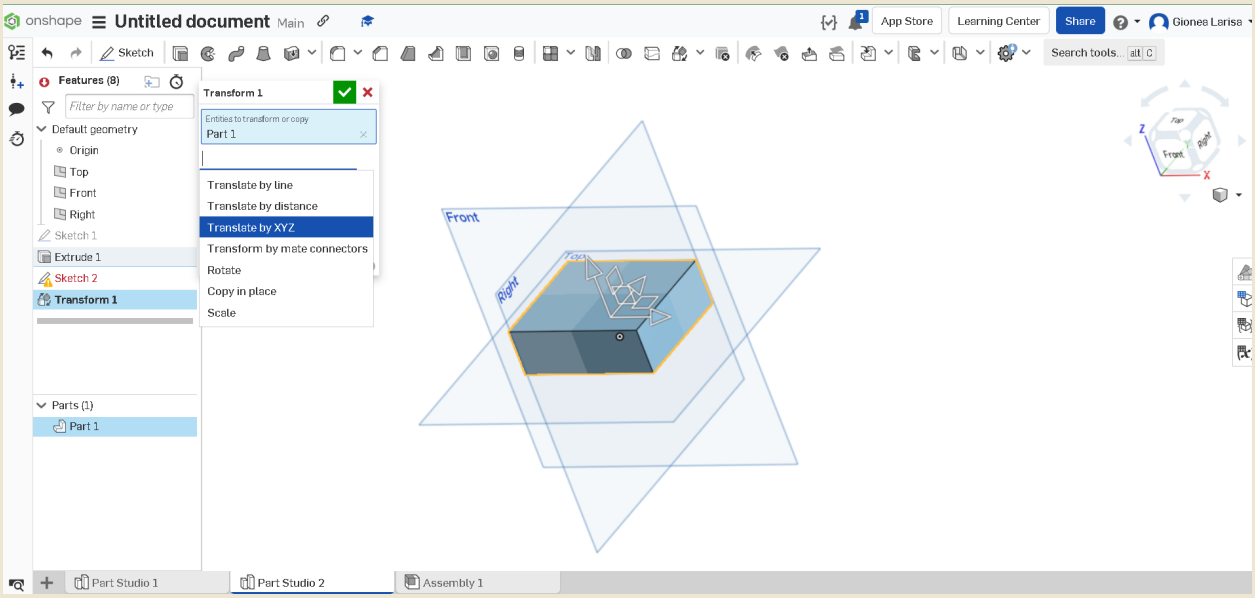
Derived

Also, the last option, *Derived*, offers the possibility to add parts of other documents to the current one, or even public tracks for all Onshape users.



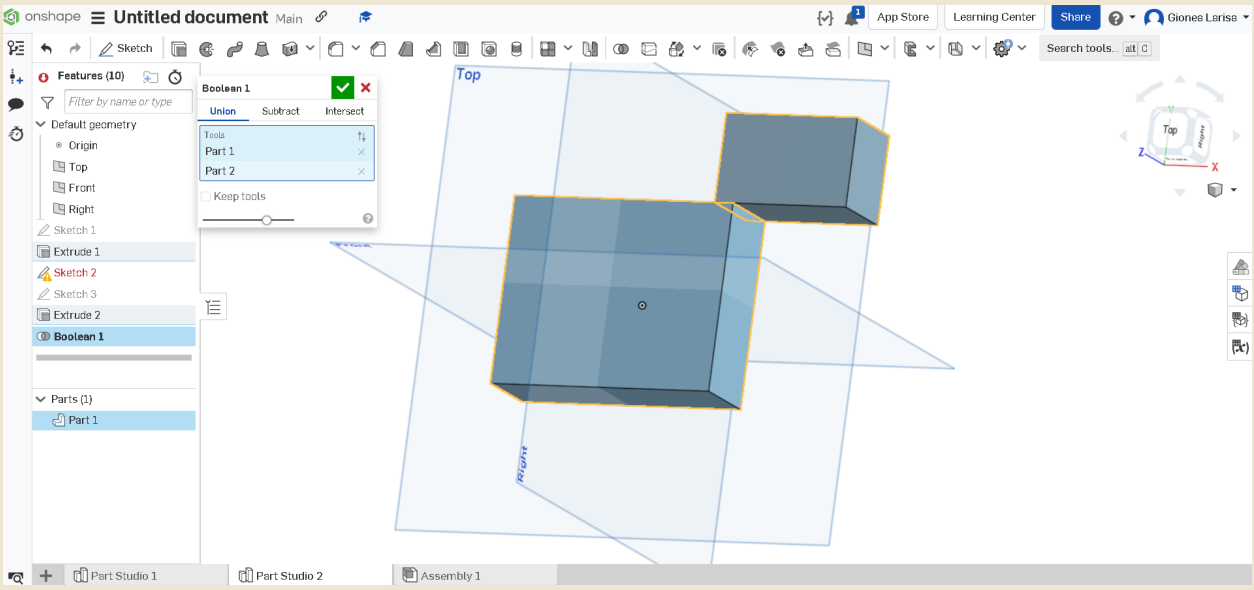
Transform

To move objects, the Transform function is used. It has 7 ways to move, but the most used and easy is *Translate by XYZ* which gives you the 3 axes to move the object.



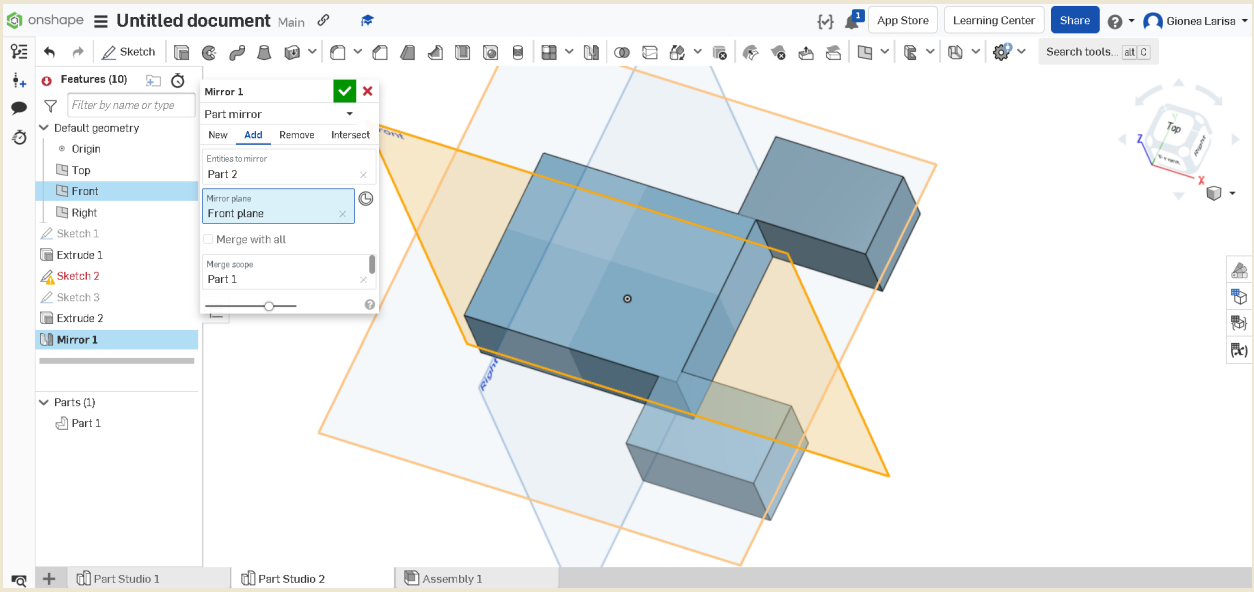
Boolean

The *Boolean* option is used for joining 2 or more objects.



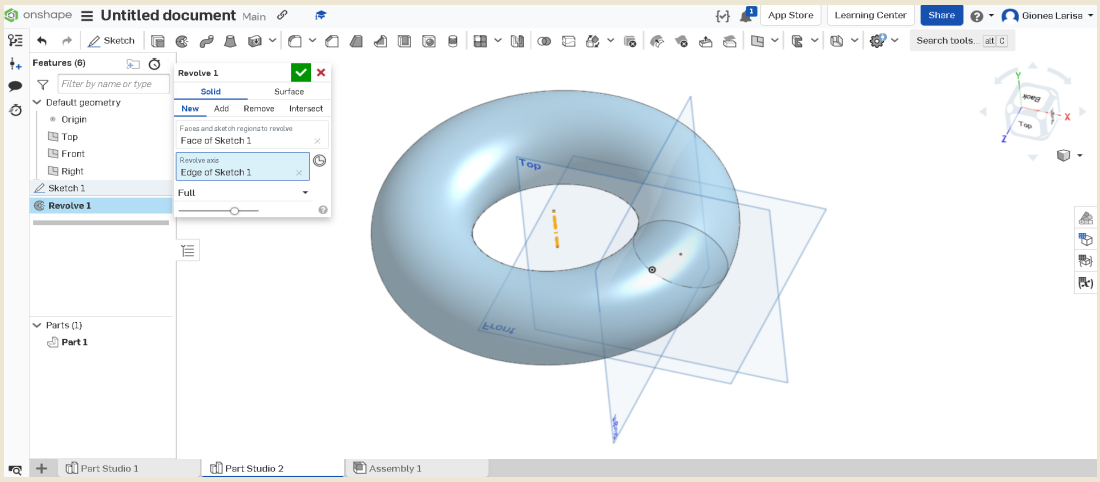
Mirror

Mirror helps you transpose the same part vis-à-vis vis. For example, I need to make pliers. I made one of the arms, given that the other arm must be identical only in the mirror, I can use the *Mirror* option to ease my work and avoid differences between the 2 halves.



Revolve

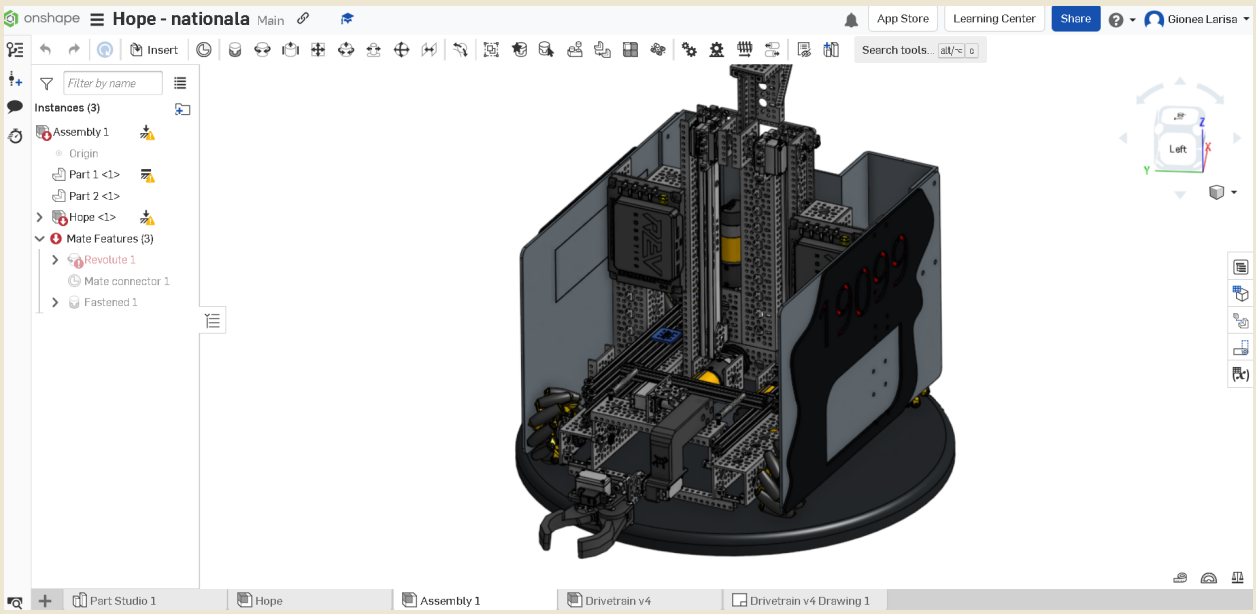
Revolve rotates Construction-made faces, regions, edges and curves around a center axis to add, remove, or intersect a part.



In addition, within this Software, you can change both the color of the object and the material from which it is made, thus managing to create much more accurate stress analyzes.

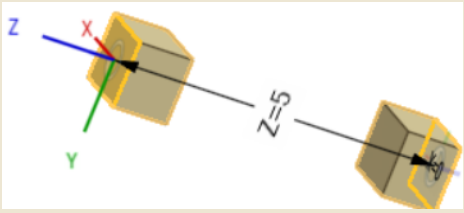
Assembly

Through such a document you can design assemblies of parts and define how they move.

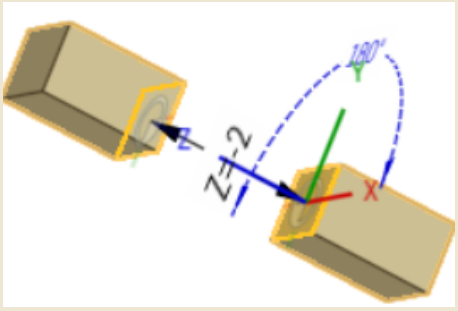


Mates:

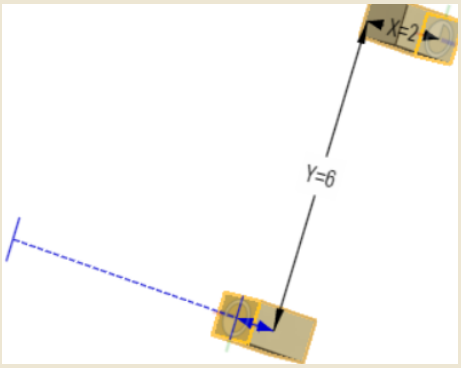
- Fastened: the union of two pieces



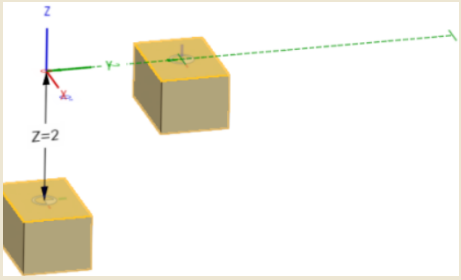
- Revolute: joining two pieces allowing rotational movement around Z



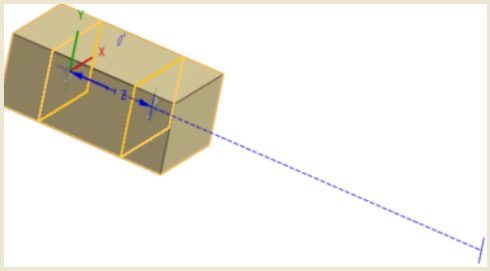
- Slider: joining two pieces allowing translational movement along the Z-axis



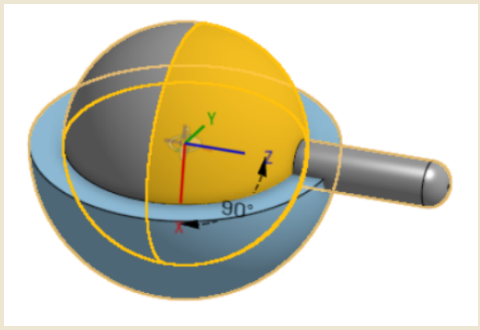
- Planar: joining two pieces allowing translational motion along the X and Y axes, and rotational motion around the Z-axis



- Cylindrical: joining two pieces allowing translational motion along the Z-axis and rotational motion around the Z-axis



- Pin Slot: joining two pieces allowing rotational movement around the Z-axis and translational motion along the X-axis
- Ball: joining two pieces allowing rotational movement around the X, Y and Z axes



- Parallel: joining two pieces allowing individual translational motion along any axis and rotational motion around any axis
- Tangent: joining two pieces tangentially to selected faces, edges or vertices

- **Fusion 360**

  It was used, in particular, in the case of 3D renderings of the robot.

  Create account:

  - Enter the Autodesk platform Autodesk

Printing

a) 3D printers

There are 3 major categories of printers: Cartesian (ex: Prusa mk3S, Ender 3), delta (ex: FLSUN QQ- S Pro), core xy (ex: Zortrax M300 Plus).

- **Cartesian printers**

They are called Cartesian because they move along a *Cartesian plan* (as in mathematics, a Cartesian coordinate system, i.e. axes *X*, *Y*, *Z*). That is, the extruder moves left, right, up, down, and the bed moves back and forth.

- **Delta printers**

Delta printers support the extruder via *3 arms* attached to *3 pillars* vertical arranged in triangle. Each of these arms can only move *vertical up* and *down*, but by their independent movement, the extruder can move in all directions.

- **Core XY printers**

It resembles Cartesian printers, but in these, the bed moves only on the axis *Z*, and the extruder only on the axes *X* and *Y*. In the printing process, the bed rises to the level of the extruder that is to move left, right, forward and backward.

b) Operation mode

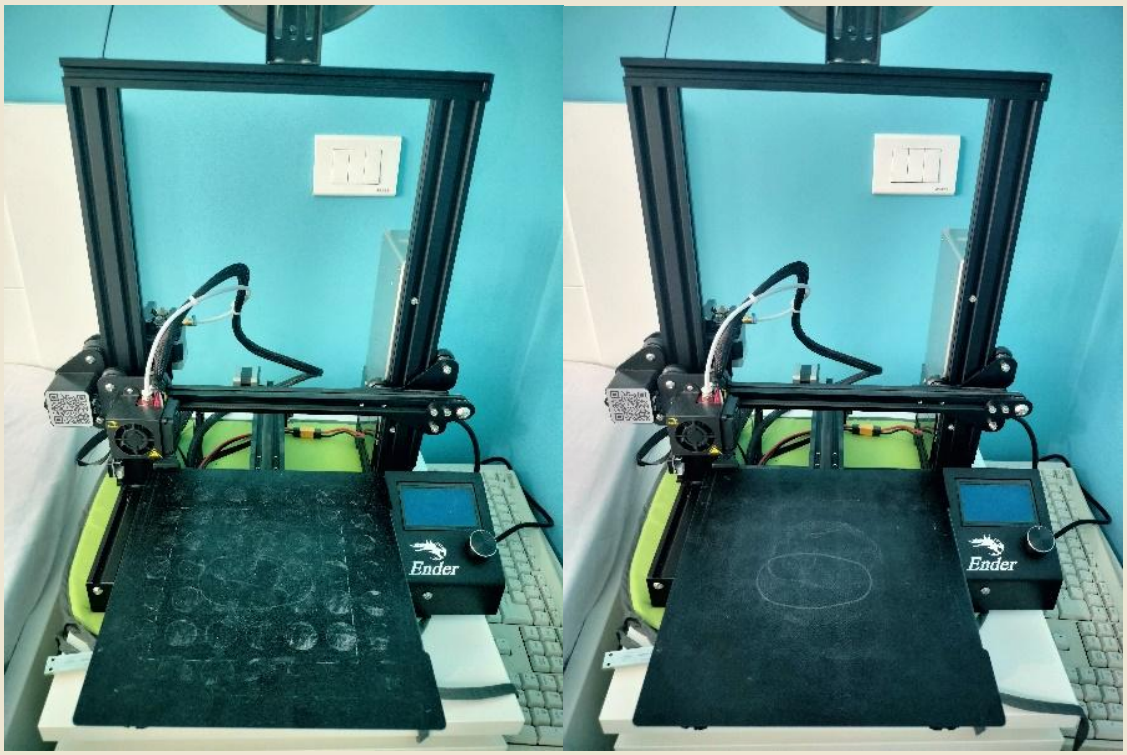
The 3D object to be printed will be processed into a Slicer (a program that converts an STL, STEP, etc. format into gcode to be read by the printer). After putting the gcode file on the printer card, if it is well calibrated it will start printing.

A 3D printer prints by stacking filament layers on top of each other until the desired object is created, so each layer supports the latter.

c) Calibration

Calibration of a printer refers to changing parameters that have a major influence on print quality. Regardless of the brand of printer, the process is similar. To illustrate the calibration process, we decided to use an Ender 3 Pro Cartesian printer.

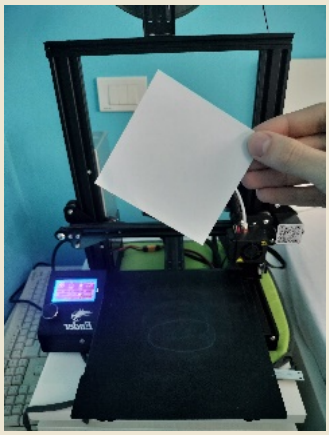
The first step in order to be able to set the appropriate parameters is to have a clean print bed and free of filament residues. For cleaning the bed it is recommended to use a lint-free cloth on which to pour a little isopropyl alcohol.

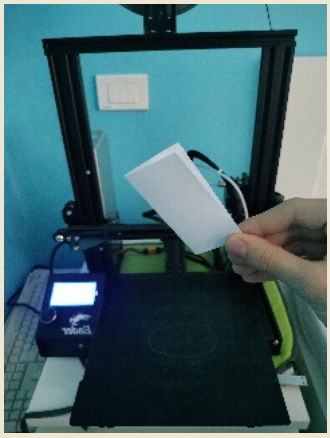


NO YES

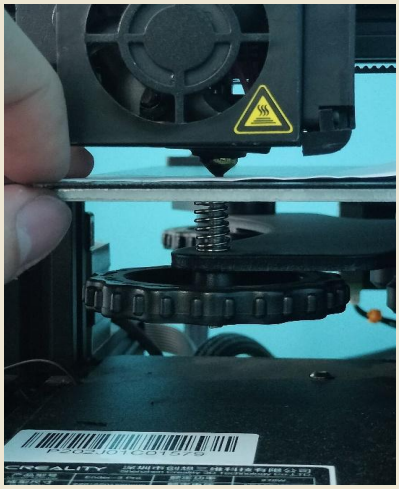
Afterwards, we will start the actual calibration process. To make it easier for us to do this, people have created special gcodes. An example of a gcode of this type can be found here: https://thangs.com/file/913963. In the following steps we will explain how you can use these files

We will start with the CHEP_M0_bed_level.gcode file. This "print" helps us set a fairly precise position of the noozle relative to the bed. We need a piece of paper about 10x10cm, which we will fold in half as in the pictures below.

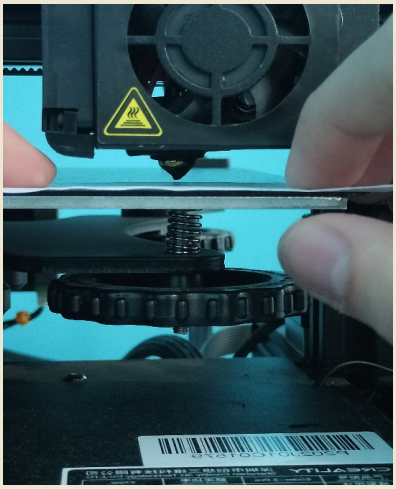




After we turn on the "print", we place the piece of paper under the hot-end. The hotend will move to the position where it should be able to print in the corner of the print bed. Now we will intervene, by making sure that the distance between the bed and the nozzle puts pressure on the paper and we feel some weight when moving the paper. We must bear in mind, however, that the paper must be able to be moved easily, feeling only a little resistance. For this we adjust the height of the bed using the wheels under each corner of it. After adjusting the height in that corner we press the button on the printer display to move to the next corner. After going through all 4 corners, the nozzle will go to the center of the bed to verify that we have done the height calibration correctly. Later we will remove the piece of paper, unfold it and press the button again to resume the process, this time with a smaller thickness of paper.



YES



NO

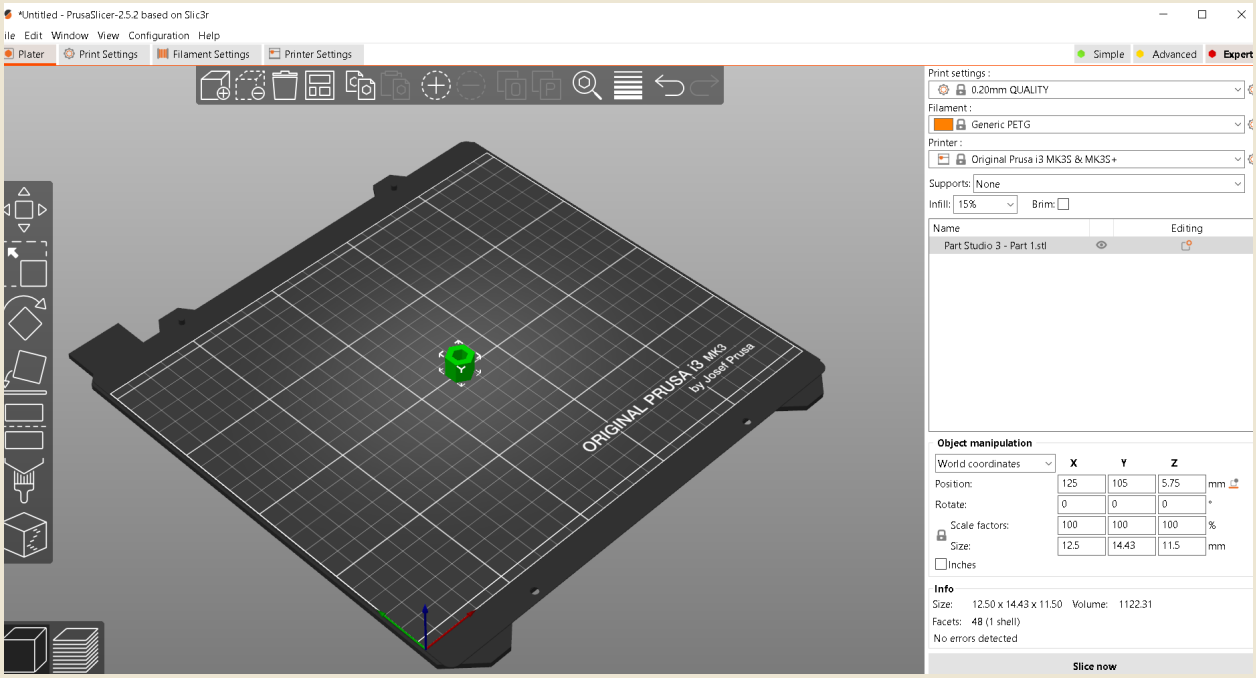
After this step we will continue with the next calibration file, the one called "CHEP_bed_level_print.gcode". This file will print squares all over the bed. Its purpose

is to ensure that the height of the nozzle is good. If we notice that the print "enters" the bed, then we must lower the bed using the corner wheel, or if the print does not stick to the printing surface, we must raise the bed to be closer to the nozzle.

In the end, we should have a calibrated and ready-to-use printer.

d) Slicing

- **Slicing Software**
  You can use: Prusa Slicer, Creality Slicer, Cura, etc. For all this, you need to select the printer you own and the type of filament you are going to print with before making advanced print-related settings.

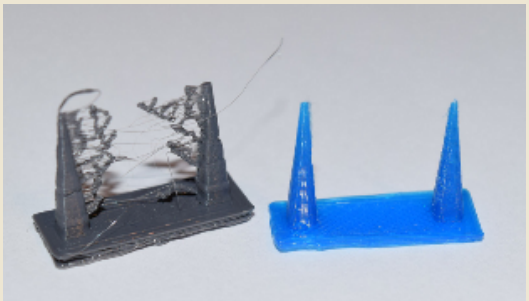


- **Types of filament+temperatures**

Types of filament	Features	Nozzle temperature	Bed temperature
HIPS	Easy to print, smooth surface finish, affordable, dissolvable, chemical resistance	230- 240°C	90- 100°C
PLA	Easy to print, biodegradable, low toxicity, high accuracy, fast cooling, wide variety of colors, limited chemical resistance	190- 210°C	60- 70°C
PETG	Durability, easy to print, limited flexibility, high printing temperature, transparency, recyclable	230- 260°C	60- 90°C
ABS	Durability, easy to print, limited flexibility, heat resistance, prone to warping and cracking	230- 250°C	90- 100°C

- **Tips**

The filament must be kept as dry as possible. PLA is a hygroscopic material (absorbs water) so from the moment of unsealing, the roll will be used as quickly as possible or stored in a box / dehydrator.



The moistened filament can create problems when extruding, in most cases it creates the phenomenon of "stringing".

- **Settings in the slicer**

Layer thickness

- It has a significant impact both for the printing time and for the strength of the part.
- It is influenced by parameters such as:

a) Layer height

✓ Minimum layer height= 25%*nozzle diameter
The thinner the layer, the more detailed the print, at the expense of the speed and strength of the part.

✓ Maximum layer height= 80%*nozzle diameter
The thicker the layer, the higher the strength of the part and the decrease in printing time. However, the quality is at a low level.

b) Layer width

It influences the strength of the part, the maximum print speed and the final appearance.

For:

- increase mechanical strength- we increase wall thickness
- Finer details – we decrease the wall thickness

c) Number of walls (wall count)

Influences the strength of the part.

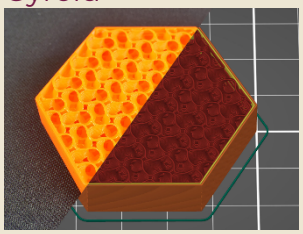
- The strength of the part is not depending on the infill, but on the walls.
- The walls are influenced by layer width.
- Tips: 5 walls provide good strength of the part, 3 walls are a good compromise between printing time and strength.

Infill and pattern

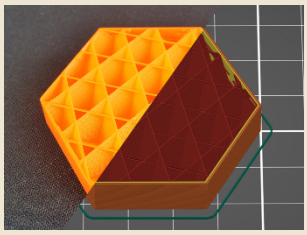
The infill represents the internal structure of a 3D printed object. Its purpose is to reduce weight and printing time, but also to improve its resistance.

This structure exists in different models such as:

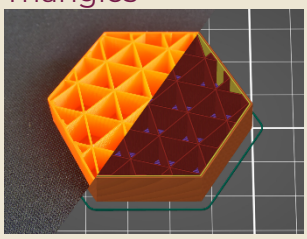
a. Gyroid



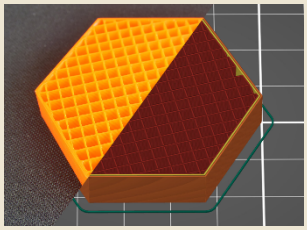
b. Cubic



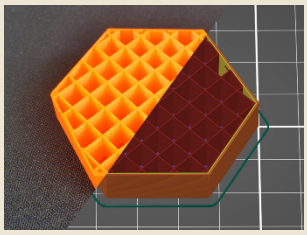
e. Triangles



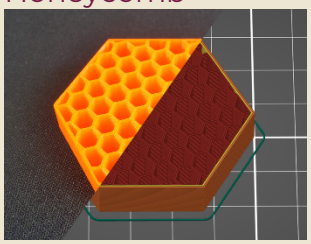
c. Rectilinear



d. Grid



f. Honeycomb



Supports

A 3D printer prints by stacking filament layers on top of each other until the desired object is created, so each layer supports the latter. For this reason, for floating or angled structures, supports are created that after finishing the print will be removed because otherwise those parts will fall or detach.

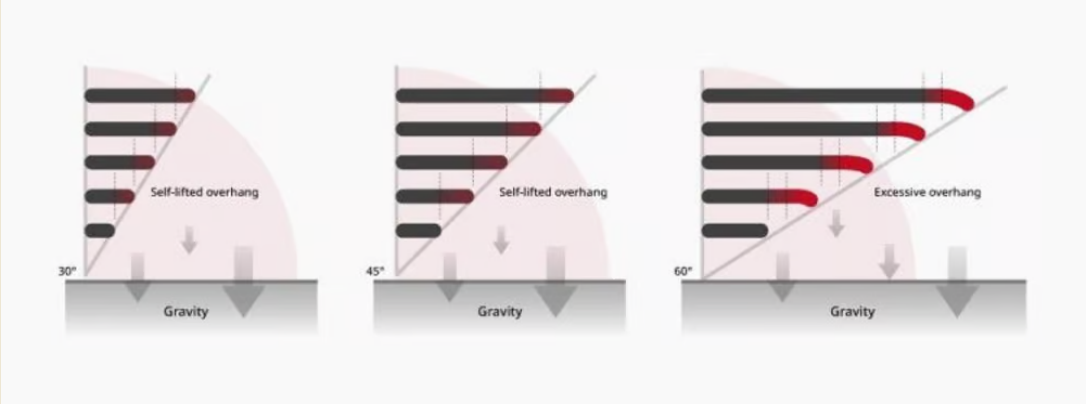
When are they used?

Usually when the object has a protruding part or a bridge that is not supported. Example:

- the letters Y and T have protruding parts
- the letter H has a bridge

The 45-degree rule

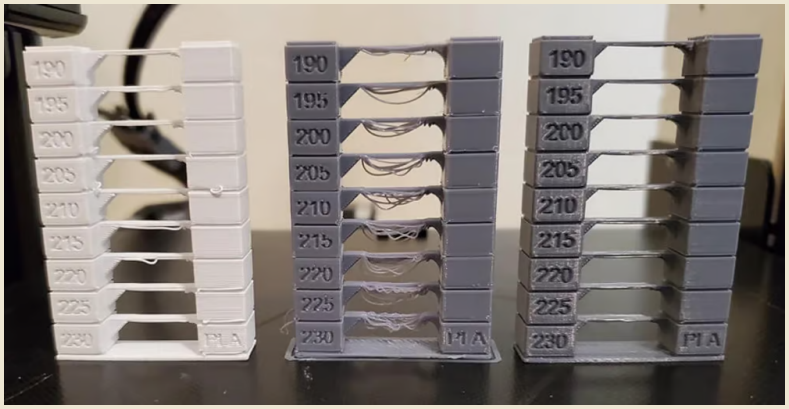
Not all protruding parts need supports. The general rule: if the angle formed by that side and the vertical is less than 45 degrees, then supports may not necessarily be necessary.



3D printers use a very small horizontal offset between layers, so one layer does not overlap the latter perfectly if there is no vertical wall. Everything below 45 degrees can be supported by the previous layer, the rest fall into the category of failures.

Example:

- the letter Y is at an angle of less than 45 degrees, so it can be printed without supports
- the letter T has an angle of 90 degrees, which is greater than 45, which means that supports will be needed;

The 10mm rule for axles

For decks smaller than 10mm, the printer can print without the need for supports, but if it is larger than 10mm, supports will be needed.