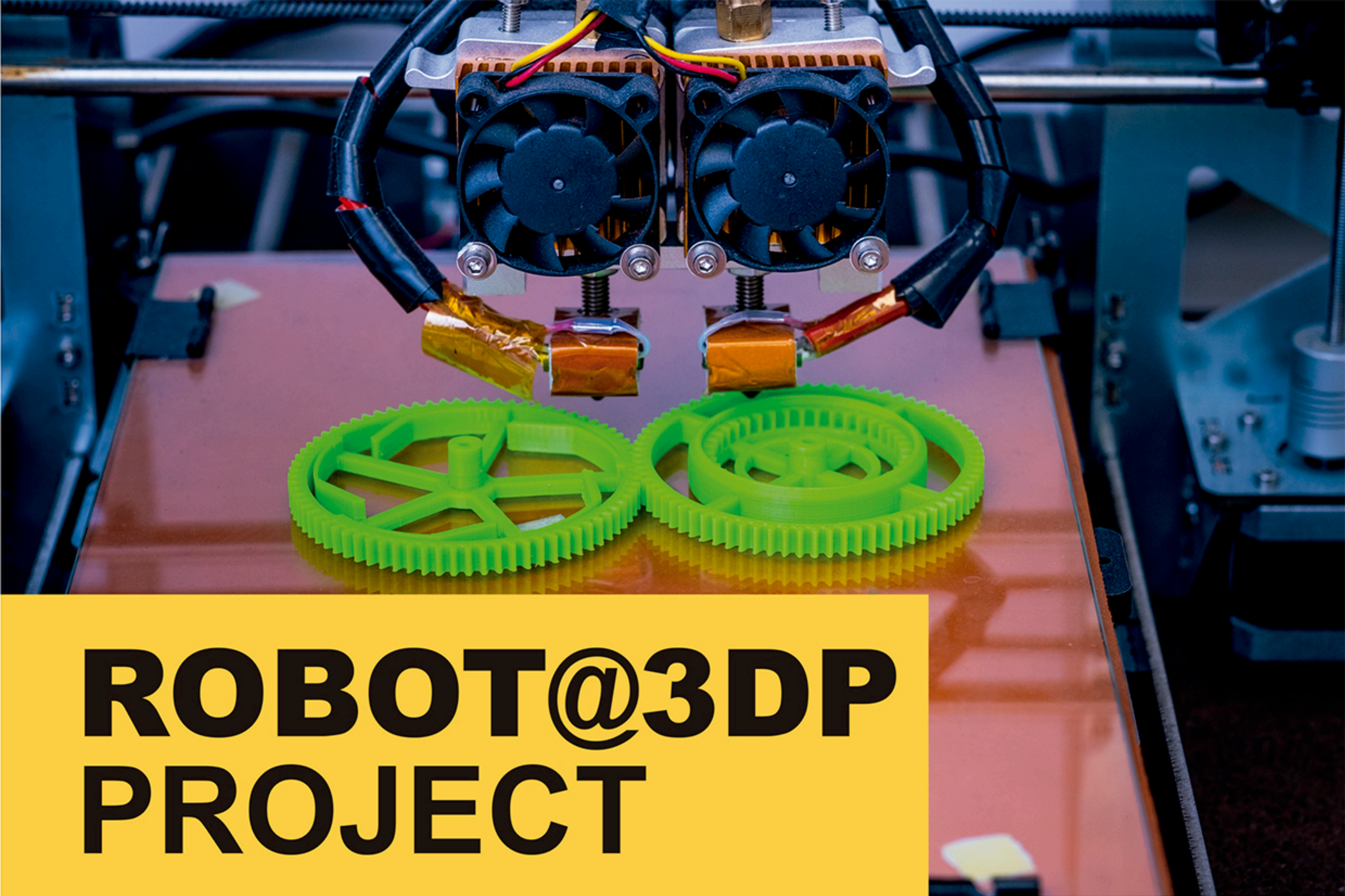


ROBOT@3DP PROJECT

NEW TRAINING RESOURCES FOR THE CHANGE OF THE INDUSTRIAL PARADIGM

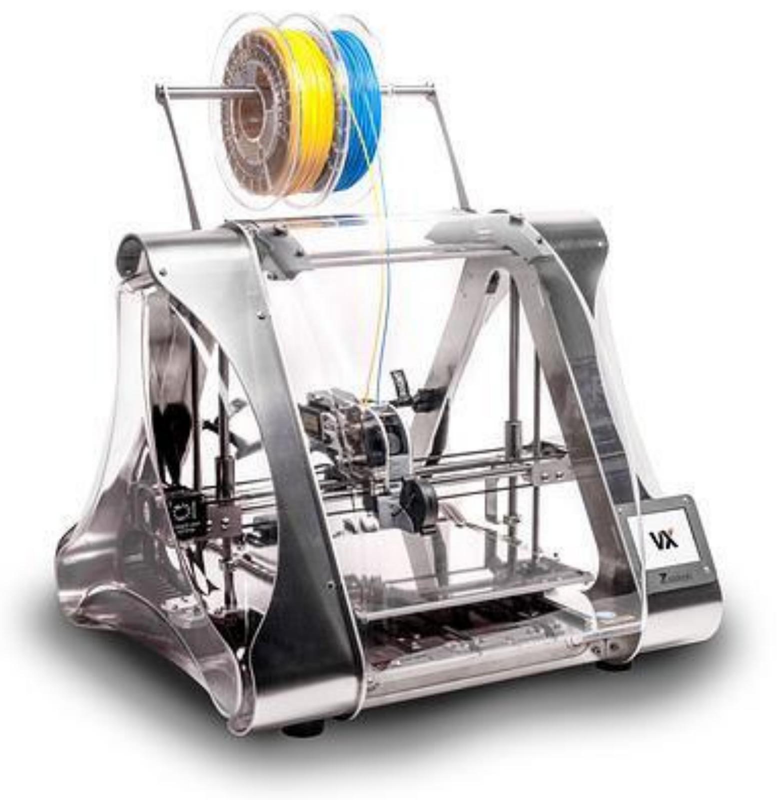
TRAINING MODULE ON THE DESIGN OF PARTS AND SUPPORTS FOR 3D PRINTING

Co-funded by the Erasmus+ Programme of the European Union

This project has been funded with support from the European Commission. This publication reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

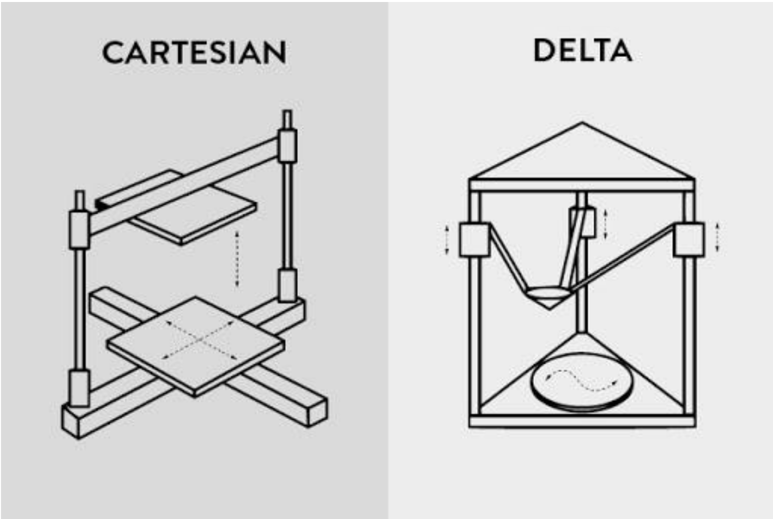
Content

1 FDM machine components



This project has been funded with support from the European Commission. This publication reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

This document will give an in-depth look at all the components that make a 3D printer run, from small parts to large ones. FDM 3D printers could be classified into two main typologies, based on the movement of the head of the extruder. In particular, if the movement of the printed is based on cartesian or polar coordinates it will be a cartesian printer or a delta printer. More specifically, this document focuses on Cartesian 3D printers, which are the most recommended for beginners.



Font: https://3dinsider.com/delta-3d-printers/

Cartesian 3D printers have a different mechanical arrangement on the frame than delta 3D printers. Cartesians have a simple XYZ arrangement while deltas have three arms that move all over the place. Delta 3D printers are, actually, a lot cooler to watch while they print.

Let's see in detail all the components that make up a 3D printer, answering the frequent question: "How does a 3D printer work?"

1.1 Print Head

Font: https://www.crea3d.com/en/ultimaker-3/531-print-core-aa-025mm.html

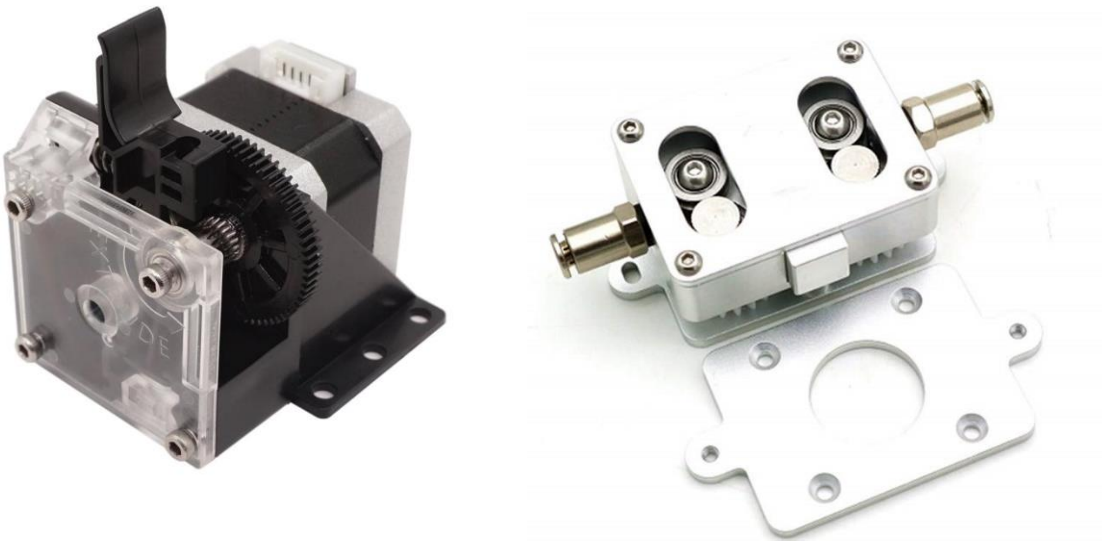
Font: https://www.alibaba.com/product-detail/Metal-Hotend-Kit-Extruder-Printing-Head_62549890215.html?spm=a2700.galleryofferlist.0.0.722f45e6uR6qg3

The print head or core is the component that melts the plastic and shapes layer by layer the structure of the product, turning the filament into a 3D model. It is made of two main parts: a cold end also called an extruder, and a hot end, where there are the thermal resistance, the thermistor, the fan, and the nozzle. To put it simply, the cold end-clamps the filament and pushes it down to the hot end while the hot end, which ends with a nozzle, melts the filament and deposits it onto the build platform.

The extruder is made up of smaller parts, each with its own dedicated function, it could be assembled near the nozzle or far away at a fixed point on the printer chassis, reducing the vibrations and improving the printing. The filament drive gear or extruder drive gear pushes the filament into the hot end. The heat sink and the heat sink fan make sure the filament doesn't get melted before it reaches the nozzle while the heater cartridge is the one that heats up the filament. The thermistor or thermocouple is the temperature sensor for the hot end. And finally, the cooling fan cools the filament as soon as it is deposited on the print bed, helping it hold its shape. The behavior of the cooling fan depends on the type of filament.

The nozzle is where the melted filament comes out. It comes in different sizes.

1.2 Extruder (cold end)



Font: https://www.alibaba.com/product-detail/Best-Price-3D-Printer-Extruder-TITAN_62547425580.html?spm=a2700.details.deiletai6.5.7f8f6412Rzsvda

Cartesian and delta 3D printers use either a Bowden feeder system or a direct feeder system. In a Bowden setup, the cold end and the end are separate from each other, and by which we mean the cold end is placed in a different location on the frame. A Bowden setup uses a filament tube to direct the filament into the hot end. Due to the lighter load, the print head moves faster, which means you get faster prints. In a direct setup, the cold end and the hot end are connected. Although a Bowden setup is also capable of producing great results when printing with a flexible material, many people often turn to a direct setup when dealing with that type of material.

1.3 Nozzle

Fonts: https://www.alibaba.com/product-detail/2020-printing-diy-Digital-3d-Printer_1600088096697.html?spm=a2700.galleryofferlist.0.0.722f45e6uR6qg3

Font: https://www.alibaba.com/product-detail/High-quality-3d-printer-nozzle-compatible_62431581657.html?spm=a2700.galleryofferlist.0.0.304a74a7orBmxt

0.4-millimeter nozzles are the default for most 3D printers. The smaller the nozzle, the higher the print detail. On the flip side, the larger the nozzle, the faster the print speed. Thankfully, you can easily swap out nozzles, so you can change your setup depending on how fast you want to print or how detailed you want the 3D models to be.

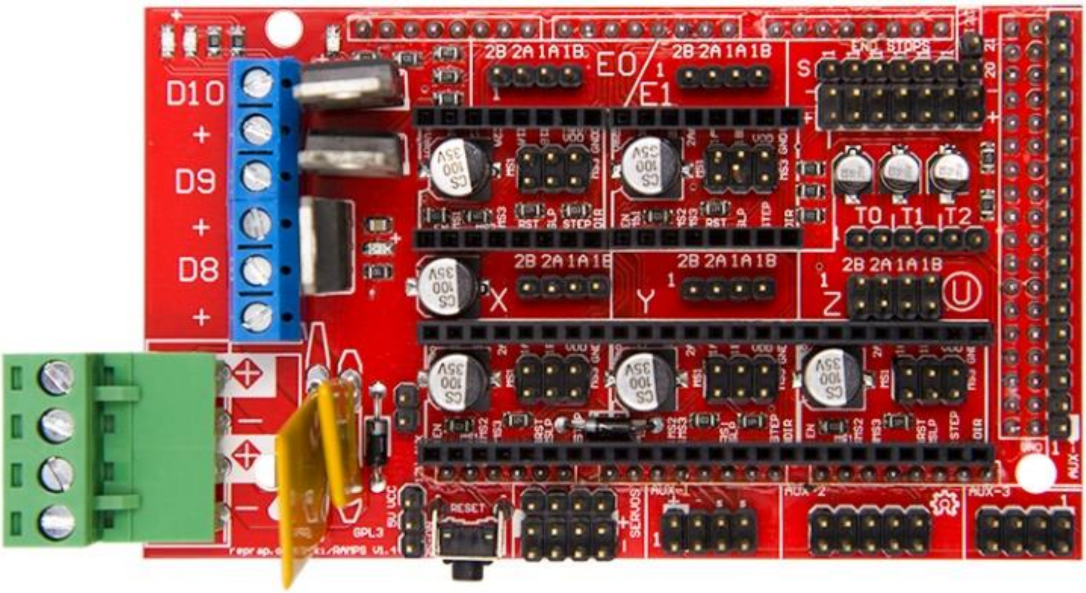
1.4 Display

A 3D printer with an LCD user interface can work as a standalone machine. The screen allows to control the printer and read the main parameters. There are different kinds of user interfaces, the most common of which is a basic LCD interface operated via a knob, a dial, or a set of buttons. Some 3D printers come with a full-color touchscreen. In other words, you can control it without a computer connection. Most 3D printers come with a frame-mounted interface, but there are some models that come with a separate controller box that houses the interface.

An onboard user interface allows you to check and adjust the machine parameters as well as initiate the filament loading/unloading process. In addition, a 3D printer with an auto-leveling system or semi-auto leveling system includes an option on the interface that activates the leveling system.

There are also 3D printers with Wi-Fi connectivity, allowing you to connect to a local network just by going through a simple setup on the onboard interface. With a Wi-Fi setup, you can start, manage, and monitor your prints from your computer or smartphone/tablet while lounging in another room.

1.5 Motherboard



The motherboard could be considered the brain of the printer. It moves components based on the instructions sent from a computer and at the same time, interprets signals from the sensors. The printing accuracy and speed are based on the quality of the motherboard. Better is this part, better will be the obtained print.

In fact, thanks to a good motherboard, is it possible to get high-performance printing from a 3D printer. Even if you choose all the good parts, but with a non-performing motherboard, your 3D printer would not be very functional.

1.6 Motors

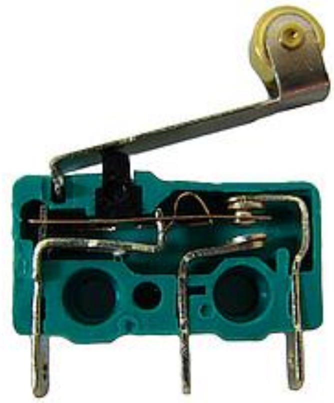
Font: https://www.alibaba.com/product-detail/3d-Printer-Nema-17-Linear-Stepper_62358045944.html?spm=a2700.galleryofferlist.0.0.7f187c2f3fl681&fullFirstScreen=true

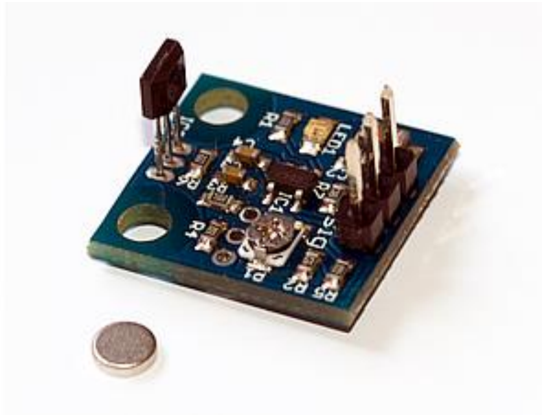
The stepper motors, which are run by stepper drivers, are the keys to the mechanical movement of a 3D printer. Stepper motors are connected to all three axes and drive the print bed, the print head, and the threaded rods or leadscrews. They make a full rotation in increments or steps, hence the name, making them more suited for 3D printers than a normal DC motor. The print head also comes with a stepper motor that drives the extruder feeding movement.

1.7 Belts

In a Cartesian 3D printer, usually, but not every time, the belts are the most common part used for guaranteeing the movement from the motor to the axes. In fact, thanks to belts, which are connected to motors, there is the movement of the X-axis and the Y-axis from side to side and are integral to the overall print speed and precision. In a delta 3D printer, belts are often used to drive the movement on the Z-axis. A loose belt can ruin an entire print. That’s why many 3D printers come with tensioners. Belt tensioning devices keep the belts in optimum tightness and provide an easy way to adjust the belt tightness.

1.8 Endstops





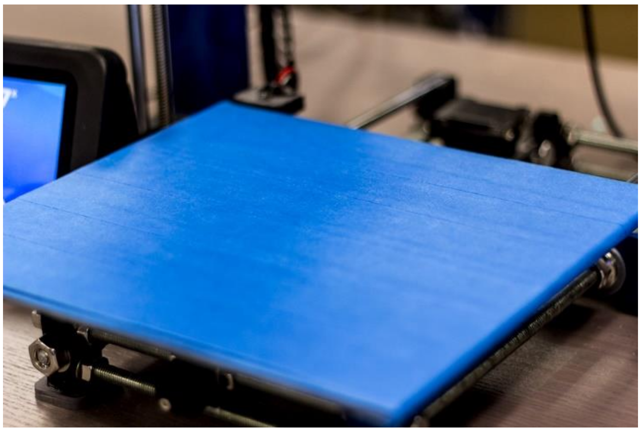
End stops are mechanical or optical sensors that allow us to understand where is the position of the axes, in particular providing the 0-position for each axe. Like markers that allow the 3D printer to identify its location along the three axes, preventing it from moving past its range, which can result in hardware damage.

1.9 Power Supply Unit (PSU)

As for every electronic device, the power supply unit supplies power to the entire 3D printer. No need for an elaborate explanation for this component. The PSU is either mounted on the frame or housed in a separate controller box together with the user interface, as for the notebooks. It's a lot better if the PSU is mounted on the frame as it translates to a smaller overall machine footprint, but at the same time, it increases the machine vibrations.

If you want to print with more advanced materials on a regular basis, make sure you have the right PSU for the job, as some are not built for high-temperature prints. Cheap 3D printers often come with an underpowered PSU good enough for PLA but not for ABS and other materials that need sustained heating for an extended period. Also, make sure the PSU is compatible with the voltage used in the country you live in. A lot of users have made the mistake of not paying attention to the voltage setting before plugging their machines into a power socket.

1.10 Bed



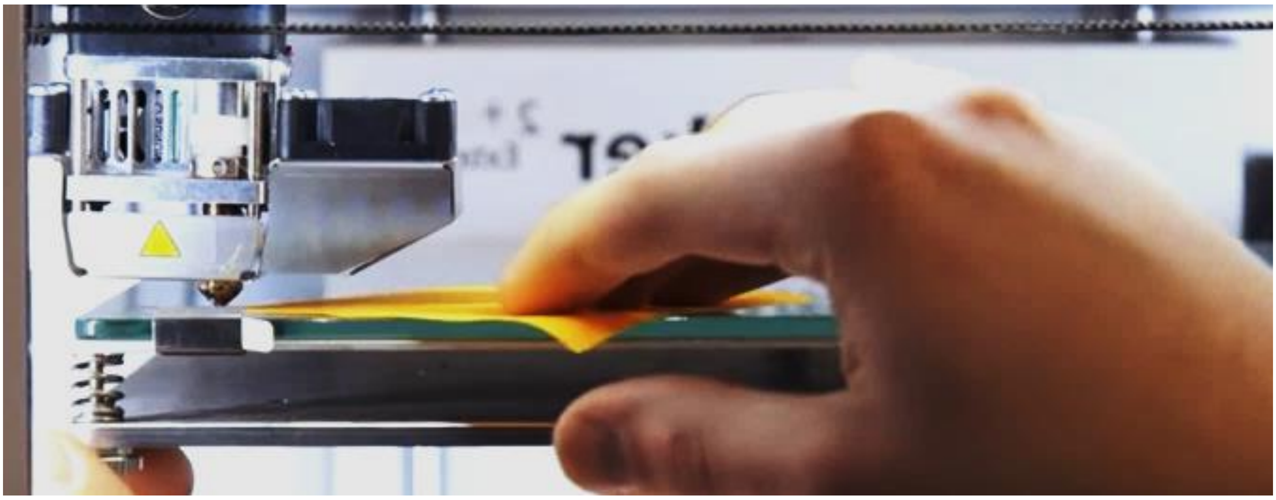
The print bed is where the extruder deposits the filament to form a solid object. Calling back to the 2D printer analogy earlier, the print bed is the equivalent of a piece of paper. It's either heated or non-heated, with the latter being common among starter 3D printers. A non-heated print bed is good enough for PLA, but for high-temperature materials, a heated print bed is a must to cut down on warping issues, improving the overall print quality.

Most 3D printers come with an aluminum print bed, but there are also models that have a glass print bed out of the box. Both types of print beds come with pros and cons. An aluminum print bed heats up faster while a glass print bed is flatter and easier to maintain. When choosing between the two, it's often a matter of personal preference.

The top part of the bed could be called the print bed surface. As the name suggests, the print bed surface or build surface is what goes on top of the print bed. It fixes the print on the bed helping the object being printed stick to the platform and allowing for easier removal of completed objects. There are different types of print surfaces, they can be in different materials, and can be fixed or removable. Also, it can be disposable as seller tape or durable as glass.

All types of print surfaces have pros and cons, so your choice depends on personal preference and, also, the type of material you want to print with. When the print surface is not sticky enough or has too much adhesion, users often resort to other materials for additional effectiveness, the most popular of which are hairspray and glue.

1.11 Bed leveling system

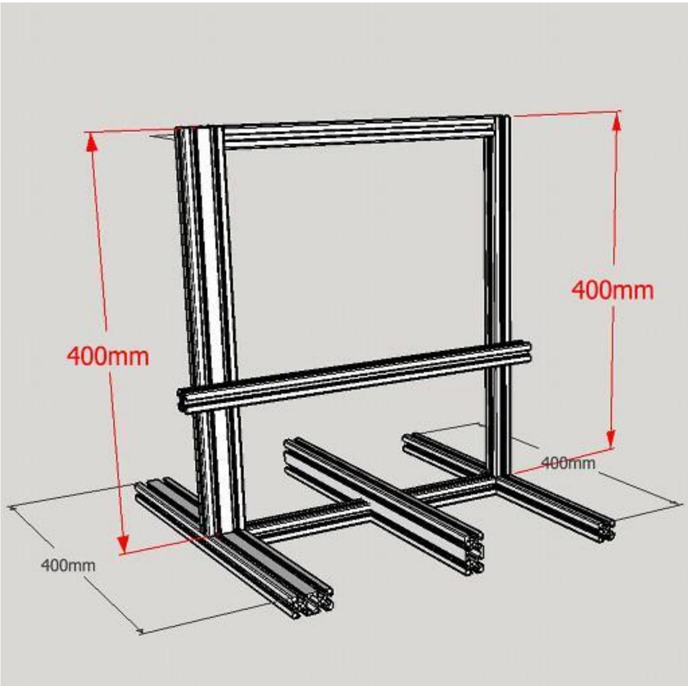


Font:https://www.matterhackers.com/articles/3d-printer-bed-leveling

To guarantee an accurate 3d printing, it must be necessaire to guarantee the orthogonality between all the axes and, in particular, between the bed and the nozzle. For this reason, the bed leveling system allows making uniform the distance between the nozzle e the printing surface. The setting of this parameter could be manual or automatic. A 3D printer with a manual bed leveling system has a set of thumbwheels under the print bed. These small mechanisms are used to adjust the alignment of the print bed. Some 3D printers are easy to level while others can be a pain, sometimes due to the poor design of the thumbwheels. On the other hand, there are 3d Printers with an automatic bed leveling system, thanks to a proximity sensor that understands, corrects, and sets a uniform distance between the parts. Usually, a correct distance is equal to the thickness of a sheet of paper.

A 3D printer with an automatic bed leveling system or an assisted manual bed leveling system also comes with a sensor or probe on the print head. An auto-level probe scans multiple points on the print bed to determine the alignment of the build platform. A 3D printer with an auto-leveling system does all the hard work, while a 3D printer with an assisted leveling system still requires a manually adjusted by the use of the thumbwheels.

1.12 Frame



The frame is the chassis of the 3D printer. It holds the other components together and is directly responsible for the stability and durability of the machine. These days, 3D printer frames are made of either acrylic or metal, but in the early days of consumer-level 3D printers, wood is often the go-to frame material.

3D printers with a metal frame are the most recommended simply because they are more stable and more durable. Going for a metal-framed 3D printer doesn't necessarily mean having to cough up a lot of money, though. There are budget 3D printers under 300 € that come with an aluminum frame.

Some 3D printers also have an enclosed frame, which protects them from dust and other particles as well as curious fingers who have no business being near the heated components. An enclosure allows for a more stable temperature in the print area, which is beneficial to certain advanced materials. There are semi-enclosed 3D printers, too, which usually come with covered sides but have an open front and/or top.

1.13 Motion Components

The motion components are the parts responsible for the movement of the 3D printer in the three axes. They are the ones that move the print bed and the print head (it depends on the type and model of the printer). Basically, the controller board directs how the 3D printer should move while the motion components are the ones that allow doing the actual moving reducing the friction effect while the parts turning or slide.

2 Materials

2.1 Introduction to Materials.

A 3D printer opens the door to a universe of possibilities. Be it something functional like prosthetics or recreational like tabletop gaming parts, there's one common need tying it all together: 3D printer filament. There is an abundance of 3D printer filament choices available. Here, we will cover the common "daily driver" filaments like PLA and PETG, plus the fancy stuff that lets you get real. Based on the technology, it is possible to print any kind of material, because it just means depositing any kind of material on a surface. In this way, it is possible to print from concrete to chocolate, from ceramic to iron. For FDM technology it is common to use plastic filaments, sometimes pure (such as PLA, ABS PETG,…), sometimes mixed with different materials (such as wood powder, aluminum, or carbenium). In addition to the thermoplastics that comprise the common 3D printer filament types (such as the aforementioned PLA and PETG), 3D printer filament can be (or consist of) the likes of nylon, polycarbonate, carbon fiber, polypropylene, and many more. There are even special blends that can conduct electricity or glow in the dark. With so much variety on offer, it's easier than ever to create functional, visually striking, and high-performing prints in a variety of exciting materials. With this in mind, the following is a list of the 3D printer filament. Split over three sections, and you'll find 25 filament material categories in total.

2.2 PLA

2.2.1 What is PLA?

In the realm of consumer 3D printing, polylactic acid (PLA) is king. Although it's often compared to ABS – arguably the next in line to the throne – PLA is easily the most popular 3D printer filament type, and for good reason.

2.2.2 More information

First and foremost, PLA is easy to print with. It has a lower printing temperature than ABS, and it doesn't warp as easily, meaning it doesn't require a heating bed (although it helps). Another benefit of using PLA is that it doesn't give off an offputting odor during printing (unlike ABS). It is generally considered an odorless filament, but many have reported smelling sweet *candy-like* fumes depending on the type of PLA.

Another appealing aspect of PLA is that it's available in a nearly endless abundance of colors and styles. As you'll see in the exotics sections, many of these specialty filaments use PLA as the base material, such as those with conductive or glow-in-the-dark properties, or those infused with wood or metal.

Finally, as a biodegradable thermoplastic, PLA is more environmentally friendly than most types of 3D printer filament, being made from annually renewable resources such as corn starch or sugar cane.

2.2.3 When should I use PLA 3d printer filament?

In this case, the better question might be, *When shouldn't I use PLA?* Compared to other types of 3D printer filament, PLA is brittle, so avoid using it when making items that might be bent, twisted, or dropped repeatedly, such as phone cases, high-wear toys, or tool handles.

You should also avoid using it with items that need to withstand higher temperatures, as PLA tends to deform around temperatures of 60°C or higher. For all other applications, PLA makes for a good overall choice in 3D printer filament.

Common prints include models, low-wear toys, prototype parts, and containers.

2.3 ABS

2.3.1 What is ABS?

Acrylonitrile butadiene styrene (ABS) typically ranks as the second most popular 3D printer filament, after PLA. But that just means it's the second most commonly *used*. Concerning its material properties, ABS is actually moderately superior to PLA, despite being slightly more difficult to print with. It's for this reason that ABS is found in many manufactured household and consumer goods, including LEGO bricks and bicycle helmets!

2.3.2 More information

Products made of ABS boast high durability and a capacity to withstand high temperatures, but 3D printer enthusiasts should be mindful of the filament's high printing temperature, tendency to warp during cooling, and intense, potentially hazardous fumes. Be sure to print with a heated bed and in a well-ventilated space (or with an enclosure).

2.3.3 When should I use ABS 3d printer filament?

ABS is tough – able to withstand high stress and temperature. It's also moderately flexible, though there are certainly better options for that further down the list. Together these properties make ABS a good general-purpose 3D printer filament, but where it really shines is with items that are frequently handled, dropped, or heated. Examples include phone cases, high-wear toys, tool handles, automotive trim components, and electrical enclosures.

2.4 PETG (PET, PETT)

2.4.1 What is PETG?

Polyethylene terephthalate (PET) is the most commonly used plastic in the world. Best known as the polymer used in water bottles, it is also found in clothing fibers and food containers. While “raw” PET is rarely used in 3D printing, its variant PETG is an increasingly popular 3D printer filament.

2.5 More information

The ‘G’ in PETG stands for “glycol-modified”, and the result is a filament that is clearer, less brittle, and most importantly, easier to print with than its base form. For this reason, PETG is often considered a good middle ground between ABS and PLA, the two most commonly used types of 3D printer filament, as it is more flexible and durable than PLA and easier to print than ABS.

Three things 3D printer enthusiasts should keep in mind when using PETG:

PETG is *hygroscopic*, meaning it absorbs moisture from the air. As this has a negative effect on the material, make sure to store the 3D printer filament in a cool, dry place.

PETG is sticky when it’s being printed, making this 3D printer filament a poor choice for support structures, but good for layer adhesion. (Just be careful with the print bed!)

Though not brittle, PETG scratches more easily than ABS.

Polyethylene coTrimethylene Terephthalate (PETT) is another PET variant. Slightly more rigid than PETG, this 3D printer filament is popular for being transparent.

2.6 When should I use PETG (pet, pett) 3d printer filament?

PETG is a good all-rounder but stands out from many other types of 3D printer filament due to its flexibility, strength, and resistance to both high temperature and impact. This makes it an ideal 3D printer filament to use for functional objects which might experience sustained or sudden stress, such as mechanical parts, printer parts, and protective components.

2.7 TPE, TPU, TPC (Flexible)

2.7.1 What is TPE?

As the name implies, thermoplastic elastomers (TPE) are essentially plastics with rubber-like qualities, making them extremely flexible and durable. As such, TPE is commonly used to produce automotive parts, household appliances, and medical supplies.

2.7.2 More information

In reality, TPE is a broad class of copolymers (and polymer mixtures), but it is nonetheless used to label many commercially available types of 3D printer filament. Soft and stretchable, these filaments can withstand the kind of physical punishment that neither ABS nor PLA can tolerate. On the other hand, printing is not always easy, as TPE can be difficult to extrude.

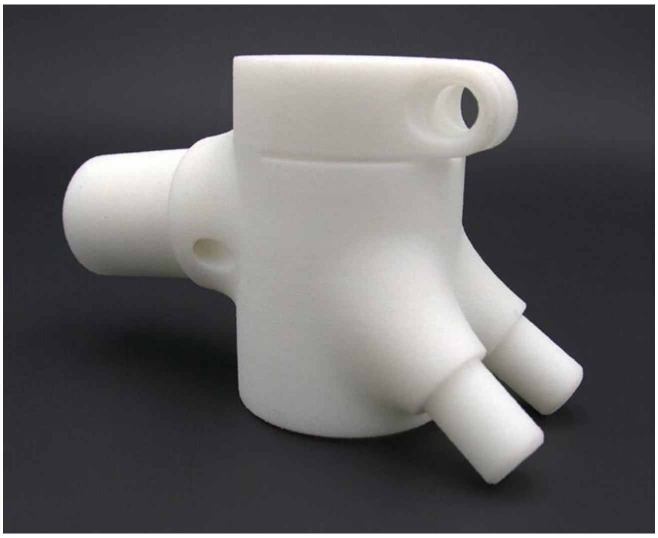
Thermoplastic polyurethane (TPU) is a particular variety of TPE and is itself a popular 3D printer filament. Compared to generic TPE, TPU is slightly more rigid – making it easier to print. It's also a little more durable and can better retain its elasticity in the cold.

Thermoplastic copolyester (TPC) is another variety of TPE, though not as commonly used as TPU. Similar in most respects to TPE, TPC's main advantage is its higher resistance to chemical and UV exposure, as well as heat (up to 150°C).

2.7.3 When should I use tpe, tpu, or TPC 3d printer filament?

Use TPE or TPU when creating objects that need to take a lot of wear. If your 3D printed part will bend, stretch, or compress, these 3D printer filaments should be up for the task. Example prints might include toys, phone cases, or wearables (like wristbands). TPC can be used for similar applications, but does especially well in harsher environments, like the outdoors.

2.8 Nylon



FONT:HTTPS://IMAGES.APP.GOO.GL/MWMSUIASBYYFKAN26

2.8.1 What is nylon?

Nylon, a popular family of synthetic polymers used in many industrial applications, is the heavyweight champion of the professional 3D printing world. Compared to most other types of 3D printer filament, it ranks as the number one contender when together considering strength, flexibility, and durability.

2.8.2 More information

Another unique characteristic of this 3D printer filament is that you can dye it, either before or after the printing process. The negative side to this is that nylon, like PETG, is *hygroscopic*, meaning it absorbs moisture, so remember to store it in a cool, dry place to keep the filament in prime condition, ensuring better quality prints.

In general, many grades of nylon exist, but among the most common for use as a 3D printer, filament are 618 and 645.

2.8.3 When should I use nylon 3d printer filament?

Taking advantage of nylon's strength, flexibility, and durability, this type of 3D printer filament can be used to create tools, functional prototypes, or mechanical parts (like hinges, buckles, or gears).

2.9 PC (Polycarbonate)

2.9.1 What is PC?

Polycarbonate (PC), in addition to being one of the strongest 3D printer filaments presented in this list, is extremely durable and resistant to both physical impact and heat, able to withstand temperatures of up to 110°C. It's also transparent, which explains its use in commercial items such as bulletproof glass, scuba masks, and electronic display screens.

2.9.2 More information

Despite being featured in similar use cases, PC shouldn't be confused with acrylic or plexiglass, which tend to shatter or crack under stress. Unlike these two materials, PC is moderately flexible (though not as much as nylon, for example), allowing it to bend until it eventually deforms.

PC 3D printer filament is *hygroscopic* and able to absorb water from the air, so remember to store it in a cool, dry place to ensure better quality prints.

2.9.3 When should I use PC 3d printer filament?

Due to its physical properties, the PC is an ideal 3D printer filament for parts that need to retain their strength, toughness, and shape in high-temperature environments, such as electrical, mechanical, or automotive components. You can also leverage its optical clarity for lighting projects, screens, and other applications that call for transparency.

2.10 Wood

2.10.1 What is wood filament?

Interested in printing objects that look and feel like wood? Well, you can! It's not really wood of course – that wouldn't make for a very good 3D printer filament – it's PLA infused with wood fiber.

2.10.2 More information

Many wood-PLA 3D printer filament blends exist on the market today. These include the more standard wood varieties, such as Pine, Birch, Cedar, Ebony, and Willow, but the range also extends itself to less common types, like Bamboo, Cherry, Coconut, Cork, and Olive.

As with other types of 3D printer filament, there is a trade-off with using wood. In this case, the aesthetic and tactile appeal comes at the cost of reduced flexibility and strength.

Be careful with the temperature at which you print wood, as too much heat can result in an almost burnt or caramelized appearance. On the other hand, the base appearance of your wooden creations can be greatly improved with a little post-print processing! The wood filament can also cause wear and tear to your 3D printer nozzle, so keep that in mind before using this material.

2.10.3 When should I use wood filament?

Wood is popular with items that are appreciated less for their functional capabilities, and more for their natural appearance. Consider using wood 3D printer filament when printing objects that are displayed on a desk, table, or shelf. Examples include bowls, figurines, and awards. One really creative application of wood as a 3D printer filament is in the creation of scale models, such as those used in architecture.

2.11 Metal

2.11.1 What is metal filament?

Maybe you're looking for a different type of aesthetic in your prints — something a little bulkier and shinier. Well, for that you can use metal. Like wood 3D printer filament, the metal filament isn't fully made from metal. It's actually a mix of metal powder and either PLA or ABS. But that doesn't stop the results from having the look and feel of metal.

Even the weight is metal-like, as blends tend to be several times denser than pure PLA or ABS.

2.11.2 More information

Bronze, brass, copper, aluminum, and stainless steel are just a few of the varieties of metal 3D printer filaments that are commercially available. And if there's a specific look you're interested in, don't be afraid to polish, weather, or tarnish your metal items after printing – a little post-processing can go a long way.

You may need to replace your nozzle a little sooner as a result of printing with metal, as the grains are somewhat abrasive, resulting in increased nozzle wear.

The most common 3D printer filament blends tend to be around 50% metal powder and 50% PLA or ABS, but blends also exist that are up to 85% metal.

2.11.3 When should I use metal filament?

Metal can be used to print for aesthetics and functionality. Figurines, models, toys, and tokens can all look great when 3D printed in metal. And as long as they don't have to deal with too much stress, feel free to use metal 3D printer filament to create parts with purpose, like tools, grates, or finishing components.

2.12 Biodegradable (bioFila)

2.12.1 What is biodegradable filament?

Biodegradable 3D printer filaments make up a unique category, as their most valuable characteristic does not lie in their physical natures. As most hobbyists can attest to, not every print turns out the way you want it to, and this results in having to throw away a ton of plastic. Biodegradable filaments seek to negate the environmental impact that plastic waste has on our planet.

2.12.2 More information

As was mentioned earlier in this article, PLA is a biodegradable filament, but others include twoBEars' bioFila line and Biome3D, by Biome Bioplastics.

2.12.3 When using biodegradable 3d printer filament?

Regardless of their primary reason for existing being environmentally friendly, biodegradable 3D printer filament types can still produce items of sound physical quality. Use them any time you don't have specific requirements for strength, flexibility, or endurance. And if you really want to take advantage of the guilt-free printing biodegradable filaments offer, try using them in projects which require prototyping.

2.13 Glow-in-the-Dark

2.13.1 What is glow-in-the-dark filament?

Glow-in-the-dark 3D printer filament – pretty self-explanatory. Leave your print in the light for a while, then flick the switch and behold that eerie green glow.

It doesn't have to be green, of course. Other glow-in-the-dark filament colors include blue, red, pink, yellow, or orange. But green tends to be the most popular and replicates that classic style of glow.

2.13.2 More information

So, how does it work? It all comes down to the phosphorescent materials mixed in with the PLA or ABS base. Thanks to these added materials, a glow-in-the-dark 3D printer filament is able to absorb and later emit photons, which are kind of like tiny particles of light. This is why your prints will only glow after being in the light – they have to *store* the energy before they can *release* it.

For best results, consider printing with thick walls and little infill. The thicker your walls, the stronger the glow!

2.13.3 When should I use glow-in-the-dark 3d printer filament?

Thinking about that eerie green glow, it almost doesn't even seem necessary to suggest using a glow-in-the-dark 3D printer filament for Halloween projects, like jack-o'-lanterns or window decorations.

Other examples of where these filaments really shine – er, *glow* – include wearables (think jewelry), toys, and figurines.

2.14 Magnetic

2.14.1 What is magnetic filament?

Are metal and conductive prints not exciting enough for you? Okay then, how about magnetic prints? This exotic 3D printer filament, a cross between PLA or ABS and powdered iron, features a grainy, gunmetal finish, and of course, it sticks to magnets!

2.14.2 More information

One thing to note: Despite the name, this 3D printer filament type is actually *ferromagnetic*, meaning that while it is attracted to magnetic fields, it has no fields of its own. In other words, the objects you print may *stick* to magnets, but they won't actually *be* magnets.

2.14.3 When should I use magnetic 3d printer filament?

Use this type of 3D printer filament whenever you want your prints to stick to something magnetic. Ornaments (especially for the fridge) are the most obvious example, but why not incorporate some magnetism into toys or tools?

2.15 Color-Changing

2.15.1 What is color-changing filament?

Remember those T-shirts from the 80s, the ones that would change color based on body temperature? Or how about mood rings? Well, this is the same idea, because color-changing 3D printer filaments also change color based on temperature changes.

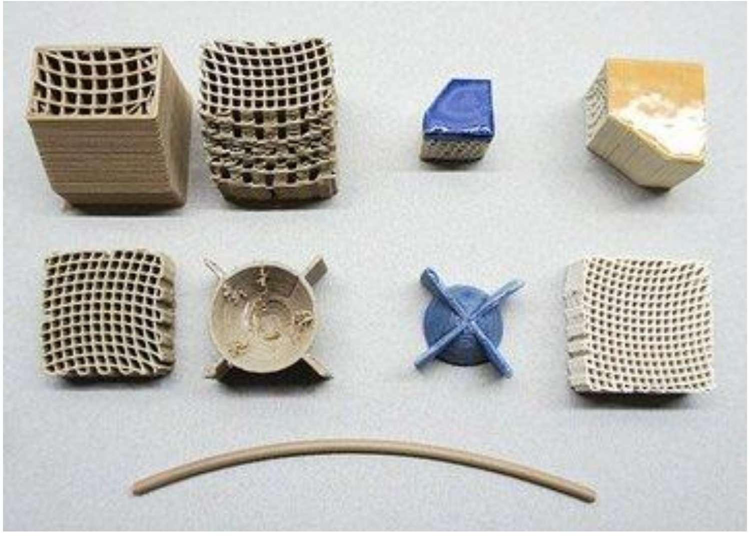
2.15.2 More information

Filaments from this category tend to change between a gradient of two colors, for example from purple to pink, blue to green, or yellow to green. As with other exotic types of 3D printer filament, color-changing filament exists in blends of both PLA and ABS.

2.15.3 When should I use color-changing 3d printer filament?

With no special physical, tactile, or functional characteristics, this type of 3D printer filament is purely designed for aesthetically-driven applications. Use it whenever you would normally use PLA or ABS, but desire that extra visual flare. Good candidate projects include phone cases, wearables, toys, and containers.

2.16 Clay/Ceramic



2.16.1 What is clay/ceramic filament?

As evidenced by this article, plastic tends to dominate 3D printing as the primary print material. We've explored some other non-plastic options already, and here's another: clay. Boasting earthenware properties, clay 3D printing filament typically contains a mixture of clay and polymer.

2.16.2 More information

There are a few different companies offering stone/earthen material-based filaments, with clay (often marketed as ceramic) being the one with perhaps the strongest use case: faux-pottery.

A common characteristic shared between these filaments is brittleness, meaning care is required to properly handle and print them.

Lay Filament's LAYCeramic is one example of a ceramic filament that achieves near-authentic results. Fireable in a kiln after printing, the polymer binding the ceramic particles within de-binds to leave behind a slightly shrunken, but the hardened, final print can be spruced up with a ceramic glaze and other post-processing effects.

2.16.3 When should I use clay/ceramic 3d printer filament?

When you're looking for a handmade earthenware look paired with the impossibly precise repeatability 3D printing gives.

2.17 Carbon Fiber

2.17.1 What is carbon fiber filament?

When types of 3D printer filament like PLA, ABS, PETG, and nylon are reinforced with carbon fiber, the result is an extremely stiff and rigid material with relatively little weight. Such compounds shine in structural applications that must withstand a wide variety of end-use applications.

2.17.2 More information

The trade-off is the increased wear and tear on your printer's nozzle, especially if it's made of a soft metal like brass. Even as little as 500 grams of this exotic 3D printer filament will noticeably increase the diameter of a brass nozzle, so unless you enjoy frequently replacing your nozzle, consider using one made of (or coated with) a harder material.

2.17.3 When should I use carbon fiber 3d printer filament?

Thanks to its structural strength and low density, carbon fiber is a fantastic candidate for mechanical components. Looking to replace a part in your model car or plane? Give this 3D printer filament a try.

2.18 PVA

2.18.1 What is PVA?

Polyvinyl alcohol (PVA) is soluble in water, and that's exactly what commercial applications take advantage of. Popular uses include packaging for dishwasher detergent "pods" or bags full of fishing bait. (Throw the bag in water and watch it dissolve, releasing the bait.)

2.18.2 More information

The same principle applies in 3D printing, making PVA a great support material when paired with another 3D printer filament in a dual extrusion 3D printer. The advantage of using PVA over HIPS is that it can be used to support more materials than just ABS.

The trade-off is a 3D printer filament that is slightly more difficult to handle. One must also be careful when storing it, as the moisture in the atmosphere can damage the filament before printing. Dry boxes and silica pouches are a must if you plan to keep a spool of PVA usable in the long run.

2.18.3 When should I use PVA 3d printer filament?

PVA filament is a great choice as support material on complex prints with overhangs.

2.19 Polypropylene (PP)

2.19.1 What is PP?

Polypropylene (PP) is tough, flexible, light, chemically resistant, and food-safe, which might explain its broad range of applications, including engineering plastics, food packaging, textiles, and banknotes.

2.19.2 More information

Unfortunately, like a 3D printer filament type, PP is notoriously difficult to print with, often presenting heavy warpage and lackluster layer adhesion. If not for these issues, PP may have contended with

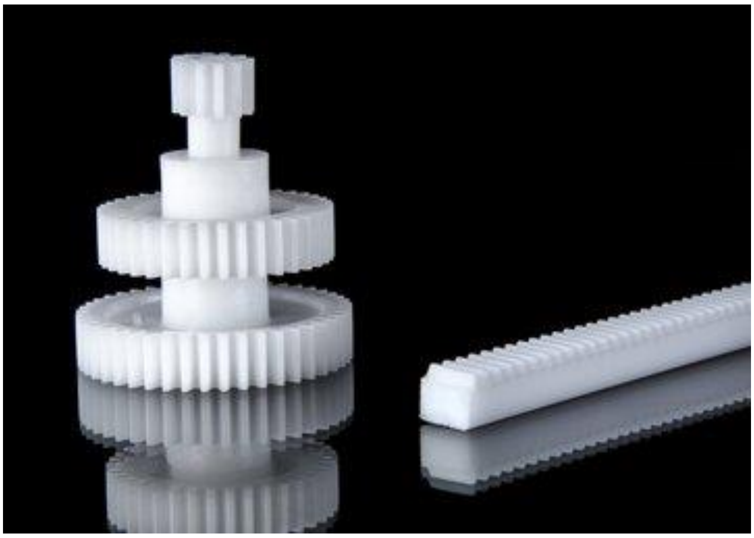
PLA and ABS for the most popular 3D printer filament types, given its strong mechanical and chemical properties.

Interestingly, since many household objects are made of PP, it's actually possible to recycle old junk and turn it into a new 3D printer filament.

2.19.3 When should I use pp 3d printer filament?

If you can wrest PP's warping under control, then most prints calling for a hardy and light material would suit PP. It's important to note however that while the material sees great use in the packaging of consumables and medicine for its food-safe properties, the process of FDM 3D printing negates this with hundreds (if not thousands) of layer lines for bacteria to hang out in — best not to try.

2.20 Acetal (POM)



2.20.1 What is acetal (POM) filament?

Polyoxymethylene (POM), also referred to as acetal and Delrin, is well known for its use as an engineering plastic, for example in parts that move or require high precision.

2.20.2 More information

Acetal as a material sees common use as gears, bearings, camera focusing mechanisms, and zippers.

POM performs exceptionally well in these types of applications due to its strength, rigidity, resistance to wear, and most importantly, its low coefficient of friction. It's thanks to this last property that POM makes such a great 3D printer filament.

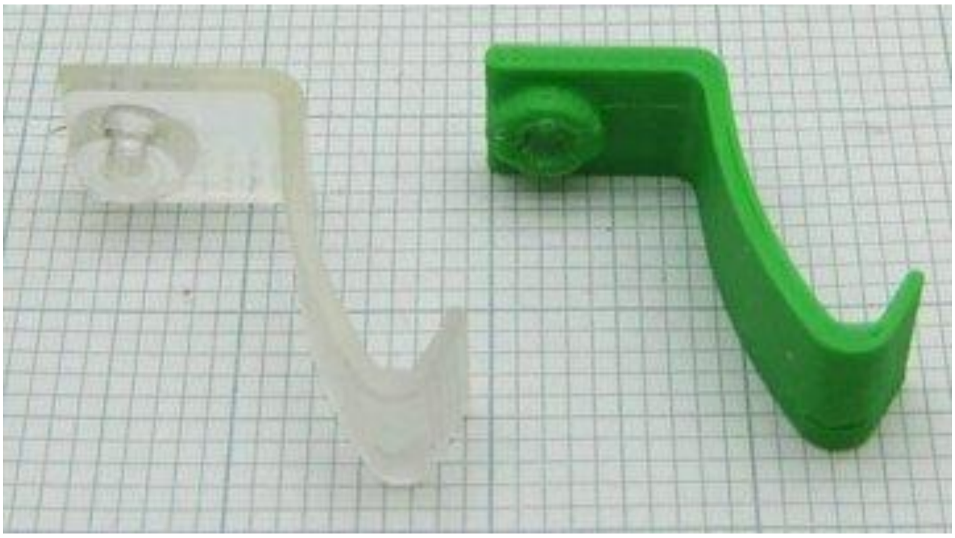
For most of the types of 3D printer filament in this list, there is a significant gap between what is made in the industry and what you can make at home with your 3D printer. For POM, this gap is somewhat smaller; the slippery nature of this material means prints can be nearly as functional as mass-produced parts.

Make sure to use a heated print bed when printing with POM 3D printer filament, as the first layer doesn't always want to stick.

2.20.3 When should I use acetal (pom) 3d printer filament?

Any moving parts need to be low friction and tough. We imagine gearing mechanisms in projects using motors (such as RC cars) could be an applicable field for POM.

2.21 PMMA (Acrylic)



2.21.1 What is PMMA filament?

Ever heard of polymethyl methacrylate (PMMA)? Maybe not. What about acrylic, or *Plexiglas*? That's right, we're talking about the same material that's most often used as a lightweight, shatter-resistant alternative to glass.

2.21.2 More information

3D printing with PMMA 3D printer filament can be a little difficult. To prevent warping and to maximize clarity, extrusion must be consistent, which requires a high nozzle temperature. It might also help to enclose the print chamber in order to better regulate cooling.

2.21.3 When should I use pmma 3d printer filament?

Rigid, impact-resistant, and transparent, use this 3D printer filament for anything that should diffuse light, whether that's a replacement windowpane or a colorful toy. Just don't use it to make anything that should bend, as PMMA is not very flexible.

2.22 FPE

2.22.1 What is FPE filament?

Flexible polyester (FPE) is a generic label given to a 3D printer filament that combines rigid and soft polymers. Such filaments are comparable to PLA but are softer and more flexible. The specific flexibility depends on the hard and soft polymers used, and on the ratio between them.

2.22.2 More information

Two notable aspects of FPE include good layer-to-layer adhesion and moderately high resistance to heat and a variety of chemical compounds. Given the wide range of FPE 3D printer filament that is available, perhaps the most useful way to differentiate between the wide range of FPE available is the Shore value (like 85A or 60D), where a higher number indicates less flexibility.

To summarise, following a table that expresses simply temperatures and costs for each of the principal materials. It is a possible understanding of how with a maximum temperature of at least 260°C it is possible printing a huge possibility of materials.

Plastic	Product name	Supplier	Color	Cost [$/kg][a)]	Print T [^{o}C]
PLA	Polylite PLA	Polymaker	True Blue	25	205
PETG	PETG	Octofiber	Natural	53	225
Eastman Amphora 3300	nGen	Colorfabb	Lulzbot green	52	230
Eastman Amphora 3300	nGen	Colorfabb	Red	52	230
PP	PP	Ultimaker	Natural	98	235
ABS	ABS	IC3D	Green	40	245
Eastman Amphora 1800	Inova-1800	Chroma Strand	Blue	80	245
ASA	ASA Extrafill	Fillamentum	Traffic Black	42	250
Polyamide copolymer-Nylon 6/69	Alloy 910	taulman3D	Black	79	255
PET	t-glase	taulman3D	Green	66	255
PC	PC-Max	Polymaker	Black	61	255

Font: https://www.researchgate.net/figure/3-D-printing-materials-arranged-by-3-D-printing-nozzle-temperature_tbl1_326697946

3 PART DESIGN

A 3D Printer without a 3D Model to be printed is not really useful. There are different ways to obtain an object to print, in general, it is possible to divide it into 3 different possibilities:

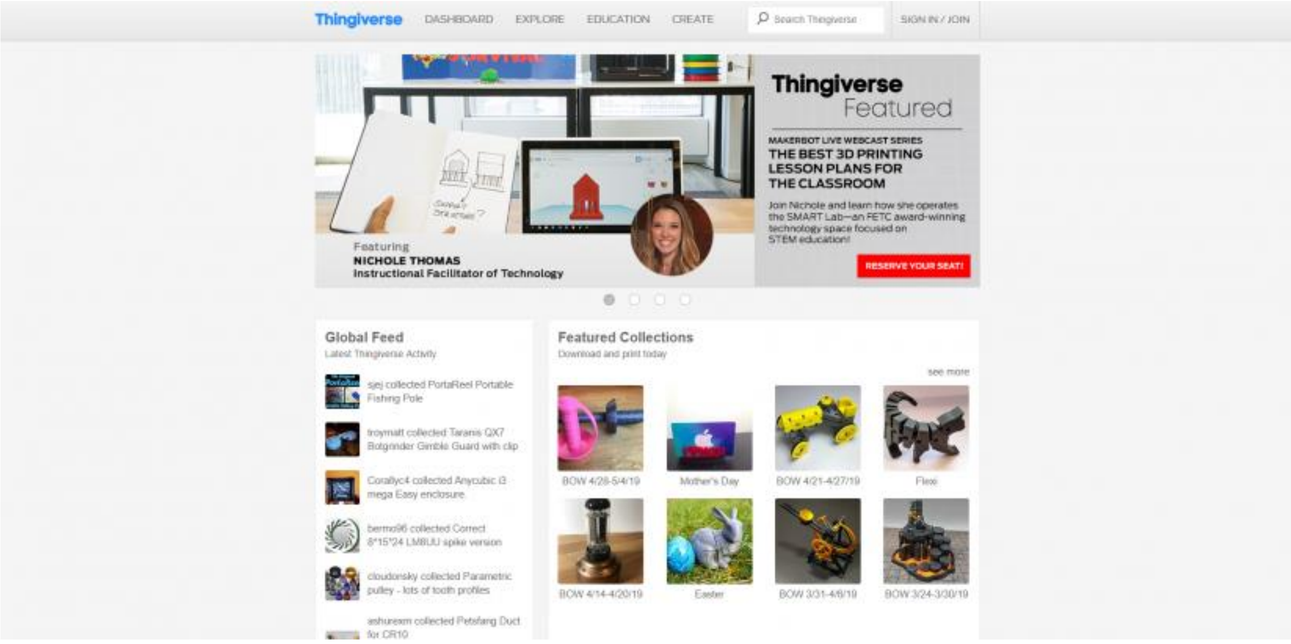
1. Download an existing object
2. Scan a real object
3. Designingign an object

Let's see deeply what the difference between each group is.

3.1 Download an existing object

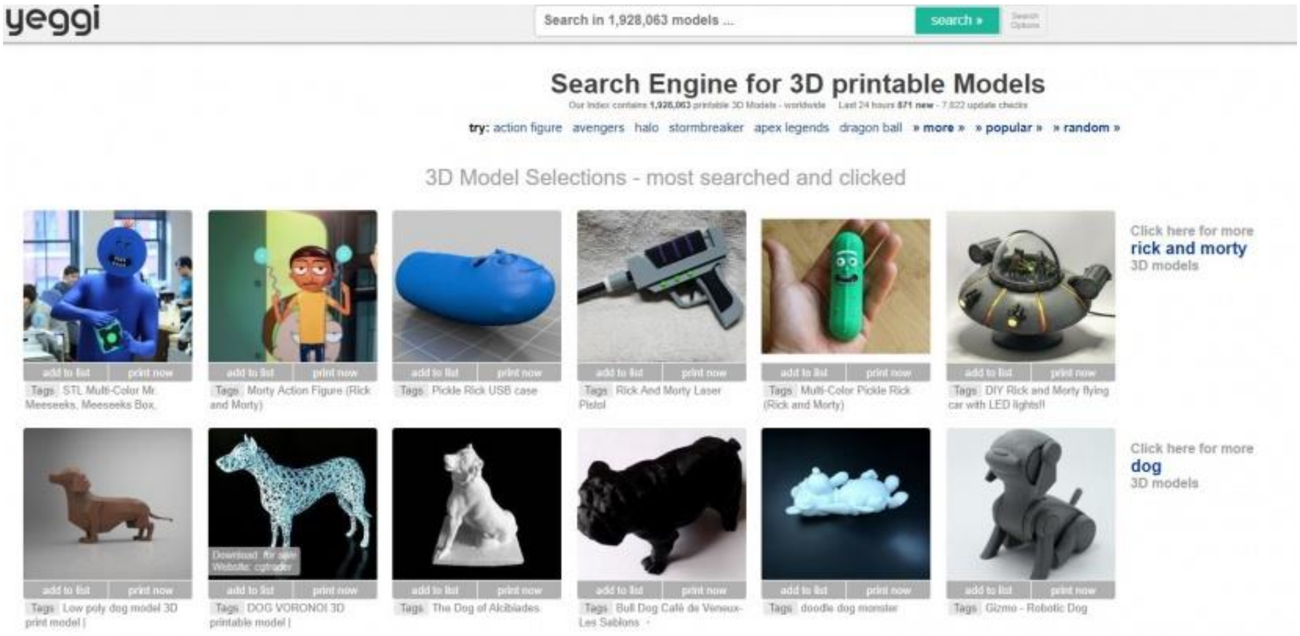
Speaking of 3D printing usually means speaking of sharing. Internet is full of 3D models to be printed and there are lots of repositories where it is possible finding different kind of 3d models in each sector, from DIY to cooking, from automotive to gadgets, from toys to tools. Anyone can upload his own model online or download a one, modify it and upload it again. There are free or payable models based on the platform and the business model of the repository where you find it. Following is a list of the most important repository for 3D Printing websites:

3.1.1 Thingiverse



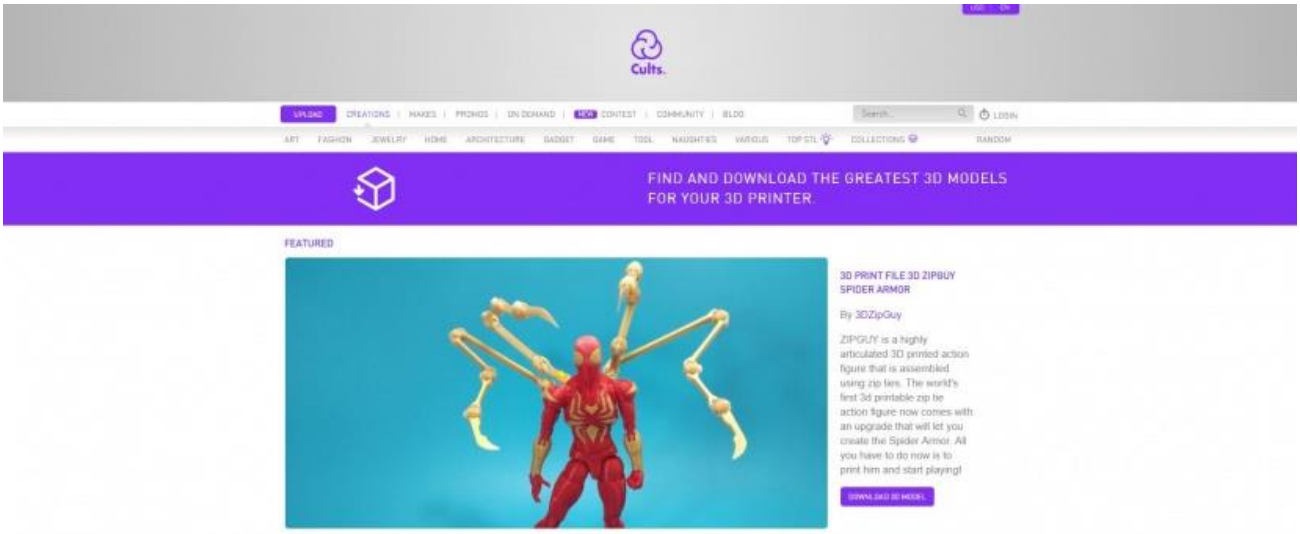
While Thingiverse isn't the only source of 3D printer files, it's easily the best website for 3D printing models. Here, you'll find an excellent collection of 3D printable downloads. It's completely free and boasts over a million 3D models for use. It's easy to navigate, and the homepage includes featured collections with curated holiday designs and more. In a few clicks, you'll be well on your way to perusing neat 3D printable projects. Although everything is completely free to download, you can, and should, tip designers. You can download files, like projects, add them to collections, and even remix creations for a fresh spin on a favorite upload. Aside from files, you'll find tutorials and educational content. Its education section is chock full of nifty resources for 3D printing DIYers of all skill levels. Since MakerBot runs Thingiverse, it's no surprise this is the best 3D source of 3D printable files. It's a must-bookmark.

3.1.2 Yeggi



Yeggi remains one of the top 3D printable model search engines available. It's a bit different from sites such as Thingiverse. Rather than a dedicated site for 3D STL files and other 3D printing downloads, it's essentially the Google of 3D printing. Merely search Yeggi, then download your files. It's a fantastic, comprehensive 3D printing search engine.

3.1.3 Cults

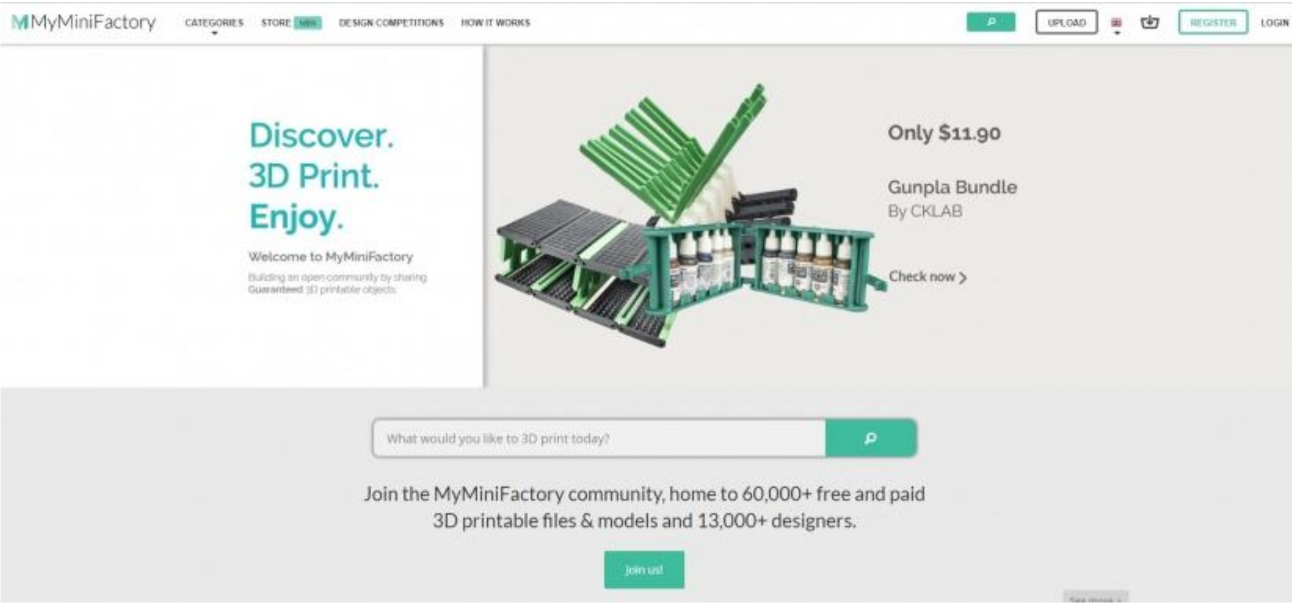


On Cults, you'll find a bevy of downloadable 3D printing assets. There's everything from basic designs to top-notch, professional-caliber designs. When uploaded, designs are reviewed for printability, and site organization is superb. You'll be able to sort by content types, such as art, fashion, jewelry, and architecture. Among the superb projects, you will find on Cults 3D.

3.1.4 Pinshape

Over on Pinshape, you'll find thousands of free and paid STL files for download. Whereas many 3D printing asset websites boast mostly free artwork, Pinshape's premium files section offers paid downloads. This is a fantastic touch that supports content creators. Moreover, its 3DPrinterOS cloud software allows creators to stream designs from the Pinshape marketplace directly to many popular 3D printers. It's a huge innovation in the 3D printing space. Like Cults, navigation is smooth, and it's a fantastic community.

3.1.5 MyMiniFactory

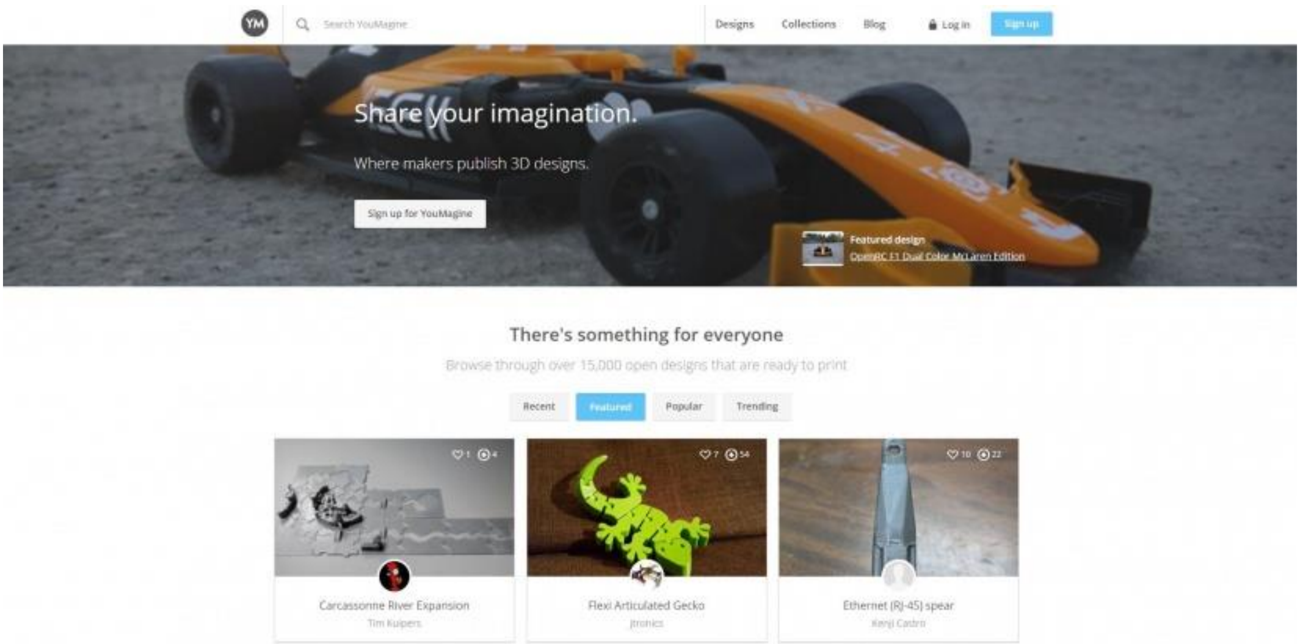


With over 60,000 free and premium paid 3D printable files from over 13,000 makers, there's a ton of utility for both printing enthusiasts and designers. Its competitions page ensures that DIYers stay on top of their game, and lends the opportunity to win prizes such as 3D printers. Plus, MyMiniFactory includes loads of categories, from jewelry to sports, upcycling, education, and even building a 3D printer.

3.1.6 Threeding

Featuring thousands of 3D printing assets to download, Threeding is one of the top 3D printing marketplaces around. Sporting both free and paid designs, you can download and print a variety of objects. The organization is spectacular, with categories such as Featured Models, New Models, Electronic & Technology, Art, and more. In addition to downloadable STL files, you'll find a robust blog with tons of 3D printing content including free 3D printing models of the week, innovations in 3D printing, and articles with the latest trends.

3.1.7 YouMagine

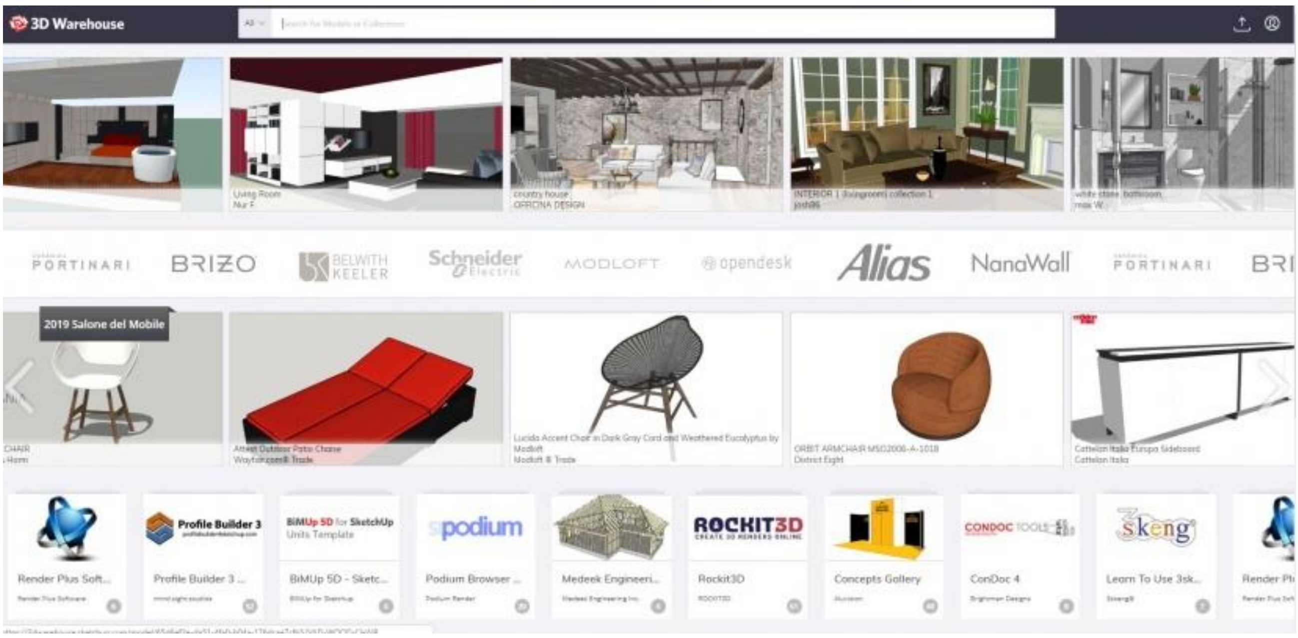


The cleverly-titled YouMagine posits itself as a space for makers to publish 3D designs. YouMagine hosts over 15,000 designs. Whereas many 3D printing sites offer both free and premium paid downloads, YouMagine hosts exclusively open files. As such, it's one of the best sites for 3D printer files. In a few clicks, you'll be on your way to churning out an OpenRC F1 dual-color McLaren Edition, Flexi articulated gecko, and tons of other neat 3D printable designs.

3.1.8 Shapetizer

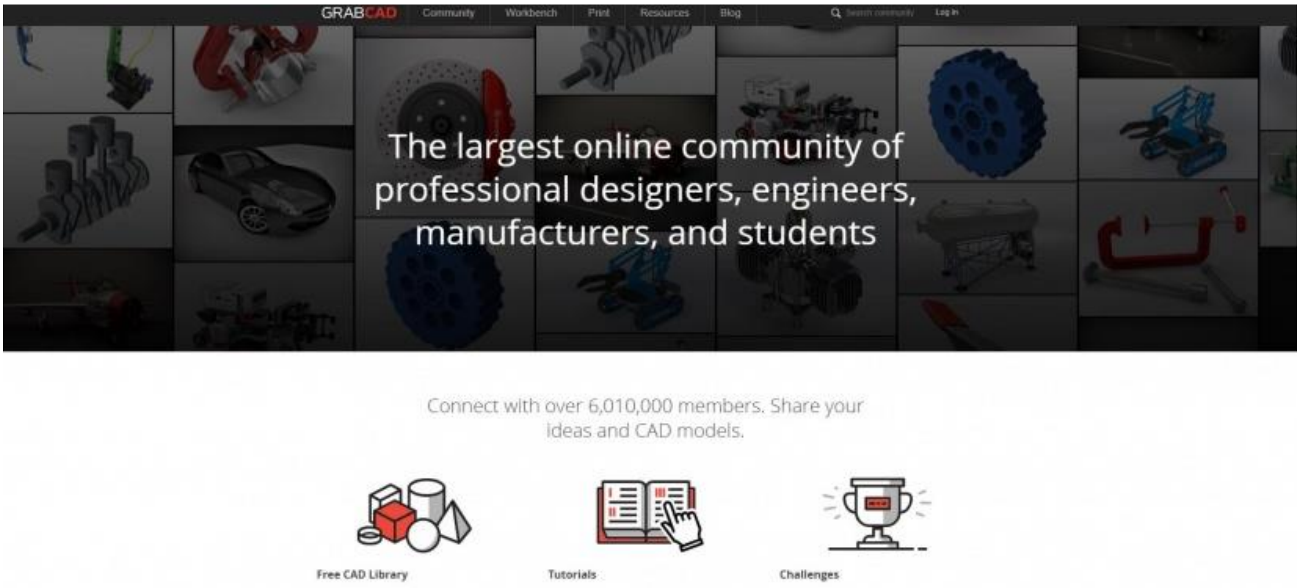
Shapetizer 3D Printing Market clocks in as one of the best sites for 3D printing models. A fusion of a free repository and premium marketplace, Shapetizer touts lush web design and high-quality 3D printing assets. While the majority of 3D printing communities highlight designs, Shapetizer includes featured designers, a refreshing twist that gives credit to creators, not just content.

3.1.9 3D Warehouse



From SketchUp comes 3D Warehouse, a comprehensive aggregate of 3D printable designs and files. Here, you'll find anything from models to collections, including keyboards, MAME cabinets, and more. Notably, 3D Warehouse features tons of designs for conceptual 3D printing assets. Check out its over 3 million 3D designs, all free.

3.1.10 GrabCAD



On GrabCAD, you'll discover millions of completely free 3D printable models and files. Easily one of the best 3D printing websites you can find, it's among the largest 3D printing online communities. Comprised of a mixed demographic, from engineers and professional designers to manufacturers, you'll find a beefy free CAD library, as well as tons of tutorials. Aside from files, you can connect with over 6 million members for ideation and collaboration.

3.1.11 STL Finder

The aptly named STL Finder is, well, a search engine for STL files. It's similar to Yeggi in that it's a search engine rather than a repository. Web design leaves a bit to be desired, but it's the Google of 3D printing. With a robust search feature including 3D model collections, search results, and the ability to favorite designs plus look at your search history, STL Finder is the definitive search engine for 3D models.

3.1.12 Embodi3D

Whereas the majority of websites for 3D printing files feature designs ranging from practical objects to pop culture miniatures. Embodi3D concentrates on fulfilling a different need. Medical 3D printing is on the rise, and Embodi3D (pronounced embodied), concentrates on just that. The biomedical 3D printing site offers loads of resources. In a few clicks, you can convert medical scans to 3D printable files, download 3D printable models of various organs, bones, and more, plus peruse medical 3D printing tutorials. Furthermore, Embodi3D boasts a robust blog page covering medical 3D printing topics such as the top free downloadable CT angiogram 3D printable models, how to create a dog skeleton model using 3D printing, and 3D printable muscular anatomy models.

3.2 Final Thoughts on Repositories

With the immense popularity, and increasing accessibility, of 3D printing, there are loads of websites for 3D printing models and assets. Whether you're seeking a 3D printable file search engine, a community for 3D printing, or something in-between, there's a website for you.

3.3 Scan a real object

Through 3D scanning, drawing it directly on a 3D modeling software, through websites with a library of files available where you can download the model to be printed.

3.3.1 How do 3D scanners work with 3D printers?

3D scanning can be an underappreciated workhorse supporting the magic of 3D printing. Some 3D printing projects don't need it, like those that use open source or purchased 3D models pre-made and ready to go. But when a project requires creating (or remixing to create) an original 3D model from a real-life object or source of inspiration, a 3D scanner can help leapfrog a ton of extra work. Without a 3D scanner, models have to be built from scratch using a design program. This means recording accurate measurements of your physical object to reference and drawing blueprints from which to work. Not to mention needing high-level 3D design skills.

3D scanners can help you:

- Digitize a physical object for mass production
- Speed up your prototyping or design process
- Model with a physical material rather than build from scratch in computer software

3.3.2 Bringing mass production to the masses with an object scanner.

Craftspeople can spend huge amounts of time hand-making individual functional or art objects for sale. Some artists will sell based on the idea of 'handmade' or enjoy the repeated process of creating their wares by hand. But there can come a time, to step up production to meet increased demand, or the desire for more time to experiment and develop new products, when an artisan or maker could stand to introduce a small-batch element of manufacturing to their business process. Scanning an object allows you to print multiple copies of a product quickly and efficiently, with the benefit of reproducing the best version of an item, eliminating the quality fluctuations that come with hand-made objects.

This is just the jumping-off point once you understand how to use a 3D scanner for 3D printing. With the help of computer programs like Autodesk and Meshlab, an artist can make variations on an existing product without needing to start from scratch. 3D scanning puts the design process in hyperdrive, eliminating work and creating new possibilities.

Let’s say it needs to duplicate or replace a small part for a project that, perhaps a rare part or a gear that can’t easily be picked up at the hardware store (and it cannot be bought). A 3D scanner can replicate the part as a 3D image file on the computer, which can then be printed in the material of your choice (with certain 3D printers you can print in metal, wood, ceramic, plastic, and more). Or it might have a slightly different part than you need, that almost fits into a project you building, but just needs a slight adjustment. Digitizing the part using a 3D scanner will allow you to make those small changes to the 3D model scan in software and print the modified part for you to use. Much faster than measuring and building the part from scratch in 3D.

3.3.3 A technological combo transforming every industry.

3D scanning and printing have unlocked a form of rapid prototyping and reverse-engineering all over the place in advanced manufacturing and production. In special cases, it’s being used to help change people’s lives in more personal ways. Ambionics, a company from North Wales currently in beta trials, creates customized prosthetic limbs for kids so young they’d outgrow their prosthetic limbs at a rapid pace. Children aren’t often outfitted with an arm with sensor technology until they’re three or four years old, though research shows that children under the age of two more easily adapt to the use of a prosthetic limb. Ambionics’ solution is to use 3D printing to provide cost-effective hydraulic prosthetics for infants and toddlers. Parents are asked to capture a 3D scan of the child while they sleep, which Ambionics uses to complete the design for the prosthetic limb, which it can produce in less than five days!

In the manufacturing world, 3D scanners can shave months-long prototyping processes down to a couple of weeks. For the hobbyist, the time savings and convenience can make the difference between giving up on an idea or following it through to completion. The best 3D scanners still usually require some knowledge of 3D modelling to clean a scan and prepare it for printing, but this is a much easier and more accessible process to learn than doing it from scratch. Time spent cleaning up a model in the software varies with the quality of your equipment and according to your project’s needs, but even a cheap 3D scanner capable of capturing basic geometry saves a huge amount of

Co-funded by the Erasmus+ Programme of the European Union

time and extra effort. Adding a 3D scanner to a makerspace or a classroom can help get a successful 3D printing project flow going in no time.

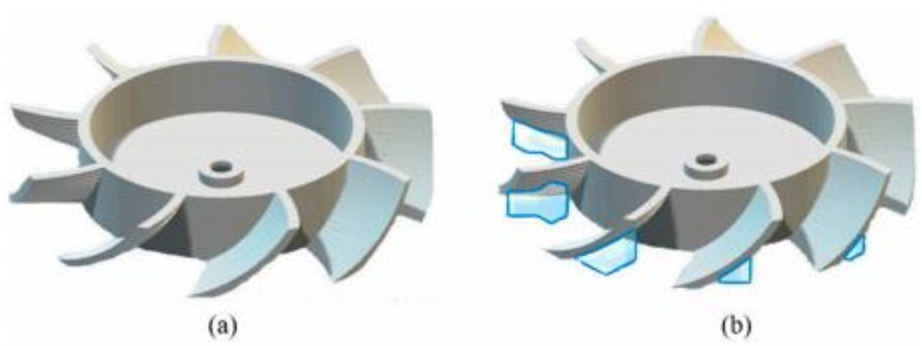
A 3D scanner is a truly amazing device that captures a physical object and converts it into a digital copy. Today, there are many types of 3D scanners out there, some of which are better suited to use with a 3D printer.

3.4 Design an object

Your choice of software affects your design process, and to an extent the way you think about design. There is free or payable CAD Design Software, but in general, they do more or less the same stuff. Based on needs, it is possible to try different solutions and choose one that fits better. For this article, there will only use open-source cross-platform tools. One popular option for 3D work in Blender. Blender is a very powerful modeling and animation tool. Blender is also a *mesh modeler*, meaning your designs are made of triangle meshes. You can then shape these meshes into your designs. A tool like Blender is a good option if you like to sculpt or "feel" our designs as you work. OpenSCAD is a CSG (constructive solid geometry) modeler. This means that you make your object by combining primitive forms. OpenSCAD doesn't sculpt. It uses a code-like design process, much like POV-RAY or other ray-tracing programs. If you like to design mathematical forms, or really like writing code, OpenSCAD is for you (it has *for* loops). OpenSCAD has one especially powerful feature: variables. Being able to assign dimensions to a variable and then generate the object from those allows you to make parametric designs. One important thing to keep in mind is making sure your designs are manifold. In the interest of time, I refer you to this excellent article on the subject. It's much easier to make non-manifold objects with a mesh modeler than with a CSG modeler.

4 Support Structures

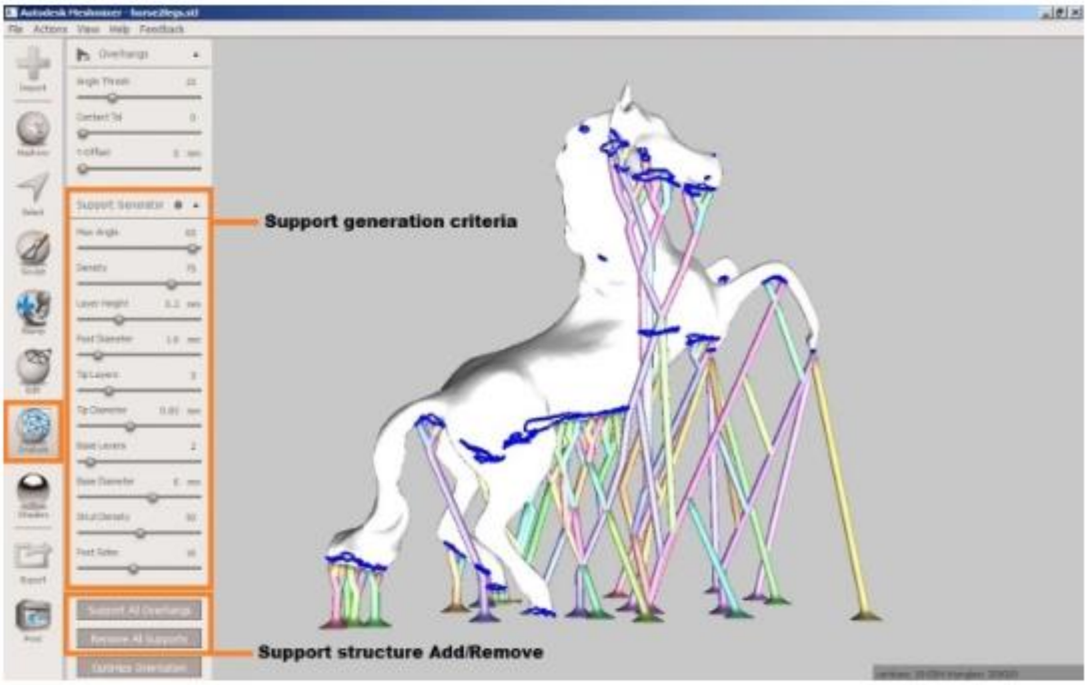
Support structures are among the most important elements for producing 3D printed parts successfully. The most important advantage of 3D printing is its ability to create freeform and intricate geometries. Much of the design freedom offered by 3D printing would be impossible without using support structures. Supports are crucial for preventing distortion and collapse within a part, among other uses. In this chapter, we are going to take a deep dive into the world of support systems, the various technology specifications, and how to reduce their use.



In 3d printing, which requires extrusion-based methodology, the support structure is required. The FDM (Fused Deposition Modeling) is a typical example of this, where the support structure is only formed when necessary during printing. The support structure necessity is defined by the item to be printed, if the object has an overhanging form, then a support structure is printed to serve as a foundation for the component that needs to be printed.

4.1 What is Support Structure?

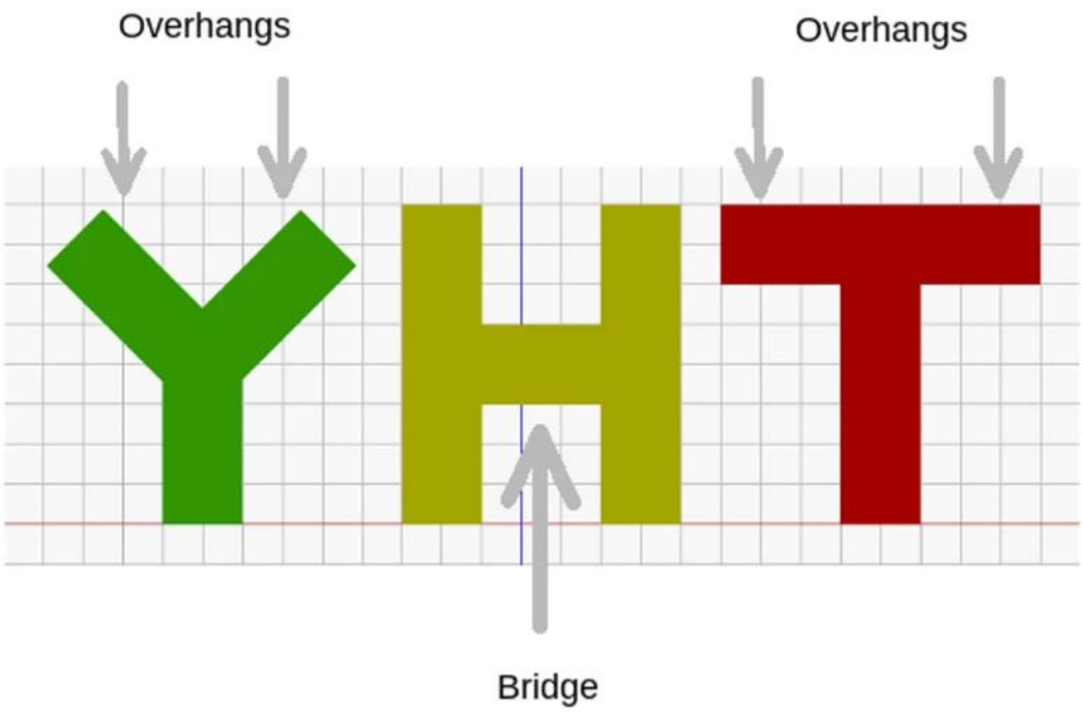
Imagine an individual standing upright model being drawn, it's possible to print that layer by layer. But when the arms of the person are spread out, the ink that is being printed to create the sections of the arm will not remain intact against gravity. This is where the support structure plays the role of acting as a platform for imprinting the required object.



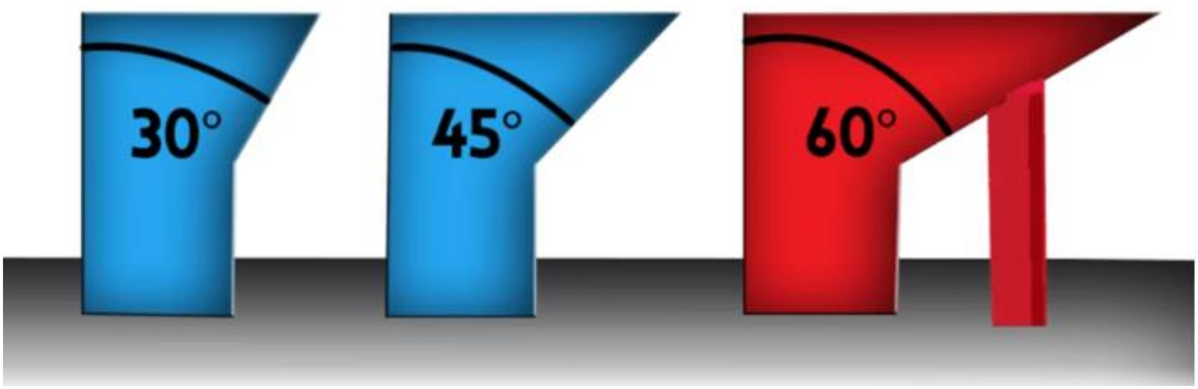
Used with almost all 3d printers, support structures seek to ensure the print quality of a part during the 3D printing process. Supports can help to reduce part deformation, ensure the security of a part to the printing bed and ensure that pieces are connected to the main body of the printed part. Much like support beams, supports are used throughout the printing process and then immediately removed. Supports can also act as heat dissipaters in processes requiring high temperatures, as is the case with 3D printing in metal.

4.2 Why are support structures needed?

Nearly all 3D printing technologies require that you consider support systems to some extent. FDM 3D Printers help to build a 3D object by depositing layer over layer of thermoplastics. In this process, the layer underneath must support each new layer. If your model has an overhang that is not supported by anything below, additional 3D print support structures need to be added to ensure successful printing.

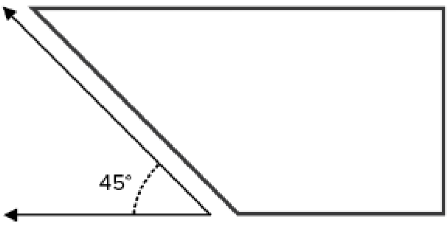


Typically we can print an overhang with no quality loss up to 45 degrees, depending on the content. At 45 degrees, 50 percent of the previous layer supports the newly printed paper. This helps to build upon adequate support and adhesion. Support is needed above 45 degrees to ensure the newly printed layer does not bounce down and away from the nozzle.

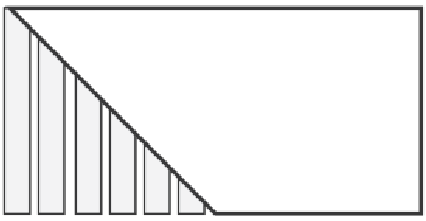


It best demonstrates this feature with the letters Y and T. In the letter Y the two overhangs have an angle of fewer than 45 degrees concerning the vertical. So if you wanted to print the letter Y, you can get away without using any support systems for 3D printing!

The overhangs in the letter T, on the other hand, have a 90-degree angle with the vertical. So you have to use 3D printing support systems to print the letter T, otherwise as shown below the result would be a mess. When printed with an overhang beyond 45 °, a feature will sag and requires material underneath to keep it up.



Overhang of less than 45 degrees
No support is needed



Overhang of more than 45 degrees
Support is needed

Not all bridges need protection, as do overhangs. The rule of thumb here is: If a bridge is less than 5 mm in length, the printer can print it without the need for 3D printing support structures. To do this, a process called bridging is used by the printer – where it spreads the hot content for short distances and manages to print it down with minimal sagging. If the bridge is longer than 5 mm, however, the technique does not work. In this scenario, you need to add support systems to 3D printing.

Co-funded by the
Erasmus+ Programme
of the European Union

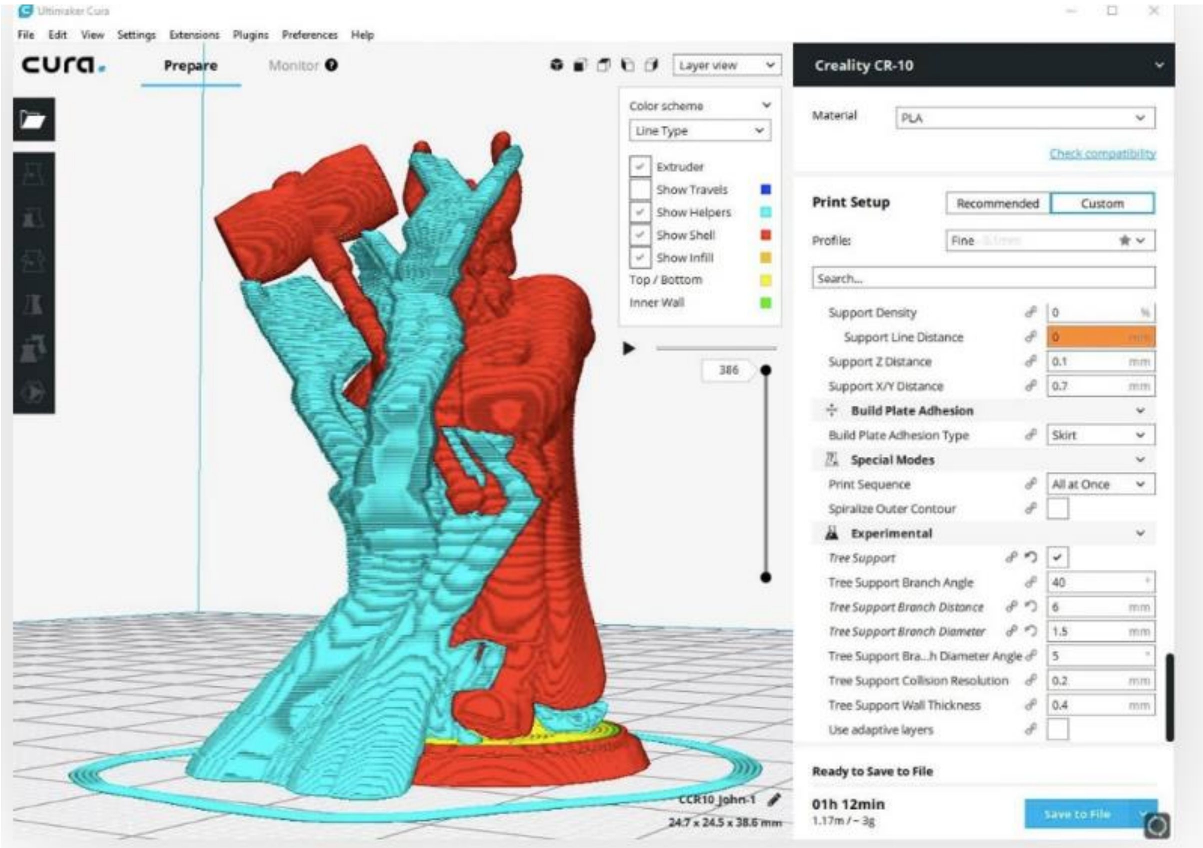
4.3 Types of Support:

The most common form of support is Lattice support. They are famous because for most 3D models they are easy to customize, fast to generate, and function well. The downside is that the supports will leave marks on the finished model if not printed correctly, which can be a problem to remove.

The default support type in Cura is lattice support for grid patterns, which is appropriate because grid patterns serve as a perfect all-purpose support form. But there are actually 7 types of support to choose from under the hood (some of which are pictured above). Pick a support pattern that matches the shape of your model.

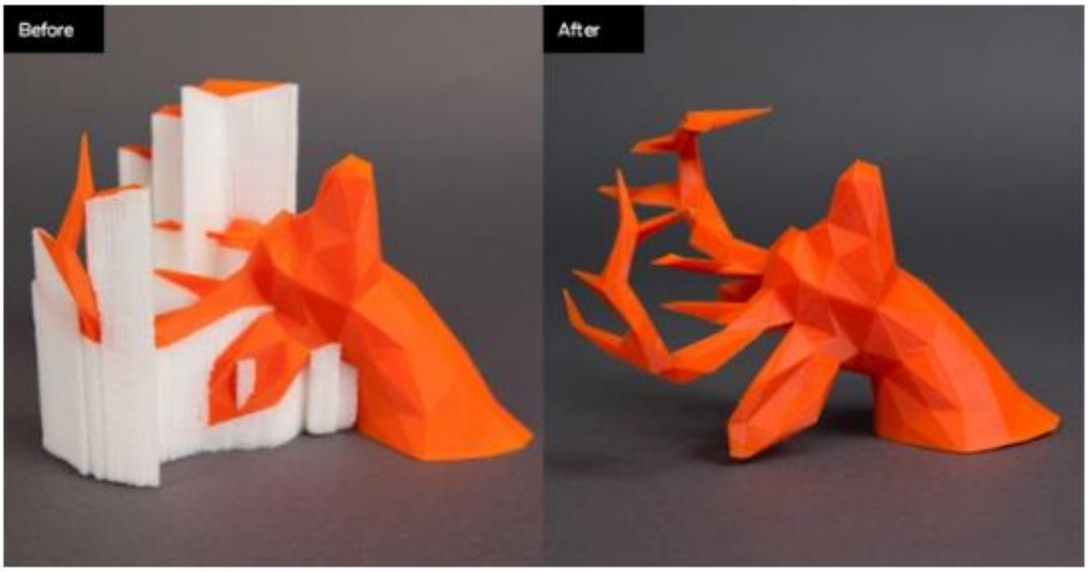
Tree supports have a trunk-like structure that branches out and up in several directions onto your model. The settings, which we will discuss later in more detail, allow you to monitor how the tree grows, and where the branch ends fall. These end tips support the structure from below effectively and can be printed hollow or with a specific density of infill. Note that these supports are not part of the default support categories. Instead, you will find them in Cura's settings section "Experimental." You'll be provided with some additional control options after activating it.

In certain cases, designs are such that they would need to stand on model surfaces if conventional supports were used. Here is where these supports have a distinct benefit. A tree support's "trunk" doesn't enter the model and since the branches come out of the main structure they're only positioned where they're required. In other words, due to the use of supports, the model itself has no artifacts left behind. In the case of organic types like humans and animals, this is especially useful.



4.4 Comments about Material Used For Printing:

As far as the FDM technology is concerned the most widely used materials on the market are ABS and PLA. In a single extruder, the type of support structure is the same as that of the frame. But in a double extruder, the content of the support system is determined solely by the material selected for the item to be printed. The properties of the material of the target and the material of the support system must complement one another, to make possible separation during the post process. The support system material usually used for ABS is HIPS, and it is PVA for PLA.

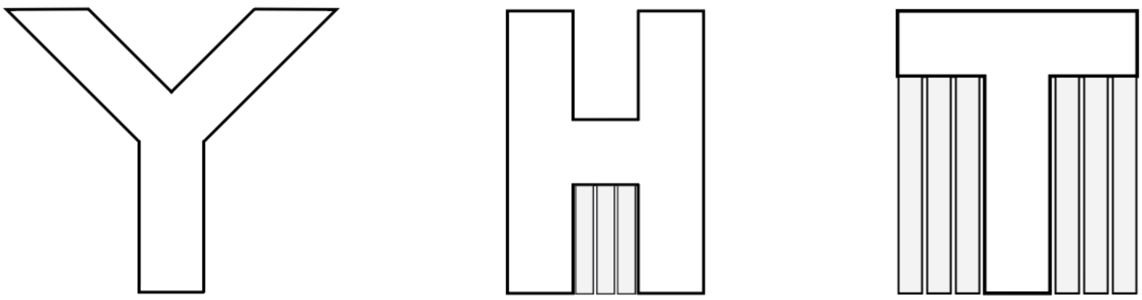


4.4.1 The ABCs (or YHTs) of FDM support

Consider the letters Y, H, and T, and a set of associated 3D models.

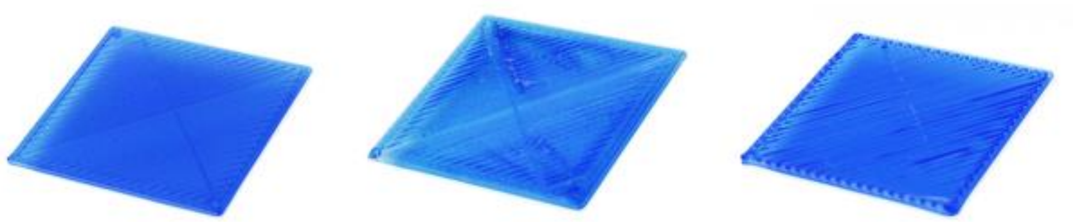
- The arms of a model of the letter Y can be printed easily. Even though the arms of the Y are outstretched, because they extend at 45 degrees or less, they do not require support.
- The letter H is a little more complicated but if the center bridge is under 5mm, it can be printed without support or any sagging. Over 5mm and support will be required. For this example, the center bridge is over 5mm, and support is needed.
- The letter T requires support for the arms of the letter. There is nothing for the outer arms to be printed on and the material will just fall without support.

The image below illustrates YHT with the support material shown in light gray.



4.4.2 Bed Adhesion:

Perfecting the first layer of 3D printing is such an important move that long and comprehensive manuals have been written on it. Too much can go wrong that will keep you from having the first layer right, making it a decent predictor of how far the rest of the print will go. The first layer is also a good time to troubleshoot since you can foresee issues that may come up throughout the rest of the print without wasting too much time and filament content.



Successful first layer — Build plate too close to nozzle — Build plate to far away from nozzle

Of course, when trying to get the ideal first layer many considerations play a role. One such consideration is the option of printing beds. The right printing bed should do two things: provide ample adhesion to hold and avoid warping of the filament material. A print not sticking to the printing bed will stick to the extruder instead, resulting in an uncharacteristic glob of molten filament. Warping happens when the material in the inner parts at the outer edges of the first layer cools at a higher rate compared with the rest, resulting in a deformed foundation.

The UI must be clean and as simple as possible for any program and Simplify3D delivers on both. The design is easy, with only a few toolbars on the main screen, making the user interface superb. But don't let the brevity fool you. Simplify3D is among the most powerful toolkits on the market for configuring customization options and settings. Rationally, the advanced control settings, perhaps its core feature, are divided into tabs that can speed up the workflow.

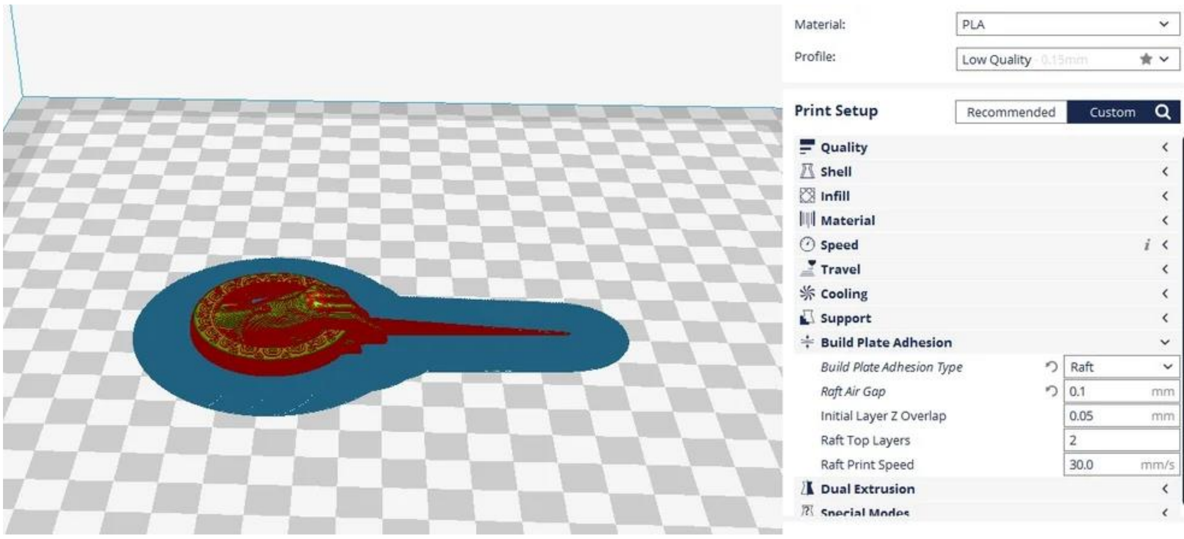
4.4.3 How to add a part to the bed:

The following points should be kept in mind:

- Getting the first layer right.
- Preventing print head collisions.
- Keeping a stable temperature.

4.4.4 3D Printing Rafts:

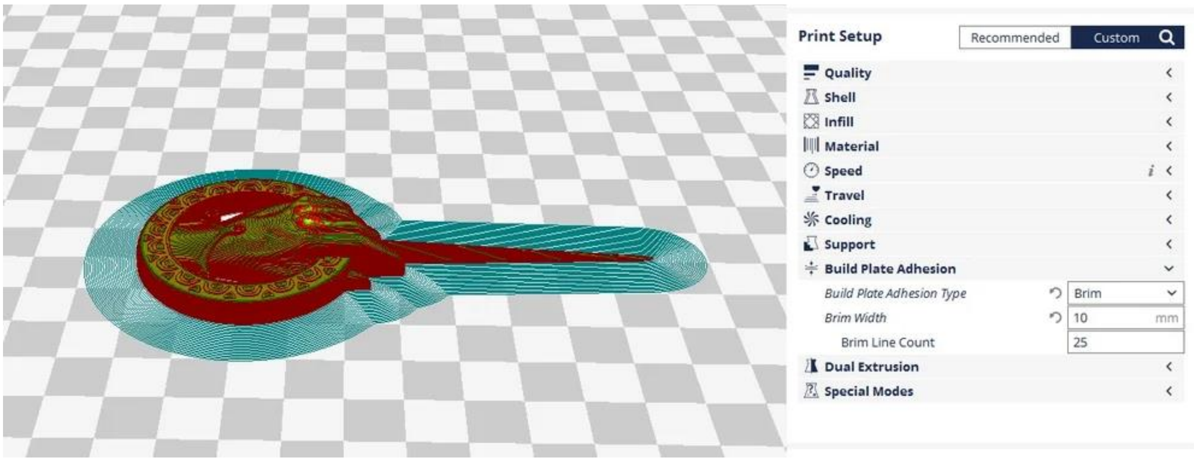
A 3D printing raft forms the first layer of a horizontal mesh of filament deposited directly on the build platform. Upon this first layer, the print is deposited. Rafts are generally used while working with ABS filament as it has a high chance of warping. A 3D printing raft is not only important to avoid warping but also to improve bed adhesion, helping the initial layers of the print to sustain the entire print. Choose the setting given in the image below for 3d printing with Raft.



4.4.5 3D Printing Brim:

A 3D printing brim is a layer of material that extends along the print bed from the edges of a 3D print. Brims help to improve bed adhesion and to prevent warping. Unlike a raft, a brim doesn't reach below the print. In this way, it can also be thought of as a skirt that doesn't touch the edge of the print.

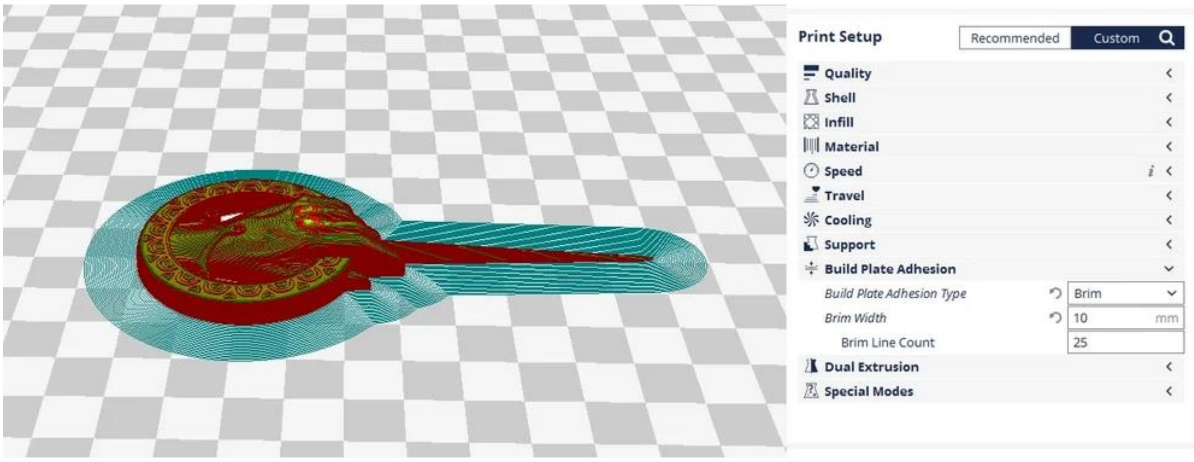
Many users depend only on a raft to improve their chances of a successful print, but a brim can be just as useful. In fact, in many cases, it is better than a raft. That's because it's easy to remove, wastes less material, and doesn't affect the bottom layer finish of the print.



Generally, we can control two important settings for brims: brim width and line count. Brim width is defined in millimeters while line count is the number of contour lines in the brim. The more the lines, the better the strength will be, up to a certain distance. However, it also becomes harder to remove the brim from the print.

4.4.6 3D Printing Skirts:

A skirt is an outline surrounding the part to be printed. The skirt does not touch the part as in the case of a raft or a brim but is controlled in more or less a similar way. It is useful in helping to prime the extruder. It ensures that the extruder starts a smooth flow of material before it actually starts printing. Though it does not support the print in any way, as with a raft or a brim, it is very useful to understand the flow of material, bed leveling, layer adhesion, and other layer properties set in the slicer.



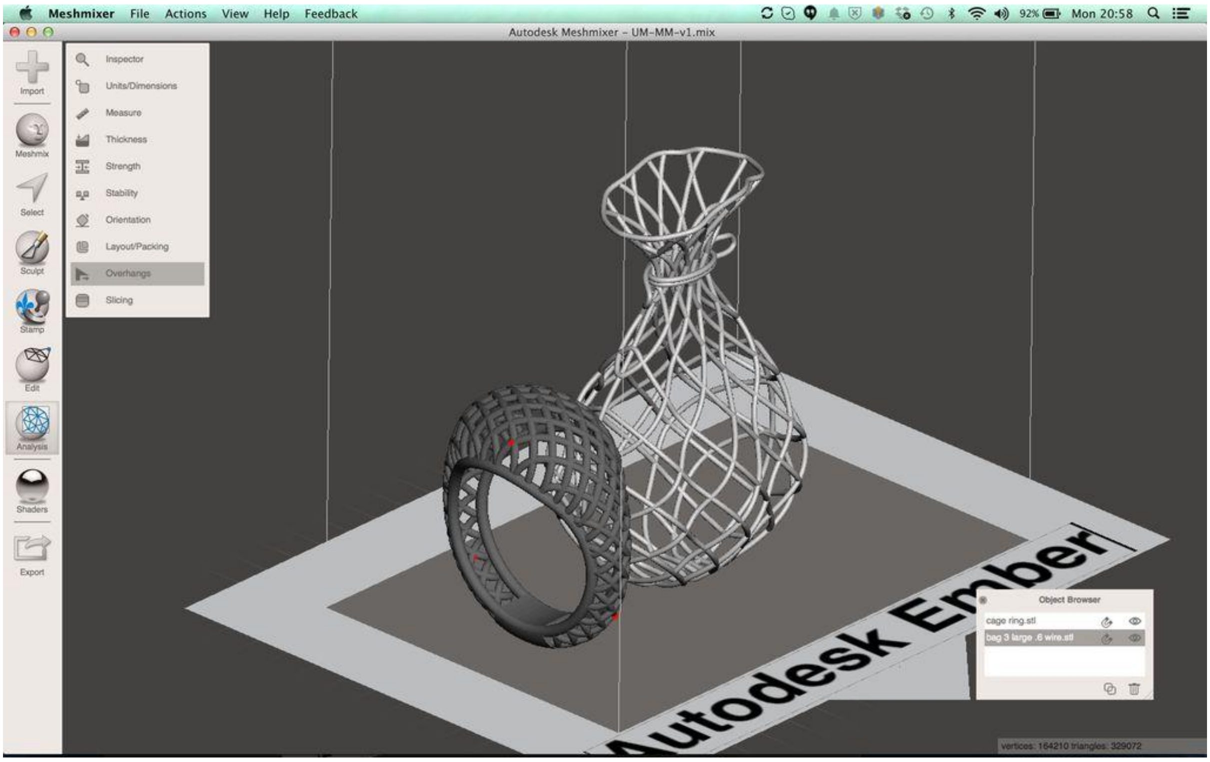
The two important settings to control a skirt are Skirt Line Count and Skirt Distance. The skirt line count defines the number of lines printed in the outline while the skirt distance defines the distance between the print and the skirt.

Generally, even a single skirt line is sufficient, but if the print area is small, then appropriate priming may not occur. In such a case, 3 lines are ideal to ensure proper priming.

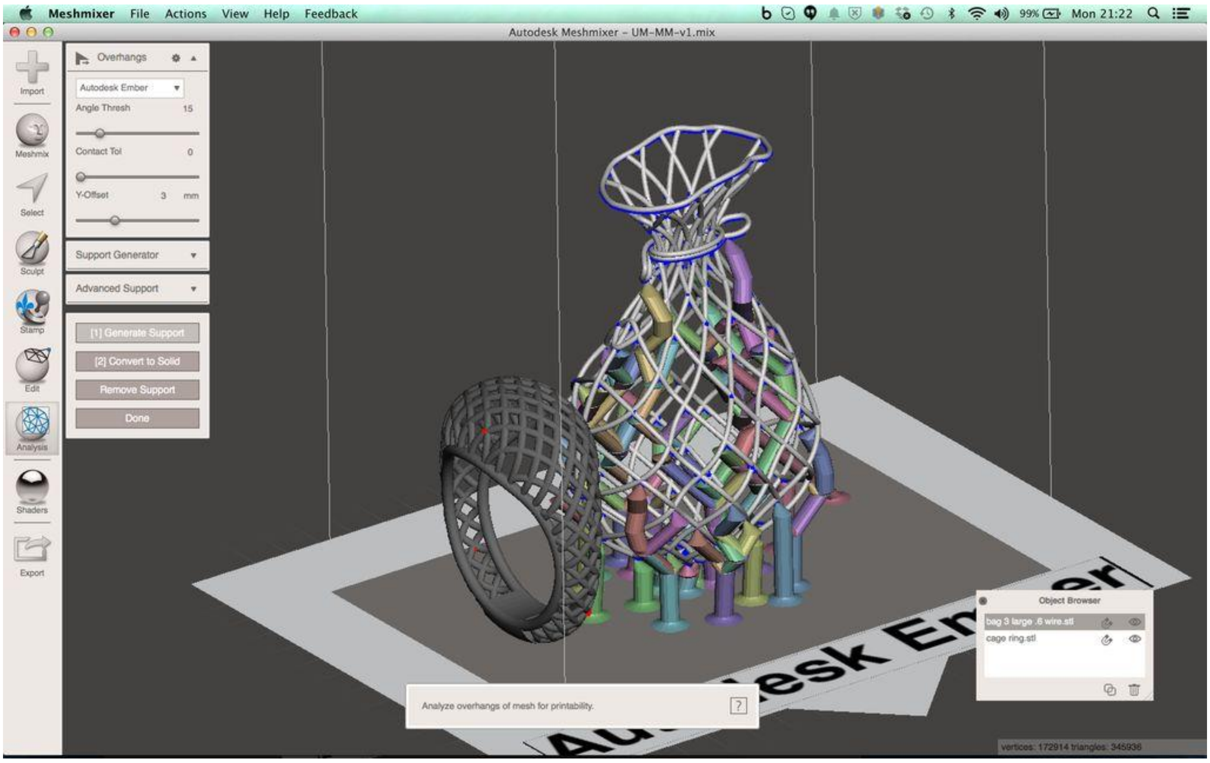
4.5 How to Create support (Meshmixer):

To automatically generate support for your model in Meshmixer.

- Click on the model you want to support.
- Click "Analysis" on the left toolbar and then "Overhangs"



- In the "Overhang" menu, make sure that "Autodesk Ember" is selected in the top drop-down menu.
- Click "Generate Support", and supports will automatically be created for your model.



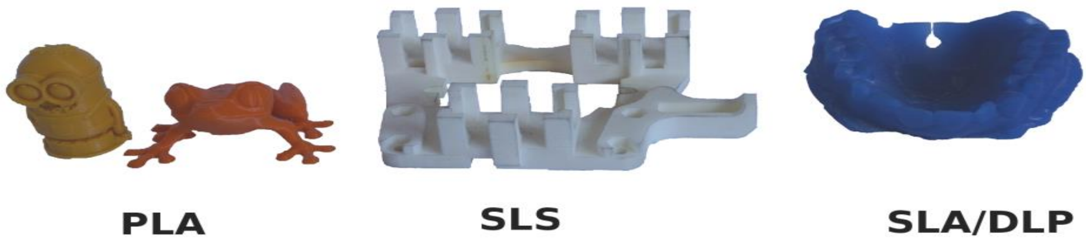
Co-funded by the Erasmus+ Programme of the European Union

- Individually supports can be deleted by secondary clicking on them
    - On Mac: CMD+click
    - On Windows: CTRL+click
- Click "Done" to save the supports
- Repeat for all un-supported models

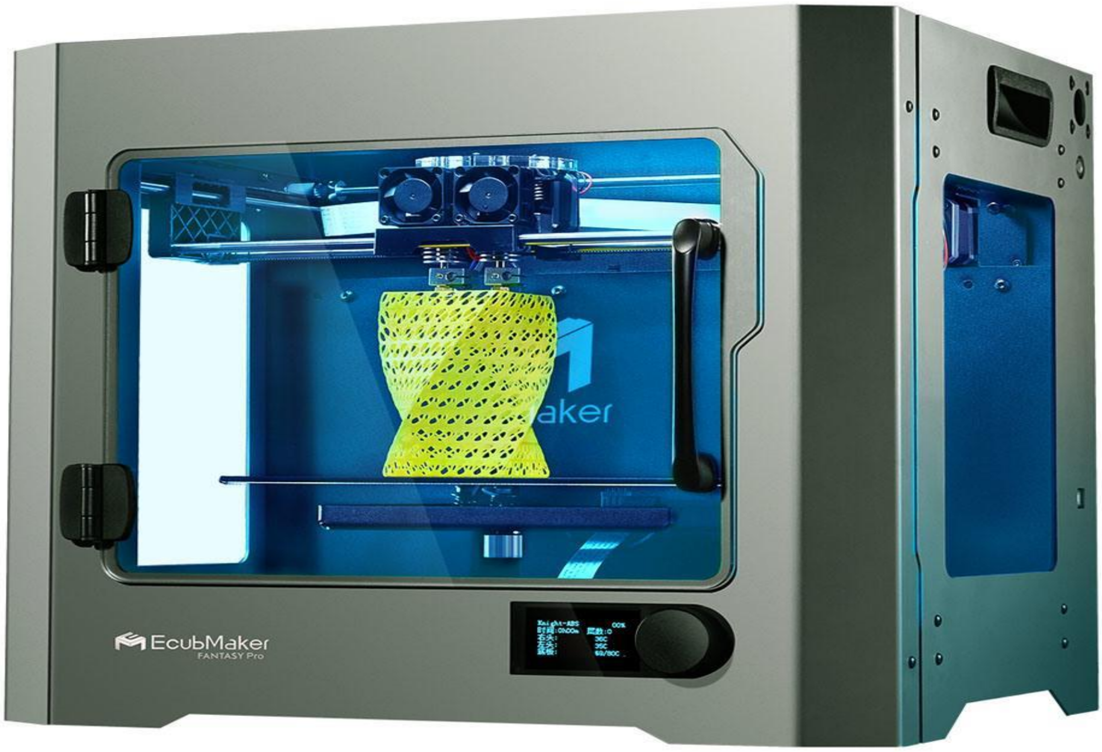
5 3D Printing

5.1 . How to print using Cura or Repetier

3D Printing is a process of attempting to make a physical object from a digital three-dimensional model, generally by laying down several subsequent thin layers of material. It involves bringing a digital object i.e. CAD representation of an object to its physical form by adding layer by layer of materials. There are various techniques for performing 3D printing. 3D Printing helps to bring with it two important advancements: the handling of objects in their digital format and the production of new shapes by adding material.

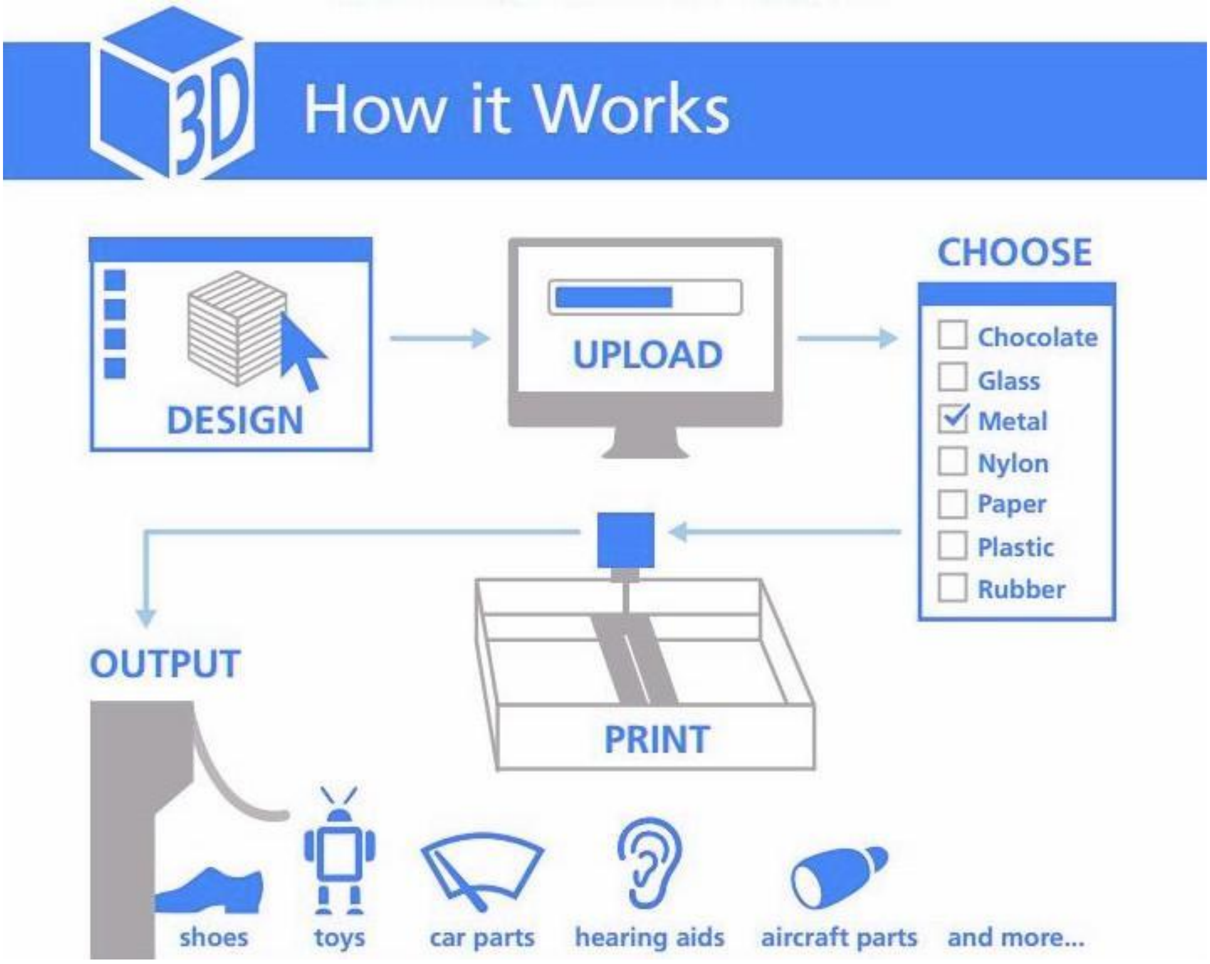


The most fundamental and distinguishing concept behind 3D printing is that it's a method of additive manufacturing. And this really is the key, since 3D printing is a fundamentally different manufacturing process based on innovative technology that creates parts in layers additively. This is radically different from all other conventional manufacturing methods that already exist.



5.2 How does 3d printing Work?

The starting point of 3d printing is a 3D model. You can build one of them on your own or import it from a 3D repository. There are several software available e.g. 3D scanner, app, haptic unit, or 3D modeling software for building a 3d model. There are various software options available for 3D modeling. Industrial-sized software can easily cost thousands per license per year and you can also get free software.

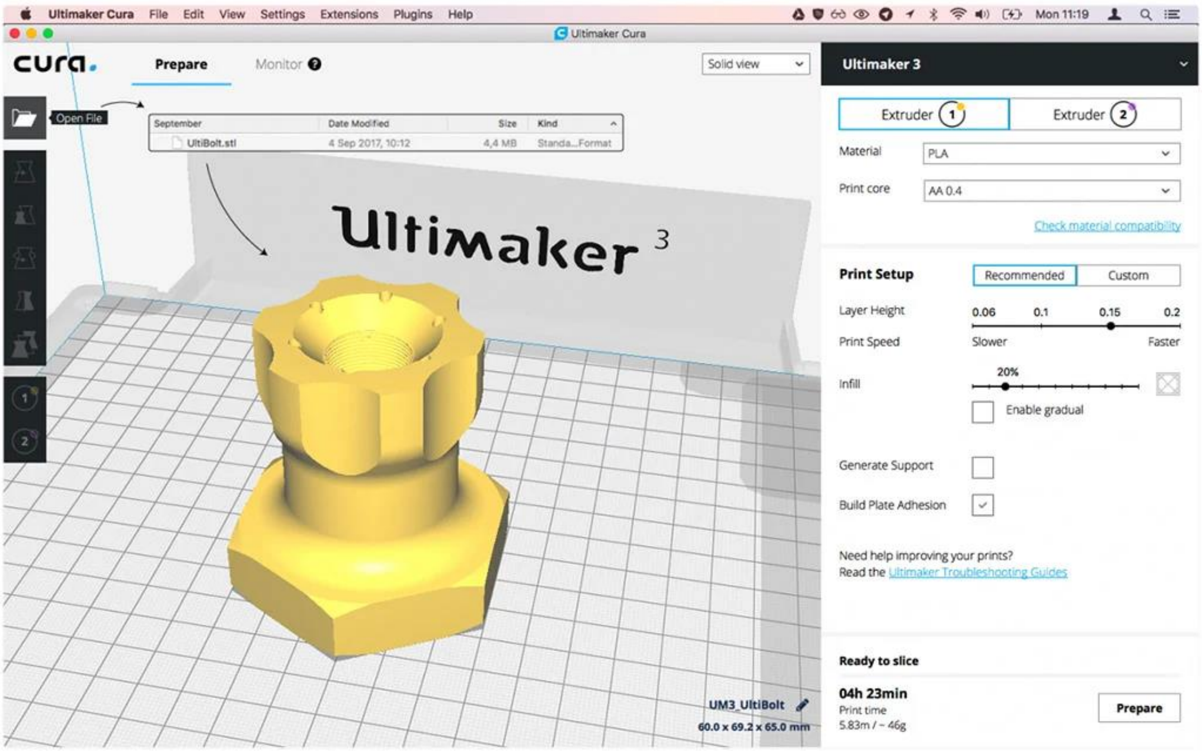


When you have a 3D model, the next step is to set up your 3D printer file for that. That is known as slicing. Slicing divides a 3D model into hundreds or even thousands of horizontal layers and is done with software. Some 3D printers have an integrated slicer that allows the feeding of raw. stl, .obj, or even CAD files. After slicing your file become ready for your 3D printer to be fed in. This can be accomplished via USB, SD, or web. Your sliced 3D model is now ready for 3D printing.

The key aspect of this technique is that even complex models can be formed with ease, utilizing fewer materials than older techniques of manufacture. There is a reduction in transport needs, as products can be printed on-site. And one-off products can be created cheaply and inexpensively, without having to worry about economy of scale – which could be a game-changer for quick prototyping, custom manufacturing, and highly customized products. The materials used during 3D printing, moreover, can be almost anything: plastic, certainly, but also metal, powder, concrete, liquid, and even chocolate.

5.3 Cura Software.

Ultimaker Cura is open-source software, and it is free of charge. It is used to slice 3d models and general G-Code, then G-Code will be sent to the 3d printer for manufacturing physical objects. Most desktop 3d printers support this application. It supports several files in the format: OBJ, STL, X3D, 3MF, etc. It supports a wide variety of software, including Repetier, Marlin, Mach3, Makerbot, Griffin, and others. Cura backs dual extrudes. Cura can be used with almost any 3D printer despite its name since it is an open-source slicer. The software is perfect for beginners because it is fast and straightforward. Best of all, it is user-friendly. More advanced users can use an additional 200 settings to perfect their prints.



5.3.1 Cura Software download and Installation:

Cura is available for all 3 major desktop operating systems. It is available for Windows (as a 32-bit and a 64-bit application), it is available for Mac, and it is available for Linux as well. A quick setup wizard can help you install Cura on a Windows-running PC. You'll be asked to add a printer

(Ultimaker, Custom, or Other) and attach it to your printer when you're finished with that and you launch Cura for the first time. If you are not connected to your printer, a portable USB drive can be used to pass files to the printer.

Cura has a good-looking and user-friendly interface, which should make it easier to figure out how to use this application. If you can't figure things out right away, then you need to know that a detailed, comprehensive Cura manual is available online. Go over it to understand everything that can be done with this free 3D printer slicing software application.

As of the writing of this guide, Cura is in version 4.6. It works on all common OS platforms: Windows, Mac, and Linux. The minimum system requirements for Cura are:

- Windows Vista or newer
- Mac OSX 10.7 or newer
- Linux Ubuntu 15.04, Fedora 23, OpenSuse 13.2, ArchLinux or newer

You can download and run older versions if your computer does not fulfill the requirements for the newest version.

To install Cura, first, download it for your OS from this page. When the download is complete, here's what you need to do on each platform.

5.3.1.1 Cura Download and Installation: Windows

Run the installer and go through the usual steps. The only non-trivial part of the installation is the following screen, which gives you the option to install additional components.

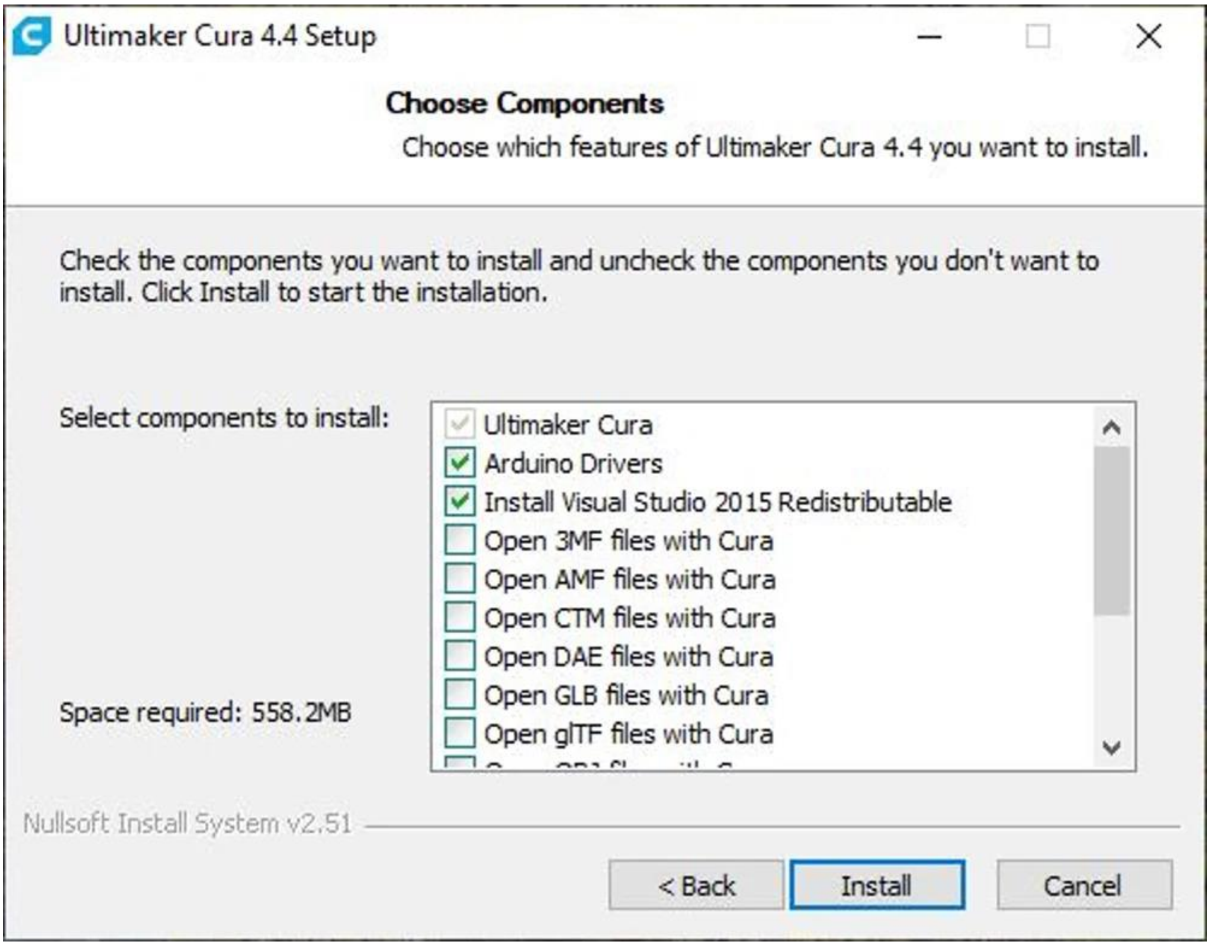
Select Cura Components

Make sure that the following items are selected:

- Install Arduino Drivers
- Open STL files with Cura
- Uninstall other Cura versions

You can select the other file types as well if you will be using them.

Click on Install to continue through the installation process. Click Next or Yes if prompted.



If you would like to import 3MF, OBJ, or X3D models into Cura 3D, check those boxes and proceed. Once the installation completes, Cura should open automatically.

5.3.1.2 Cura Download and Installation: Mac OSX

After downloading the software installer, open the installer and run the installation wizard to complete the installation. Easy peasy. You can find Cura 3D in your programs folder.

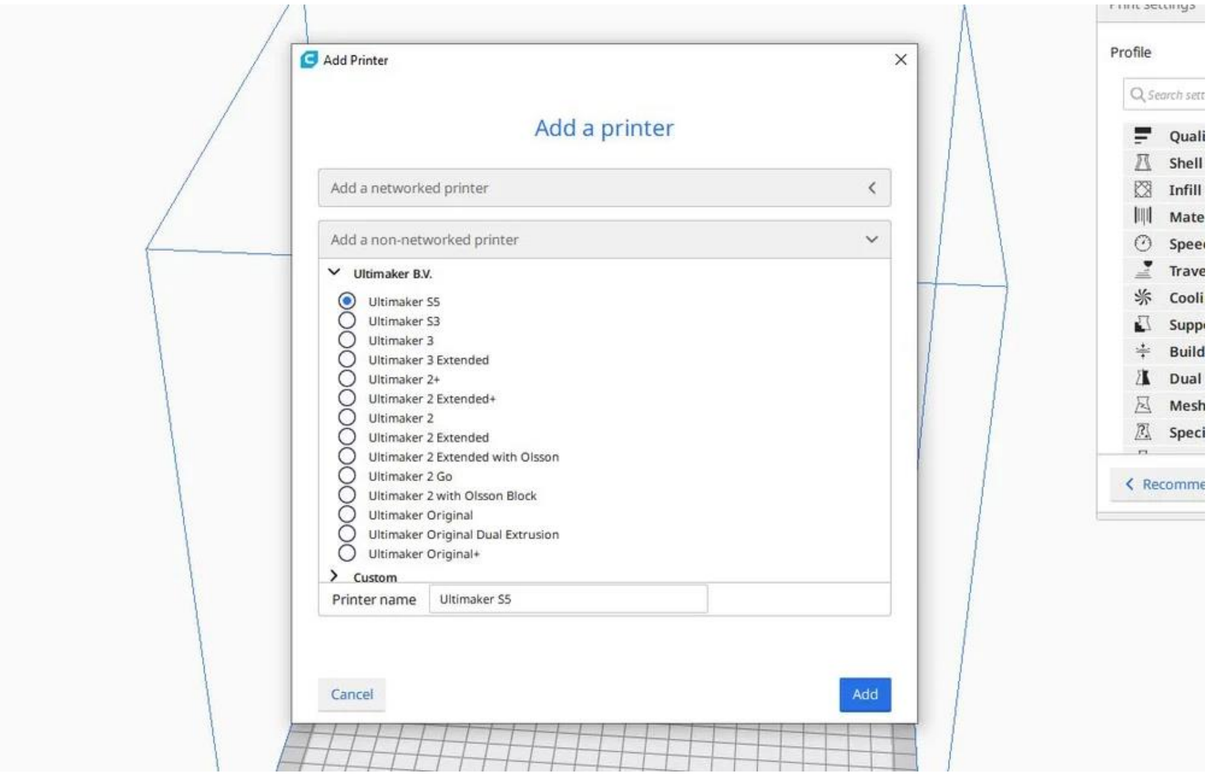
5.3.1.3 Cura Download and Installation: Ubuntu

For Ubuntu, the downloaded file is called Cura-4.4.1.AppImage. This is a binary executable. You should copy the Cura installer to a convenient location and give the current user the right to execute the file.

5.3.2 Cura Quick Start Guide:

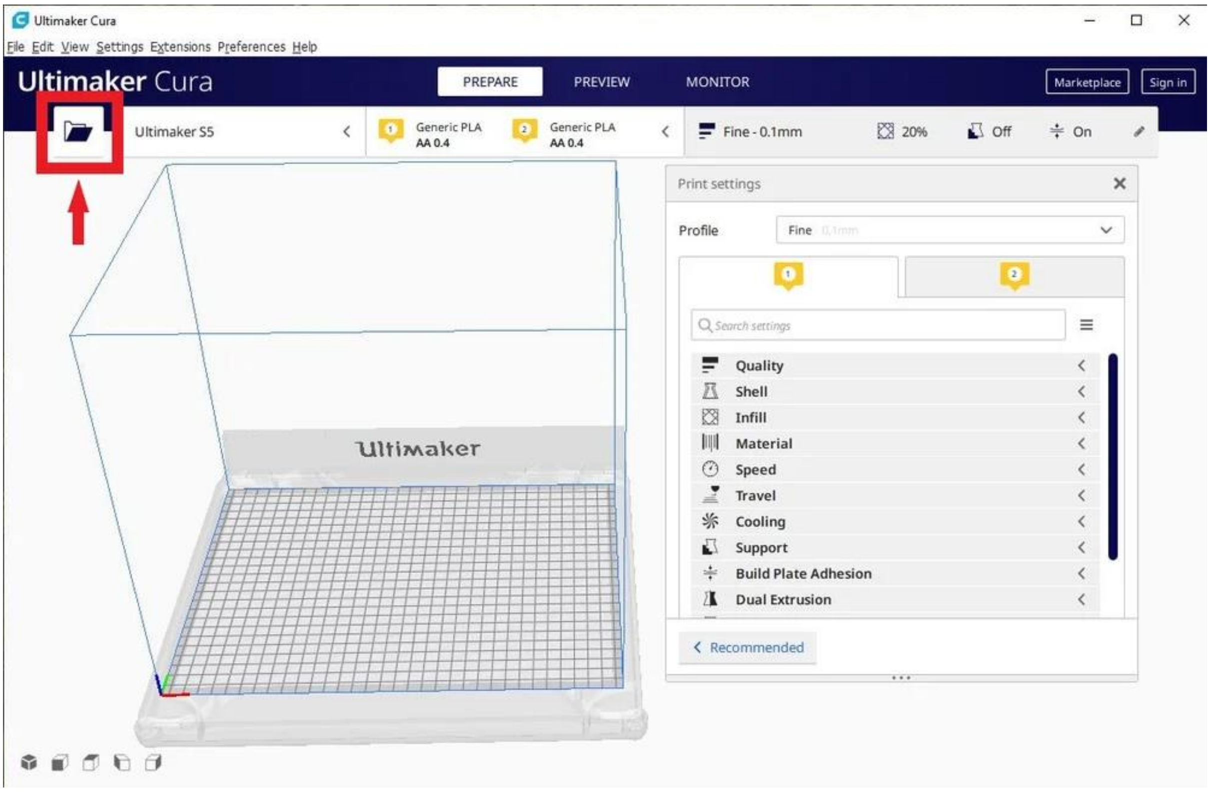
5.3.2.1 Cura 3D: Set up your 3D Printer

On first loading Cura, you'll be asked to select a printer. If not, or if you want to set up a new printer, then select Settings > Printer. You'll now be confronted with a selection of many printers. If you downloaded through the link at the top, then the listed printers will all be Ultimaker. For all other printers click Other and if you're lucky then your printer will be listed.



5.3.2.2 Adding a 3D printer model to Cura (Adding Part to Bed)

Once you have set up Cura for your printer, it's time to import a model into the Cura software. To import a model, you can either click on the floating folder icon on the left or select File > Open File(s) from the top menu. Select an STL, OBJ, or 3MF file from your computer and Cura will import it.



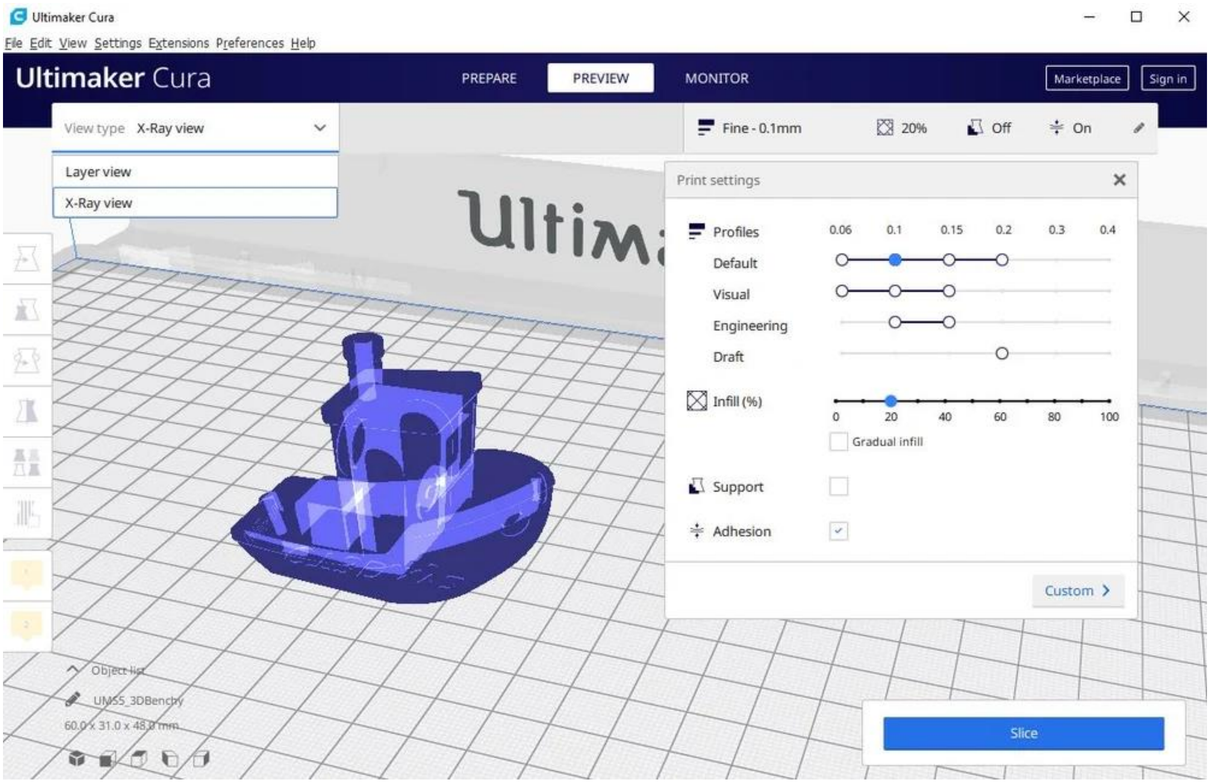
5.3.2.3 Model views in Cura

In the Cura software, there are three basic ways to view the model. Each is useful for different reasons, especially when a problem arises with your prints.

Solid: Cura's default view enables you to get a good idea of how the model will look when printed. It will show you the size and shape related to the print platform.

X-Ray: Found under Preview, this feature is great for when prints go wrong and quickly enables you to see parts of the internal structure of your print.

Layers: Also under Preview, if a print is failing every time at a certain point, or you've done something clever and just want to check that part of the print is OK, you can switch to Layer view.



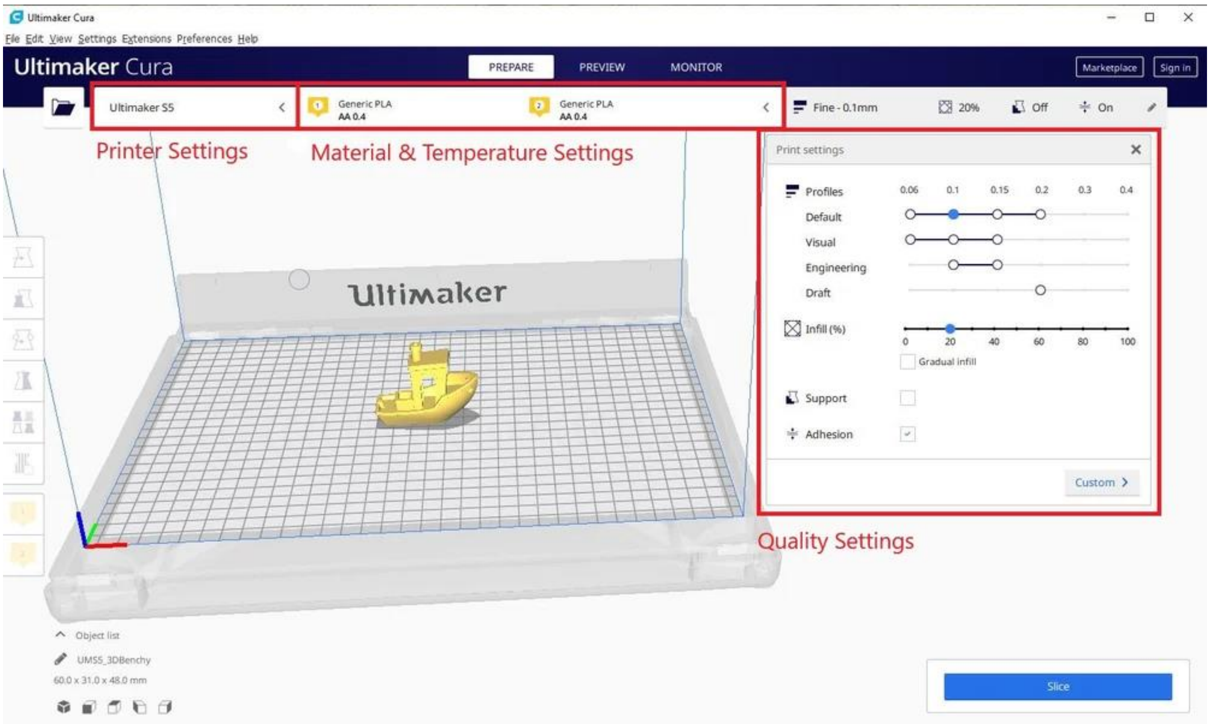
5.3.2.4 Cura's settings panel

Perhaps the most important part of the Cura window is the settings panel on the right. You need to choose the correct settings in this panel to get your desired print quality. Cura's settings panel is divided into two sections. The topmost section is the Printer Settings and the next section is called Print Setup.

5.3.3 Printer Settings

This section lets you select the right printer and material.

Printer: This is the printer that you selected in the first step. If you have more than one printer, then these can be set up, and then selected from this dropdown menu.



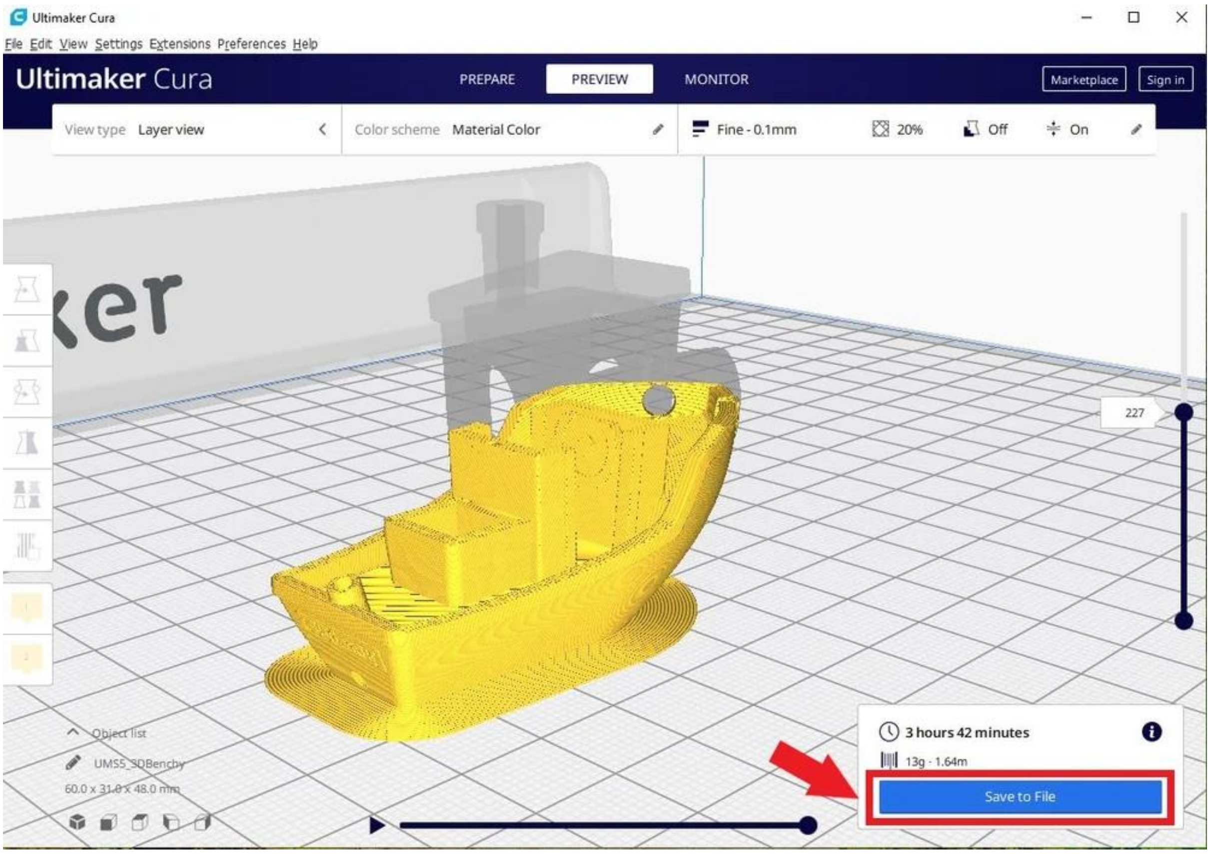
5.3.3.1 Material & Temperature:

Quickly select the material and nozzle that your printer is using, and temperatures will be automatically adjusted.

5.3.3.2 Generate a G-code file with Cura

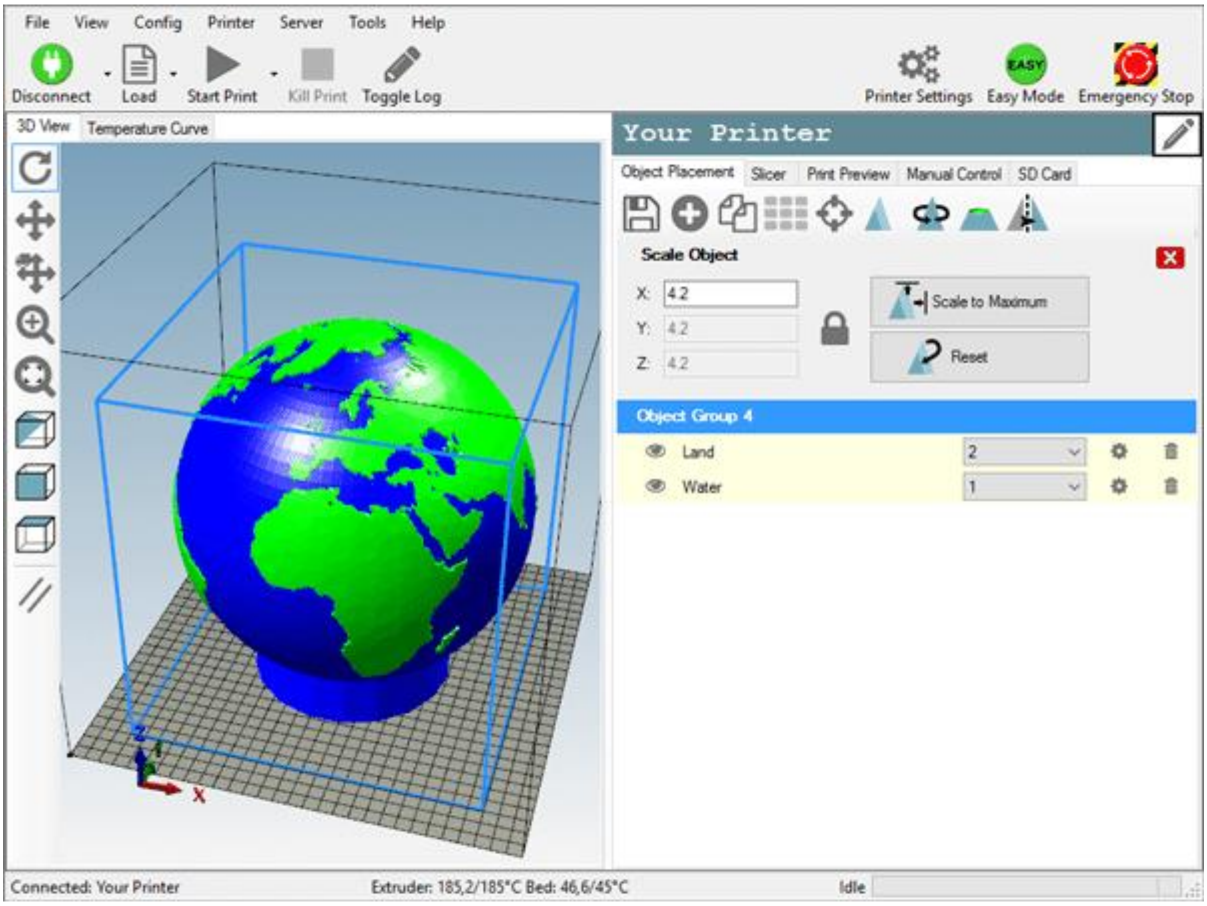
The model is now print-ready and all you need to do is to export the file from Cura to either an SD card or send it directly to the printer. Cura will now handle everything converting the 3D STL or OBJ into the G-code file required by the printer.

- Save the 3D print file: Click either Save to file, Save to SD or Send to Printer button on the bottom right of the window.
- Estimate of time for 3D print: Cura will give you a rough estimate of the length of time it will take for your printer to print the piece.
- Start the 3D print



5.4 Repetier Software:

This open-source slicer software supports three different slicing engines; Slic3r, CuraEngine, and Skeinforge. Repetier can also handle up to 16 extruders with different filament types and colors simultaneously, and you can visualize your end result before printing. There is a lot of customization and a lot of tinkering involved, making Repetier ideal for more advanced users. You also get remote access to your printers with Repetier host.



5.4.1 Prerequisites for Installation:

Before you start with the installation, you should check if your computer meets the requirements. Currently, available computers should have no problems at all. If you have an old computer running Windows XP you may have difficulties. The Host works on Windows XP and later and on Linux. If you have a Macintosh computer, check for the Repetier-Host Mac on this site. All you need is .NET framework 4.0 or a recent Mono installation if you are running Linux. The only other requirement is a graphic card with OpenGL. For a good rendering performance, OpenGL 1.5 or higher is needed. With lower versions, you may have speed issues with live preview.

Getting the software

Go to the [download page](#) and fetch the latest version for your os.

5.4.1.1 Windows installation

The windows version comes with an installer. After downloading run the installer and you are done. The installer already contains Slic3r and Skeinforge for slicing and python and pypy, which are needed to run Skeinforge.

5.4.1.2 Linux installation

The Linux version comes as a gzipped tar file. Move it to where you want your files and unpack its contents and run the post-installation script:

tar -xzf repetierHostLinux_1_03.tgz

cd RepetierHost

sh configureFirst.sh

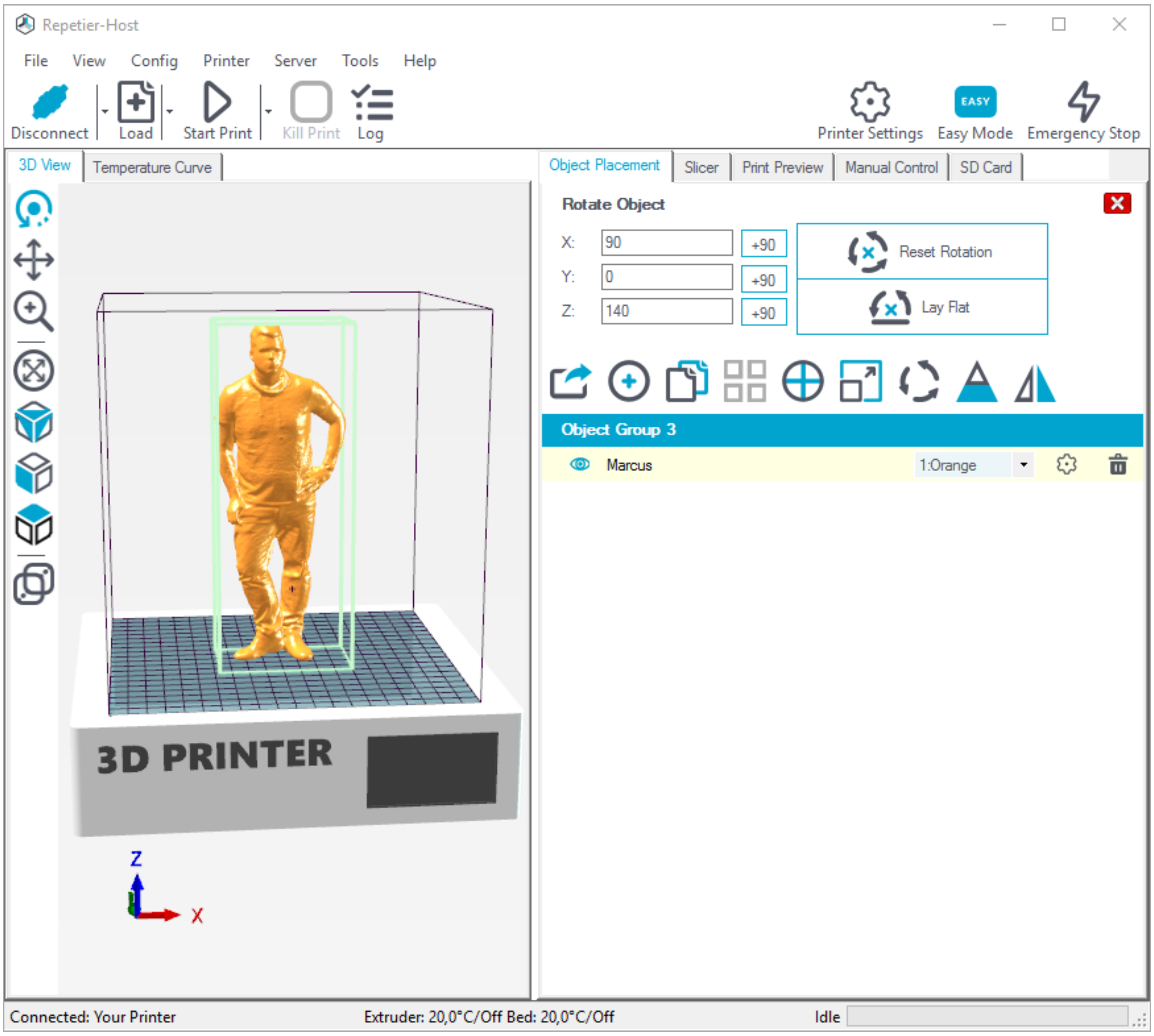
After that you have a link in /usr/bin to the installation, so you can start it with repetierHost. Make sure you have all required Mono libraries installed. If you are in doubt, install Mono develop, which has all the needed libraries as dependencies. One problem that most Linux distributions have is that normal users are not allowed to connect to a serial console. You need to put your user into the right group. On Debian you can call:

usermod -a -G dialout your username

to add your user to the group dialout.

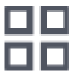
5.4.2 How to add a part to a bed (Repetier)

Prepare all objects you want to print, so your printer can print them. Learn how to arrange them on your print bed. Rotate and scale them to your likes.



 Here you can export all displayed objects at once. If you save them as .amf file, the object grouping and material assignments remain intact, if you save it as a .stl or .obj file, everything gets combined into one object.

Add ObjectHere you can add objects in .stl, .obj, .amf and .3ds format.

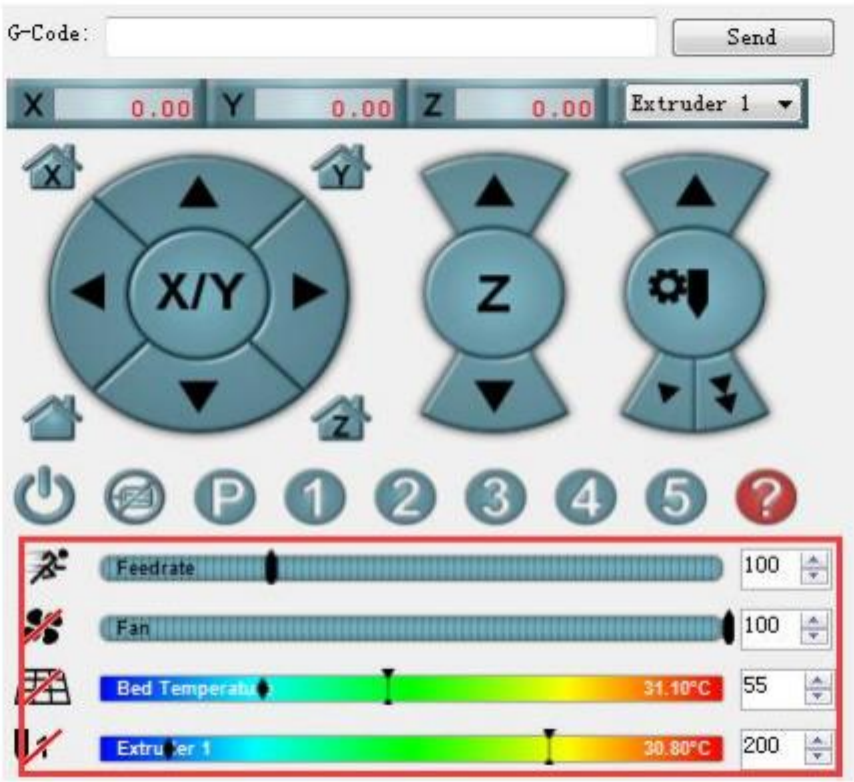
 Copy Object(s)Here you can duplicate the marked object(s) as many times  Click here to place all objects so that they fit on the bed.

Center ObjectThis function centers the marked object in the center of the bed.

 Scale Object (S)With this function you can scale the marked object.s you want.

5.4.2.1 How to Fix Temperature (Repetier):

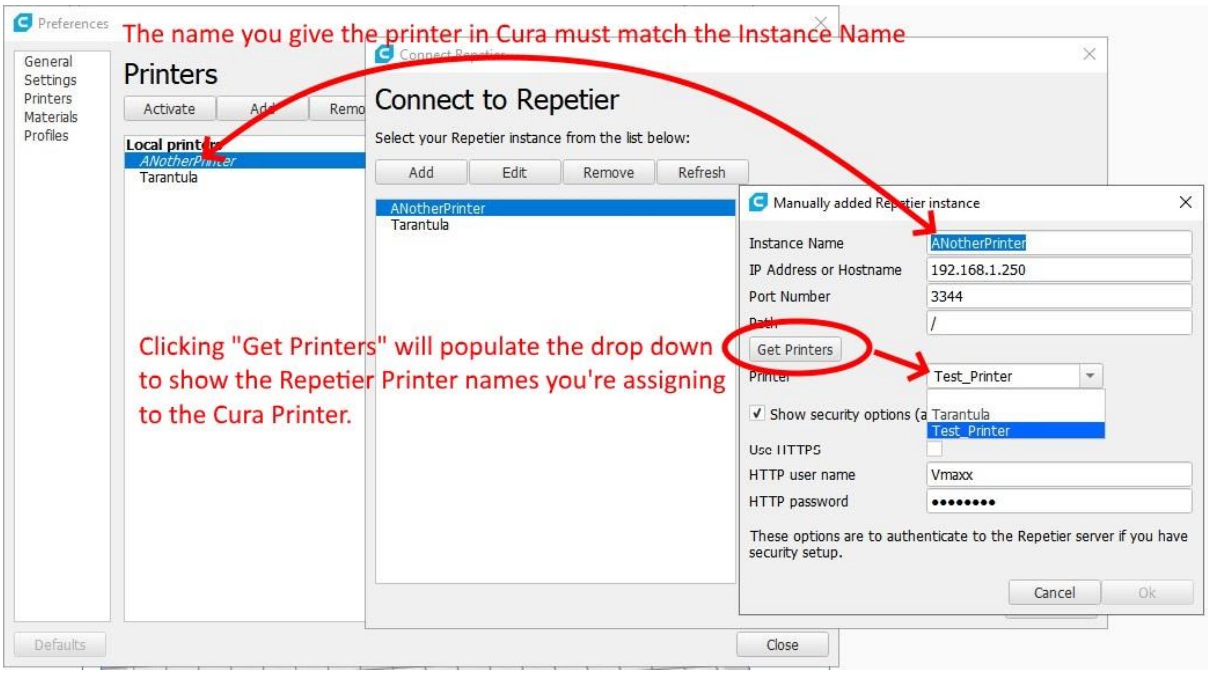
Manual control is the most important function for Repetier-Host. Now please switch to the "manual control" section from the "function area". There are two modes in a simple or complicated way, you can switch the "Easy Mode" button on the toolbar easily, let's take the non-simple pattern for example. Before the printer is not connected, the buttons of manual control are so gray and inactive. The manual control section consists of four parts, "Send G-Code", "control shift & extruder", "set temperature, fan speed and override" and "adjust option".



5.5 Using Cura with Repetier-Server:

- Make sure Repetier is up and running
- In Cura, under Manage printers select your printer.
- Select "Connect to Repetier" on the Manage Printers page.

- Click add and make sure you match the Name you give it in the plugin, with the name of the Printer in Cura.
- Fill in the IP and Port, if you have security turned on, click the advanced checkbox and enter that information
- Click the Get Printers button, it should populate the dropdown to select your printer.
- Click OK this will show the printer in the Printers list again but then ask for your Repetier API key. Once that is filled you can check the extra options if you have a webcam and need to rotate it.
- If you do not want to print immediately but have your print job stored uncheck "Automatically start the print job after uploading"
- From this point on, the print monitor should be functional and you should be able to switch to "Print to Repetier" on the bottom of the sidebar.



5.5.1 USB Machine Connection (Connecting Printer to Computer)

For 3D printing, you need to complete a few steps with your computer. As 3D printers for home users are relatively new, the machines are often not plug-and-play machines. In general, the following steps need to be fulfilled:

- To connect your printer, you need to plug it in using USB.
- Your computer needs to install the printer's driver software just as it does when using other USB devices like a USB mouse.
- A printing software should be installed that either comes with the printer or has to be downloaded. There is a software that can preprocess your 3D model for printing called Repetier-Host.
- As this Repetier-Host can be used for any 3D printer, it needs to know the specifics of your printer.

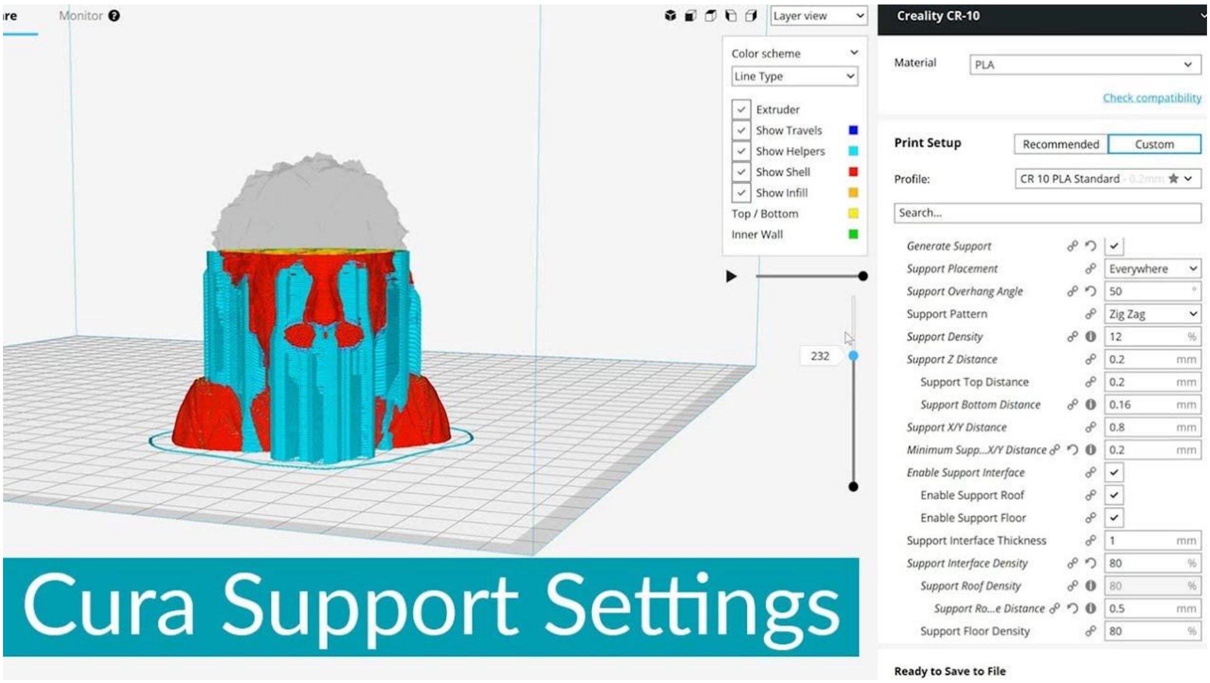
Depending on your operating system, printer drivers may be installed automatically. Often, newer operating systems like Windows 10 can do that. Also, there might be a 3D printer driver software that has also been shipped with your printer. Install it like you used to install driver software for computer mouse a while ago.

In case you have an old operating system and no driver software was shipped with your printer, it needs to be installed manually. Two drivers usually work:

- Arduino drivers
- CH340/CH341 drivers

5.5.1.1 Automatic Support Setting in Cura:

Some models have overhanging parts, which means that parts of the model float mid-air when you would print the model. In this case, you must print a support structure under the model to prevent the plastic from falling. This can be achieved by enabling "Generate support".


Cura Support Settings
Monitor
Layer view
Color scheme
Line Type
Extruder
Show Travels
Show Helpers
Show Shell
Show Infill
Top / Bottom
Inner Wall
232
Creality CR-10
Material
PLA
Check compatibility
Print Setup
Recommended
Custom
Profile:
CR 10 PLA Standard
Search...
Generate Support
Support Placement
Everywhere
Support Overhang Angle
50
Support Pattern
Zig Zag
Support Density
12
Support Z Distance
0.2
Support Top Distance
0.2
Support Bottom Distance
0.16
Support X/Y Distance
0.8
Minimum Supp...X/Y Distance
0.2
Enable Support Interface
Enable Support Roof
Enable Support Floor
Support Interface Thickness
1
Support Interface Density
80
Support Roof Density
80
Support Ro...e Distance
0.5
Support Floor Density
80
Ready to Save to File

6 Part Quality

6.1 Introduction

Many companies use 3D printing, a form of additive manufacturing, for prototyping or turning out production parts. It is a computer-based process that lays down layer after layer of a product until it is complete. The process uses metal or plastics and starts with a 3D digital model of the final object.

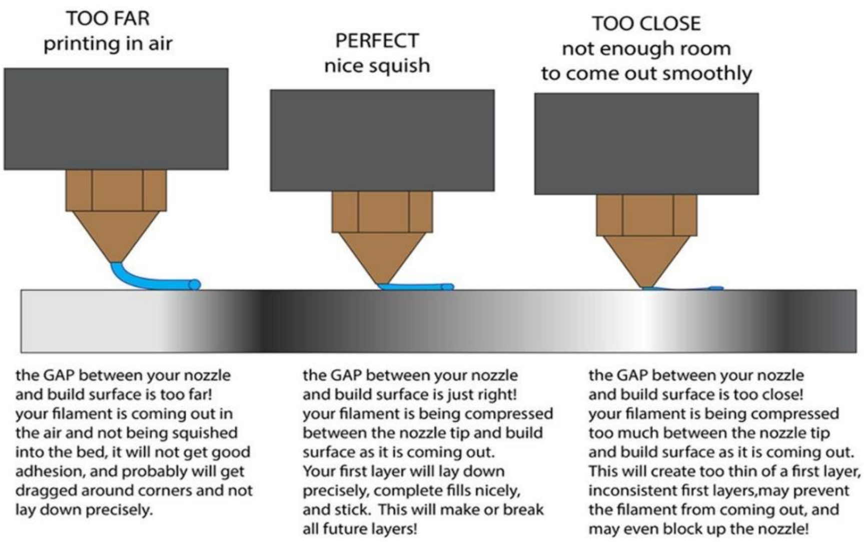
The most fundamental and distinguishing concept behind 3D printing is that it's a method of additive manufacturing. And this really is the key, since 3D printing is a fundamentally different manufacturing process based on innovative technology that creates parts in layers additively. This is radically different from all other conventional manufacturing methods that already exist.

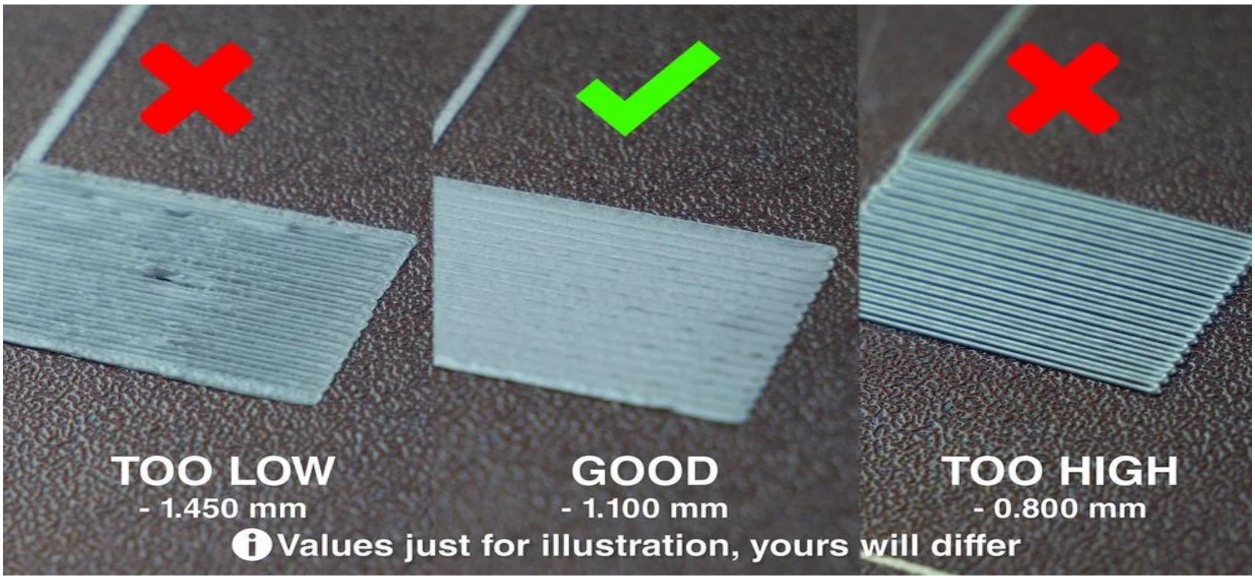
In this chapter, we are going to look at common 3d printing issues that should be resolved to increase the print quality. Each issue has a clear high resolution photograph, a detailed explanation of the subject, and a problem-solving checklist for how to improve 3D print quality. This includes instructions for software settings and even best practices for specific prints and materials, where applicable.

6.2 First layer Issues

The most significant layer possibly is the first layer of your print. As a base for your entire print, the proper adhesion to the build plate is important. Many common issues with 3D printing originate from a weak first layer. There are a few things that can go wrong when you print the first layer.

If the nozzle is too close to the printing bed, there would be little space for the plastic to get out of the extruder. You can effectively obstruct the opening by making the nozzle too close to the surface of the print so that no plastic can be extruded. You can quickly identify this problem when the first of two layers of plastic are not extruded by the printer. Use Live Change Z and First Layer Calibration options to tweak the nozzle height.

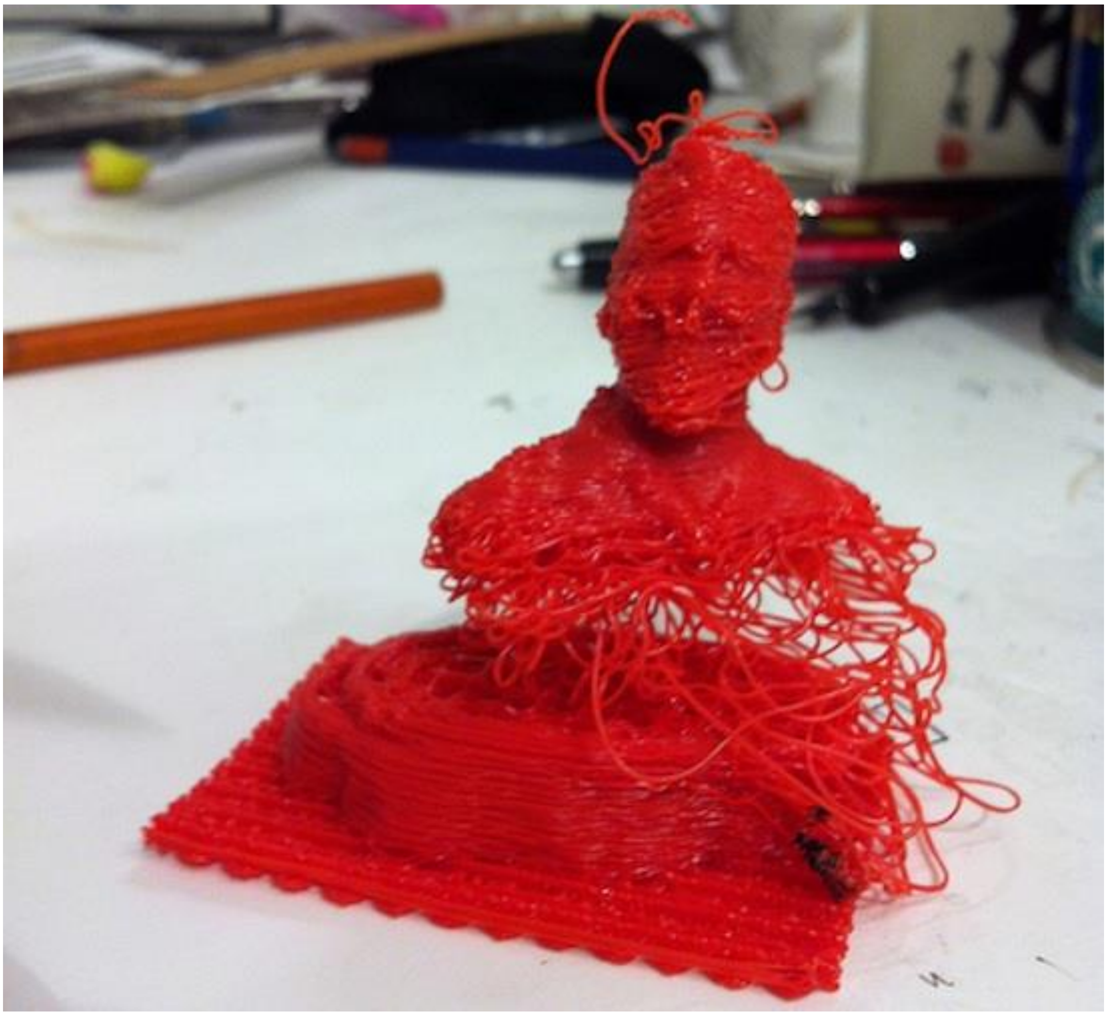




If the steps described above didn't help, then try decreasing the printing speed. The easiest way to do it is by rotating the knob during the printing process. Anti-clockwise = decrease speed, Clockwise = increase speed. We suggest decreasing the speed to about 75% for the first three layers, then returning it to normal. Make sure to use the recommended nozzle and heated temperatures – PrusaSlicer will configure them correctly based on the selected material, so you don't need to adjust the temperatures manually on the printer itself. If you are experimenting with new materials that don't adhere well, you can try to bump up the heatbed temperature by 5-10 °C. This way the plastic will stick a bit better.

6.3 Tips for getting print to stick

For many 3D printers, this is one of the most frequent problems. If your adhesion is missing, you can end up with twisted printing – or no printing at all besides a big mess of tangled filament on your bed. The following are the different reasons for the failure of bed adhesion before or during a print.



Co-funded by the
Erasmus+ Programme
of the European Union

6.3.1 Bed Leveling

If your printer has an adjustable bed and you're having issues with sticking, check to make sure your bed level is flat. An uneven bed could mean that one side is closer to the nozzle while the other side is too far, creating a difficult print environment. Additionally, if your bed is uneven it can result in your print warping or breaking. The process for leveling your bed depends on your printer.

6.3.2 Nozzle to Bed Distance

There is a certain sweet spot between the bed and the nozzle. It's like Goldilocks – not too close, and not too far, but just right. If your 3D print is not sticking to the bed, check the distance between the bedplate and the nozzle. You will have to experiment to see what works best for the filament you are printing. Ideally, the nozzle should be close enough to the bed so that the filament is slightly squished onto the bed's surface.

6.3.3 Nozzle Speed

Nozzle speed can also play an important role in your 3D print not sticking to the bed. Similar to nozzle distance, you must find a certain sweet spot for the nozzle speed, especially when printing the first few layers.

Slowing down the nozzle's speed gives the plastic more time to bond to the bed and get a better grip. If you print too quickly, the filament may not stick to the bed because the plastic cools off too quickly.

6.3.4 Bed Temperature

The final thing you can look at is the temperature of your bed. If you are using a heated bed on your prints, double-check that you are using the proper temperature for your specific filament. Different 3D printing materials require different bed temperatures.

6.3.5 Adhesives

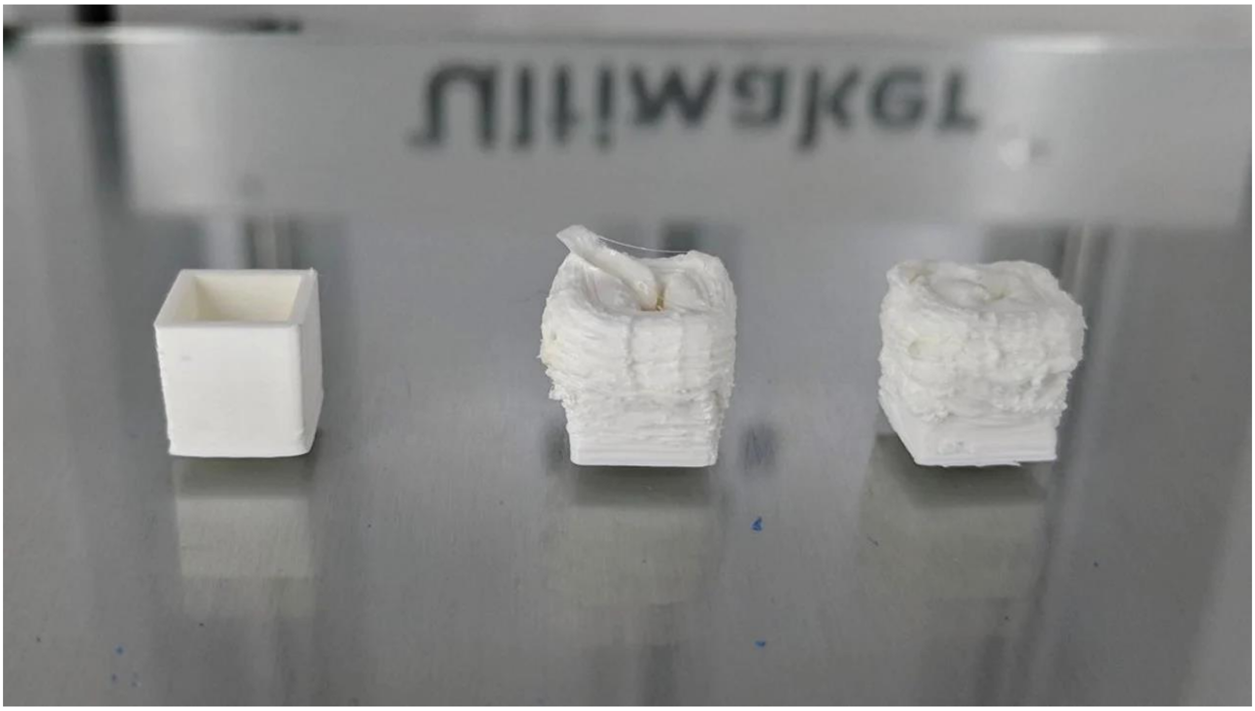
If you've double and triple-checked all of your bed and nozzle settings and you still have bed adhesion problems, then it's time to bring in some backup tools. If you still cannot get your 3D print to stick to the bed, use an adhesive right on the bed where the filament will land.

There are a few different options you can consider, including glue sticks, painter tape, hairspray, or 3D printing specific adhesives like Magigoo. Using an adhesive specifically designed for 3D printing ensures that you will be able to properly wash off the adhesive from your print once it is finished.

6.4 Inconsistent Extrusion: Under Extrusion and Over Extrusion:

3D printing under extrusion is one form of inconsistent extrusion (the other being over extrusion). Unfortunately, it can have a myriad of causes. No 3D printer troubleshooting guide would be complete without the full list of causes. The signs of under extrusion are easy to spot: you end up with weak prints that crumble, crack or tear under even slight stress, you have visible gaps in your objects, and walls start becoming see-through because solid areas show spongy patches instead.

Whereas, in case of over extrusion, too much plastic is coming out of the nozzle. You will be able to see this on your print (if not when coming out of the nozzle) the lines will be thick, uneven, and 'blobby' in some areas.



6.4.1 Tips for removing under extrusion:

- If your extruder isn't pushing enough filament, the most obvious course of action is to increase the extrusion multiplier (or flow) setting in your slicer. Tweak this setting by 2.5% until you find the right spot.
- Nozzle temperature is an extremely significant factor when trying to fix under-extrusion. Increase your print temperature by 5-degree increments until you find the proper temperature for your machine and material.

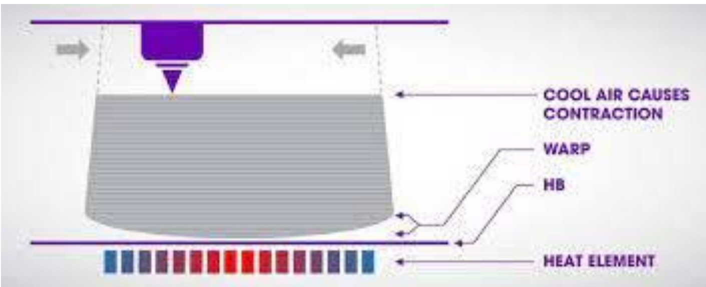
6.4.2 Tips for removing over extrusion:

Recalibrate the steps/mm value of your extruder to ensure your extruder delivers the proper amount of filament requested. Once the steps per mm are set correctly, the next step to a proper calibration is to set your extrusion multiplier (also called feed rate) properly to combat extrusion 3D printing. Both steps/mm as well as extrusion multipliers are filament dependent and might even change over time. Or it could simply be a matter of you changing the extrusion multiplier for a recent print and forgetting to reset it in your slicer. Temperature can play a factor in 3D printing over extrusion, so always ensure you're printing at the cooler end of the spectrum for your material.

6.4.3 Warping:

Warping can be caused by a few different variables but is fairly easy to recognise and resolve. Read below for more information on warping and how to fix it. Usually, it starts in the corners and can progress if the print isn't stopped. The print will start to lift and appear to peel away from the bed.

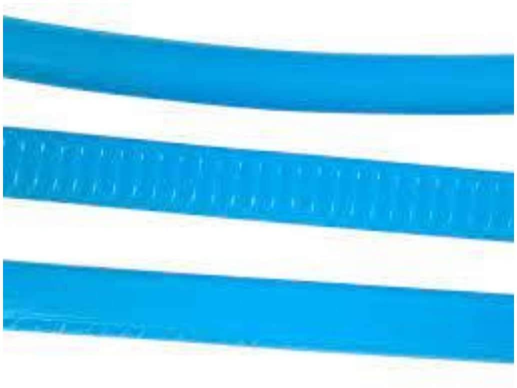
- Balance the printing and room/chamber temperature. Print a little cooler depending on the initial printing temperature. To print the cooler - start at your current nozzle temperature and work your nozzle temperature down in 5°C increments.
- Print thinner layers. For example, starting at a typical 0.2mm layer height, try 0.15mm or even 0.1mm layer height. This will reduce stress on your part during printing.
- Make sure you're not setting your bed temperature close to the glass transition temperature of your filament (stay 10°C below).



6.4.4 Filament Issues:

The quality and state of your filament play a vital role in the success and quality of your prints. Here are some common issues with filament to look out for:

- Extruder is crushing filament: A tell-tale sign of filament being crushed is that it appears deformed. If your extruder is equipped with some form of idler tension adjustment, decrease the tension. Lacking that you might be able to modify the feeder in some other manner (for example shortening the idler spring or replacing it with a softer spring). Or try a different, harder filament type or brand.



- Extruder is grinding filament: Grinding filament is never welcome and not what you want to see but keep reading on how to spot and fix the issue. Use good calipers or a better yet a micrometer screw gauge to measure the diameter of the filament coming off the spool and check if it is round or has been flattened. If it is thicker than it should be or no longer perfectly round, return the spool to the manufacturer/seller for a replacement. Calibrate your extruder and reduce your material flow. Especially when switching to smaller nozzle sizes your extrusion settings need to be spot on.

6.4.5 Overheating:

One of the ugliest forms of bad 3D prints is overheating. Here's some information on how to spot it, why it happens, and how to fix it: Your print has the correct general shape but has deformed where it has overheated. This can be at the start of your print or partway through. Following are the tips to avoid or reduce overheating:

- Increase the part cooling fans. If you're not using 100% part cooling fans you can try increasing up to 100% which should help. This may not be suitable for all filaments though so be sure to check the manufacturer's recommendations.
- Try printing colder, to begin with. Sometimes you can get away with a cooler printing temperature for the rest of the print with the same result. Alternatively, reduce the temperature when the printer is approaching the problem area.
- Print slower, giving the print more time to cool down. Consider the 15-second rule: the time for the nozzle coming back to the same point on your object should not be less than 15s.
- If your slicer supports it, set a minimum time per layer to ensure proper cooling. This usually results in the slicer dynamically slowing down the print speed to ensure the minimum layer time, which, in this particular case, won't help all that much.

6.4.6 Misaligned Layers:

Some 3D printing issues like this, appear as completely random and isolated events. There are some checks that can be done to fix it though so keep reading to learn more. Unfortunately, issues like this can happen randomly and often partway through a print. The 'How to fix:' section below can also be used as a check-list before you set your print going to prevent this from happening. Following are the tips to avoid or reduce layer shifting or misaligned layers:

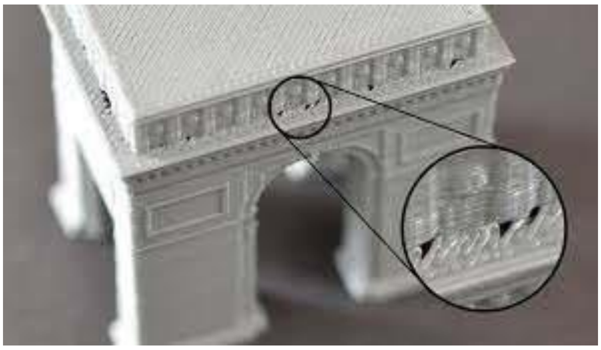
- Trying to print too fast will cause the motors to skip, resulting in 3D printing layer shifting. Try reducing your speed and running test prints.
- If you're also experiencing warping or curling, it could be that the HotEnd simply crashed into a section that has curled upwards. This picture however does not show any signs of warping or curling.
- Check if your printer moves freely along this axis, clean and lubricate the mechanical parts like smooth rods, lead-screws, or rails and check for broken bearings.
- Losese belts or not properly tightened pulleys will cause layers to shift as the required amount of travel will not be achieved. So, check the belt(s) for the axis affected and if necessary tighten it according to the specifications of your printer manufacturer. Also, check if all pulleys are properly tightened to the motor shafts for the axis in question.

6.4.7 Gaps and Holes:

This is a common fault and there are many reasons for gaps in your print, depending on where they're located. It's worth checking everything mentioned in this section so that you don't miss the exact cause. Probably one of the most obvious problems to spot in a print. Following are the tips to avoid or reduce gaps and holes:

- Print colder or increase fan speeds. Better cooling improves bridging performance, and that's basically what we're doing when printing over infill.
- Use a higher infill percentage. More infill means smaller gaps, which are easier to cover.

- Another possible cause for gaps in your top layer is under-extrusion.
- Not enough top layers - increase the top layers count so you are printing at least 1mm thick.
- Printing too hot - print at a lower temperature so the plastic sets in position faster.
- Printing too fast - slow down the print speed. This allows the extruded plastic to cool more before the next pass of the nozzle. If kept too warm, the layer will peel back from where it was printed.
- Thin wall options - A lot of slicers today have special thin wall options, make yourself familiar with them and see where they can take you.
- Adjust line width - Adjust (not necessarily increase) line width or the number of outlines to force your slicer to construct your wall differently.



6.4.8 Failing Supports:

Failing supports are not what anyone needs when using support. Unfortunately, it happens, and can even happen to several independently failed support like in the image displayed. Usually, you're using supports because they are needed for your print. If your supports fail it will be very visible as your print will not be complete. Support pillars, especially when setup using a low support density are not the most stable things and will be in increasing danger of toppling over the taller they get. Following are the tips:

- Avoid isolated towers, place your supports in bigger groups.
- Reduce printing speed for support.
- Use higher support density and if your slicer supports it - a different support pattern.

- If your slicer supports it, have a brim or a solid bottom layer added to your supports.

6.4.9 Poor Bridging:

Bridging, i.e. printing (more or less) long distances unsupported over thin air is a tricky business. It requires different settings than regular printing, usually speeds and cooling is key to success. Saggy lines in the picture show poor bridging performance. Advanced slicer software detects when bridging is required and will allow you to apply different settings for the bridge.

- Increase extrusion multiplier for the bridge.
- Try different speeds, slower is usually better, but results may vary so experimenting is key.
- Increase your fan speed for bridges. We want the material to harden quickly without drooping. It may not be suitable for all materials, check with the manufacturer/seller.
- Make sure your slicer is actually using bridging mode. If you're using Simplify3D, make sure bridging of outlines is enabled.
- More advanced bridging options like the direction of the lines that make up your bridge or increasing the start and end zone of a bridge can help as well.
- Better than trying to optimise bridging performance is trying to avoid bridges, to begin with. If possible, reorient your part on the build plate so fewer bridges are required or add supports to your bridges. With supports under your bridge, it won't be able to drop as much.

Co-funded by the
Erasmus+ Programme
of the European Union

Font: https://help.prusa3d.com/en/article/poor-bridging_1802

7 Machine maintenance.

7.1 Introduction to maintenance

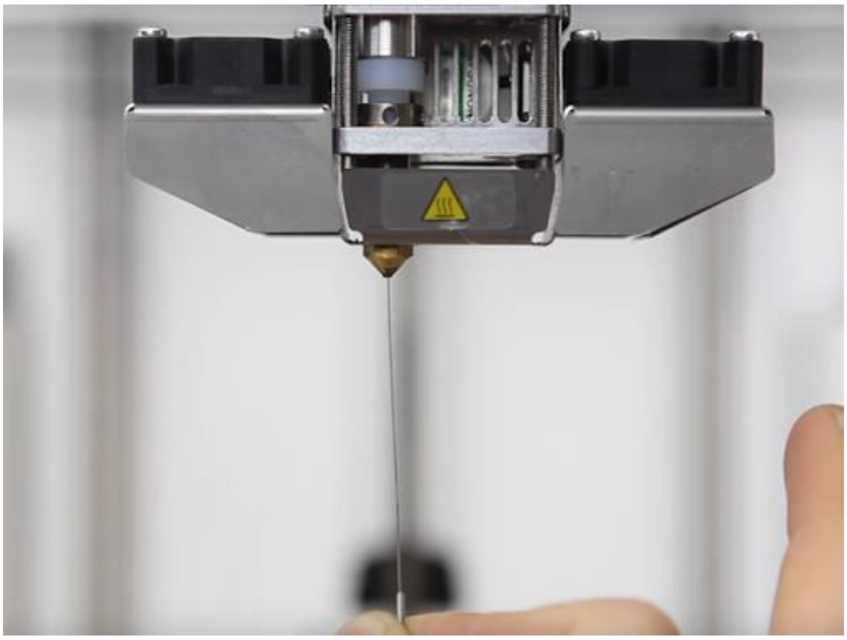
A 3D printer can be a finicky piece of hardware, and you don't want to be dealing with a filament problem or breakdown right in the middle of making your latest work of genius. Like any machine, you need to take care of it. This rule applies to 3D printers more than most, because – let's face it, the quality of manufacture of most consumer 3D printers isn't quite where we'd like it to be.

This is where 3D printer maintenance comes in. To get the cost of the hardware down, some (not all, but some) companies have tried to get away with lower-quality components, requiring owners to do far more regular maintenance and modding than on your standard household appliances (or even on your old non-3D printer – can you remember the last time you put any maintenance into that)? The quality of 3D printers is coming up, but right now, it is what it is.

Not to fear. Let's go over some of the best ways to take care of your 3D printer and keep it happy. Every 3D printer is different, and online forums can be helpful for particular manufacturers, but here are some general 3D printer maintenance tips that apply across the board.

7.2 How to replace the nozzle

The nozzle is one of the most important parts of a 3D Printer. If it is not clean the material will not get out and there will be a serious problem with your final print. If the printer is not working well, and it is clearly visible that the filament is not coming out in a correct flow amount, most probably the nozzle is dirty or occluded. First stuff it is necessary to try to clean it with a needle, after warming ut the printer.



Font: ultimaker.com

Clogs can impact the quality of your 3D printed products pretty substantially. Even a tiny clog can foul up the design or make the piece less structurally sound than it might otherwise be. If you see even a bit of a curve as the filament comes down from the nozzle, that's a sign you've got a problem. Fortunately, this is a fairly straightforward bit of 3D printer maintenance. Take your nozzle out of your 3D printer. You'll need some tape, hand tools a razor blade, a glass jar, and some high-quality acetone from your hardware store.

Font: https://youtu.be/SfACwC9diQY

This part can get sabotaged by debris that builds up after several 3D printing jobs. First, unscrew the extruder from the printer. Take off the extruder cover. You'll likely need some hex keys to take off the screws holding the fan in place. Depending on the model, you'll need to do a bit more dismantling before you get to the extruder, where you can use a sharp tool to scrape off the gunk. Following are some general instructions to keep in mind for replacing a nozzle:

1. Gain better access to the nozzle by moving the extruder (Z-axis) as high as possible. Usually, the procedure is the next:

Go to *LCD Menu - Settings - Move Axis - Move Z.* Spin the **Knob** to adjust the height.

2. Unscrew the two screws on the print fan and the single screw securing the fan shroud. Remove both parts (picture below).

Font: https://help.prusa3d.com/en/article/changing-or-replacing-the-nozzle_2069

3. Preheat the nozzle to the fusion temperature of the material is inserted into the printer

Of course, this operation must be done with the highest level of accuracy because heated parts can cause severe burns.

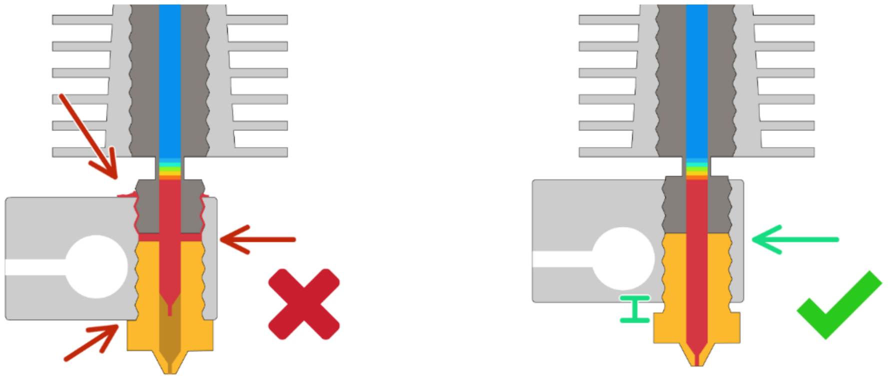
4. Unload the filament from the LCD Menu - Unload the filament or manually if there is no possibility on the printer.

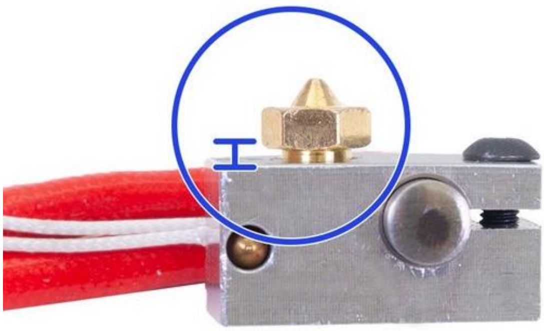
5. Hold the heater block with a wrench.

Be extra careful around the fragile hot end heater and thermistor wires can be broken.

1. Unscrew the nozzle using the supplied pliers. Do it carefully, the nozzle is still hot! So for this reason it must be placed out of the way on a non-flammable surface.
2. Make sure that the set temperature didn't change. Holding the heater block with a spanner, carefully screw the new nozzle in and tighten it firmly.

There must always be a gap (~0.5mm) between the nozzle and the heater block (left picture). The nozzle must be tightened/secured in the heater block, and locked against the heat break, while heated. Failing to do so will cause leaks (right picture).





Co-funded by the Erasmus+ Programme of the European Union

7.3 How to change material

Changing material on a 3D Printer is something frequent. For this reason, it happens problems like the below:

- Filament stuck in the hot end.

- Need to use excessive force to pull out the filament.

- Having a hard time feeding the filament back in the right place.

- Having bad print result after changing filament.

If not done correctly, it can damage the hotend. Not only there will be poor quality prints, one day the hotend will give up and stop extruding. It is suggested having to throw out a few hot ends. To avoid all this from happening all you need is to follow our step-by-step guide for changing filament. Before starting it is suggested to have the below information and tools ready.

7.3.1 Information:

The extrusion temperature settings for both the current & replacement filaments (based on the manufacturer's recommendations)

7.3.1.1 Material - Recommended Extrusion Temperature

- ABS - 150 to 260C
- PLA - 200 to 220C
- NEO-PLA - 188 to 200C
- Filaglow Glow in the Dark - 205 to 225C
- Filastic Flexible - 220 to 240C
- Filatron Conductive - 200 to 220C
- reFilactive Reflective - 230 to 240C

7.3.1.2 Tools:

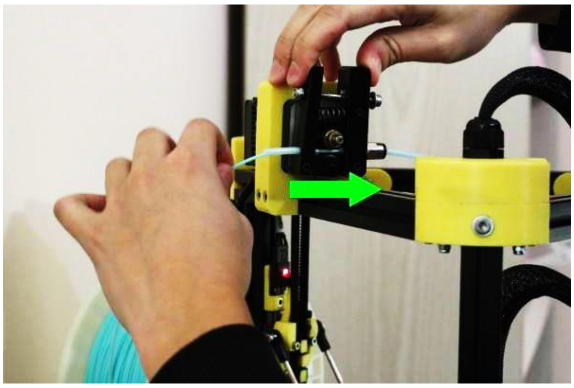
- A pair of scissors
- A pair of tweezers
- Current filament
- Replacement filament

7.3.1.3 Removal of the current filament

Step 1: Preheat your hot end based on the temperature guidelines of your current filament.

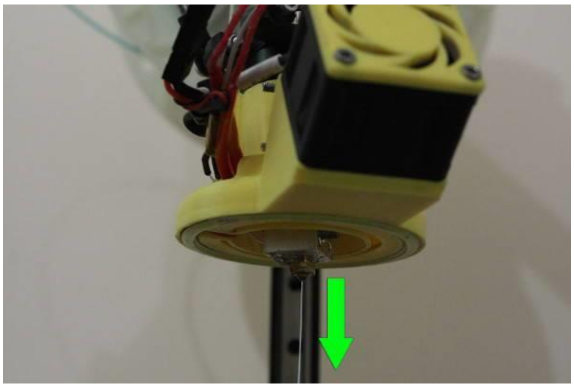
Step 2: Wait until the hot end heats up to the required temperature.

Step 3: Manually extrude a small portion of the filament.



Step 4: Unclamp the filament

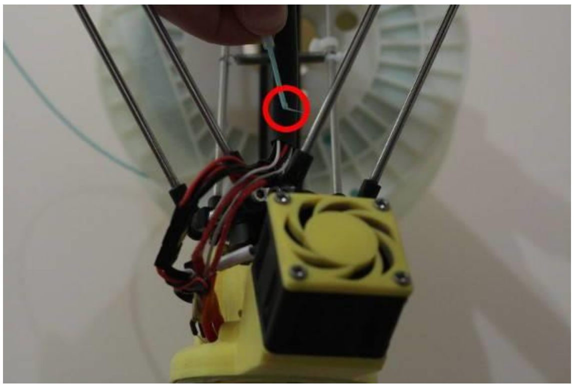
Step 5: Push the filament through the hot end until melted filament squeezes out from the nozzle. This process ensures easy extraction of the filament.



Step 6: Push down the coupling to release the filament from the hot end.

Step 7: Gently unplug the filament from the hot end.

Step 8: Clip off the blob-like end of the filament.



Step 9: Unwind the filament back to the spool holder.

Step 10: Unclamp the filament

Step 11: Slowly wind them back onto the spool holder. Please note: secure the loose end at all times.

Step 12: Secure the loose end through the hole of the spool holder or by binding it with a filament clip or tape.

- Step 13: Remove the current spool.

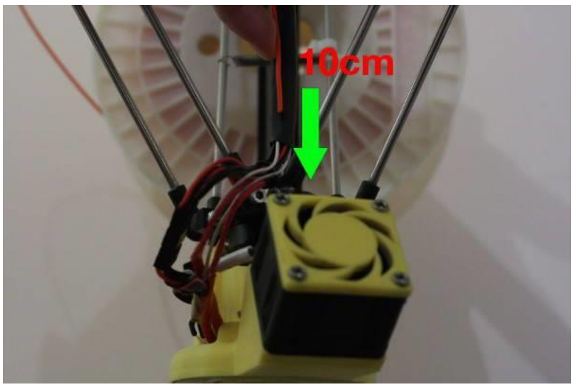
7.3.1.4 LOAD REPLACEMENT FILAMENT

Step 1: Load the replacement spool on the filament slot.

Step 2: Preheat your hot end based on the temperature guidelines of your replacement filament.

Step 3: Unclamp the filament and feed the filament through and up to the hot end.

Step 4: Prepare approximately 10cm of filament ready to be fed into the hot end.



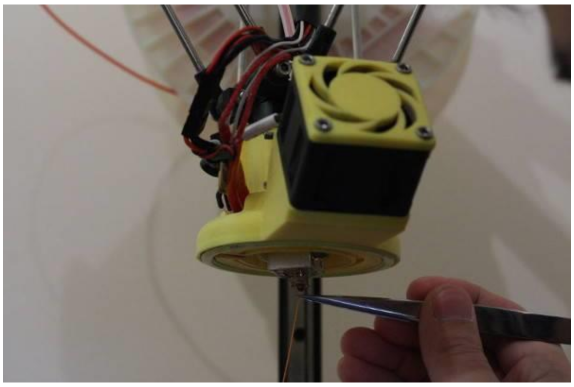
Step 5: Wait until the hot end heats up to the required temperature.

Step 6: Start feeding the filament into the hot end until the melted filament begins to squeeze out of the nozzle.

Step 7: Secure the coupling.

Step 8: Manually force down 3-4cm of filament through the hot end to flush out the old filament. Please note: flexible material may take more filament to flush out completely.

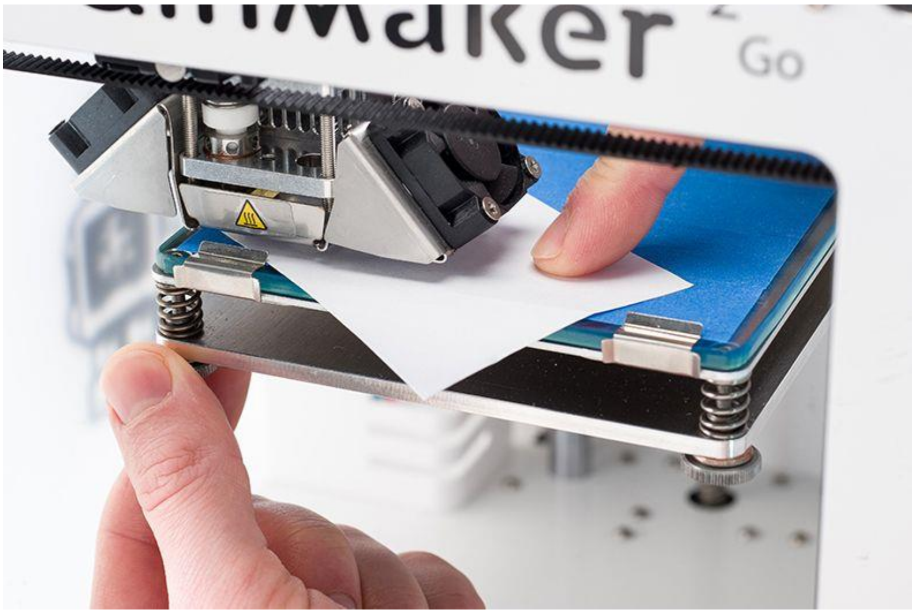
Step 9: Carefully clip off any excess filament from the nozzle with a pair of tweezers. Please note: do not touch the brass nozzle tip.



Step 10: Cool down your hot end.

Font: https://botfeeder.ca/blogs/tips-tricks-and-guides/how-to-properly-change-3d-printer-filament

7.3.2 How to calibrate the build plate



If the printer ahs, not have an automatic calibration system, it is possible using the knob to fix the Z height. As said before, it is suggested to use a paper sheet (0.1 mm height) to understand what is the correct distance between the nozzle and the bed surface. Try to move the sheet, if it is simply sloping the distance is too much, if it is stuck is too little. There is a correct distance when there is a friction but that allows however a movement of the sheet.

If the printer has an automatic Z heigh distance measurement, it will run a calibration phase. During this operation, the printer will automatically set a Z Calibration part, measure the distance between the nozzle and the plate, and calibrate the Z-Axis to the correct distance.

It is really important keeping a correct distance between the bed and the nozzle: from one site it allows to keep the first layer fixed on the bed, in this way it will not detach accidentally during the printing. On the other hand, the nozzle cannot be really close to the bed to guarantee a correct flow of material while printing.

7.4 General Advice

7.4.1 Keep Your 3D Printer Lubrificated

Just like with a car engine, lots of metal moving parts can lead to stoppages if you don't keep the rods and linear bearings from seizing up. You're not using motor oil, though. The sewing machine oil works well. Just a drop or two on your rails and rods will do the trick. Other kinds of lubricants can work – just make sure they're safe to use with plastic. Don't overdo it – too much grease can actually gum up the works by attracting dust and grime. Here's a good primer on how to do it courtesy of Jimmy Younkin on YouTube.

7.4.2 Replace Worn-Out Kapton Tape or Build-Surface Area

The area you're building on can get scratched up, affecting the look and integrity of your 3D creations. Easy fix, here.

7.4.3 Recommended routine maintenance for 3D printers

For guaranteeing a correct 3d printing functionality, it is suggested to carry out the following basic maintenance on a 3D printer so that it's always ready to print

Daily

- Clean out any dust and remains from the machine's interior before using it.
- Check the smooth rods and axis movement.
- Check that the printing profiles are correct.
-

Weekly

- Clean the printing surface (place the bed in hot water and use a spatula to remove any hairspray residue).
- Perform calibration and offset adjustment. This should also be done if an axis has been modified, if the printer has been shipped or if the Hot-End has been removed.
- Use the wire brush and cleaning needle to quickly clean the Hot-End.

Monthly

- Comprehensive calibration of the machine.
- Check if there are any firmware updates available for the machine.
- Clean the smooth rods (X and Y) with a microfibre cloth.
- Lubricate the smooth rods (X and Y) and the flanged bearings (if there are any) with sewing machine oil.

- Use a paper towel to remove grease from the Z-axis leadscrew, then lubricate with industrial lithium grease.

Quarterly

- Carry out an in-depth clean of the extruder by removing the fan and heat sink. Clean out any PLA remains with a paintbrush.
- Check the extruder wiring given that it undergoes a lot of movement when the printer is in operation. Also, check the connections to the motherboard. Only applicable to DIY kits
- Check threads, bolts, and springs (if there are any).
- Check belt tensions (X and Y axes). Only applicable to DIY kits
- Make a backup and format the SD card.
- Check the PTFE tube (heat to 200ºC then use thermal gloves or pliers to remove it) and replace it if necessary (only applicable to DDG extruders). Every 250h of printing

8 MESHMIXER –

8.1 HOW TO DIVIDE A 3D MODEL INTO DIFFERENT PARTS

1. To divide a 3d model into different parts there are different software. Many of these could be expensive, but one of the best software available for free (an open Source) is undoubtedly Meshmixer. The first thing to do to proceed with the division into parts of the model is to import it into the Software. Figure 1 Meshmixer shows how the model looks once loaded.

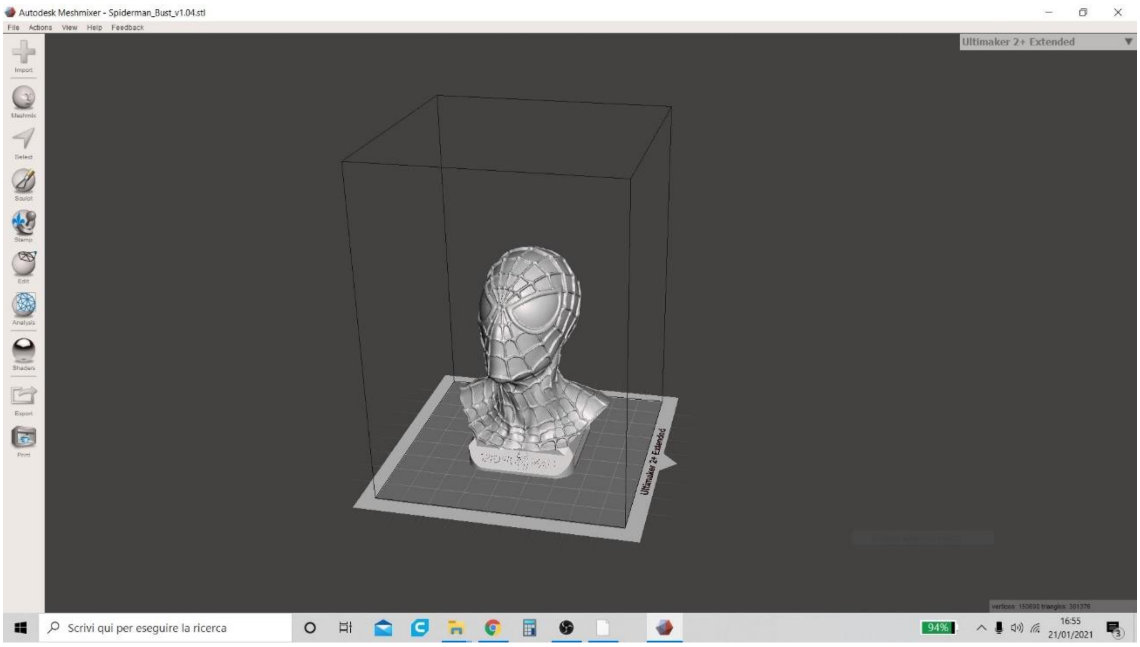


Figure 1 Meshmixer

2. Once the model has been imported into the Software, click on the “Edit” menu (present on the left as shown in Figure 2 Meshmixer) and choose the “Plane Cut” function. This function allows you to generate a preview of the cutting plan and the user can start the procedure for cutting the chosen model.

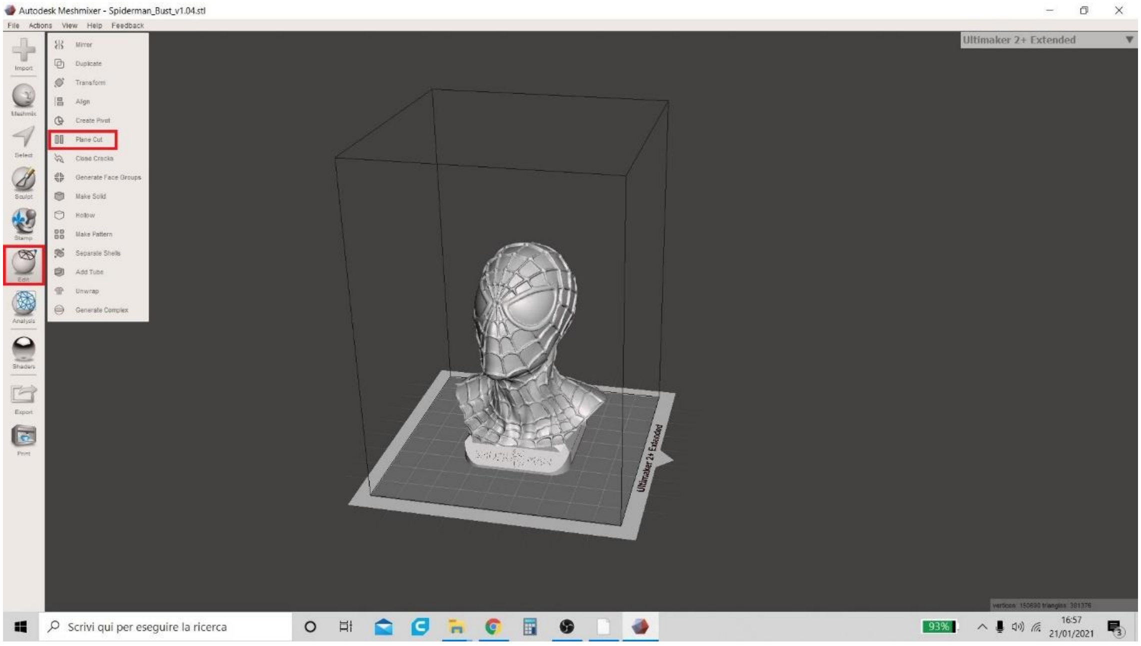


Figure 2 Meshmixer

3. Meshmixer offers the possibility to move the cutting plane along the three main axes x, y, z by means of arrows (in Figure 3 Meshmixer they are indicated with the three colours blue, red and green) or by holding down the triangle with the mouse (in figure green and red). In addition, it is possible to rotate the top in order to obtain cuts that have a certain inclination (at your choice) with respect to the horizontal by intervening on the icon in the shape of an arch (green or red in the figure).

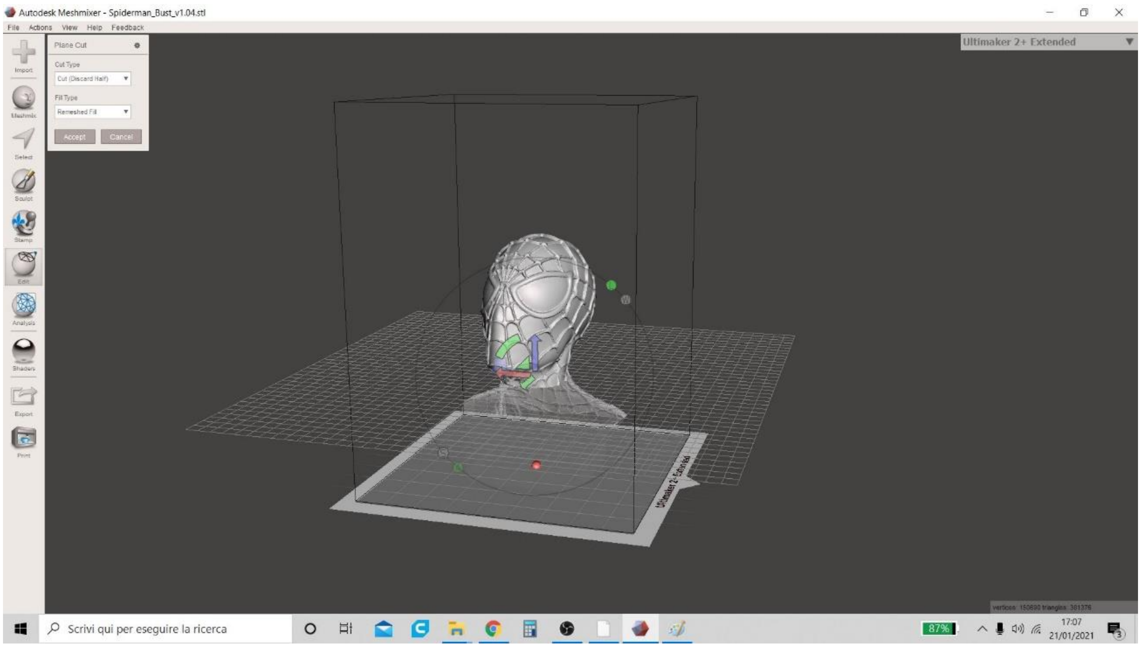


Figure 3 Meshmixer

4. Once the definitive position of the cutting plane has been decided, the user must choose the "Slice (Keep Both)" item in the" Cut Type "pop-up menu and then click on the button "Accept" (as shown in Figure 4 Meshmixer).

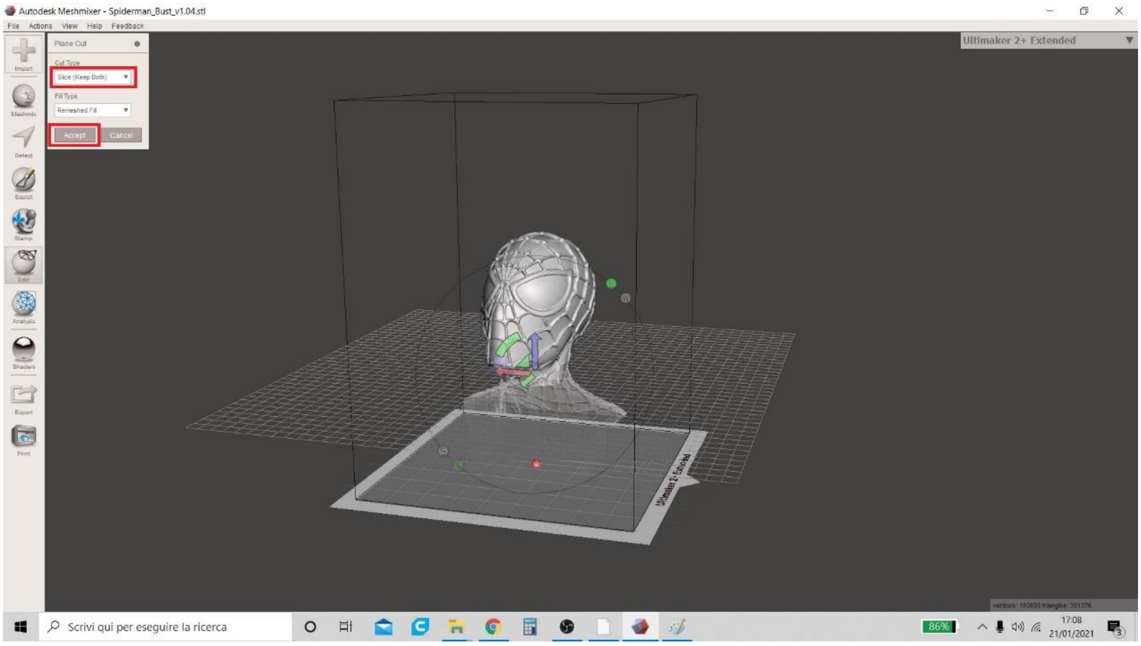


Figure 4 Meshmixer

5. After this operation, the user must return to the "Edit" menu and click on the "Separate Shells" function. This function will bring up a table indicating the two parts that were created with the cut (indicated with the original name of the starting model followed by the wording respectively of "shell1" and "shell2") as indicated in Figure 5 Meshmixer.

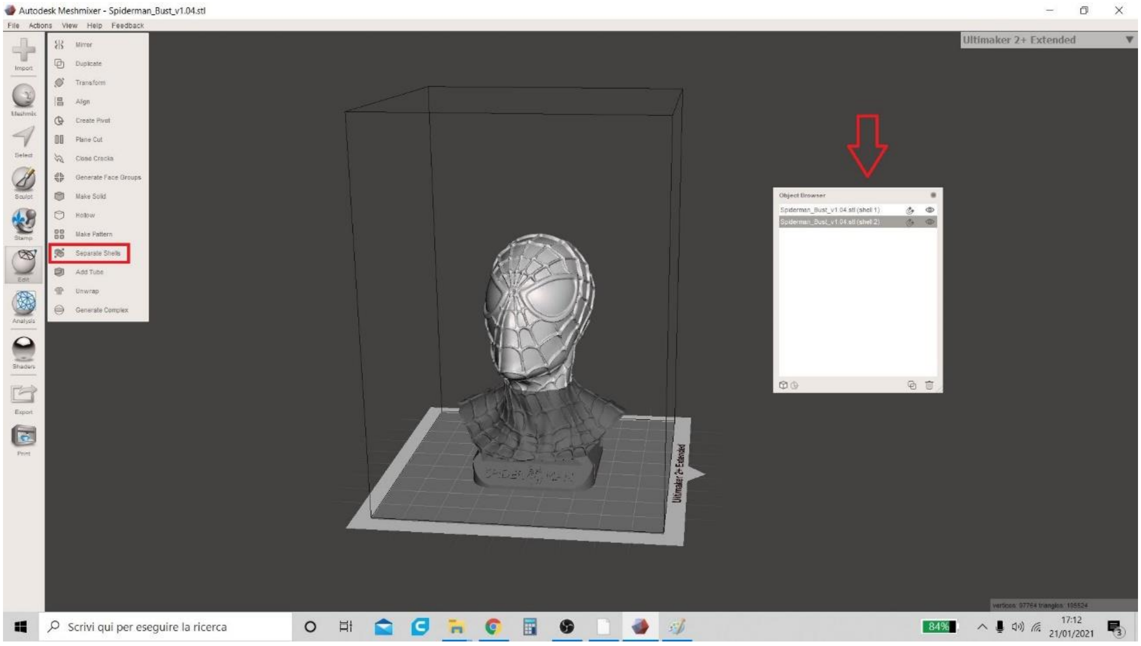


Figure 5 Meshmixer

6. By clicking on the eye icon in the table to the right of the names of both models (as shown in Figure 6 Meshmixer), the user can choose to hide one or the other part respectively (Figure 7 Meshmixer) in order to save the models in different formats separately. To save, click on the "Export" menu at the bottom left and then choose the format in which you prefer to save (e.g. the "STL ASCII Format" which is the one that appears by default).

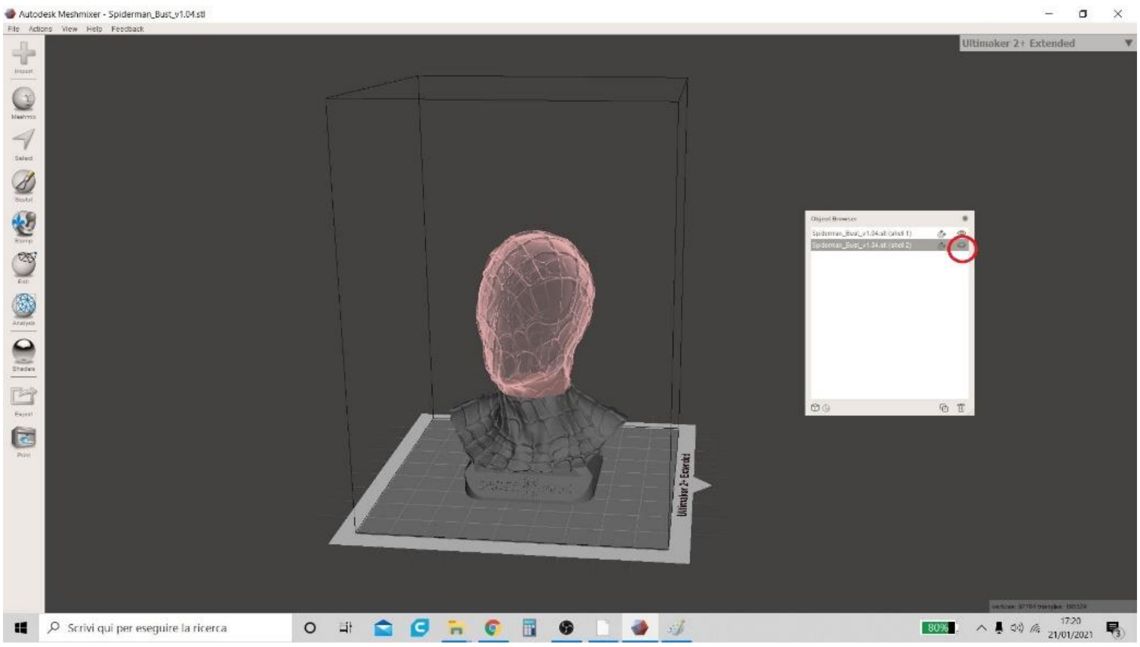


Figure 6 Meshmixer

Co-funded by the Erasmus+ Programme of the European Union

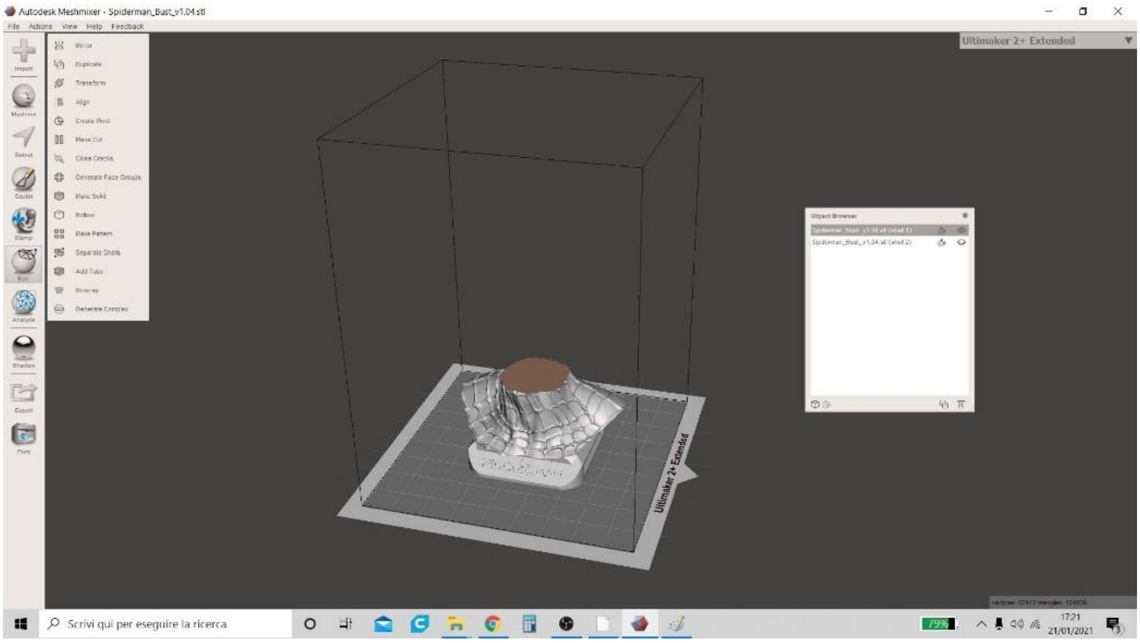


Figure 7 Meshmixer

8.2 HOW TO ADD ELEMENTS TO ASSEMBLE THE SEPARATE PARTS (HOLES AND PIVOTS)

1. Meshmixer offers the user another important feature. After dividing the model into parts, by creating holes on both parts (both on "Shell 1" and on "Shell 2") and a common pivot, it is possible to assemble the starting model before printing it.

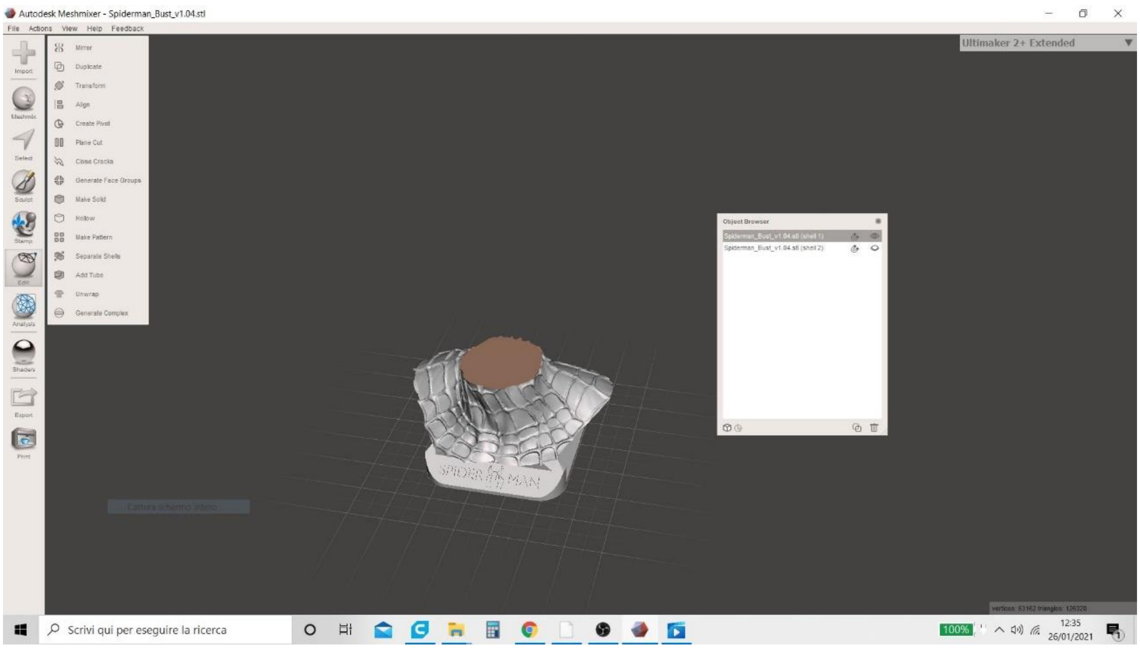


Figure 8 Meshmixer

2. First you make the "Shell 1" model visible and hide the "Shell 2" model. In the "Meshmix" menu, select the cylinder-shaped geometry (defined as "Pivot" - Figure 9 Meshmixer)) and drag this geometry on the model with the mouse. The software allows to scale (both as base diameter and as height) the Pivot and to move it to the correct point.

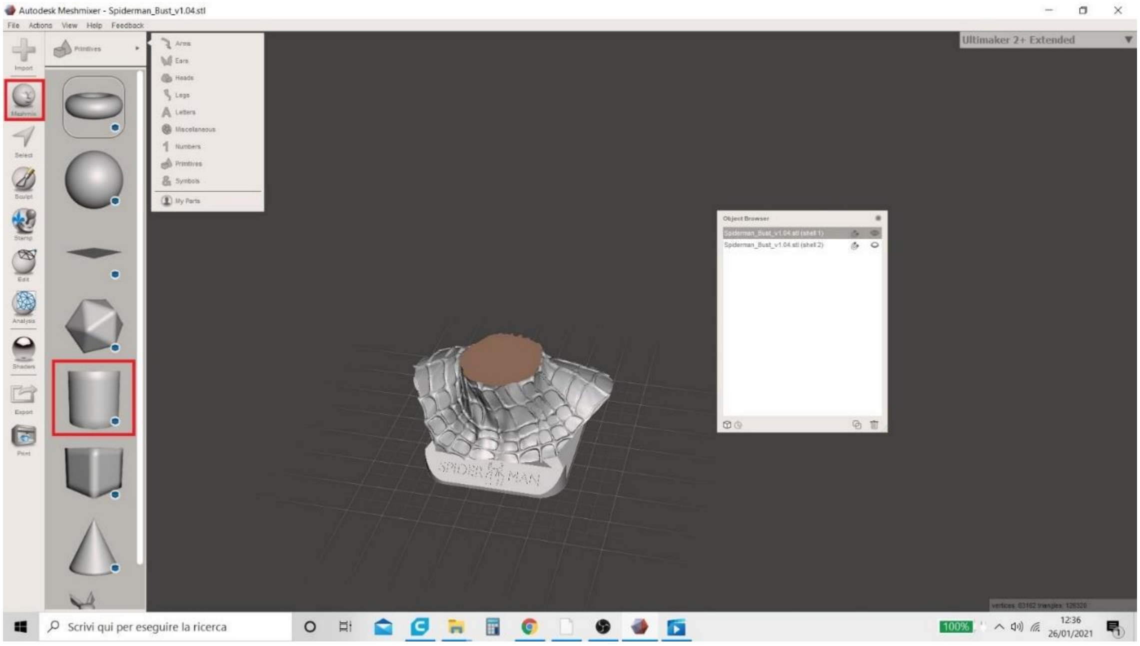


Figure 9 Meshmixer

3. After scaling and positioning the Pivot, the user must choose "Create New Object" from the "Composition Mode" pop-up menu (Figure 10 Meshmixer).

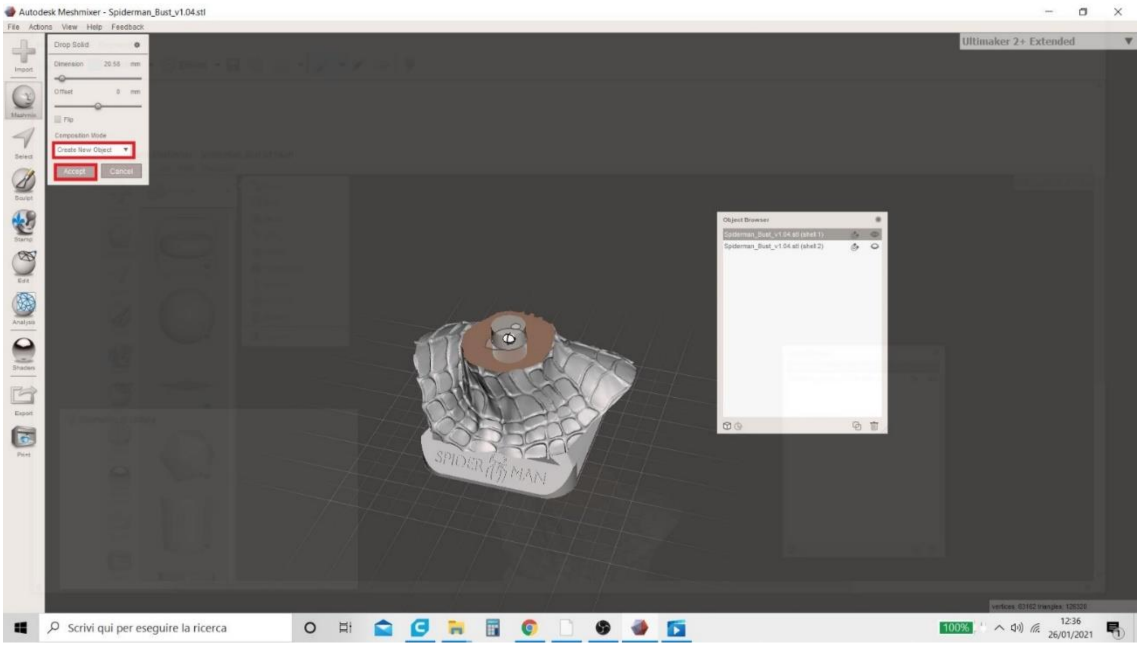


Figure 10 Meshmixer

4. At the end of this operation, in the table that provides the summary of the models present in the work plan, a third model will appear automatically called “Dropped Part 1” (our “Pivot” - Figure 11 Meshmixer).

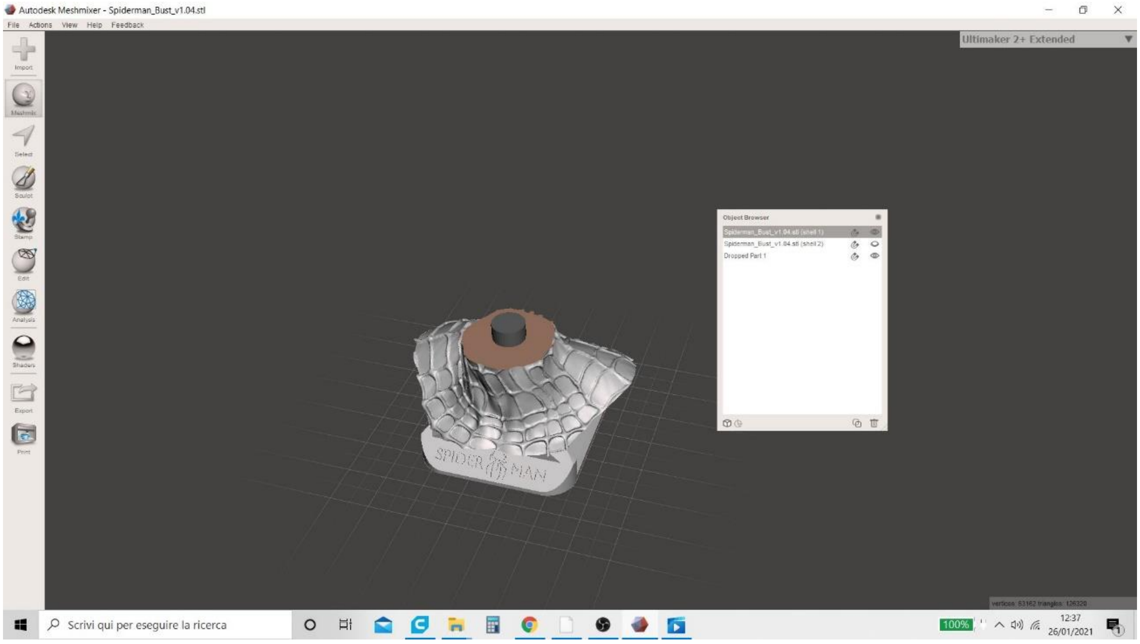


Figure 11 Meshmixer

5. To proceed, we need to create 2 copies of the Pivot (a copy to be used to create the hole on "Shell 1", a copy to be used to create the hole on "Shell 2" and finally a copy to be used as a real Pivot). The user must select the "Dropped Part 1" model with the mouse (which will become highlighted in gray) and click twice on the icon highlighted with a red circle in Figure 12 Meshmixer.

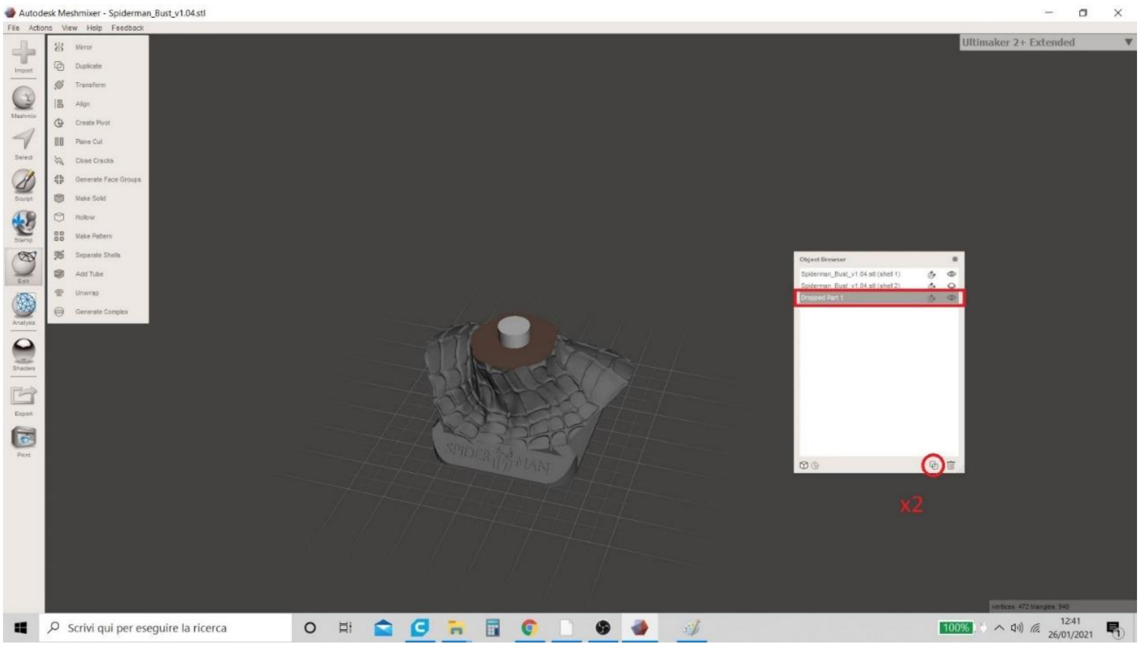


Figure 12 Meshmixer

6. This operation generates two other models in the summary table (Figure 6 Meshmixer) called "Dropped Part 1 (copy)" and "Dropped Part 1 (copy 1)" which are the two copies of the Pivot needed to drill the models. But it is necessary to hide the two copies just created (by clicking on the eye icon in the table as shown in Figure 13 Meshmixer) because they will be needed later.

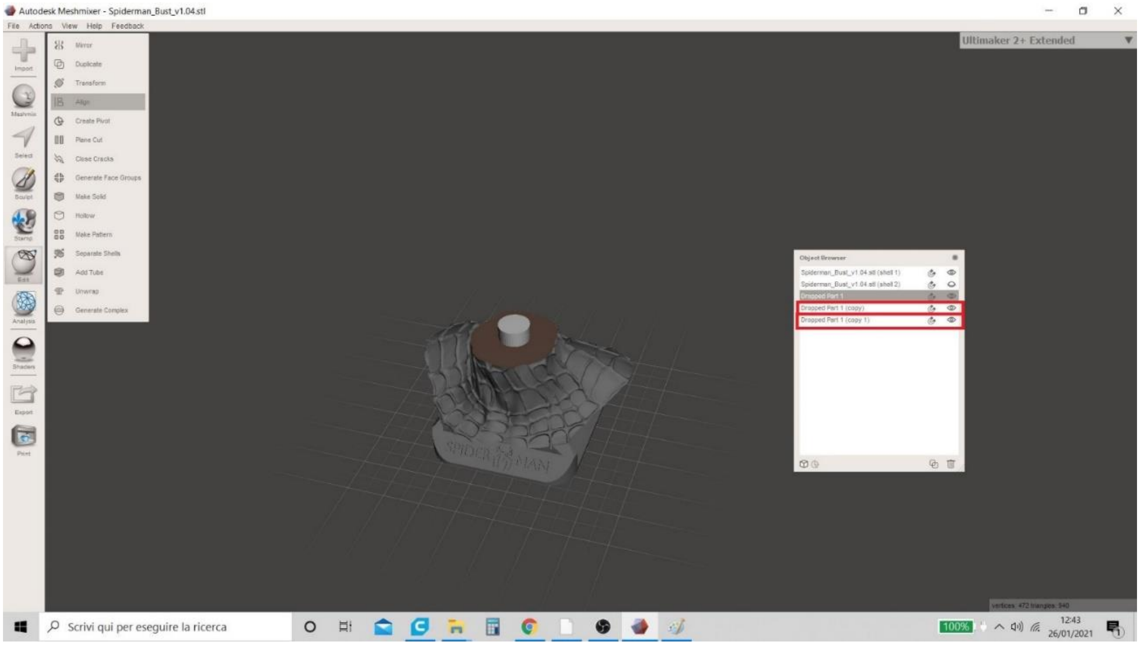


Figure 13 Meshmixer

7. At this point, the user must simultaneously select the "Shell 1" and "Dropped Part 1" models. To perform this operation correctly, you must select the "Shell 1" model, hold down the "Shift" key, and also click on the second model "Dropped Part 1" (OSS: the selection of the models must not be done by clicking on the respective names in the table, but directly in the work area). Once selected, a table will be displayed and the user must choose the "Boolean Difference" function (Figure 14 Meshmixer).

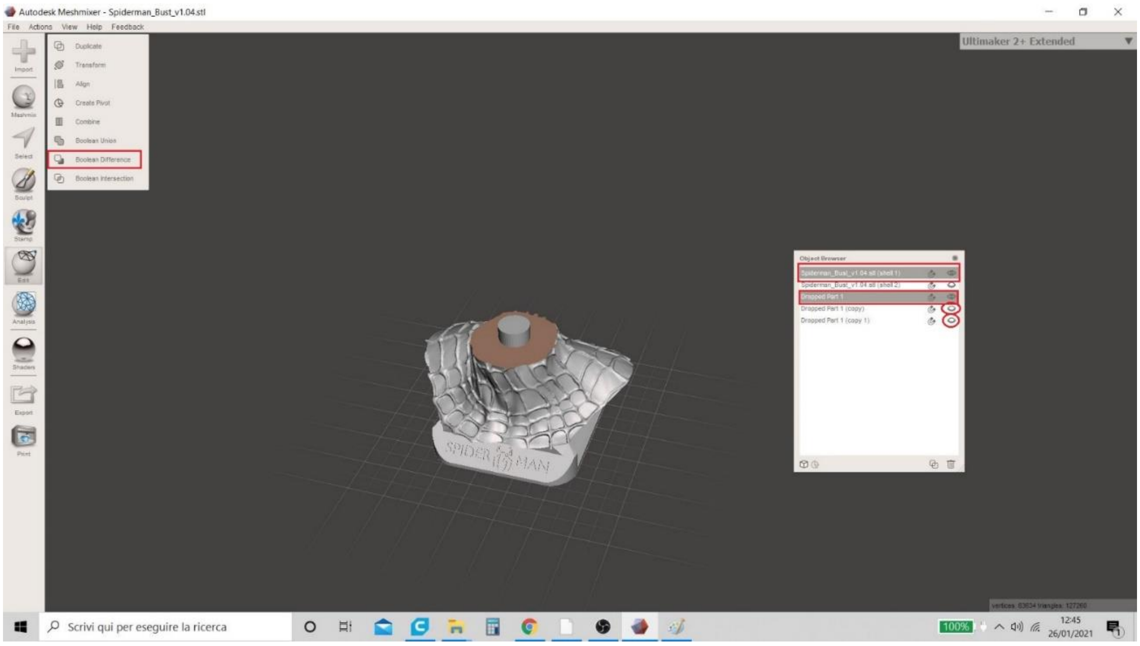


Figure 14 Meshmixer

8. The "Boolean Difference" function allows the user to view the hole on the model (Figure 15 Meshmixer) and generates a popup table at the top left.

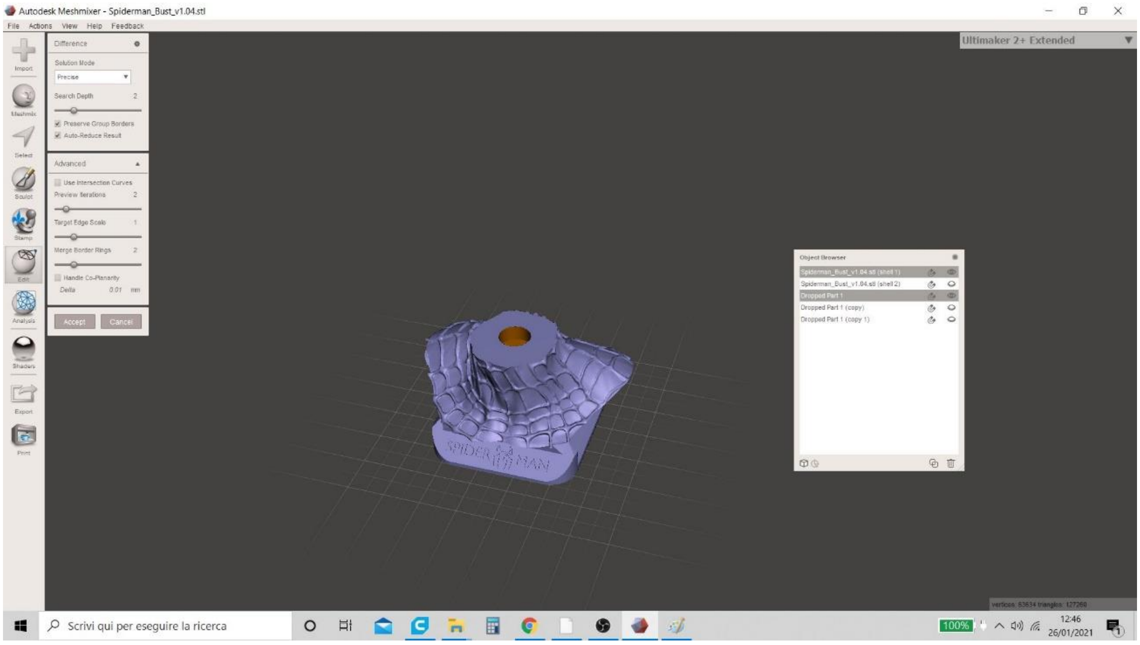


Figure 15 Meshmixer

9. To conclude the procedure, in the popup screen that has been generated, the user must:

- uncheck the item “Auto - Reduce Result” and check the item “Use Intersection Curves” instead;

- enter the value 0.5 in the “Target Edge Scale” item and click on “Accept”. The procedure is shown in Figure 16 Meshmixer.

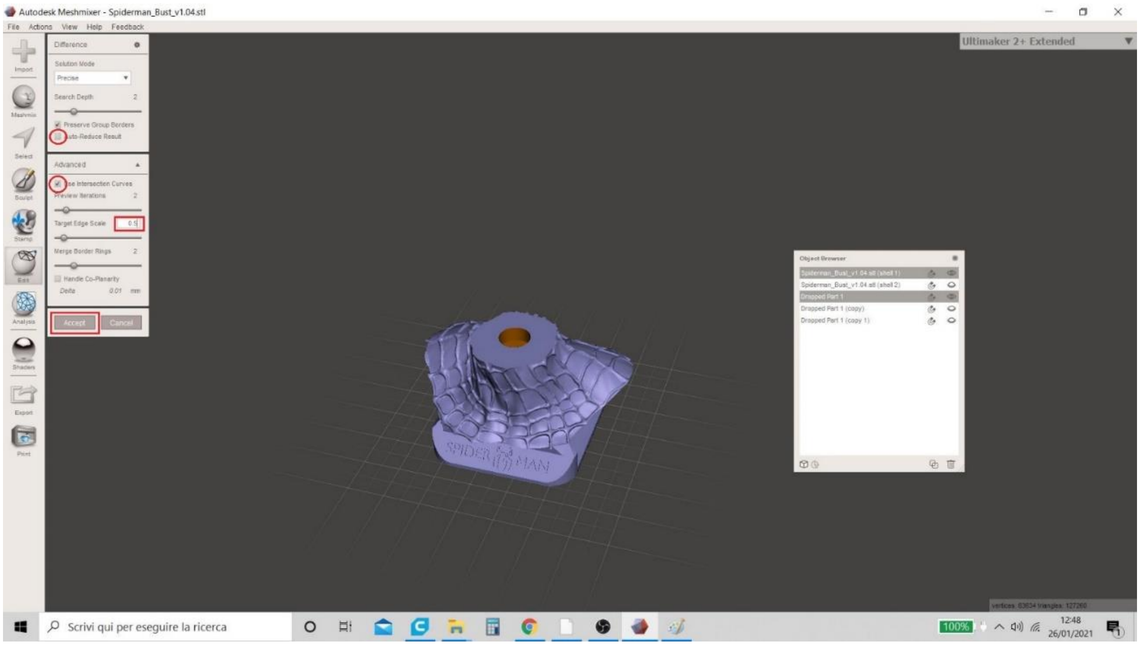


Figure 16 Meshmixer

10. The procedure for creating the hole in the “Shell 1” model is finished. In the table that summarizes the models in the work area, the user now sees 4 and no longer 5 (the "Boolean Difference" function has combined the two parts "Shell 1" and "Dropped Part 1" creating a single model - Figure 17 Meshmixer).

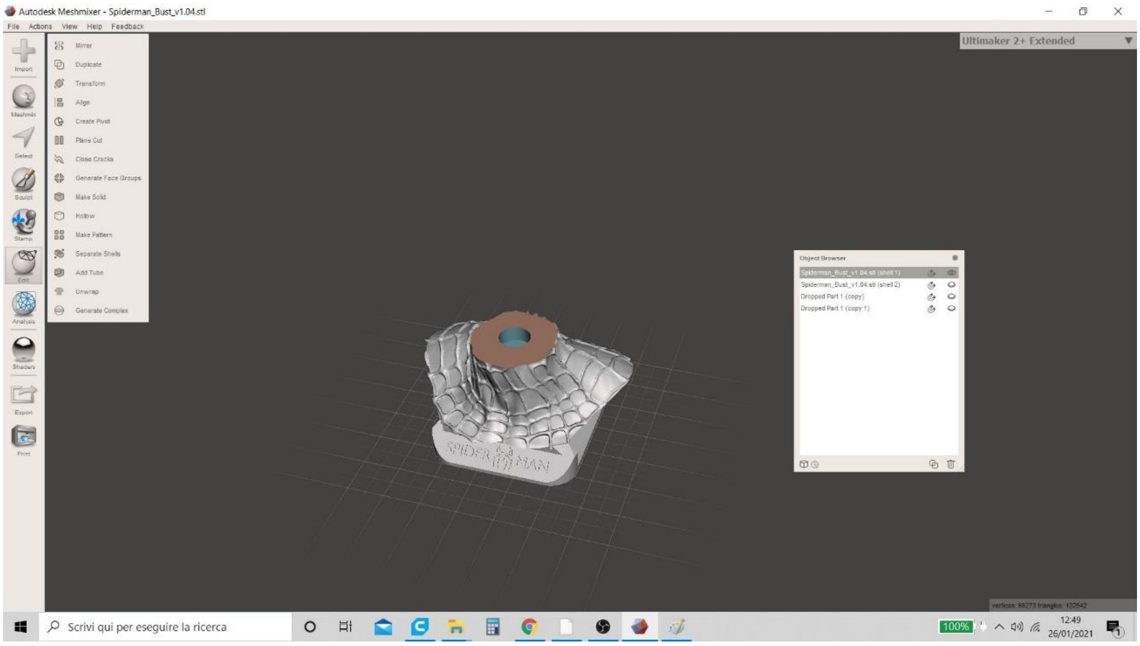


Figure 17 Meshmixer

11. It will proceed by hiding the combined model just created, and making visible "Shell 2" and "Dropped Part 1 (copy 1)". To do this, click on the eye-shaped icon in the table as shown in Figure 18 Meshmixer.

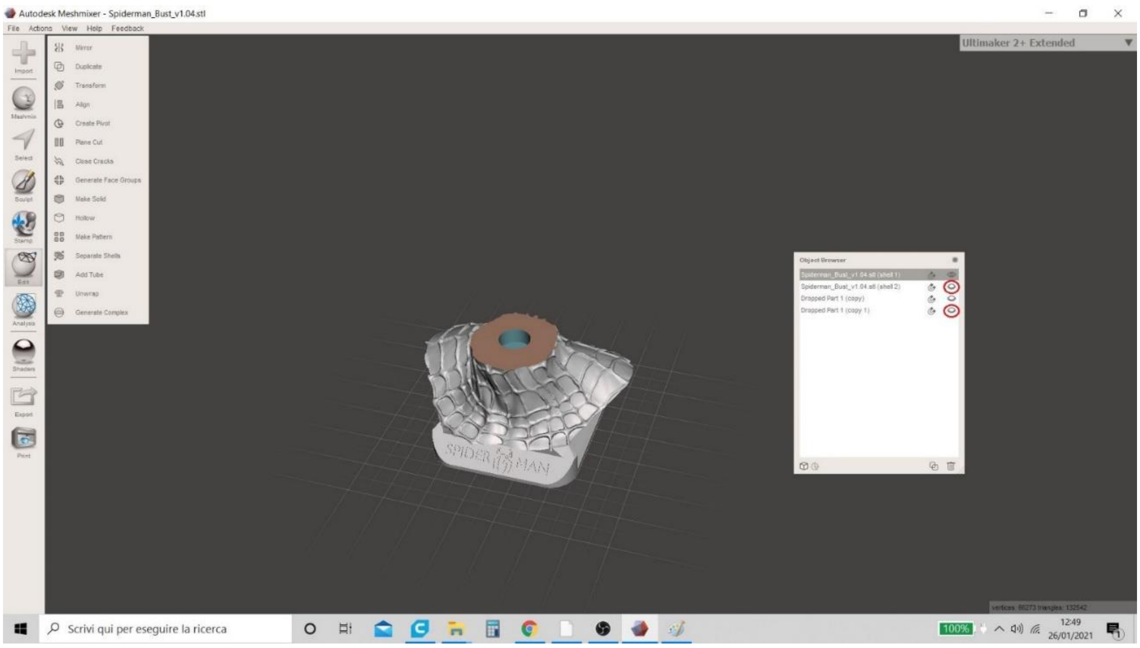


Figure 18 Meshmixer

Co-funded by the
Erasmus+ Programme
of the European Union

12. The two models "Shell 2" and "Dropped Part 1 (copy 1)" are now visible (Figure 19 Meshmixer)) and the user can repeat the same procedure to create the hole.

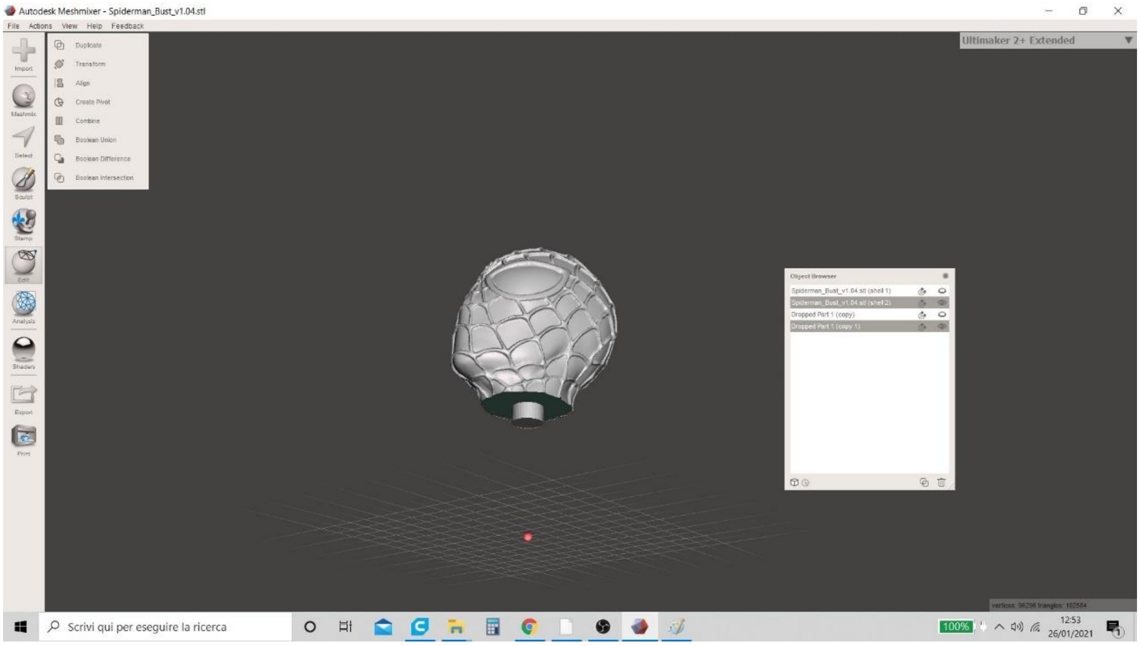


Figure 19 Meshmixer

13. Select the two models at the same time (by holding down the "Shift" key) and in the "Edit" menu click again on the "Boolean Difference" function (Figure 20 Meshmixer).

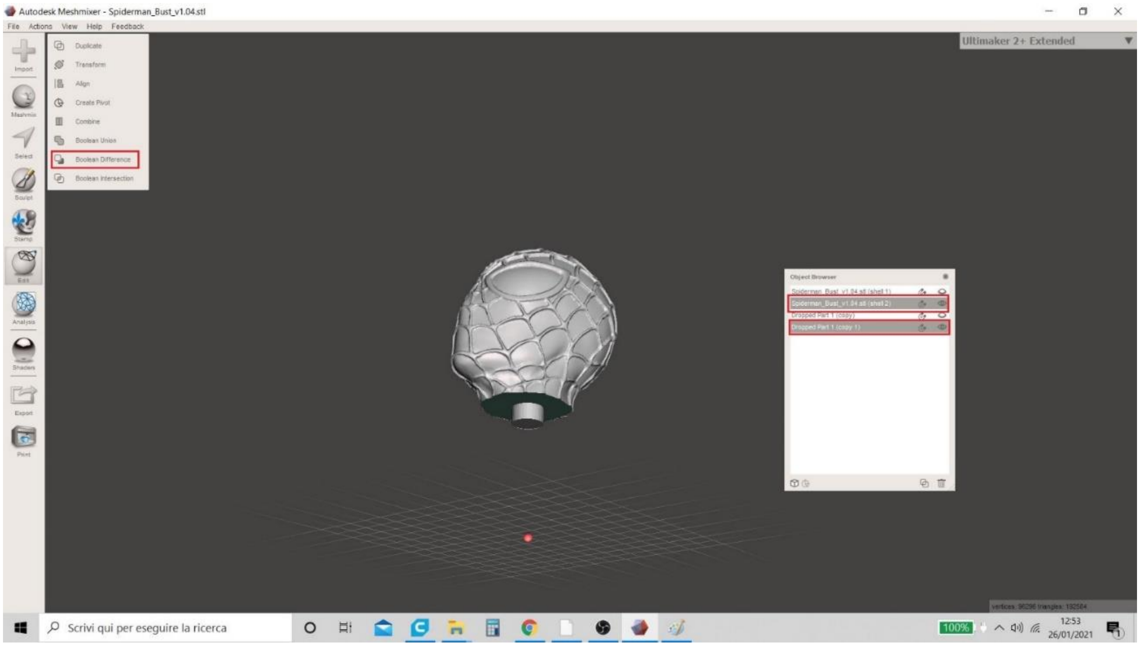


Figure 20 Meshmixer

14. All the operations are repeated and once clicked on "Accept" the user also will view the hole in the "Shell 2" model (Figure 21 Meshmixer).

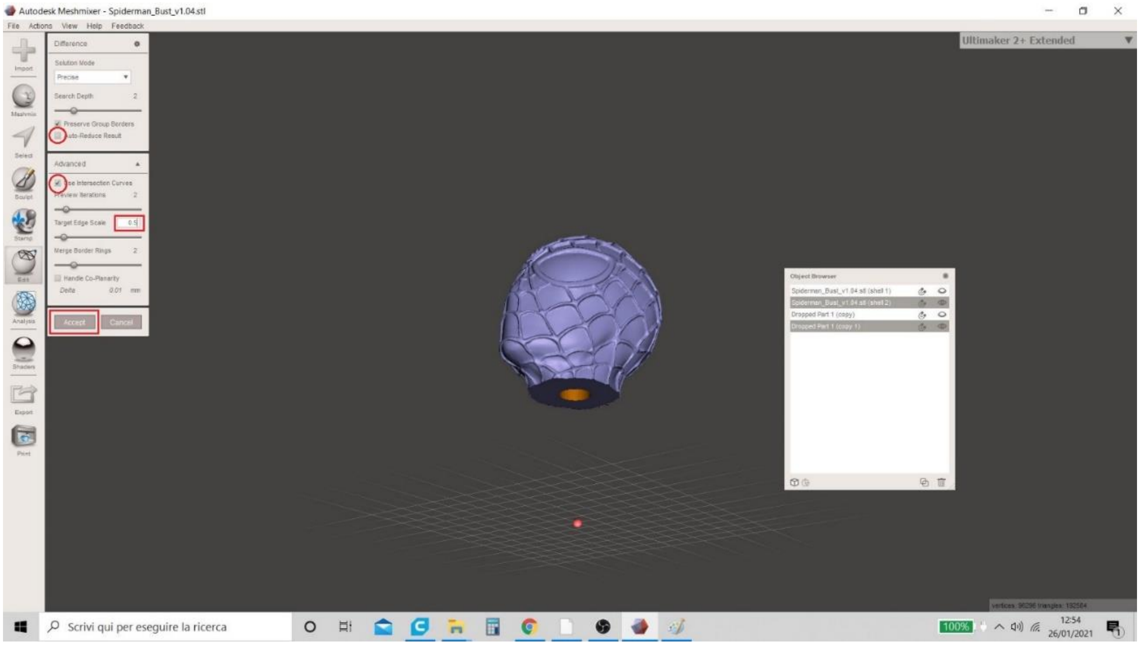


Figure 21 Meshmixer

15. The procedure for creating the hole in the "Shell 2" model is finished. In the table that summarizes the models in the work area, the user now sees 3 and no longer 4 (as before, the "Boolean Difference" combined the two parts "Shell 2" and "Dropped Part 1 (copy 1) " creating a single model - Figure 22 Meshmixer).

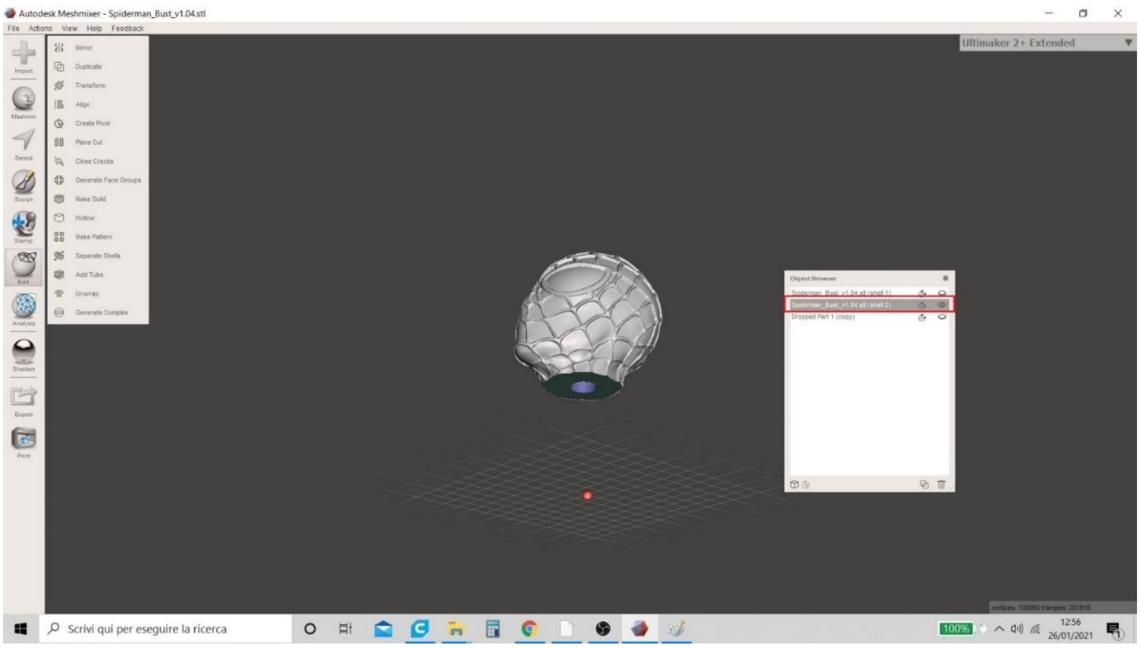


Figure 22 Meshmixer

16. The user can proceed to save the files in .stl format. By clicking on the eye-shaped icon in the summary table (as shown in Figure 23 Meshmixer), the user can choose to hide one or more parts respectively in order to save the models separately in different formats. To proceed with saving, the "Shell 1" model is made visible (and the others are hidden), click on the "Export" menu at the bottom left, and then select the format in which you prefer to save (e.g.: ". STL ")

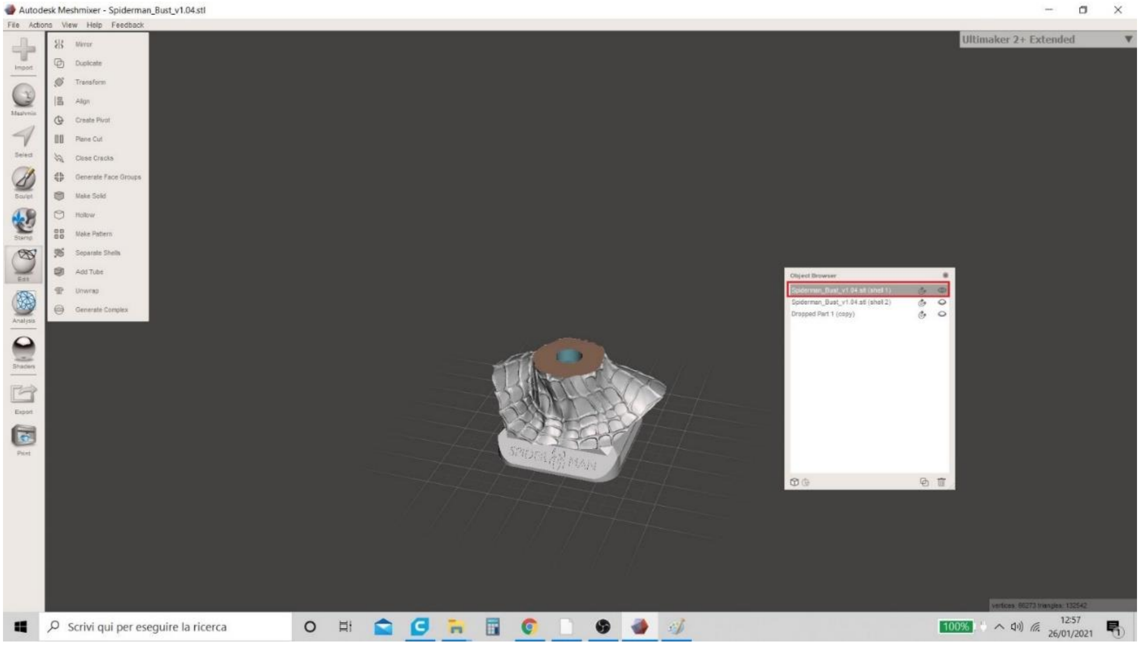


Figure 23 Meshmixer

17. The “Dropped Part 1” model is made visible (hiding “Shell 1” and “Shell 2”) and the file is saved in “. STL " format.

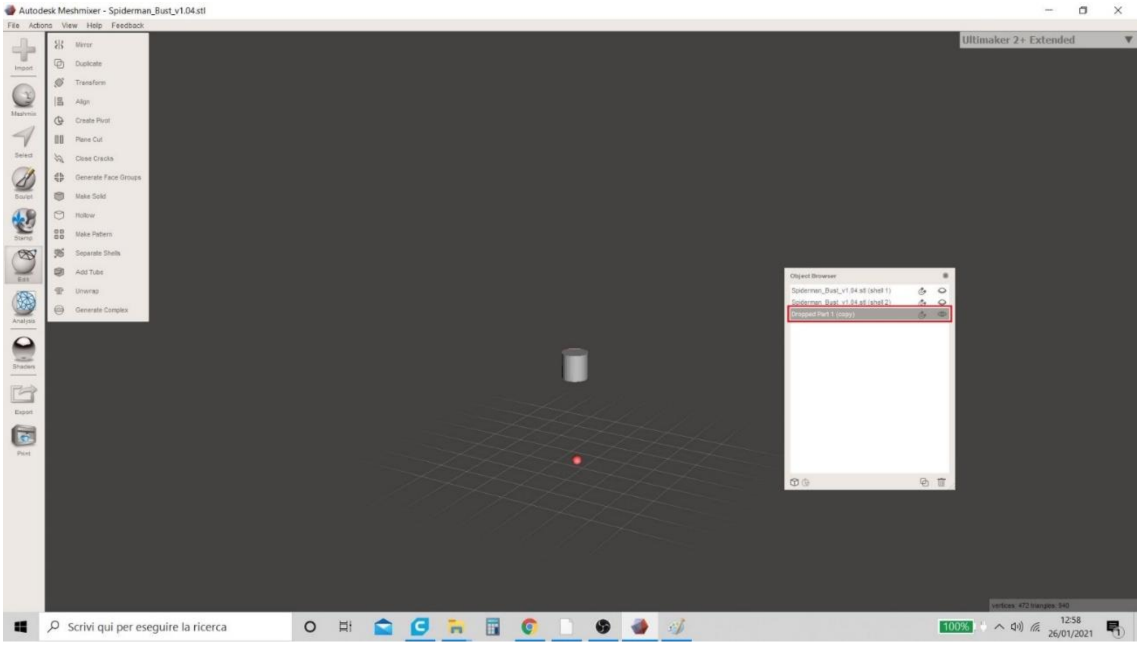


Figure 24 Meshmixer

18. Finally, the “Shell 2” model is made visible (hiding “Shell 1” and “Dropped Part 1”) and the file is saved in “. STL " format.

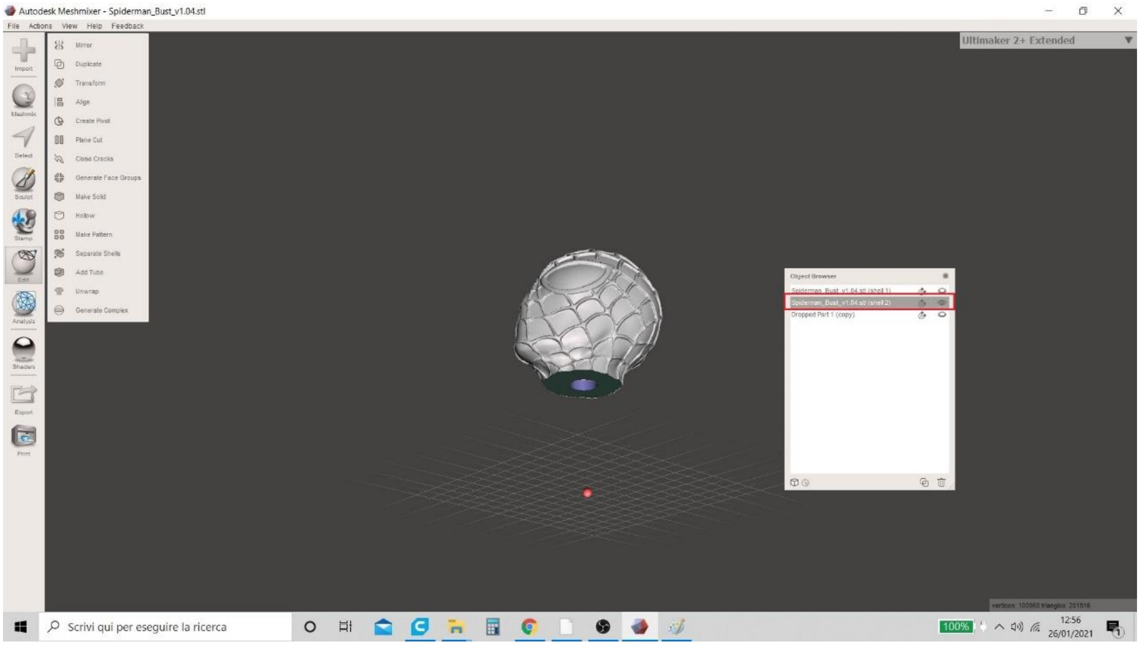


Figure 25 Meshmixer

8.3 HOW TO CREATE SUPPORTS WITH MESHMIXER

1. Meshmixer is the most used software for the creation (in an optimized way) of the support material necessary for the success of a 3D print. To start the procedure, the user must import the model into the software work area (Figure 26 Meshmixer).

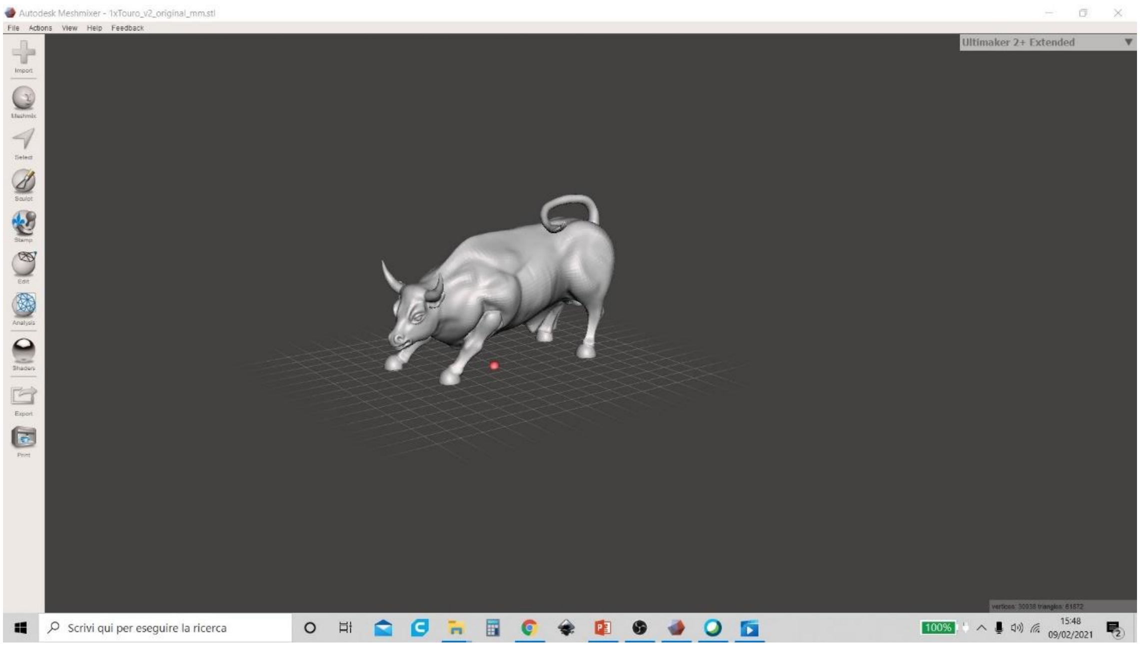


Figure 26 Meshmixer

2. Once the model has been loaded, click on the “Analysis” menu (on the left as shown in Figure 27 Meshmixer) and choose the “Overhangs” function. This function allows you to generate a preview of the critical parts of the model where it will be necessary to insert the support.

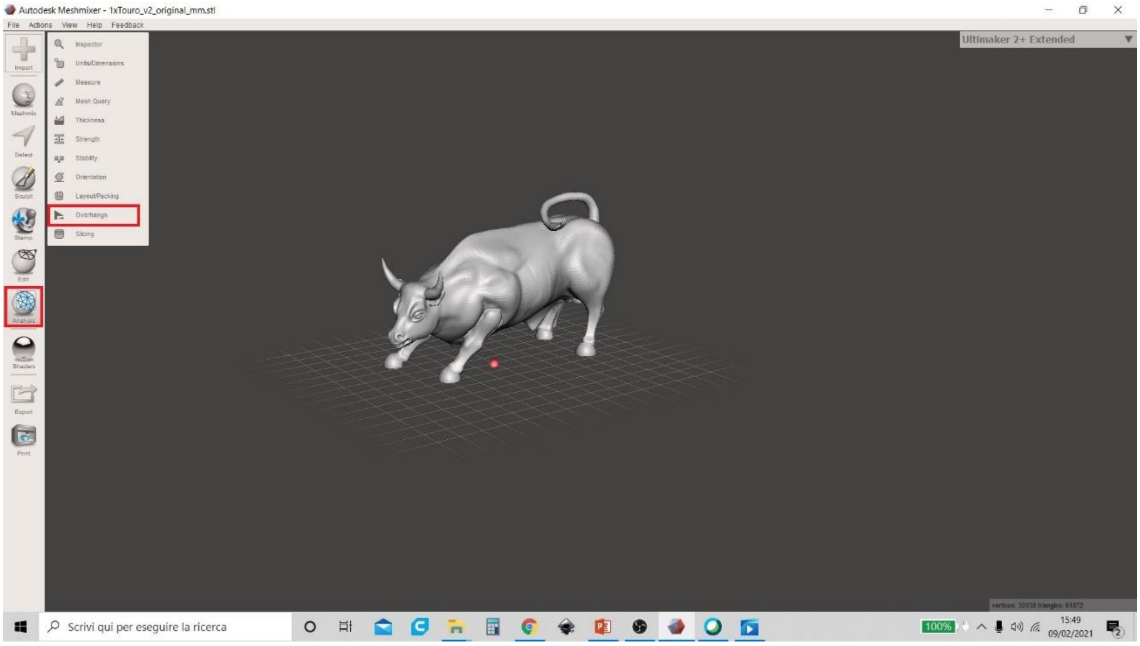


Figure 27 Meshmixer

3. The higher the value entered in the “Angle Thresh” item, the more critical parts will be indicated on the model (they are indicated by red halos with a blue outline). The user can choose an angle between 0 ° to 90 °. An optimal value to enter is 25 ° (Figure 28 Meshmixer). After choosing, the user must click on the "Generate Support" command.

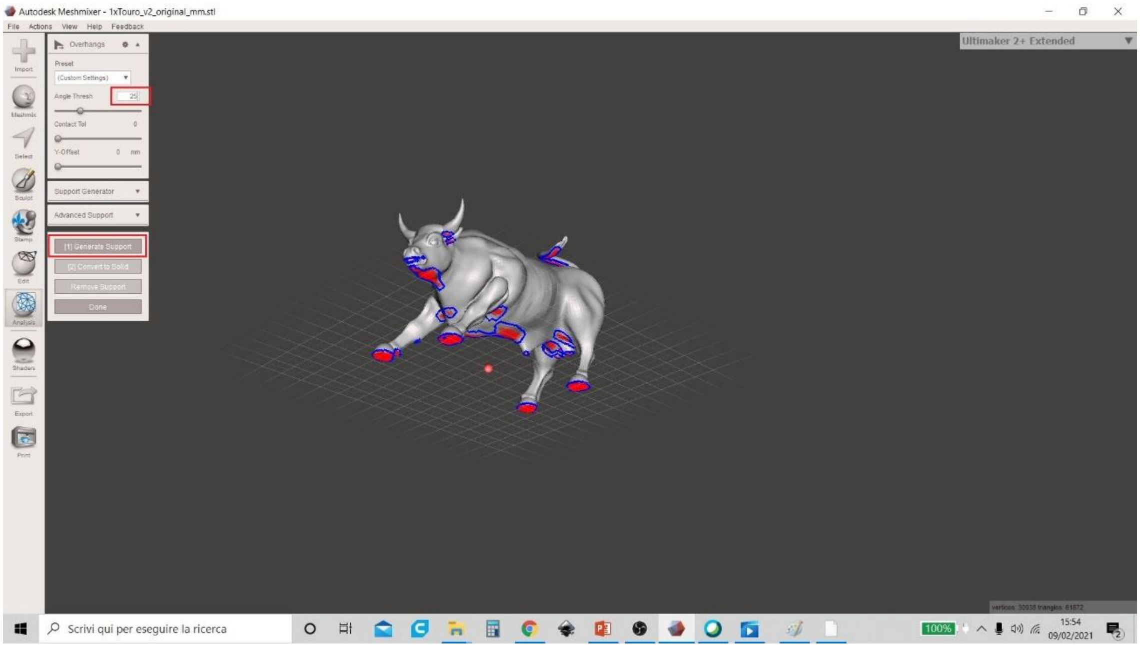


Figure 28 Meshmixer

4. This operation provides a preview of the structure of the support material. To be able to make other changes, however, it is necessary to click on the "Remove Support" command (otherwise the changes to the values of other parameters will not be correctly displayed) as shown in Figure 29 Meshmixer.

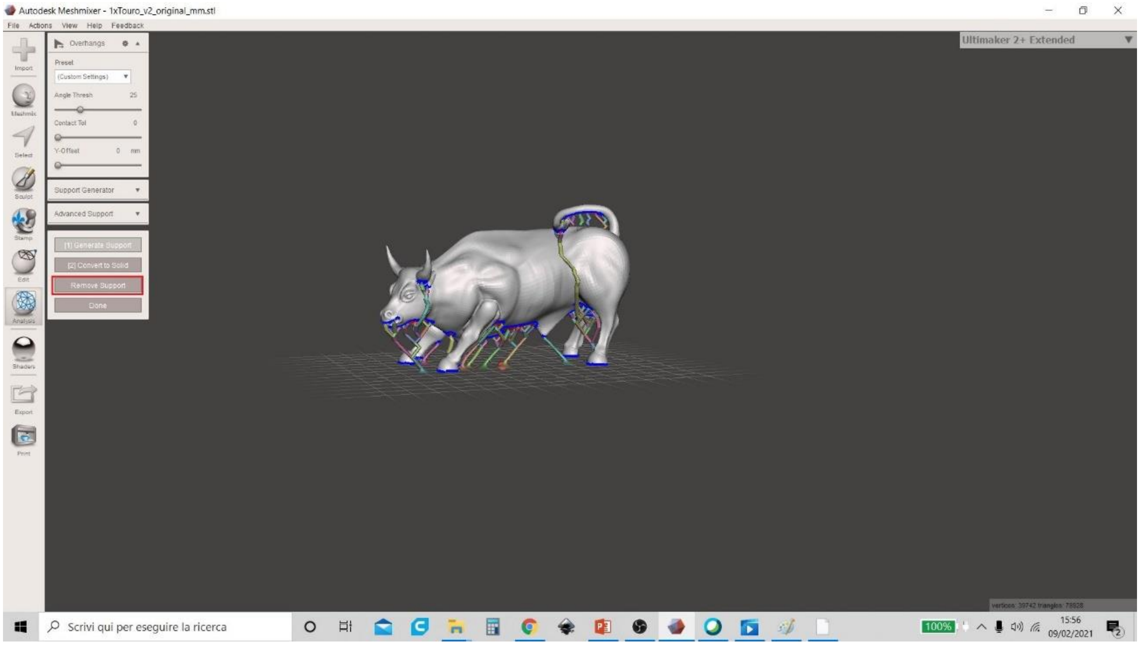


Figure 29 Meshmixer

5. The user has the ability to change many parameters. To view them all and interact with the Software, the user must click on the arrow in the "Support Generator" Menu and a drop-down menu will appear with all the items: "Max Angle", "Density", "Layer Height", "Post Diameter" "," Tip Diameter "and" Base Diamenter "(Figure 30 Meshmixer).

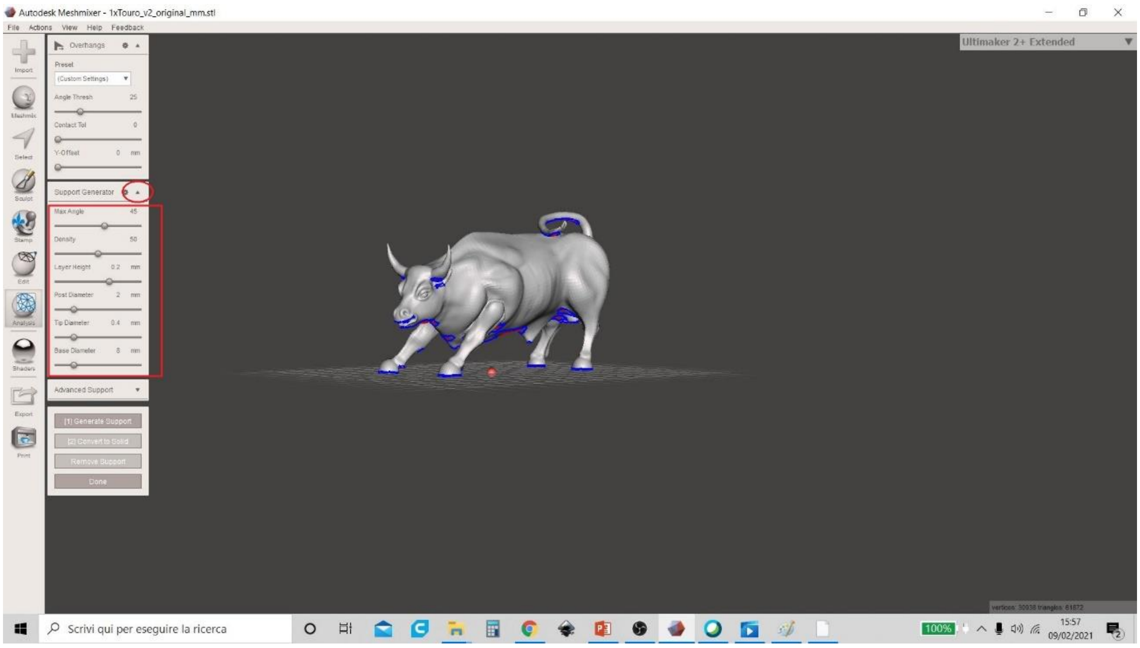


Figure 30 Meshmixer

6. One of these parameters (which most influences the printing time and the amount of material used) is the density of the support material. The user can optionally change the value in the "Density" menu item (the range of values goes from 0 to 100 €%) but the optimal value is 50%. Figure 31 Meshmixer provides a preview of the "Density" value of 100%.

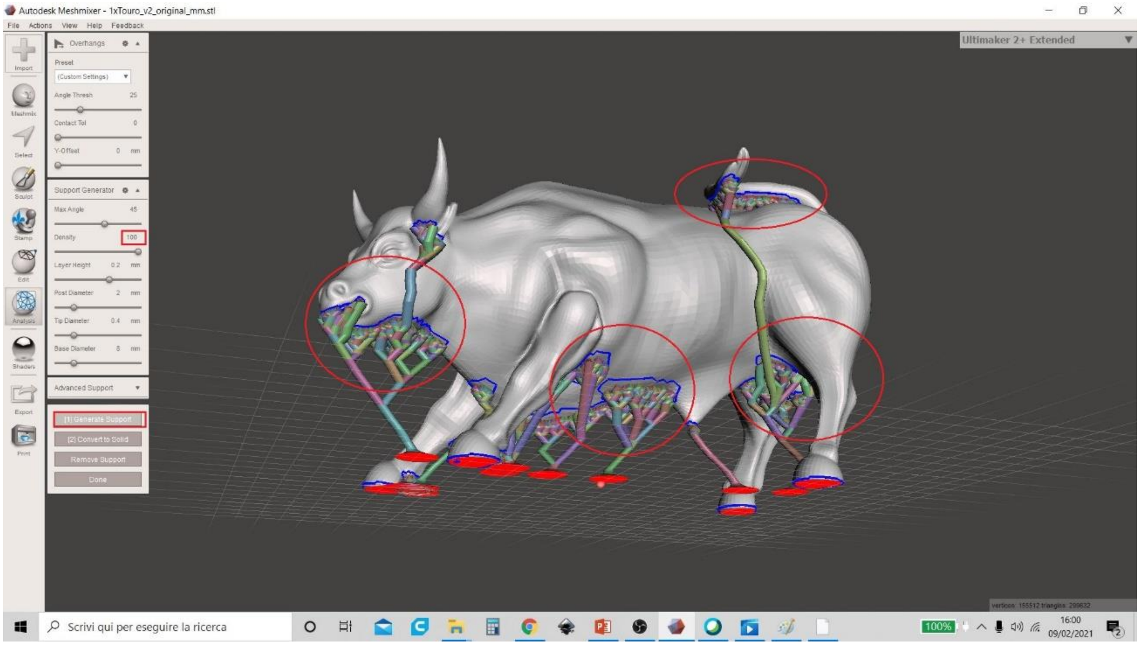


Figure 31 Meshmixer

7. Figure 32 Meshmixer instead shows a density of the support at 50% (the difference with a density at 100% in terms of material used is evident).

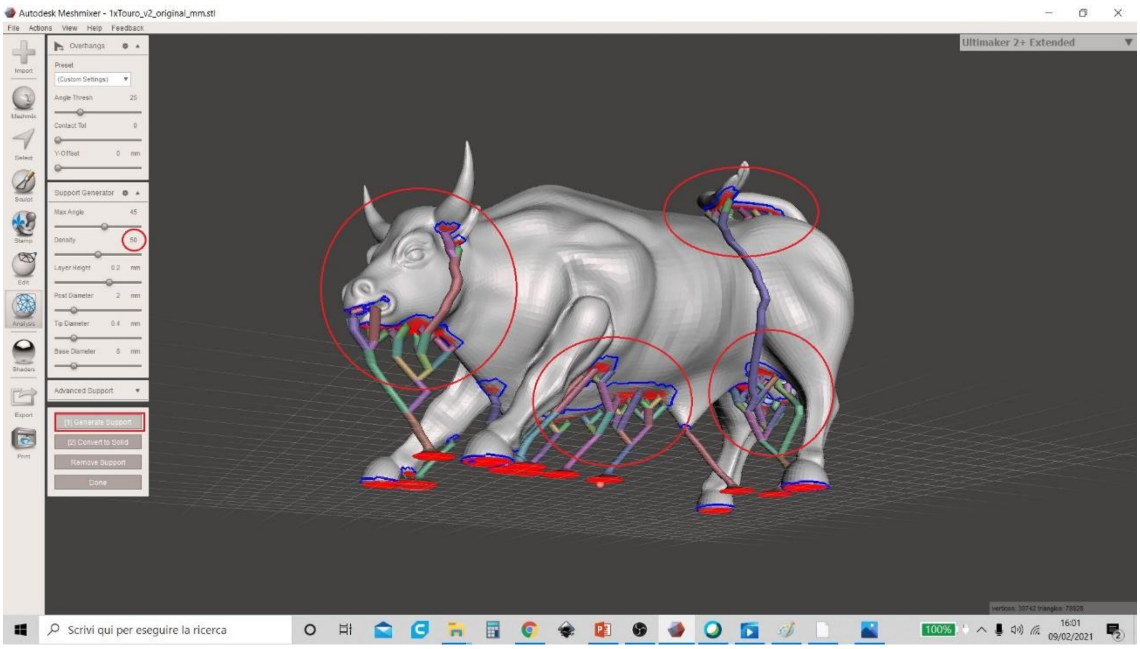


Figure 32 Meshmixer

8. Once all the parameters have been decided, the user must click on "Convert to Solid". In the popup window that is generated click on "New Object" (Figure 33 Meshmixer).

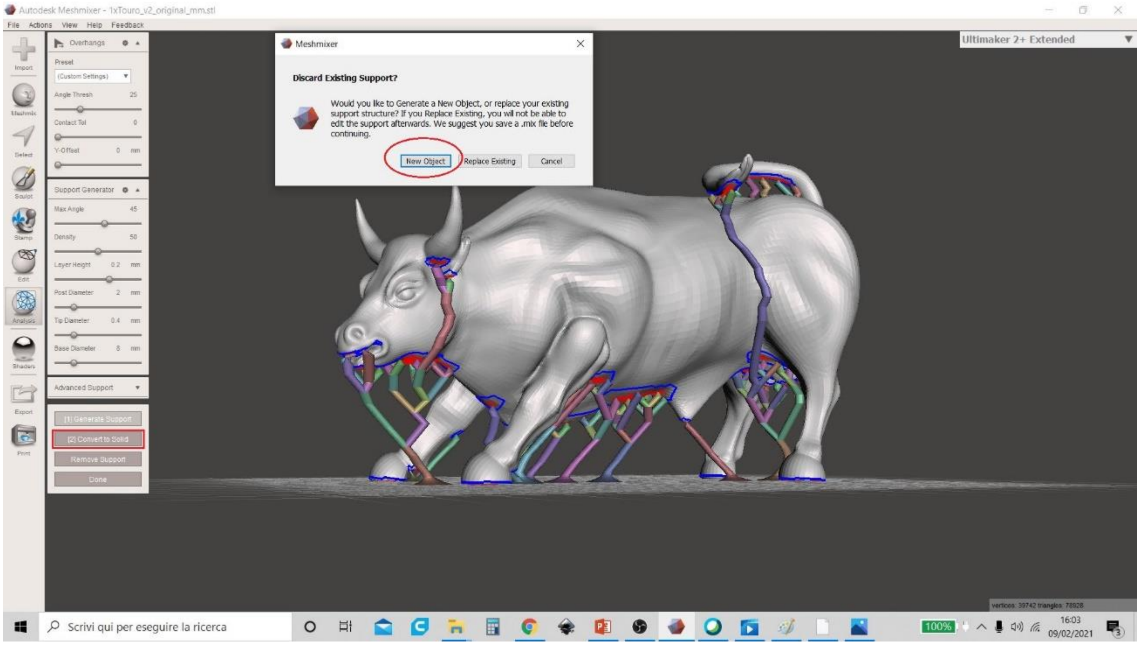


Figure 33 Meshmixer

9. At the end of the operation, a table will be displayed showing the models present in the work area. It can be seen that at this point the models become two: the original model and the support material (indicated with the name of the original model followed by "Support" which is considered by the software as a separate entity). Figure 34 Meshmixer shows the table with the names of the two models.

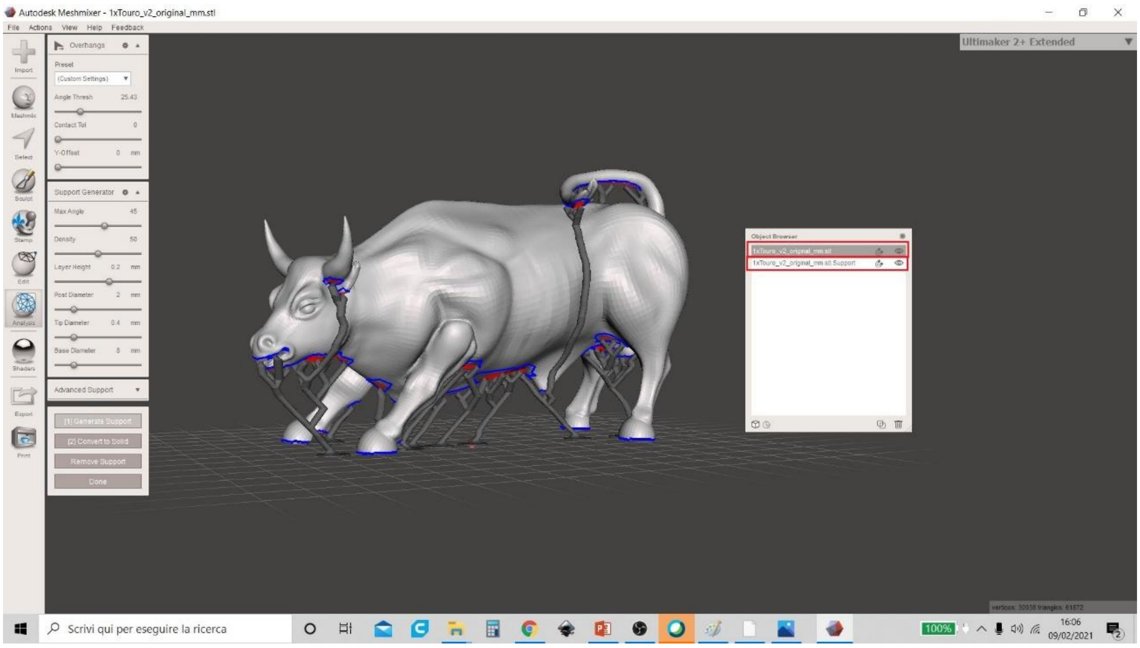


Figure 34 Meshmixer

10. It will be up to the user to decide whether to export the models together or separately (by making the models visible or hiding them by choosing the eye icon to the right of the names as shown in Figure 35 Meshmixer).

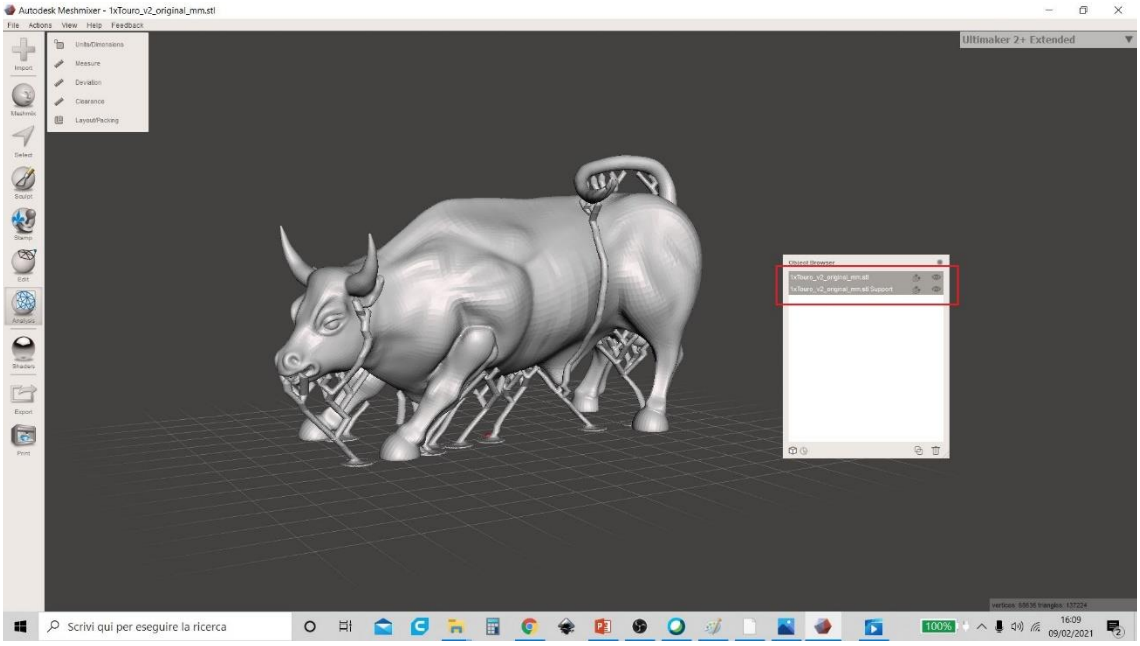


Figure 35 Meshmixer

11. To save both models, select them together (as shown in Figure 36 Meshmixer) and click on the “Export” menu. By default, the extension used by Meshmixer is ". STL ". Rename the model just created and click on "Save".

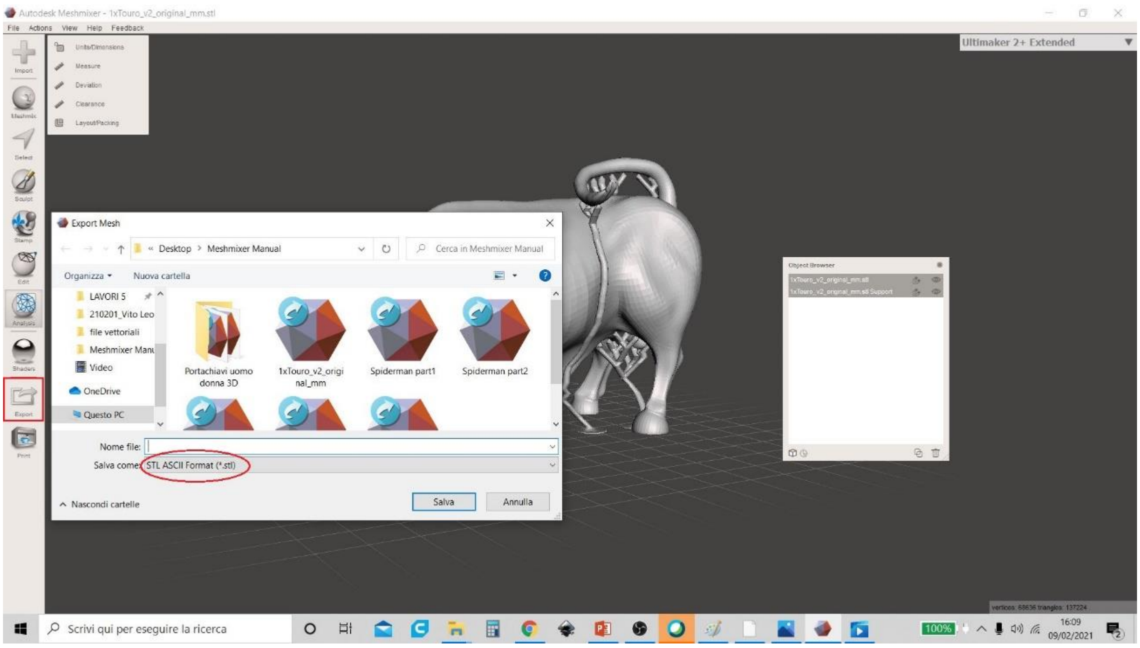


Figure 36 Meshmixer

9 Basic design with Tinkercad

9.1 Introduction

Tinkercad is possibly the simplest tool to get started in 3D design for those in vocational training or those who are not familiar with 3D design for additive manufacturing. The 3D design is a necessary complement to be able to have prototypes adapted to the mechatronic elements that are developed. Normally, the aim is to design parts to house the electronics, in the form of housings, which do not require a great deal of technical knowledge.

Logically, if the training cycle involves working in 3D design, more powerful software will be available to carry out the necessary designs. But the use of Tinkercad has the advantage that it does not require a licence, being free to use, without the need for installation on any device, which is not limited to the different operating systems and has therefore been selected as the initiation tool for these developments.

In the same way, a list of 40 basic pieces is proposed for the development of sufficient knowledge in Tinkercad to be able to train in the basic knowledge in design for the development of prototypes to be used combined with electronics for Industry 4.0. For each of the proposed parts designed, the settings necessary for additive manufacturing are established by the experts.

9.2 What is Tinkercad

Tinkercad is 3D prototyping software and consists of a set of applications that make up the Autodesk Apps group (formerly 123D Apps). These applications allow you to do the following:

- Designing 3D parts from pre-designed figures.
- Creating 3D objects using code.

In the 3D world, it is ideal for creating objects and then exporting them to 3D printers where they will be manufactured with additive manufacturing.

TinkerCAD is online and works from any browser.

It is limited in that it doesn't allow you to do the same as a professional 3D program, but for 90% of the designs we do it is more than enough.

TinkerCAD is basically a 3D Design program that works through the creation of parts by Boolean operations through the union of polygons.

Technical characteristics and advantages:

- It is an online application.
- It is free. You only need to register to access and use it.
- It is made up of 3 main modules: 3D Design, Electronics and Codeblocks. We will focus on 3D Design.
- We will be able to create reusable models. Once a part has been designed, we can save it as a block and create more complex parts from multiple blocks.
- It supports the most common formats for 3D printing. All designs can be imported and exported to .stl, .obj and .svg formats.

Tinkercad community gallery. You can download already made designs from the Tinkercad gallery: https://www.tinkercad.com/things/featured .

Tinkercrafting. You can import Minecraft models and edit them in the Tinkercad editor or export to Minecraft using MCEdit the part created from scratch or downloaded from the gallery.

9.2.1 Registration in Tinkercad 3D

Steps to create a Tinkercad account:

- Access the website www.tinkercad.com.
- Click on the "Start Tinkering" button.
- Create an Autodesk account.
- To create the account we must enter our country of residence and date of birth.
- Enter your email address and create a password.
- Once the account is created, we can click on the "Done" button. Remember to verify the account from your email address.
- Once the account is created, we can click on the "Login" button and we will have immediate access to our Dashboard (control panel).

As it is online, there is no need to download TinkerCAD.

9.2.2 Change Language in Tinkercad

When entering Tinkercad we usually find the need to change the language you desire:

- Logging into Tinkercad session, we must go to the bottom and look for the Autodesk logo.
- There you will see a drop-down list of languages.
- Click to display the available languages.
- Select your preferred language.

9.2.3 Main Tinkercad Controls

The controls of TinkerCAD are very simple and with them you will be able to move everywhere:

- Right click: Rotate.
- Move mouse wheel: Zoom.
- Press mouse wheel: Translation.
- Press left mouse button: Multiple selection.

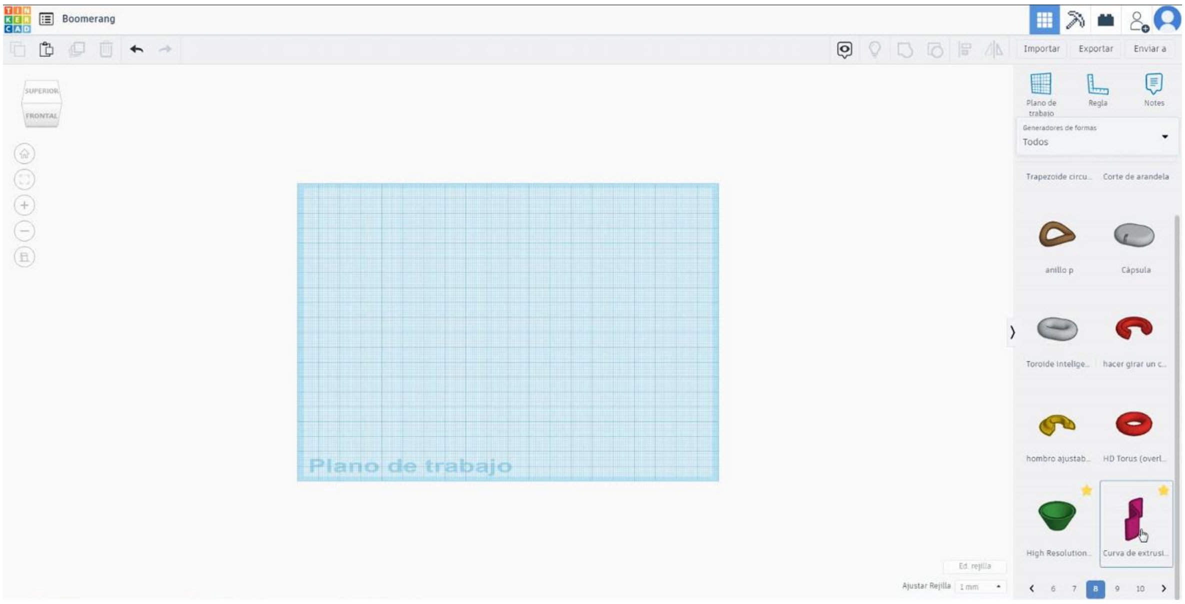
In addition, you can move a cube at the top left to move the whole object and if you double click on its faces you can position your view perpendicular to that plane.

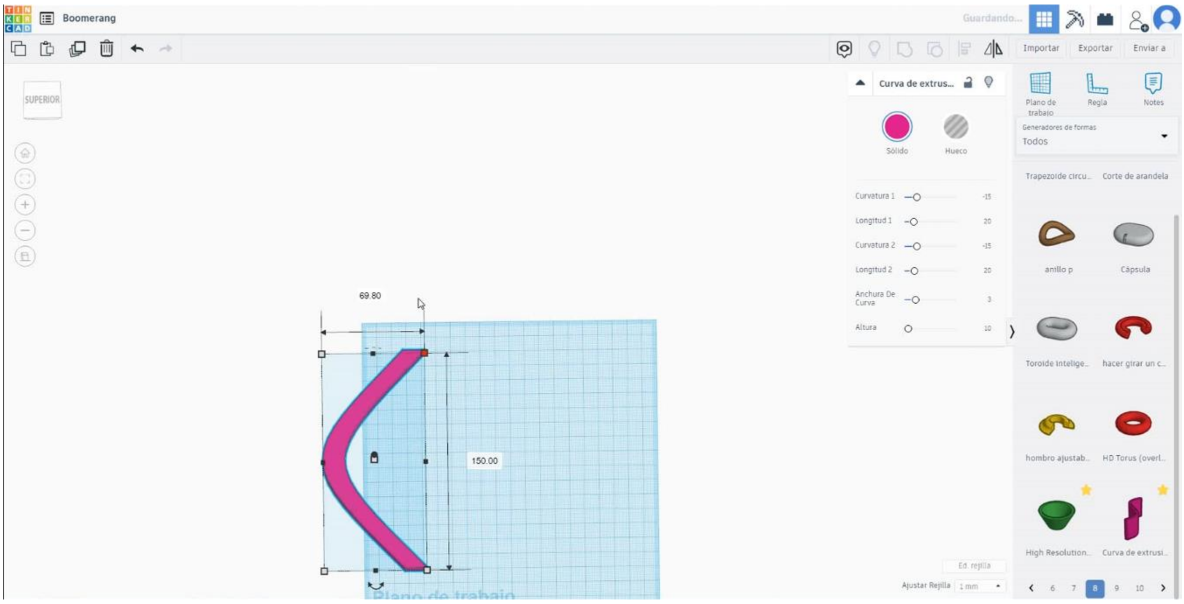
9.3 40 models designed and set up for 3D printing

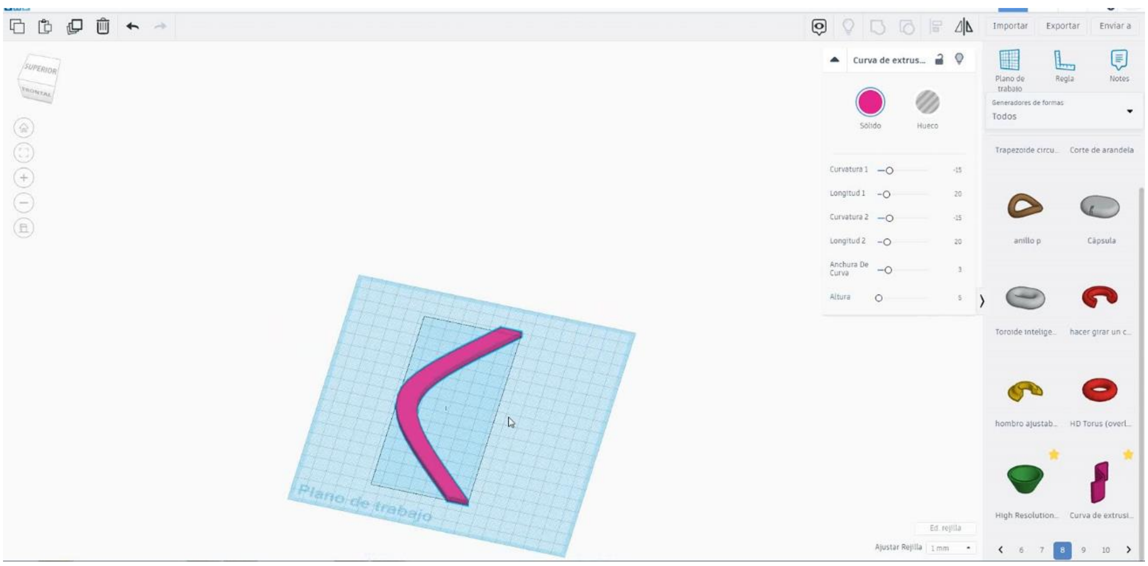
9.3.1 Part 1: Boomerang

9.3.1.1 Boomerang Design

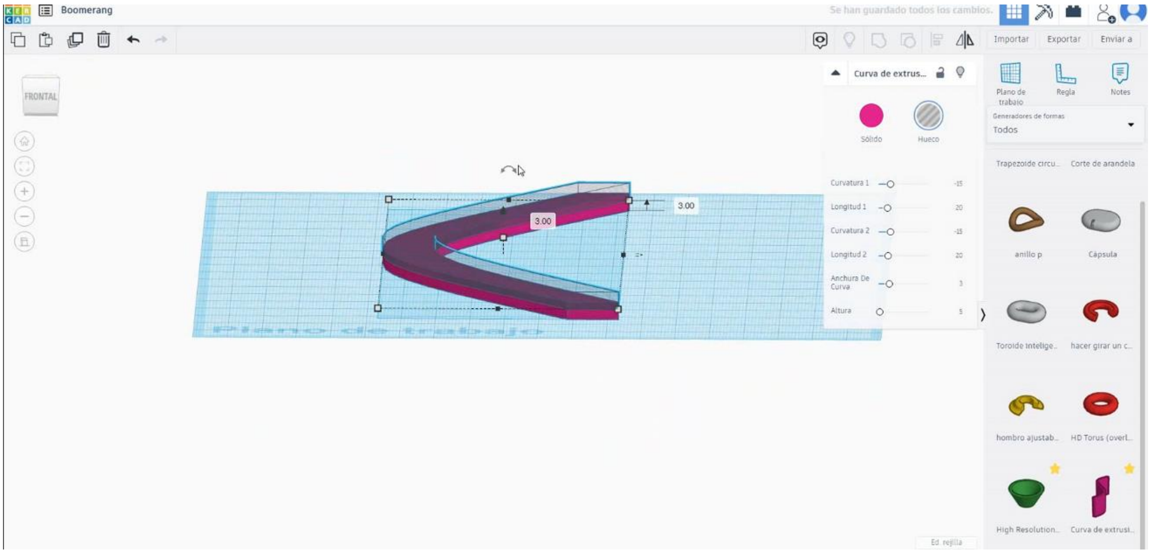
1. Choose the curved extrusion shape from the 8th page of all categories. Introduce -15 mm in curvatures 1 and 2. Introduce 20 mm in lengths 1 and 2 with a curve width of 3 mm. Size it to 150x69.8x5 mm and center it in the workplane.

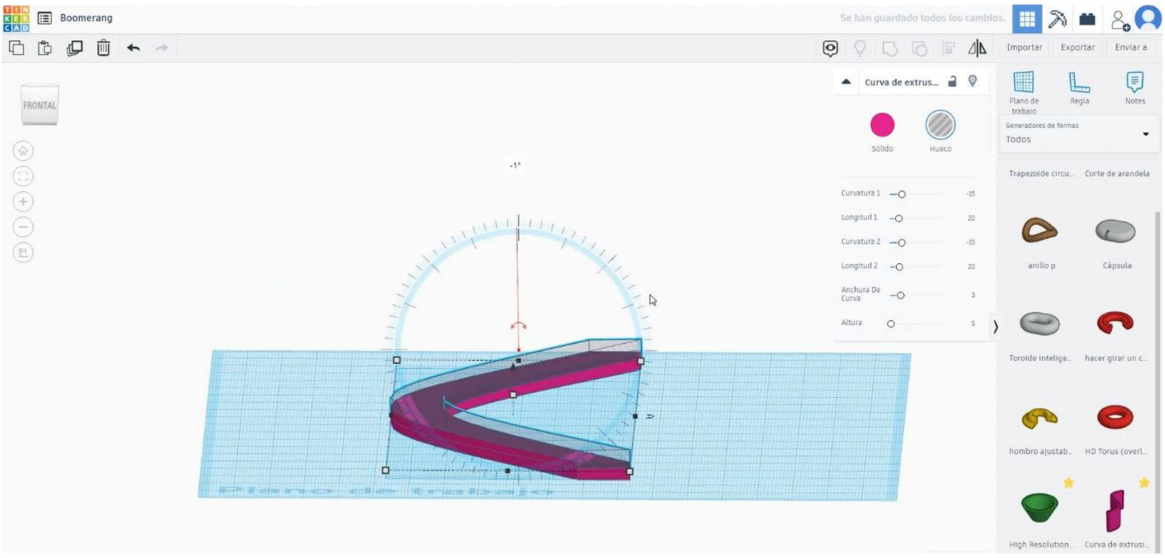

Boomerang
Importar
Exportar
Enviar a
Plano de trabajo
Regla
Notes
Todos
Trapezoide circu...
Corte de arandela
anillo p
Cápsula
Toroide intelige...
hacer girar un c...
hombro ajustab...
HD Torus (overl...
High Resolution...
Curva de extrusi...
Plano de trabajo


Boomerang
Guardando...
Importar
Exportar
Enviar a
Curva de extrus...
Sólido
Hueco
Curvatura 1
Longitud 1
Curvatura 2
Longitud 2
Anchura De Curva
Altura
69.80
150.00

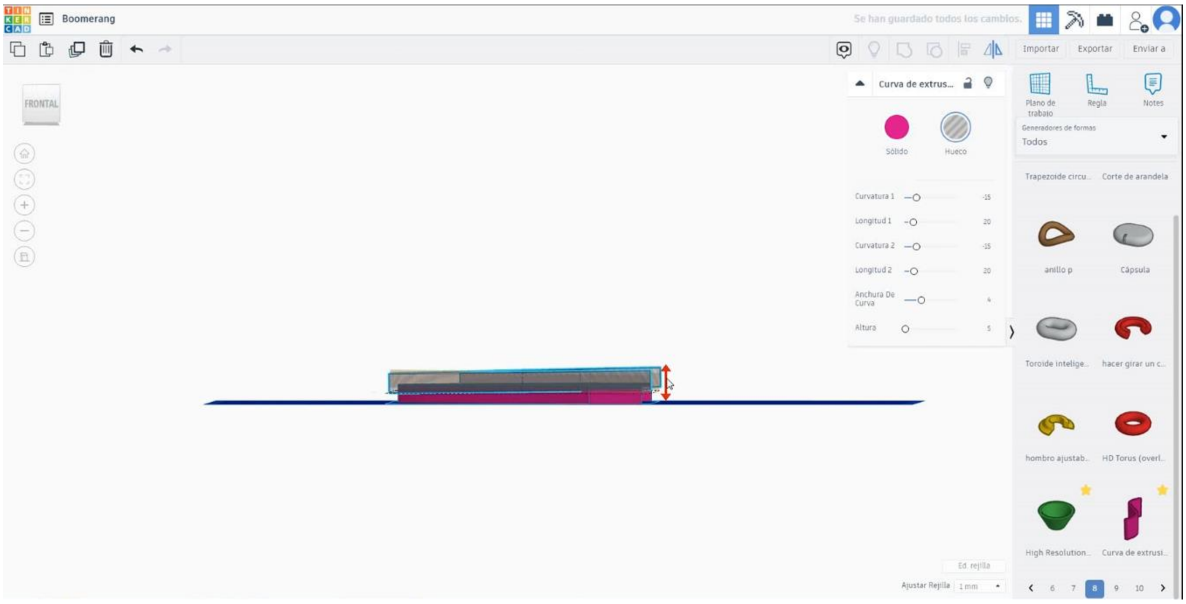

Importar
Exportar
Enviar a
Curva de extrus...
Sólido
Hueco
Curvatura 1
Longitud 1
Curvatura 2
Longitud 2
Anchura De Curva
Altura
Plano de trabajo

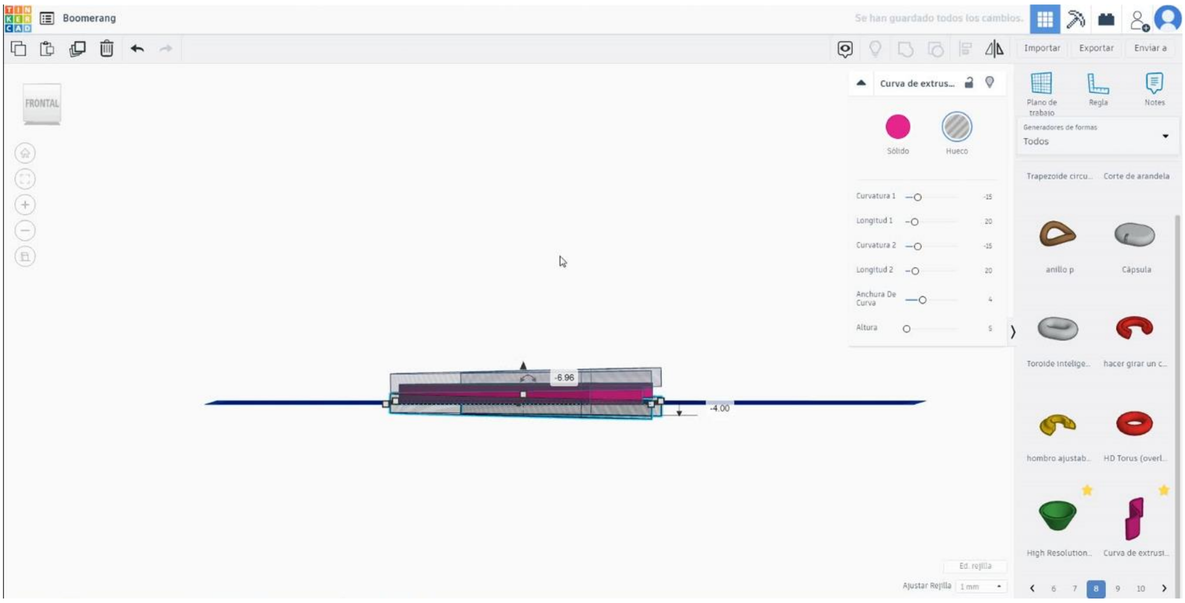
2. Duplicate the boomerang and select hole mode, move to 3 mm height and turn it -1 degree. Change the curve width to 4 mm.



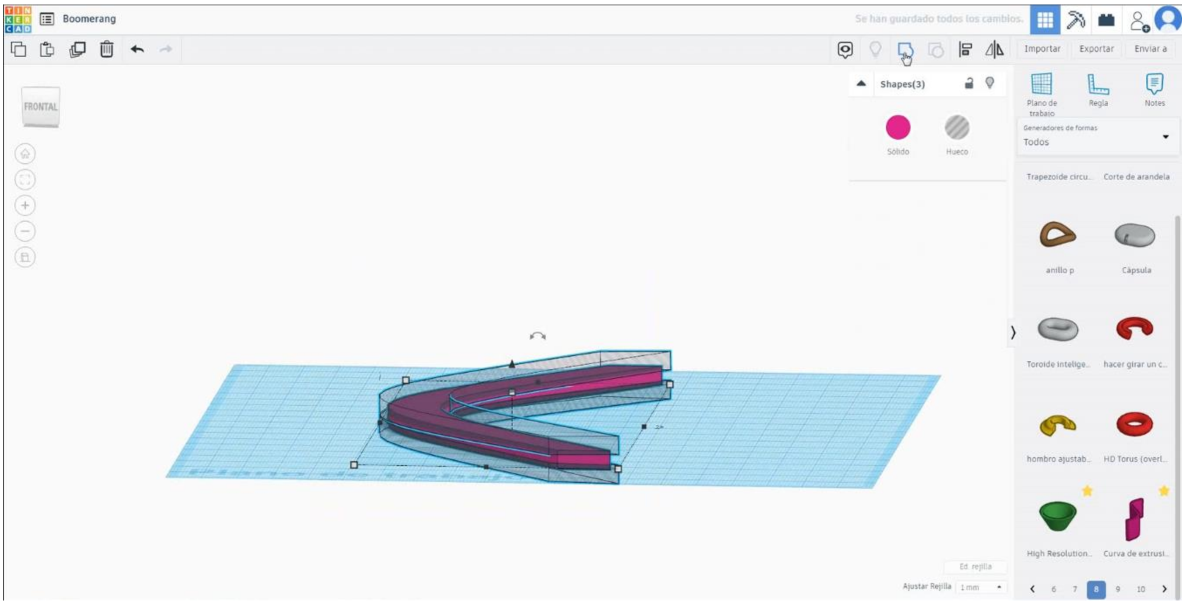


3. Duplicate the boomerang in hole mode doing asymmetry in the vertical direction. Then move it to the height of – 4 mm.

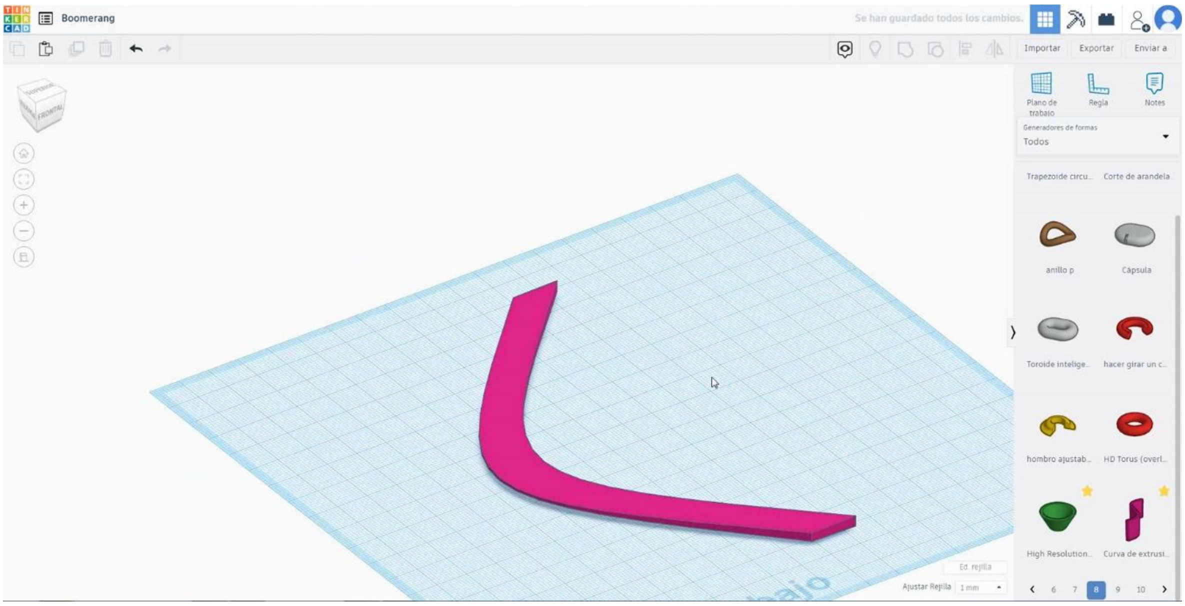




4. Select all the objects and press group.



5. Now, the boomerang is finished.



9.3.1.2 Boomerang 3D printing seetings.

<u>Filament</u>

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑ ...

Quality

Initial layer thickness (mm)	0.3
Initial layer line width (%)	115
Cut off object bottom (mm)	0.0
Dual extrusion overlap (mm)	0.15

Speed and Temperature

Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)

Speed and Temperature

Print speed (mm/s)	75
Printing temperature (C)	210
Bed temperature (C)	60

Speed

Travel speed (mm/s)	60
Bottom layer speed (mm/s)	30
Infill speed (mm/s)	0.0
Top/bottom speed (mm/s)	0.0
Outer shell speed (mm/s)	35
Inner shell speed (mm/s)	50

Support Type

None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

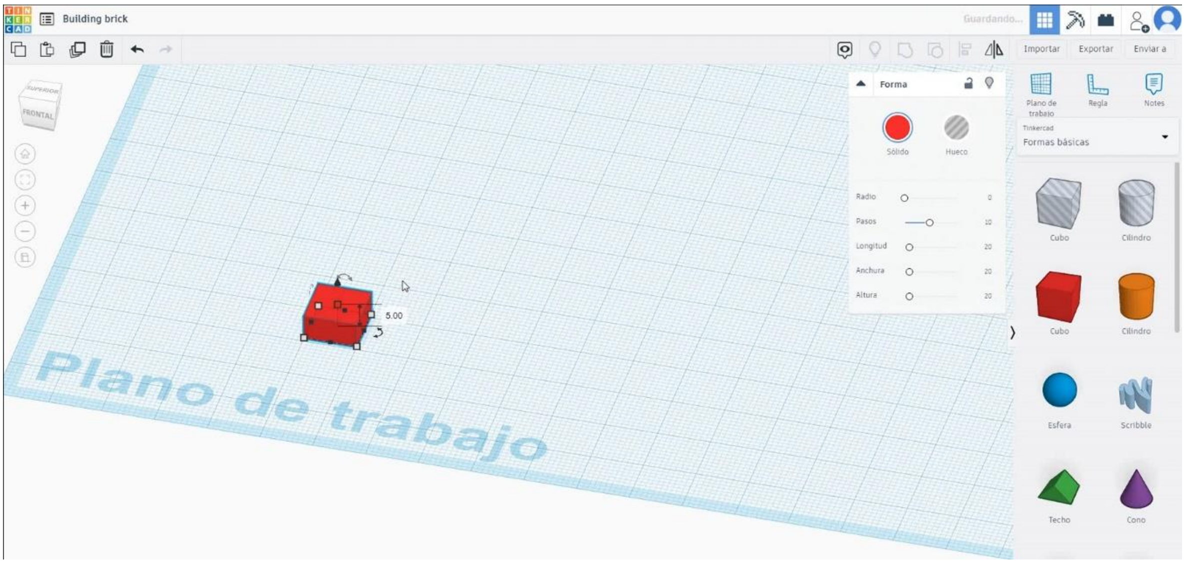
Support

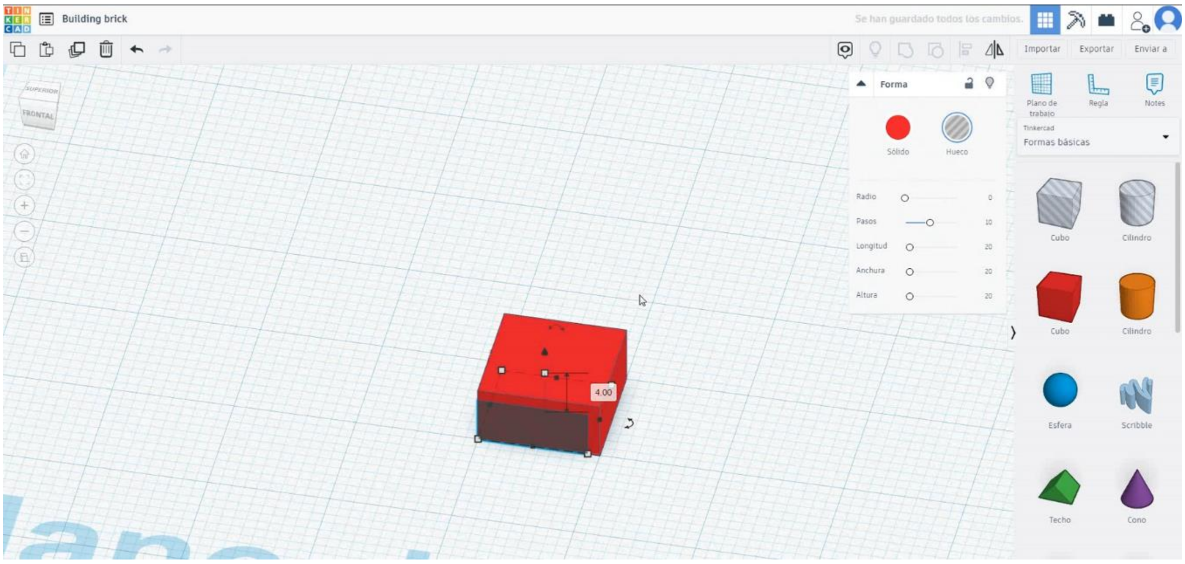
Support type	Touching buildplate ...
Platform adhesion type	None ...

9.3.2 Part 2: Building Brick

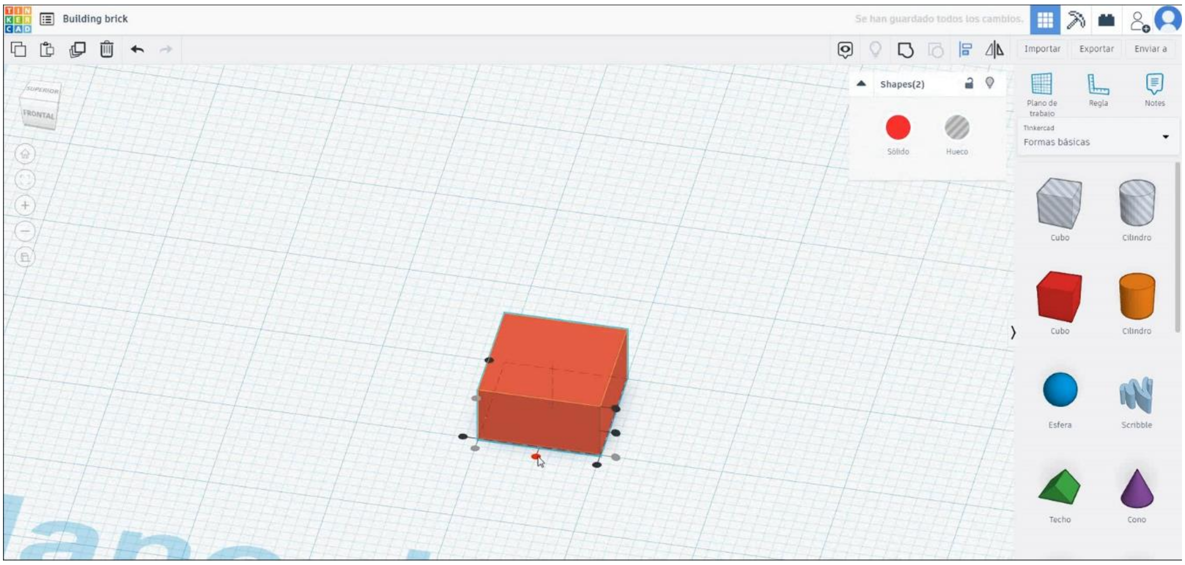
9.3.2.1 Building Brick Design

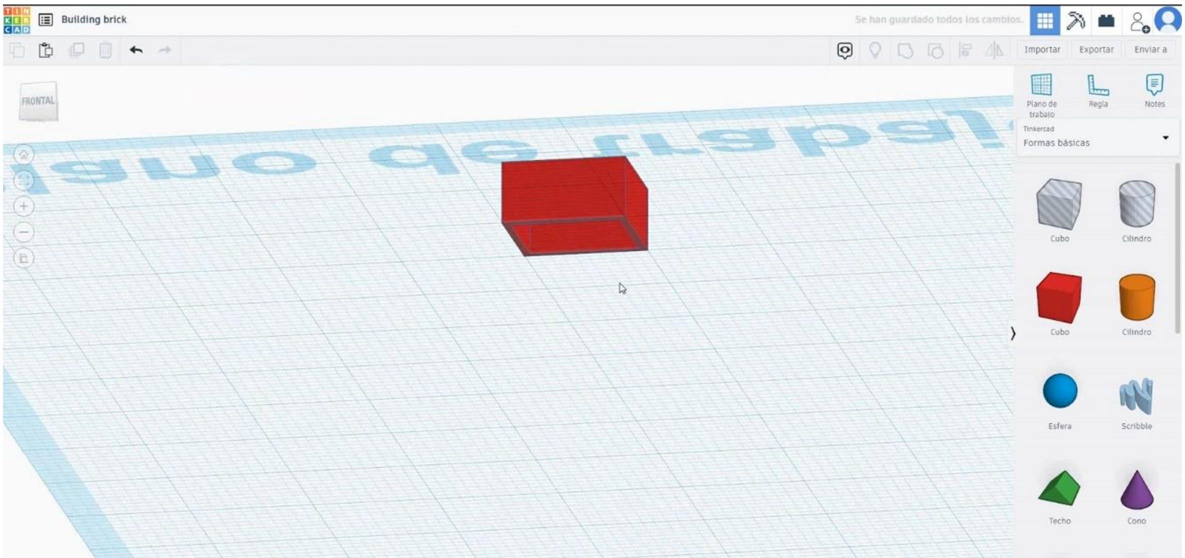
1. Choose the cube and size it to 10x10x5 mm. Duplicate it, select hole mode, and size it to 9x9x4 mm. Align in the center of the first cube, select both objects, and press group to empty the core of the cube.



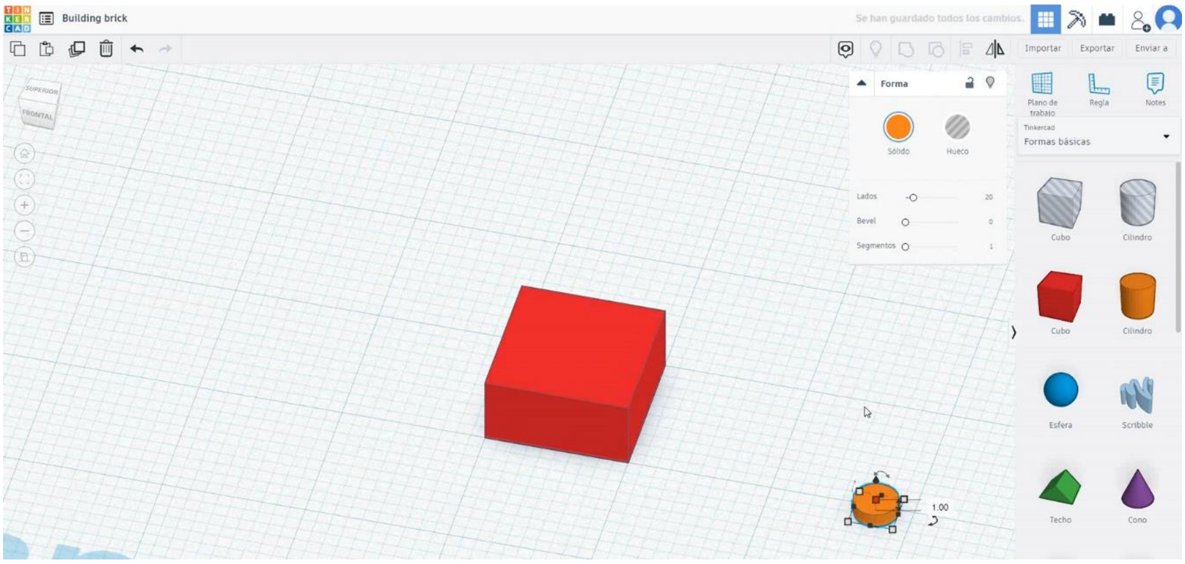


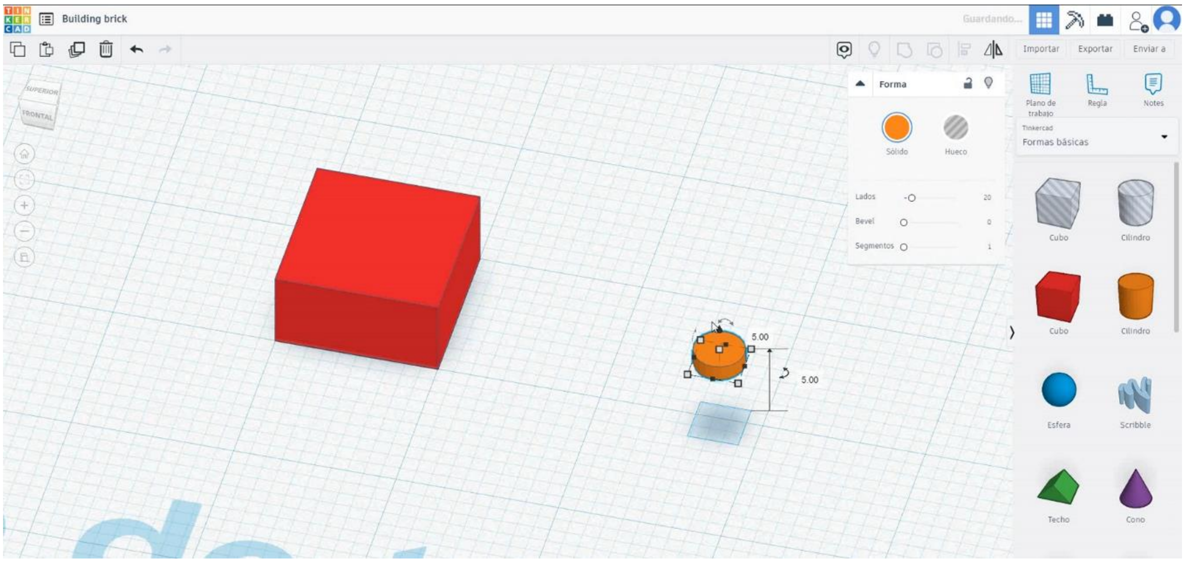
Co-funded by the Erasmus+ Programme of the European Union

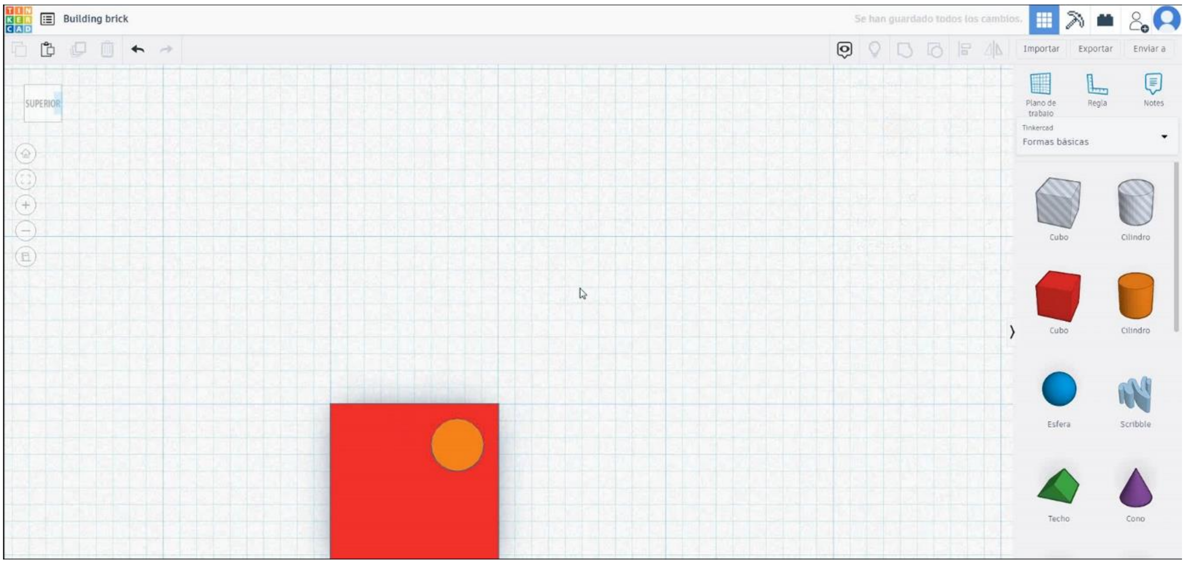


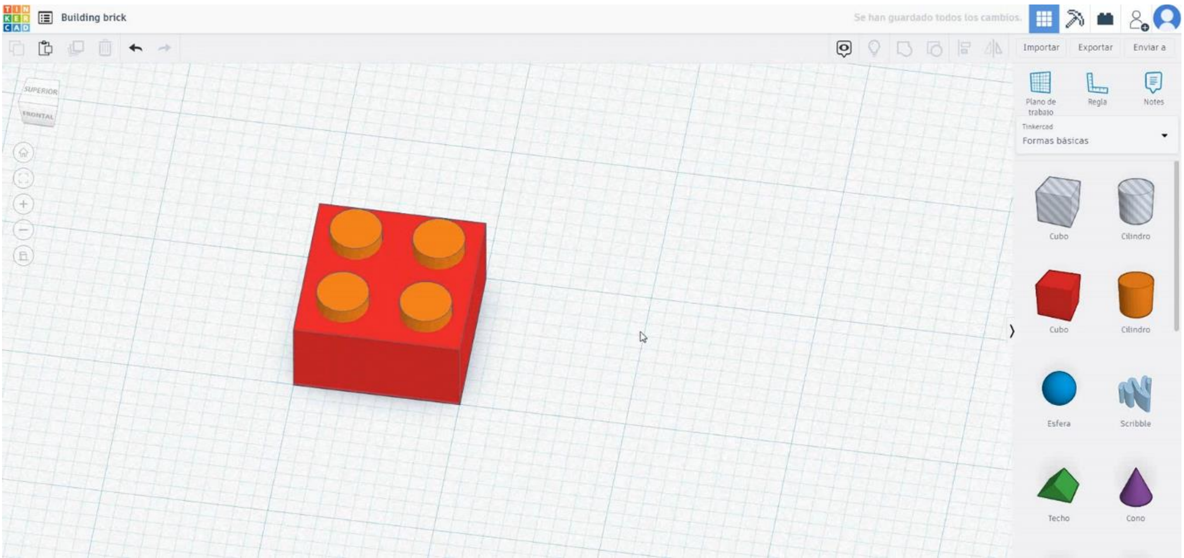


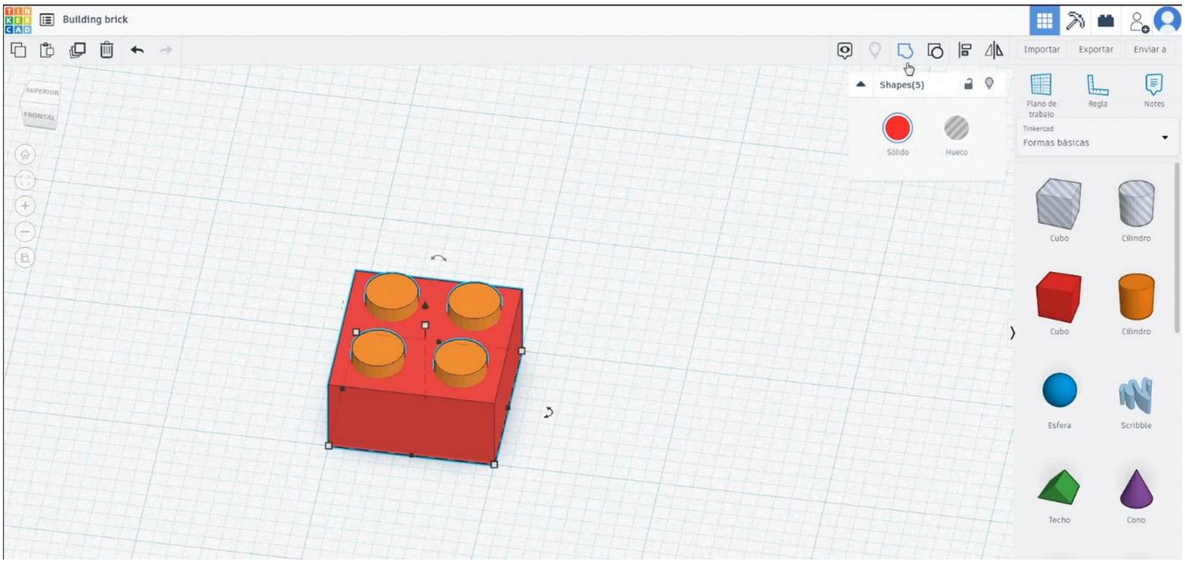
2. Select the cylinder shape, size it to 3.12x3.12x1 mm, and move it to the height of 5 mm. Move it to make the center of the cylinder match with one corner of the cube, then move 2x2 mm to the center. Copy this cylinder three times moving 5 mm in each direction. Select both objects and press group.


Building brick
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
1.00

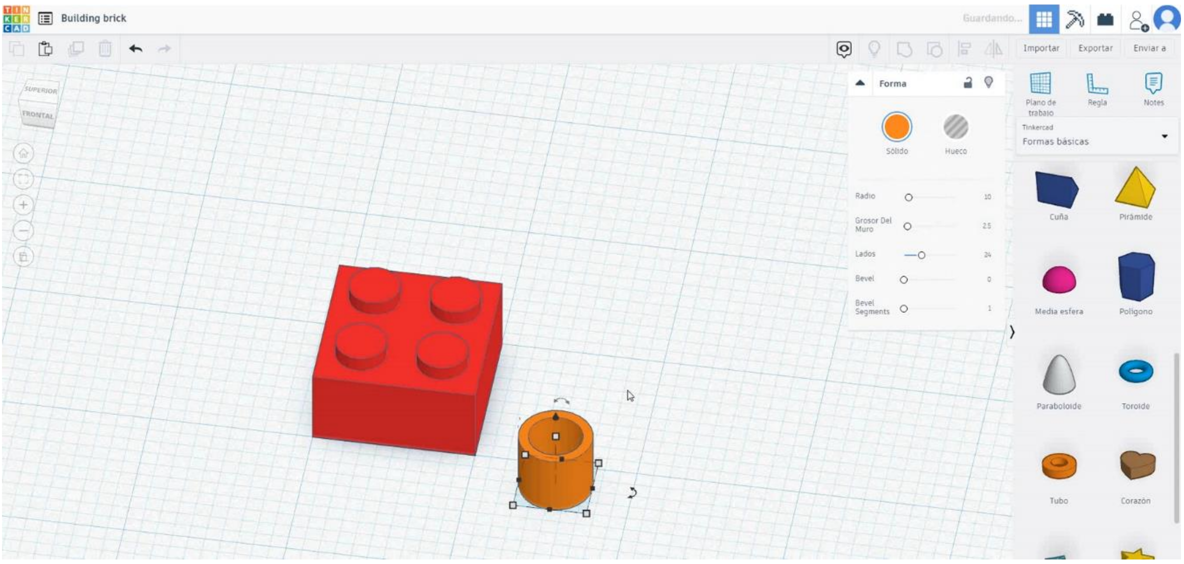

Building brick
Guardando...
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
5.00
5.00

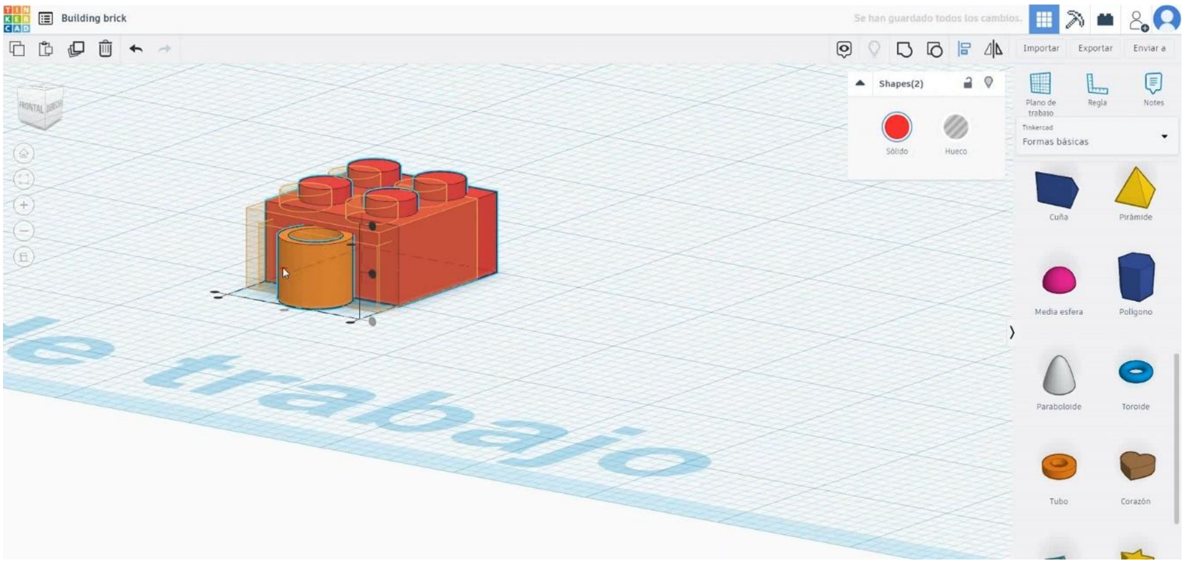

Building brick
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
SUPERIOR
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono

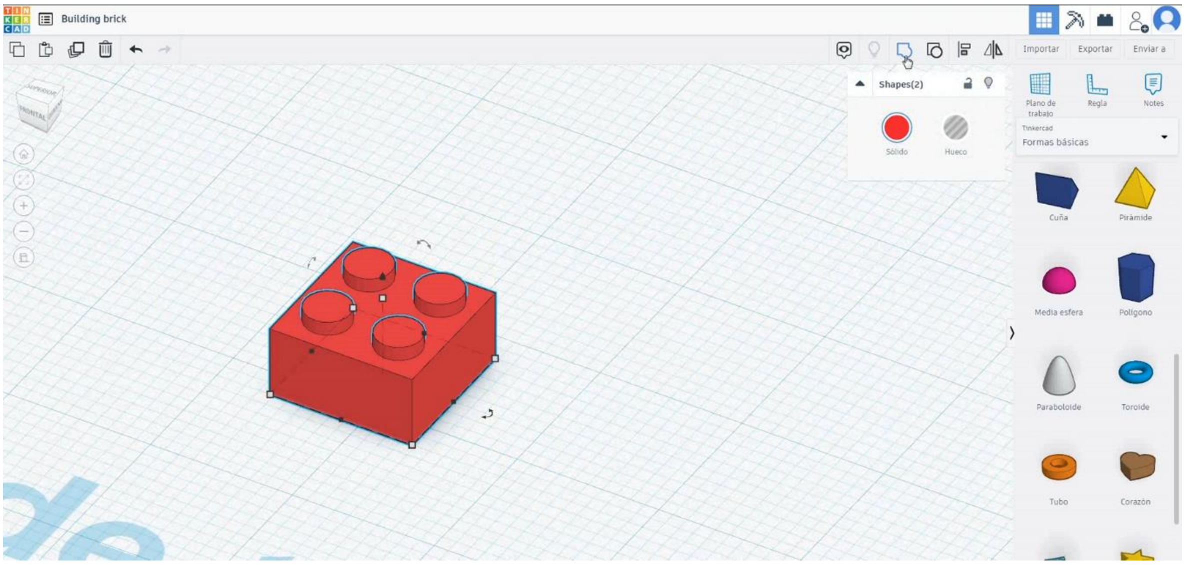




3. Select the tube shape, size it to 4.53x4.53x4 mm, and align it in the center of the cube. Select both objects and press group.

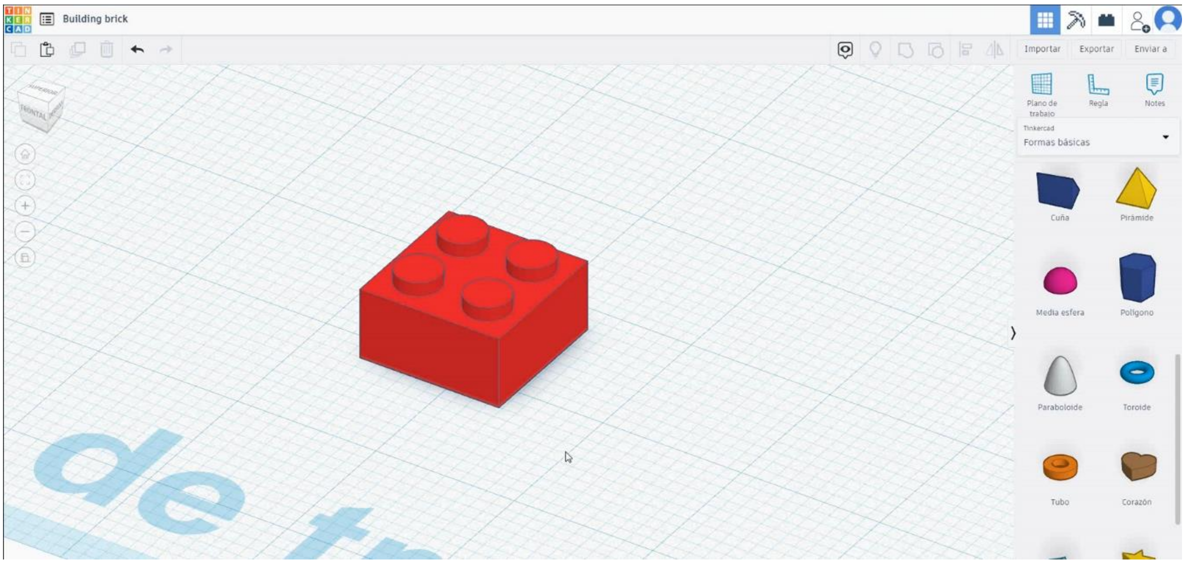

Building brick
Guardando...
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Radio
Grosor Del Muro
Lados
Bevel
Bevel Segments
Plano de trabajo
Regla
Notas
Formas básicas
Cuña
Pirámide
Media esfera
Polígono
Paraboloide
Toroide
Tubo
Corazón

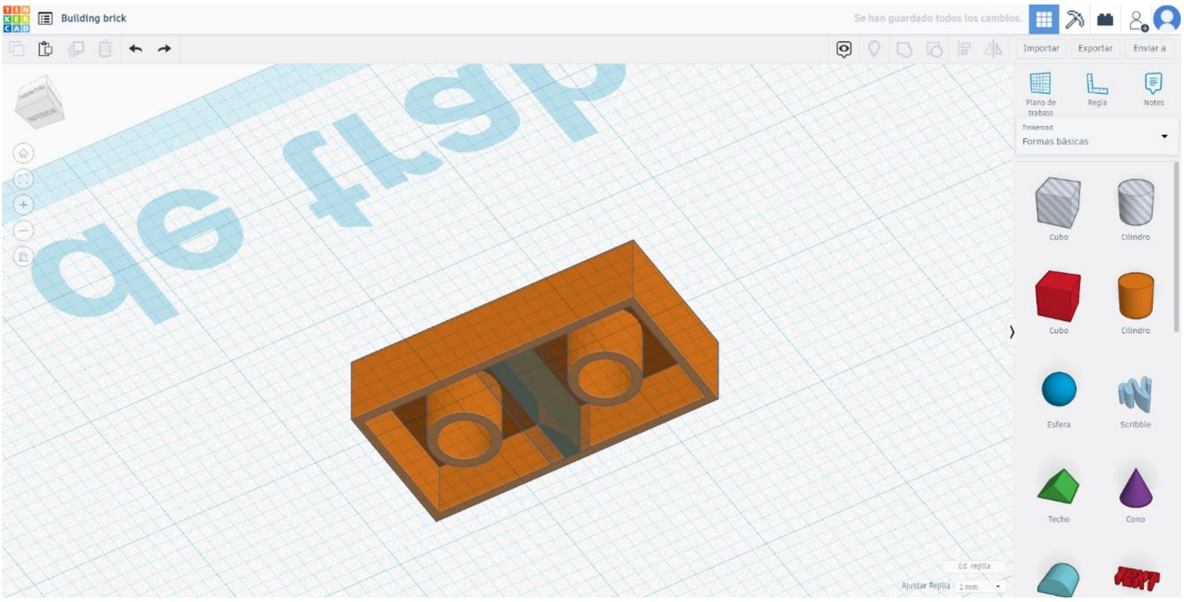

Building brick
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Shapes(2)
Sólido
Hueco
Plano de trabajo
Regla
Notas
Formas básicas
Cuña
Pirámide
Media esfera
Polígono
Paraboloide
Toroide
Tubo
Corazón

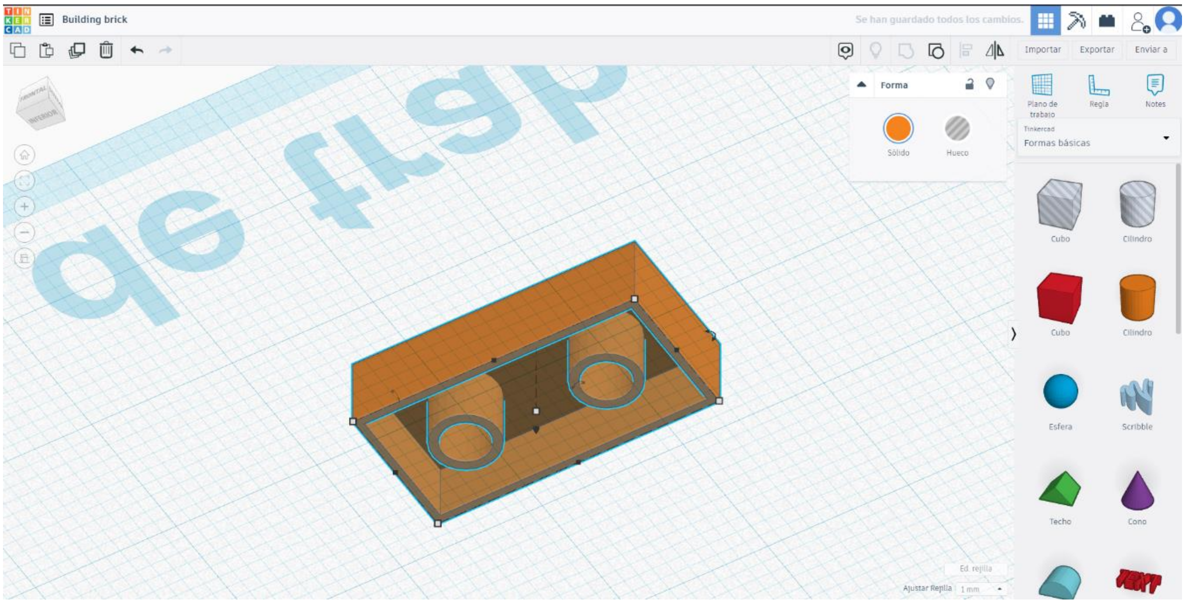

Building brick
Importar
Exportar
Enviar a
Shapes(2)
Sólido
Hueco
Plano de trabajo
Regla
Notas
Formas básicas
Cuña
Pirámide
Media esfera
Polígono
Paraboloide
Toroide
Tubo
Corazón

Co-funded by the Erasmus+ Programme of the European Union

4. Now, the building brick is finished. This shape can be scaled to make it bigger or you can duplicate it to make the double its size, but erasing the central wall like the images below.







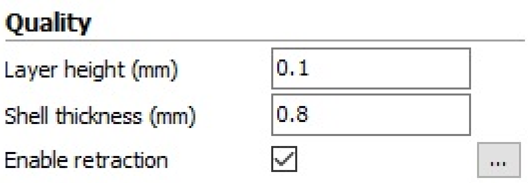
9.3.2.2 Building Brick 3D printing seetings.

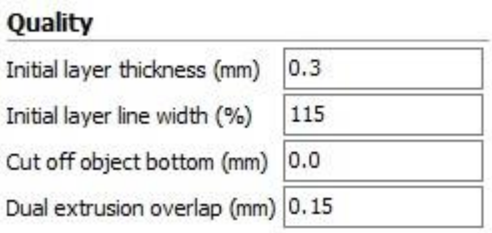
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

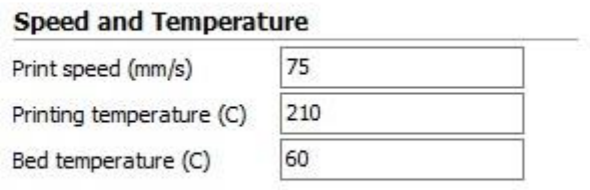
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

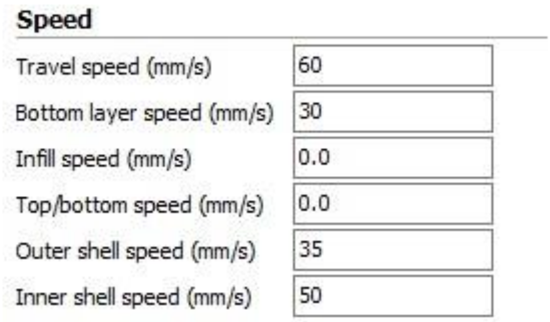




Speed and Temperature

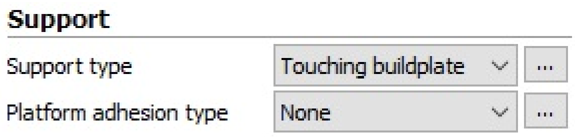
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

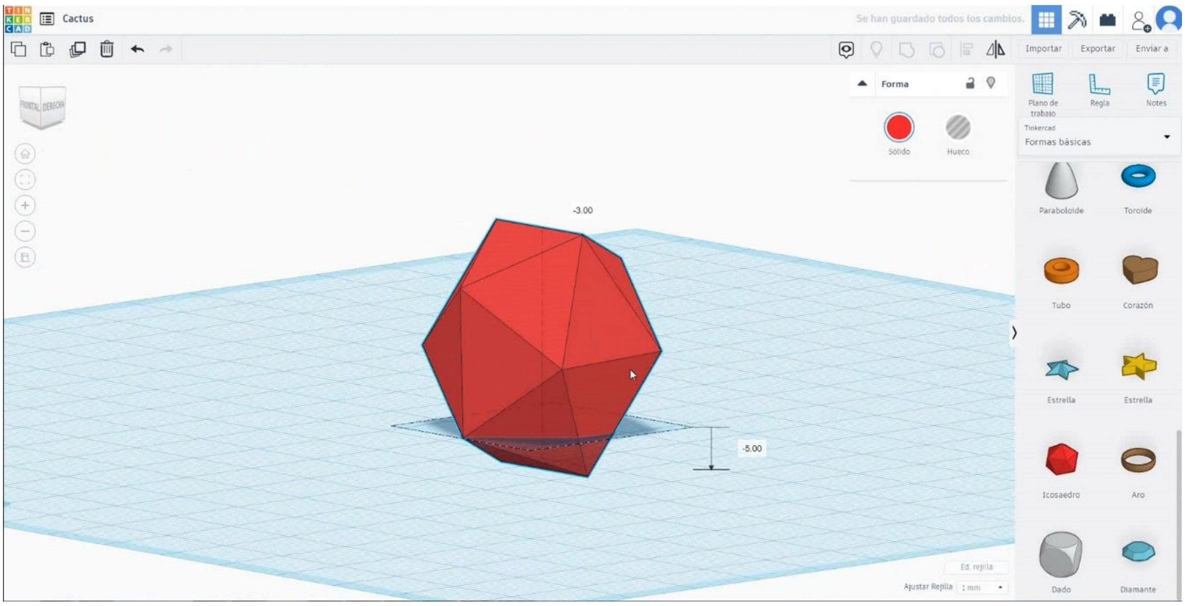
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



9.3.3 Part 3: Cactus

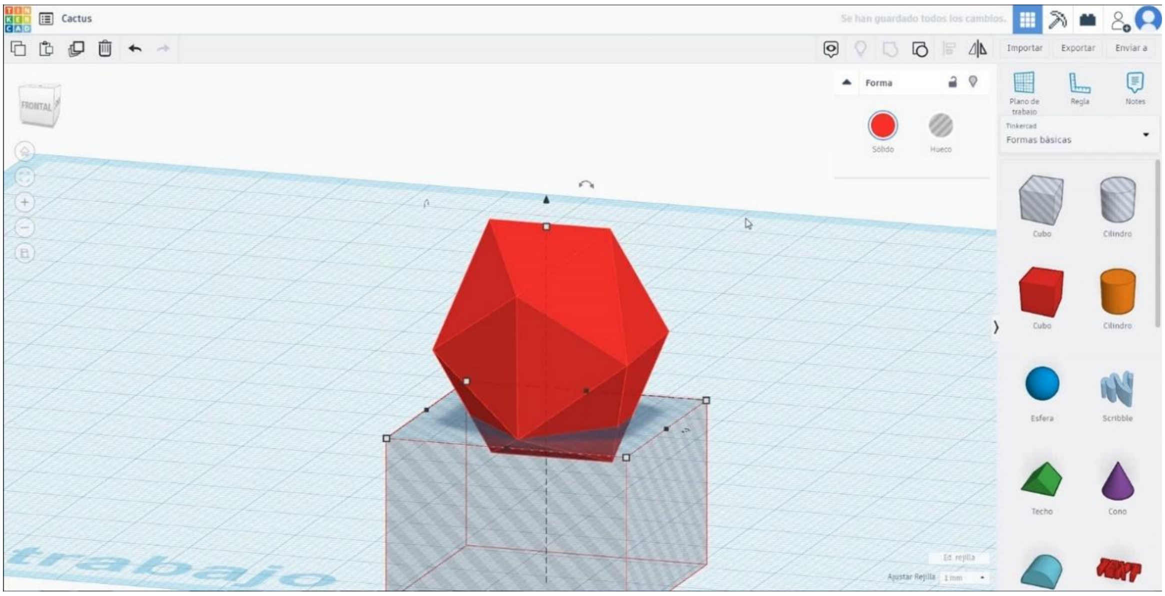
9.3.3.1 Cactus Design

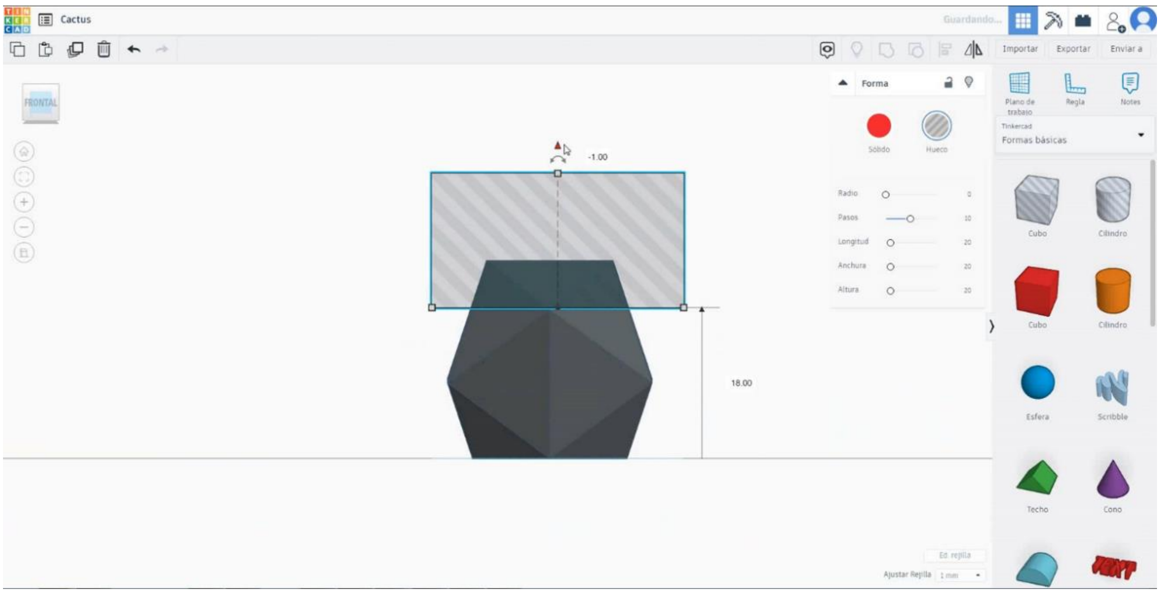
1. Choose the icosahedron shape, size it to 24.27x25.51x28.53 mm, and move it to Z 0 mm.

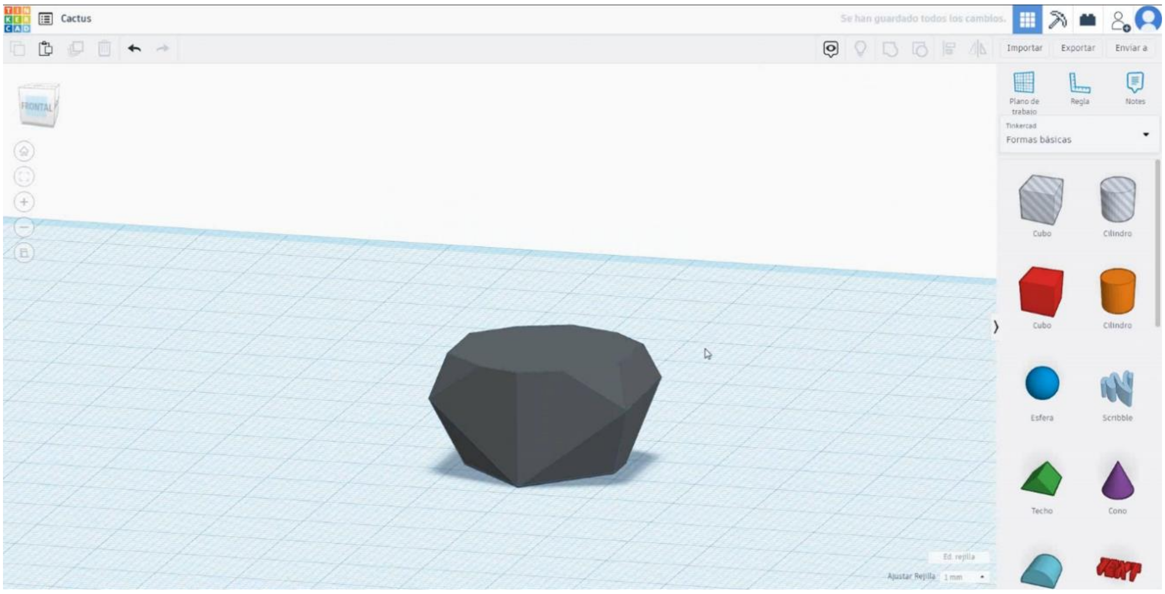


Co-funded by the Erasmus+ Programme of the European Union

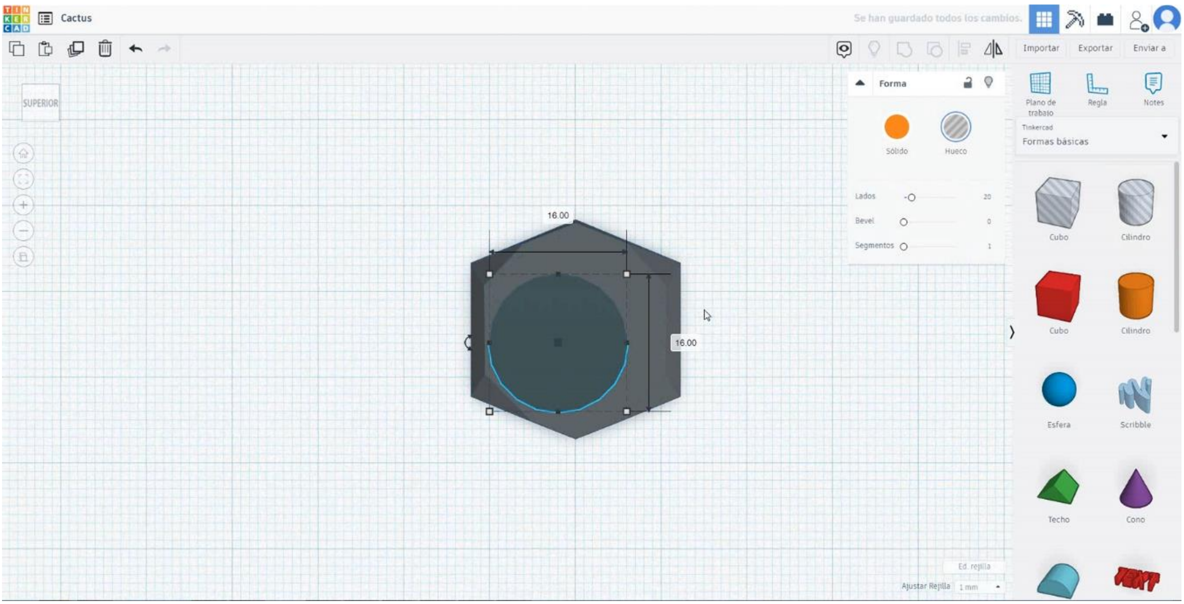
2. Choose the cube shape in hole mode, size it to 30x30x20 mm and move it to a height of -20 mm and align it in the center. Select both objects and press group. Then choose a cube shape in hole mode, size it to 30x30x16 mm and move it to a height of 16 mm and align it in the center. Choose both objects and press group.

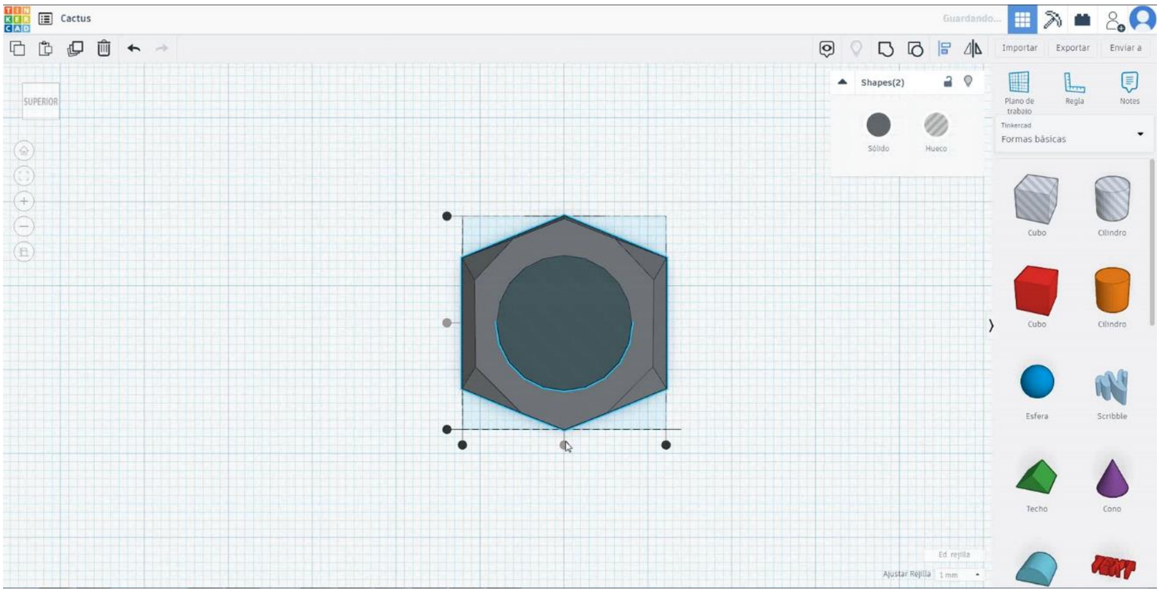


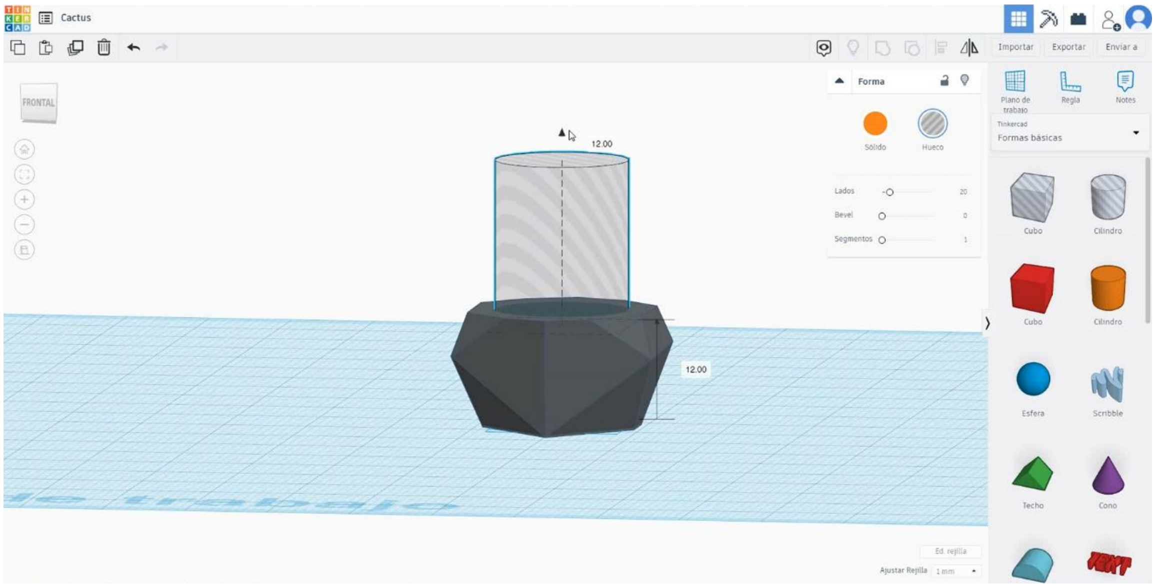


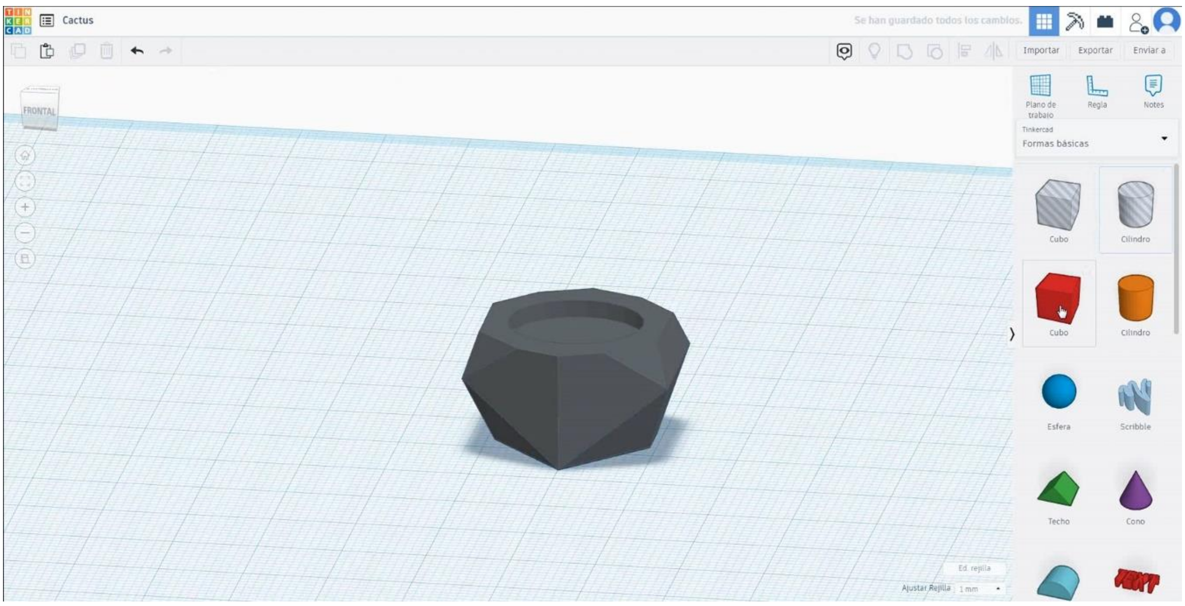


3. Choose the cylinder shape in hole mode and size it to 16x16x20 mm and move it to the height of 12 mm. Then align in the center. Select both objects and press group.



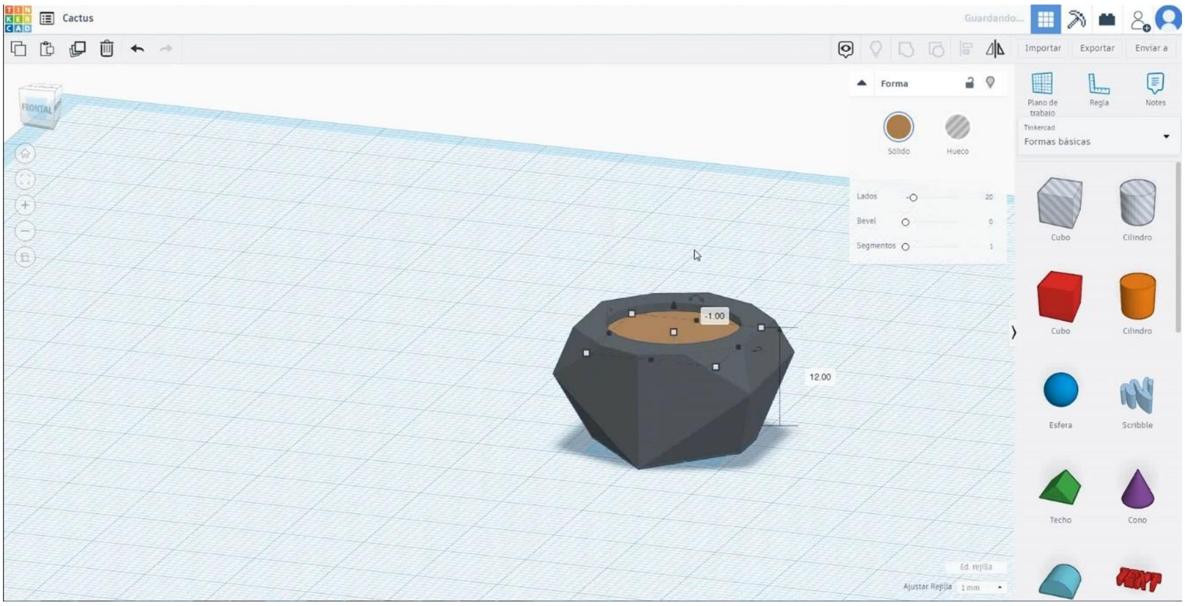

Cactus
Importar
Exportar
Enviar a
Shapes(2)
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono

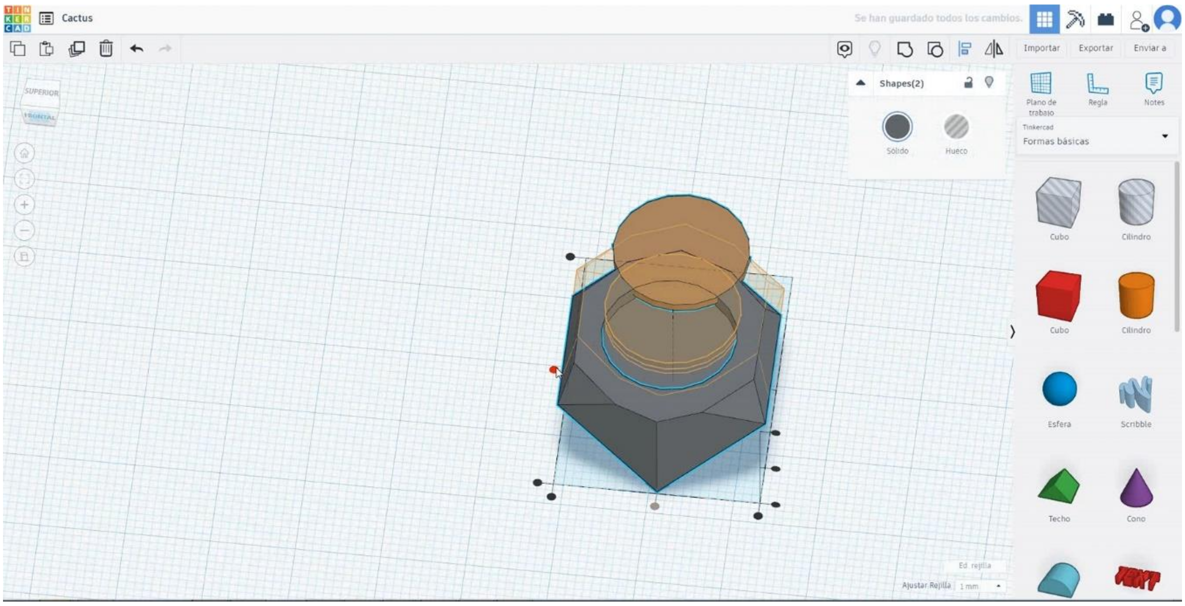

Cactus
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
12.00
12.00
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono


Cactus
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono

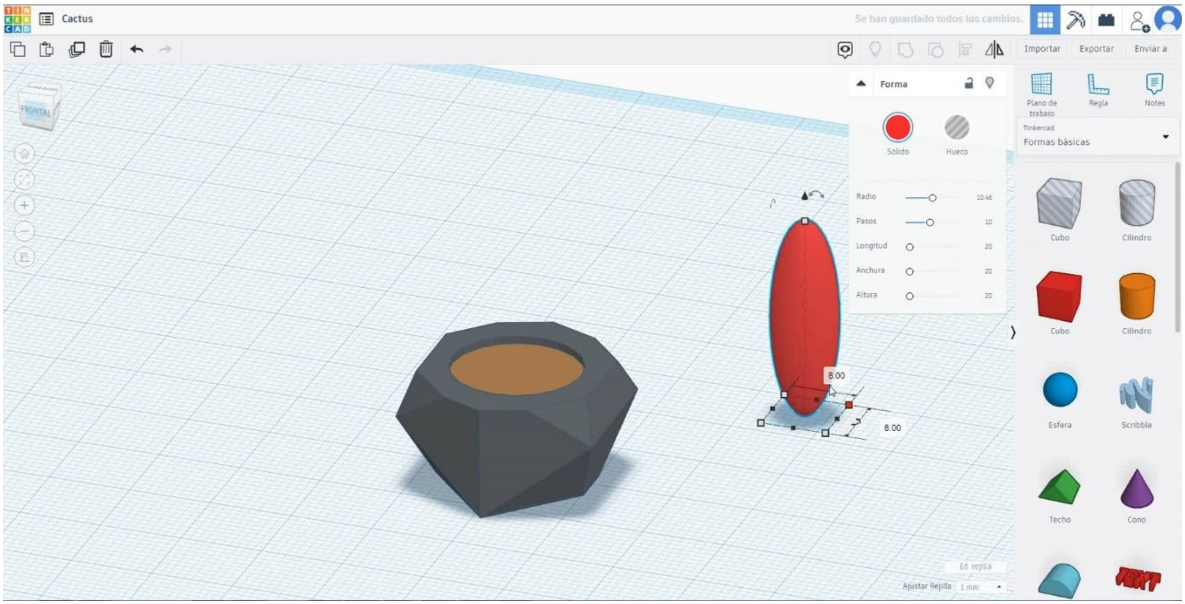
Co-funded by the Erasmus+ Programme of the European Union

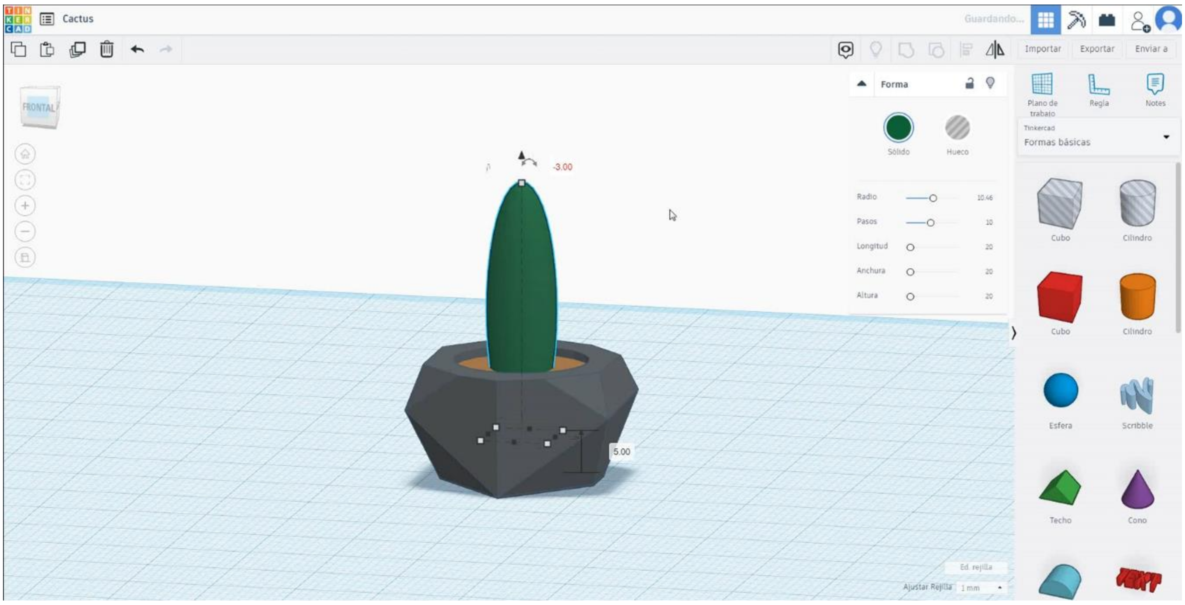
4. Choose cylinder shape and size it to 16x16x1 mm and move it to the height of 12 mm. Then align in the center of the cube.



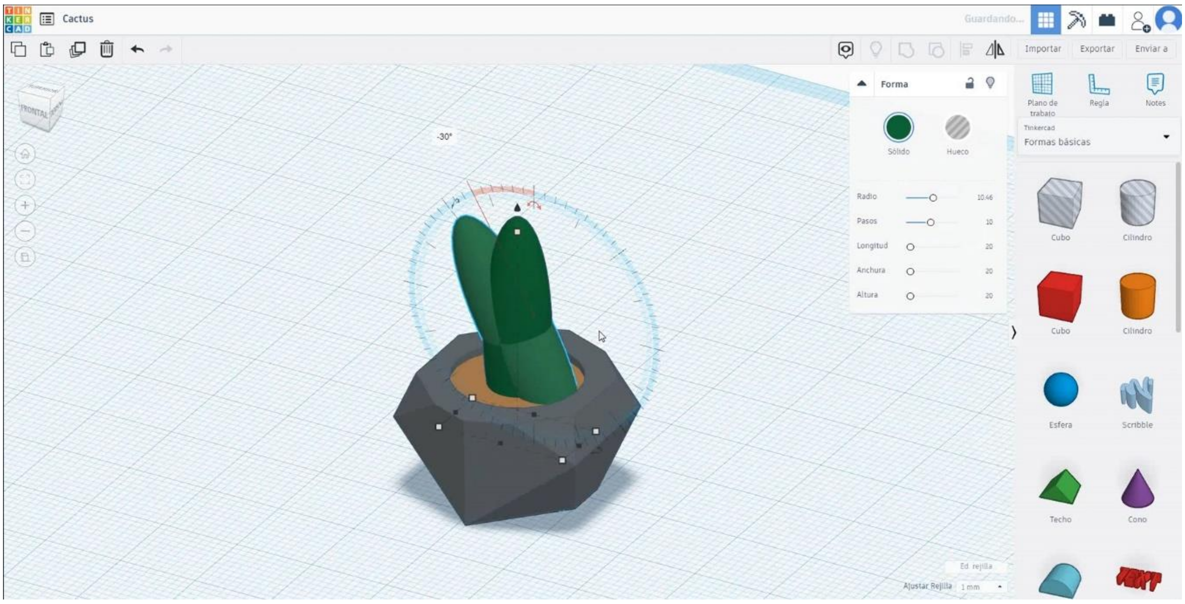


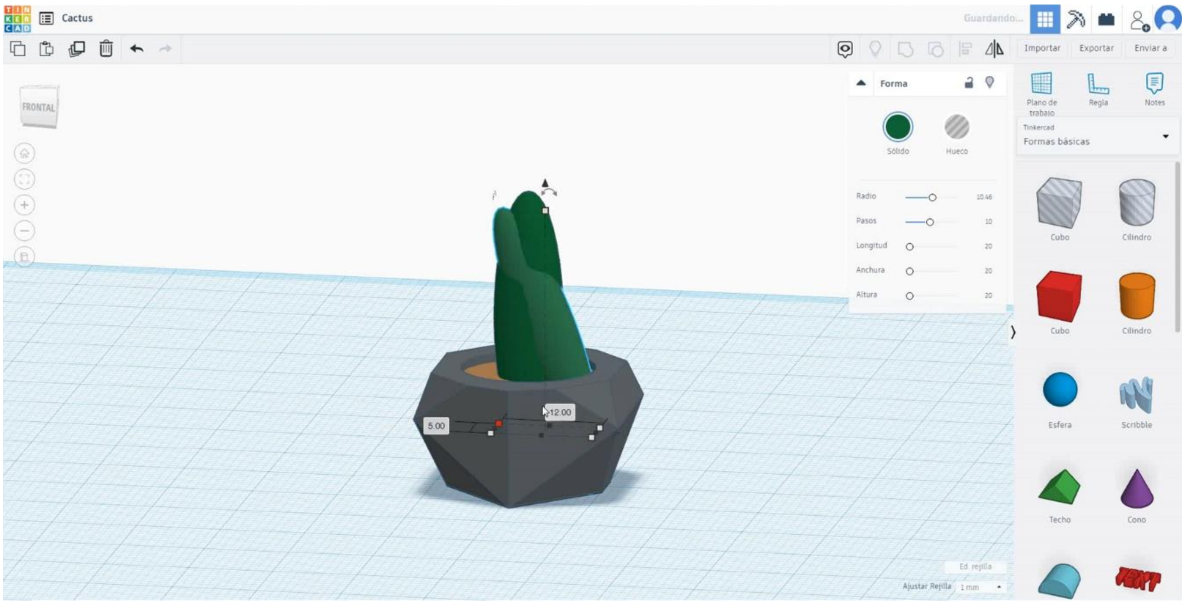
5. Select the shape cube, and size it to 8x8x30 mm with 10 mm radius. Move it to the height of 5 mm in height and alit ign in the center.

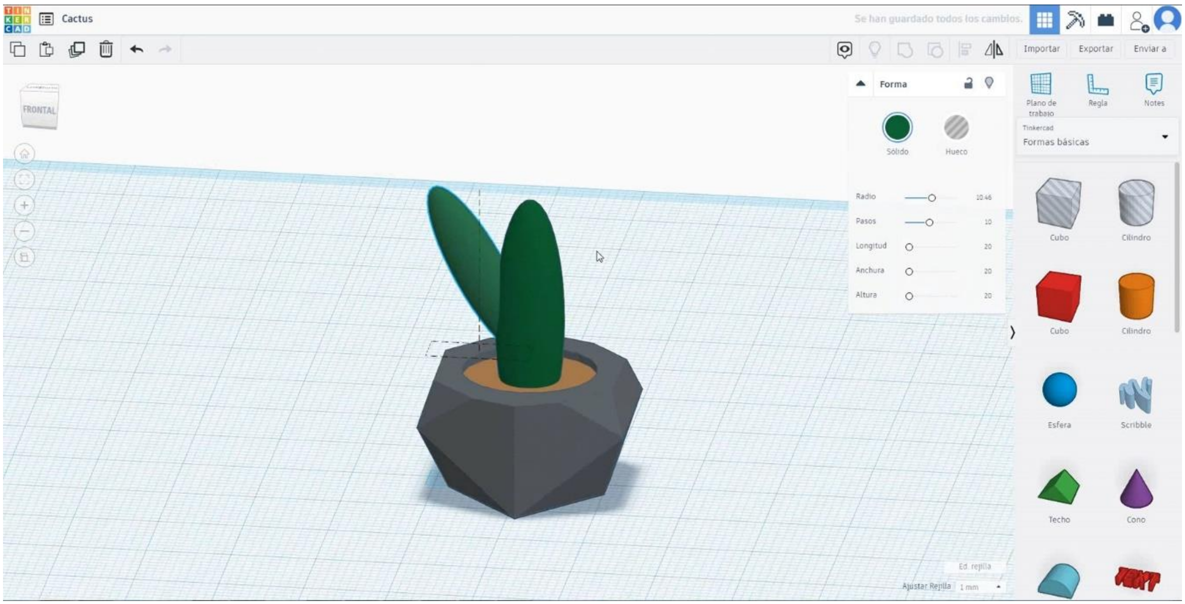




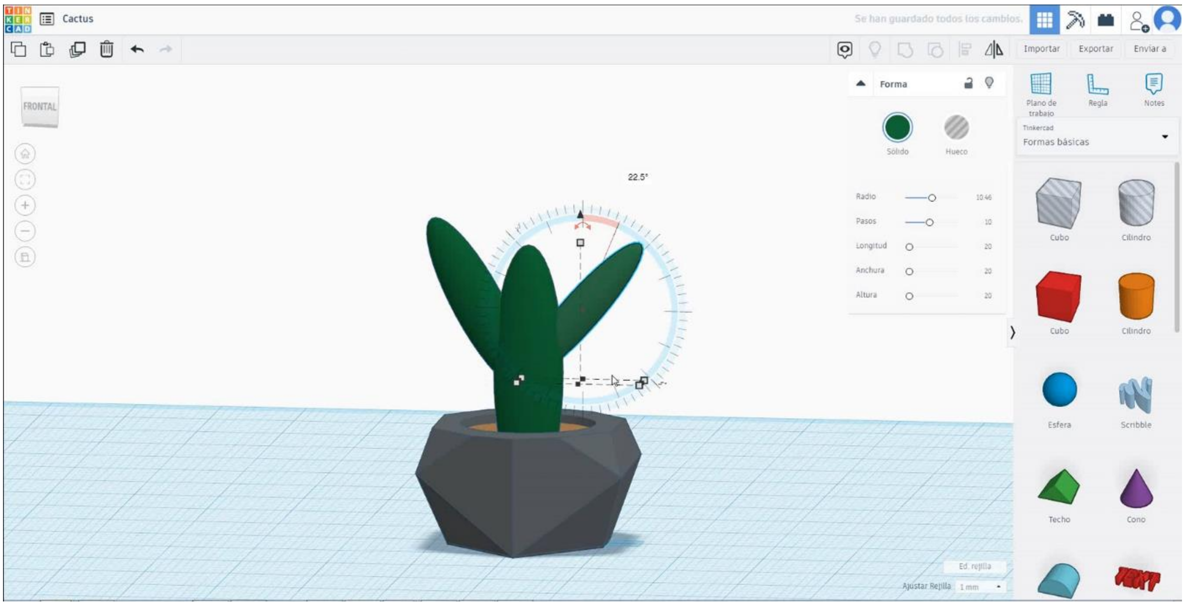
6. Select the cube and duplicate it turning -30 degrees. Size it to 5x12x20 mm and move it to the height of 18 mm aligning it with the first cube.

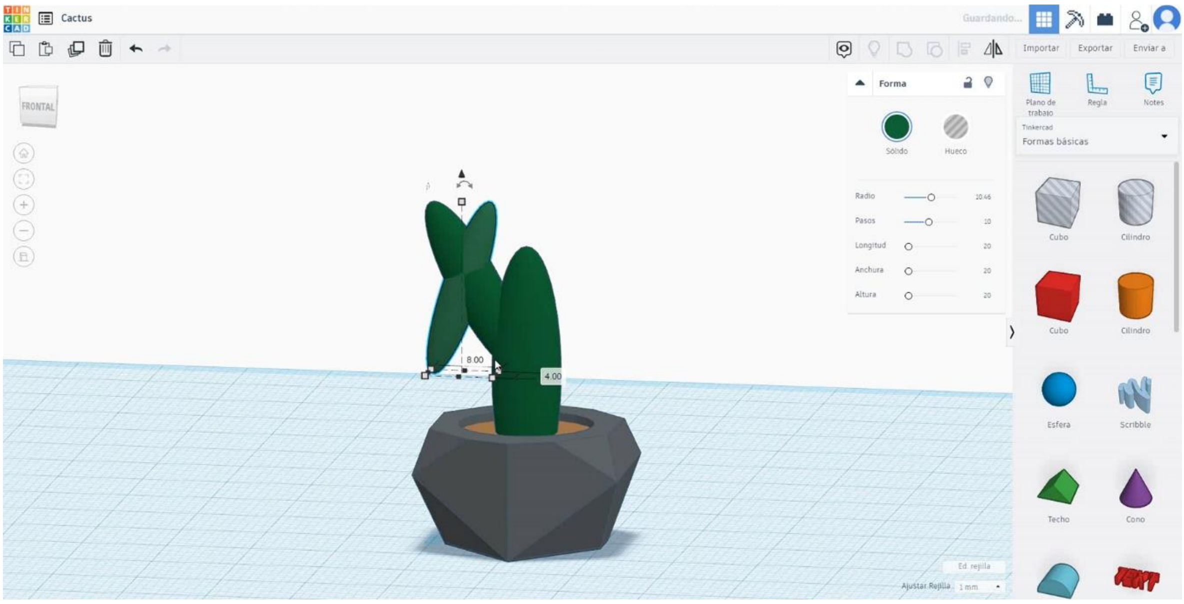

Cactus
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Radio
Pasos
Longitud
Anchura
Altura
Plano de trabajo
Regla
Notas
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
-30°
Ajustar Rejilla
1 mm

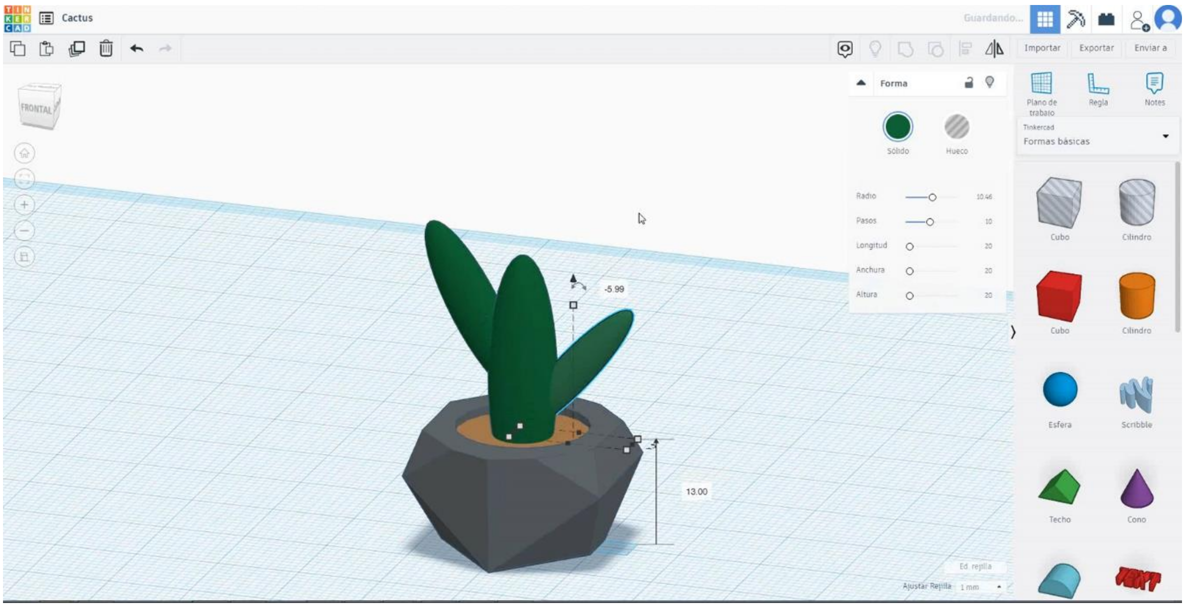

Cactus
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Radio
Pasos
Longitud
Anchura
Altura
Plano de trabajo
Regla
Notas
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
5.00
12.00
Ajustar Rejilla
1 mm



7. Duplicate the second cube turning it 22.5 degrees, size it to 8x4x20 mm, and move it to the height of 13 mm. Move to the opposite side and align with the first cube.

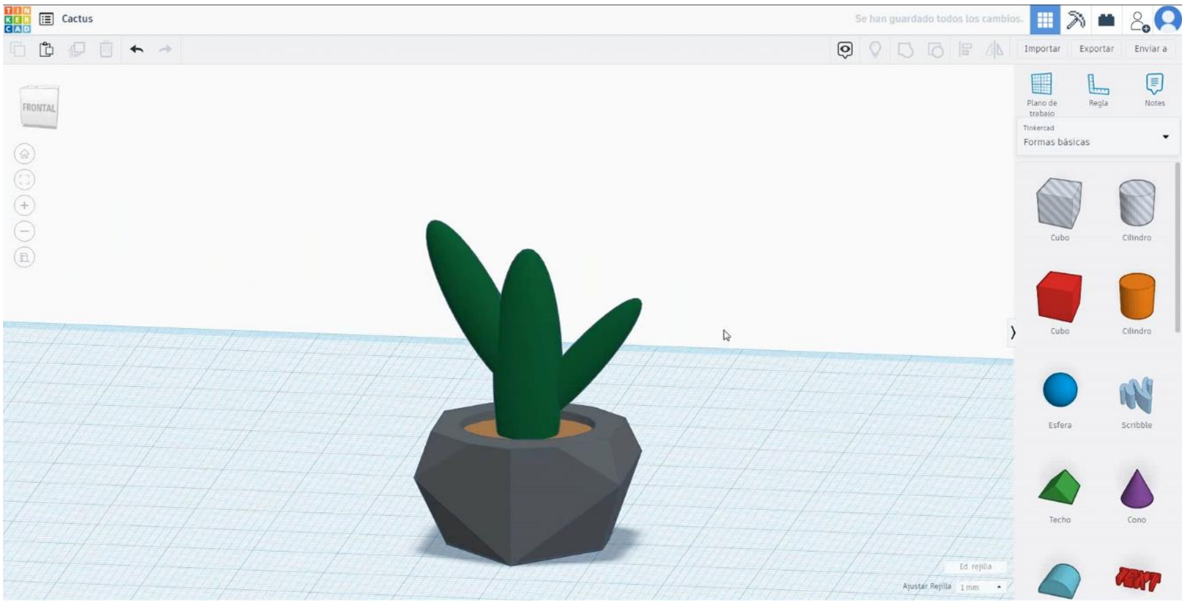






8. Now, the cactus is finished.

Co-funded by the Erasmus+ Programme of the European Union



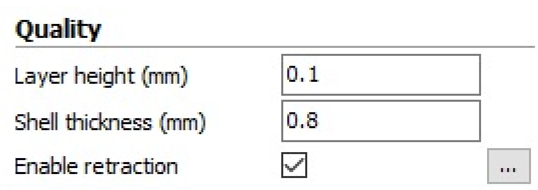
9.3.3.2 Cactus 3D printing seetings.

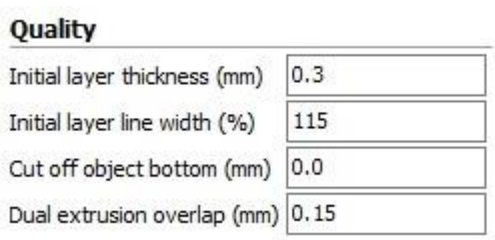
<u>Filament</u>

PLA
Diameter - 1.75 (mm)
Flow - 100%

<u>Quality</u>

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

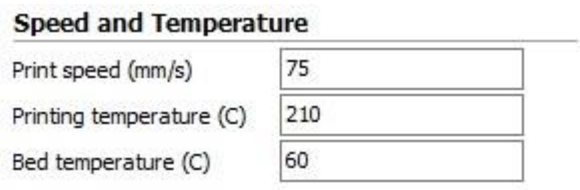


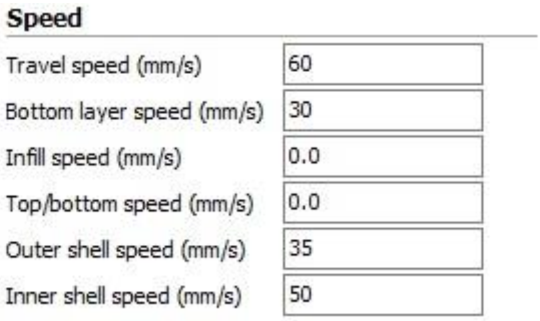


<u>Speed and Temperature</u>

Print Speed - 50 (mm/s)

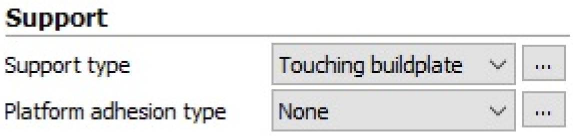
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





<u>Support Type</u>

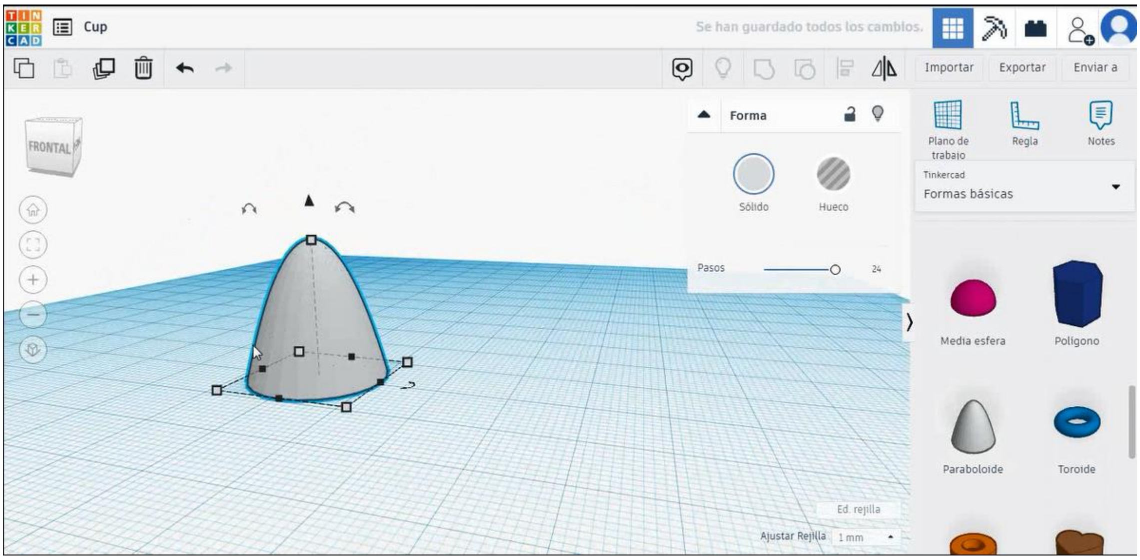
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



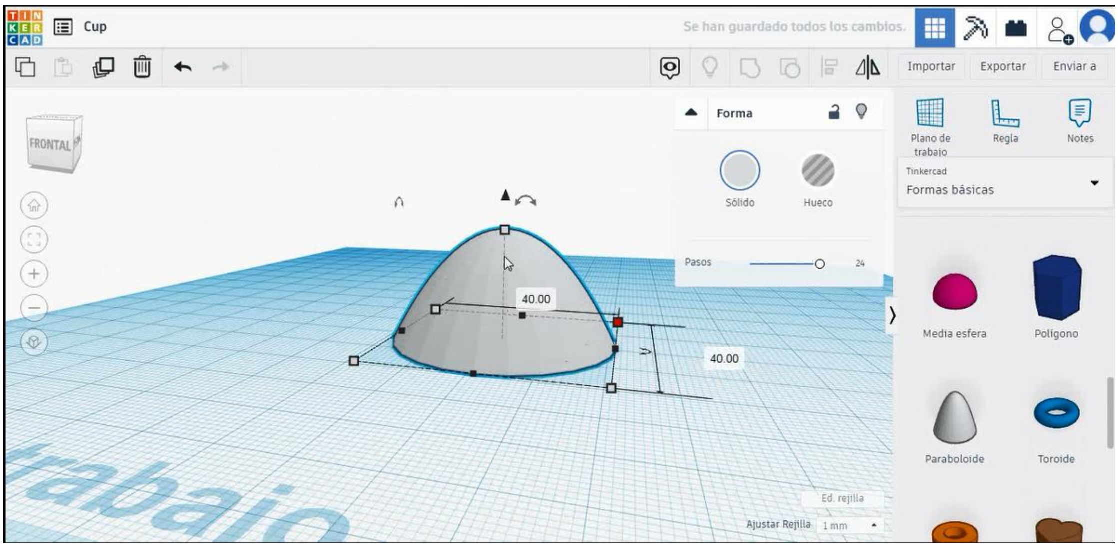
9.3.4 Part 4: Cup

9.3.4.1 Cup Design

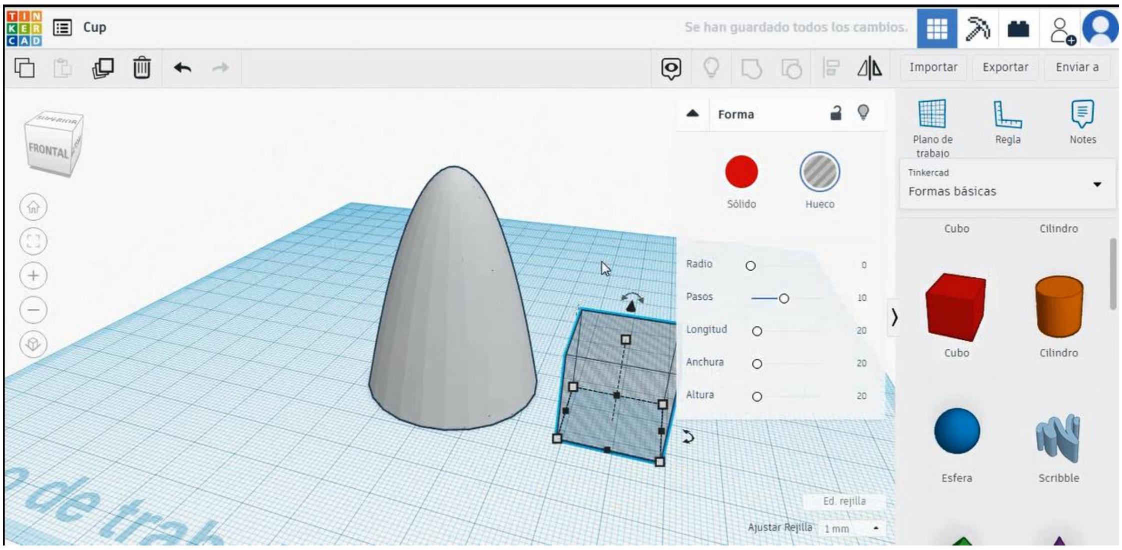
1. Choose our base shape: Paraboloid



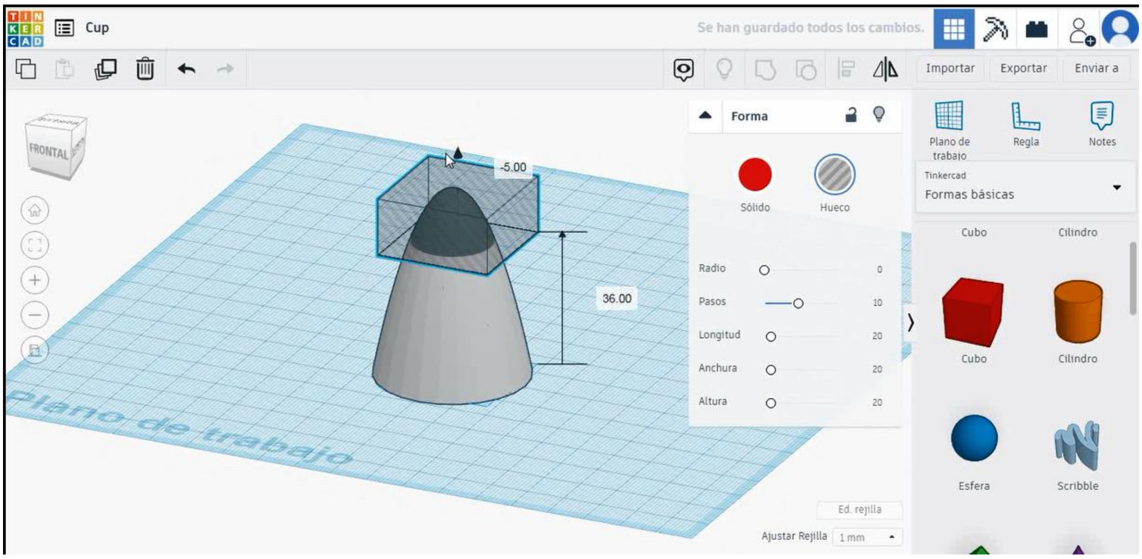
2. Sixe it to 40x40x50 mm (x,y,z)



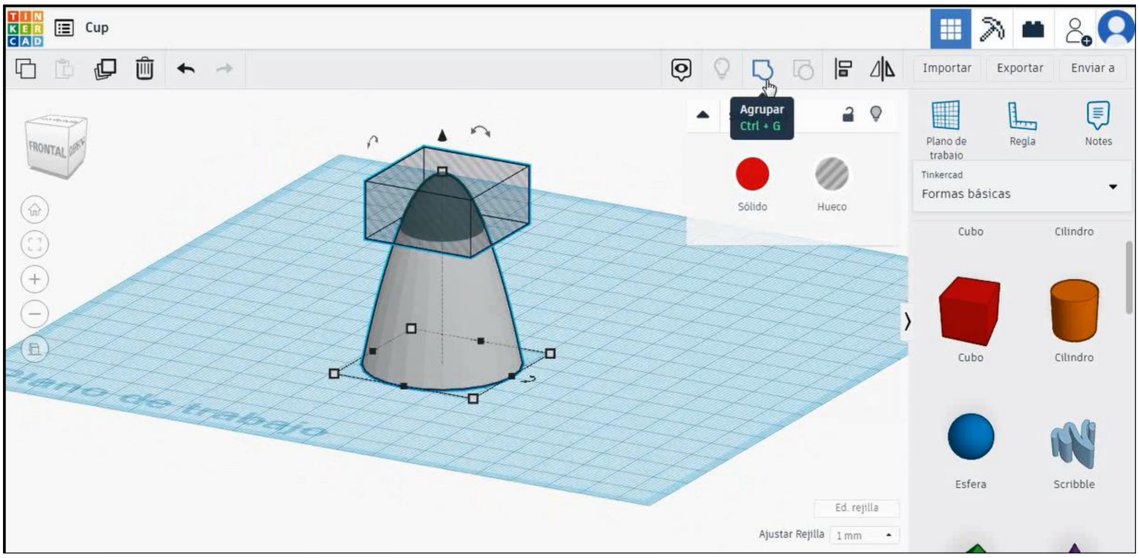
3. In order to create a flat base for the cup, we choose the cube and put it in hole mode

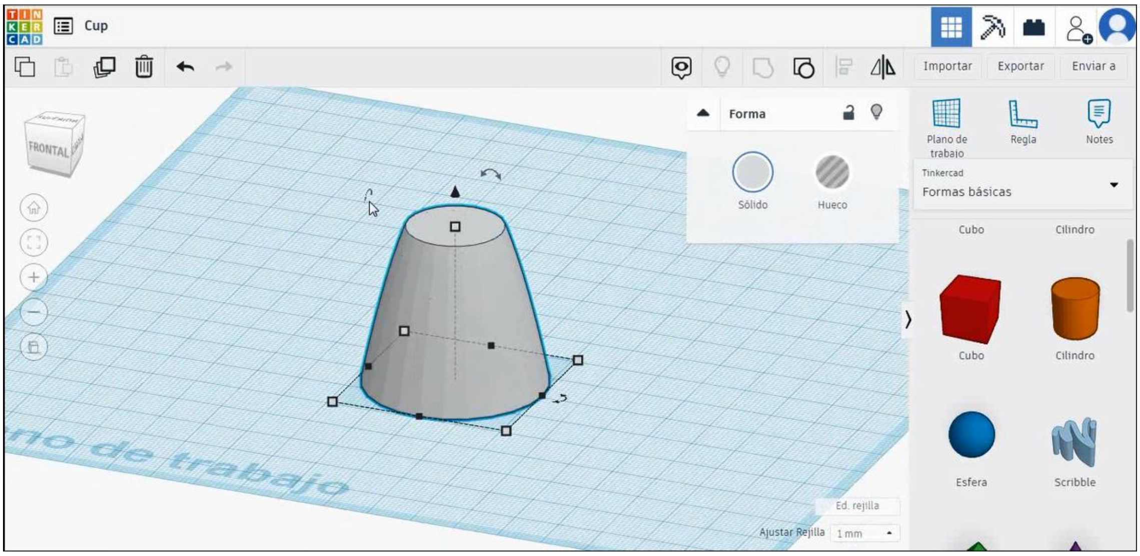


4. Change the measures to make it bigger and place it in the top of the paraboloid.

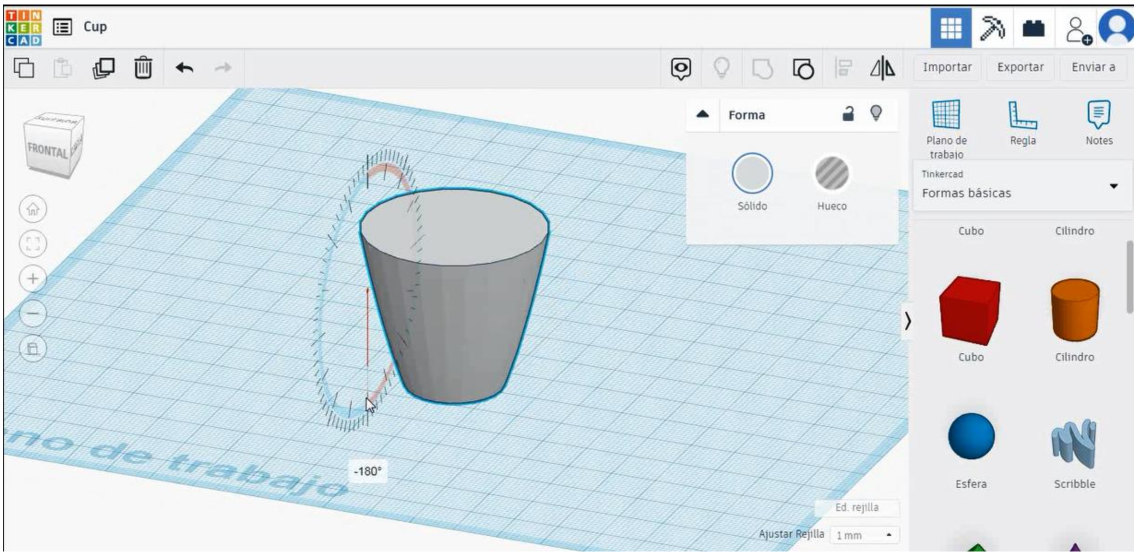


5. Pressing Shift and selecting the paraboloid and the cube, select group

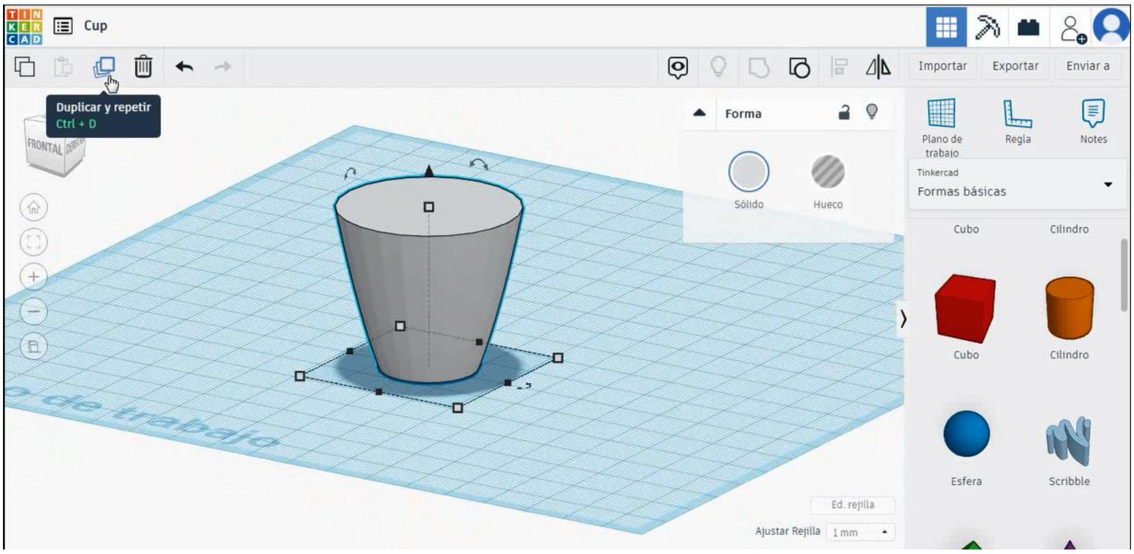


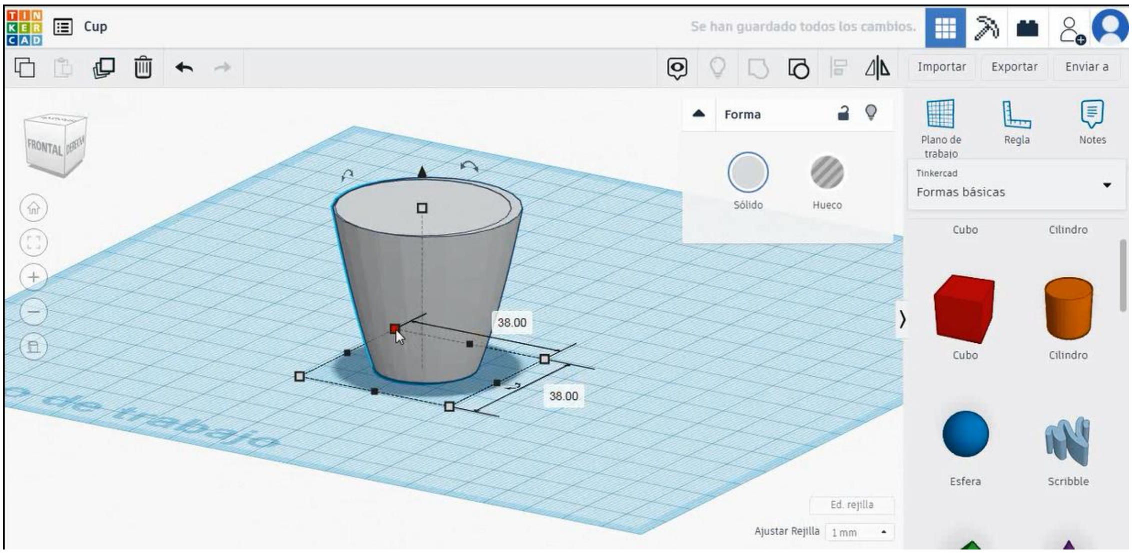


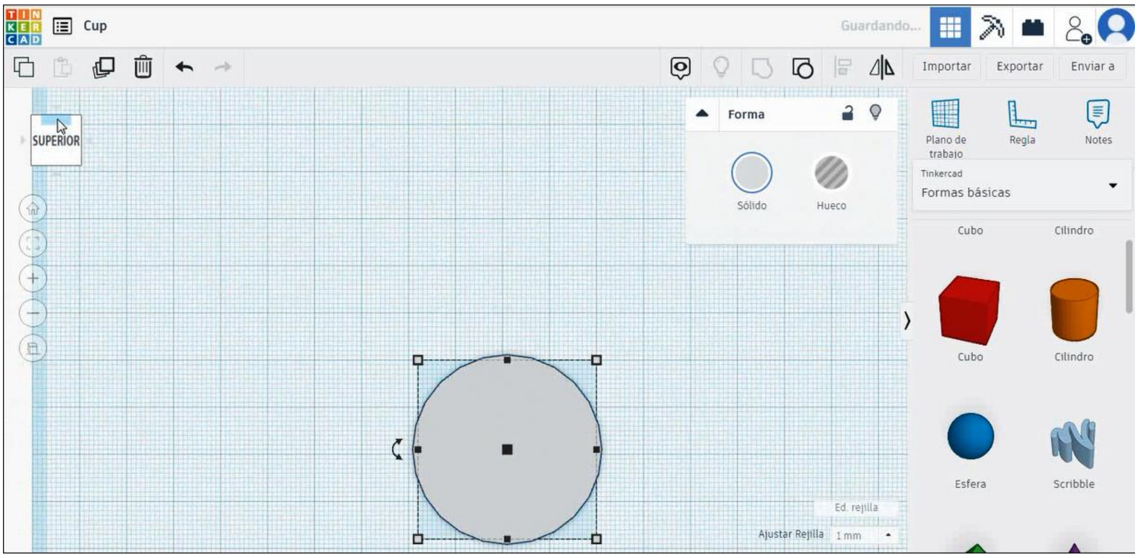
6. Now, we are going to rotate the cup 180 degrees

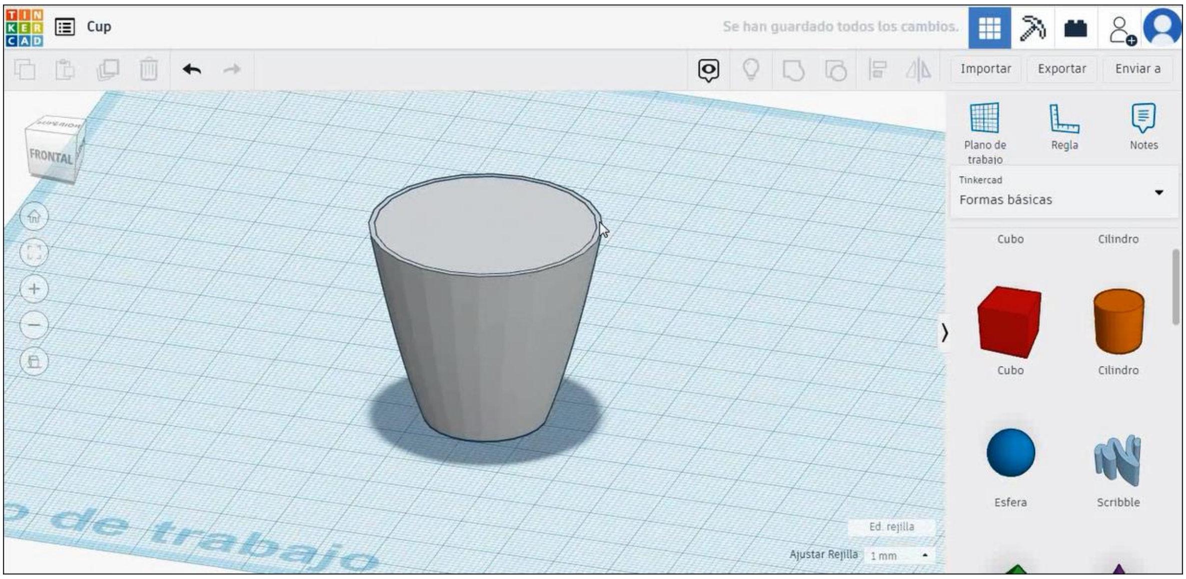


7. Copy the paraboloid shape and size it to 38x38x34 mm, then we put in the center of the original shape. We will use it later to empty the core.

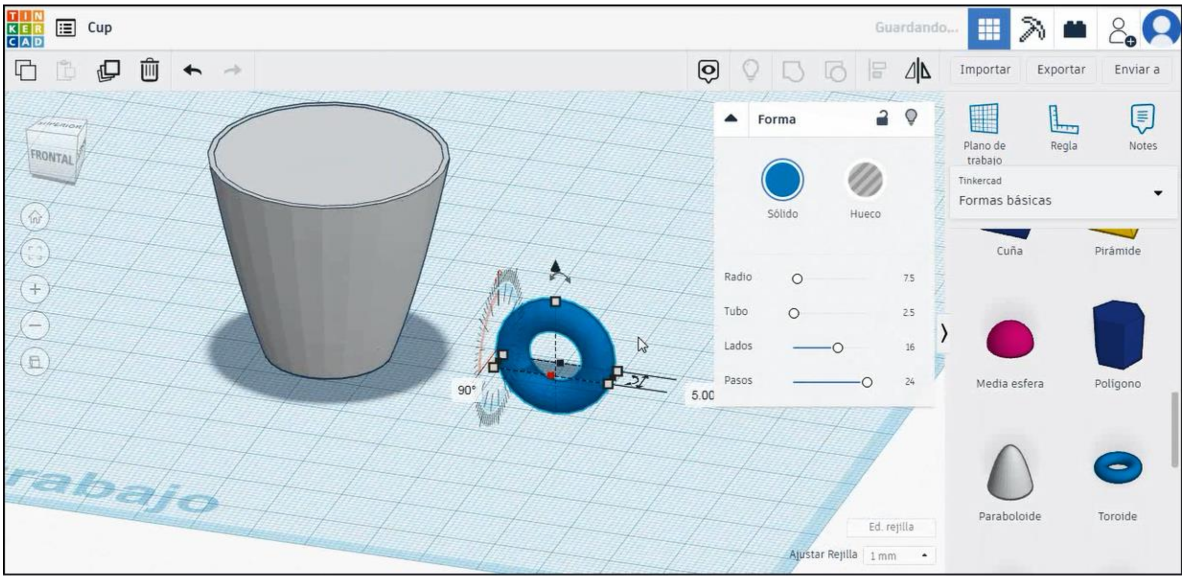


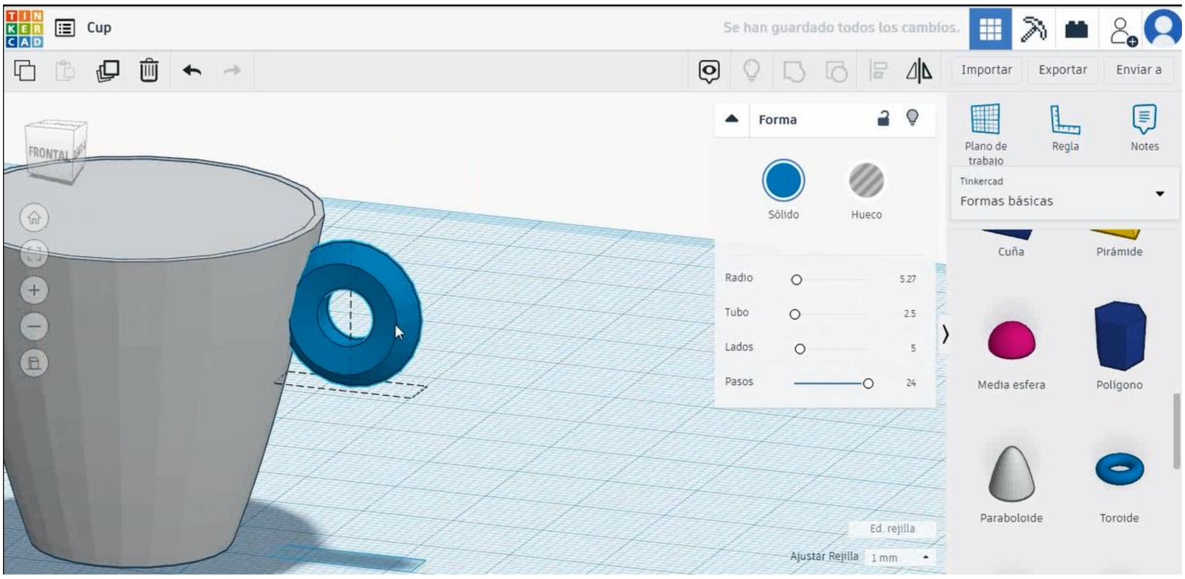


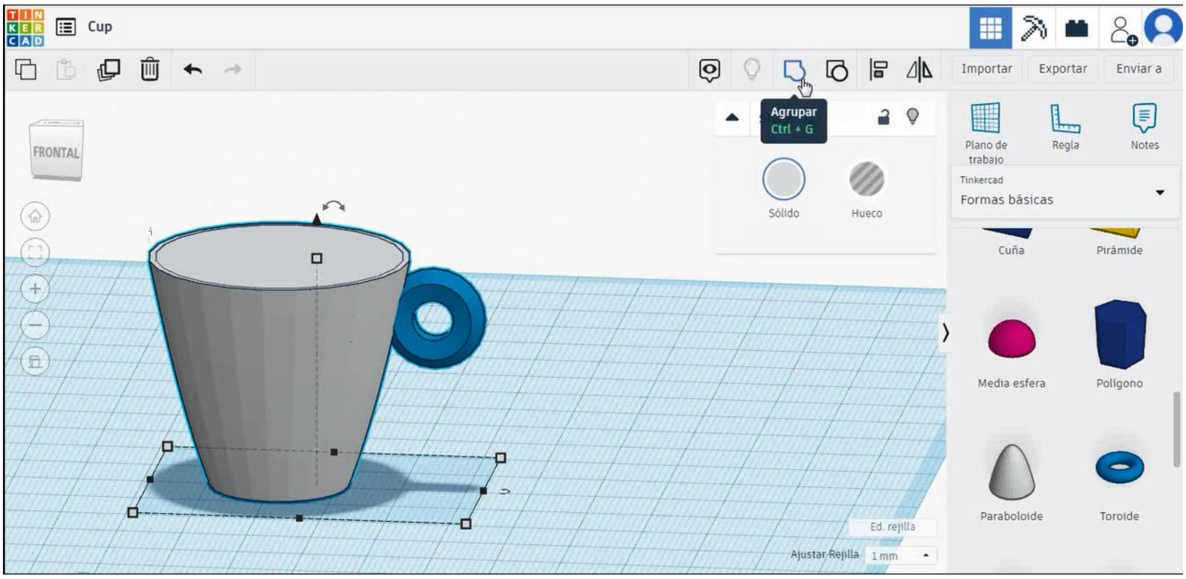




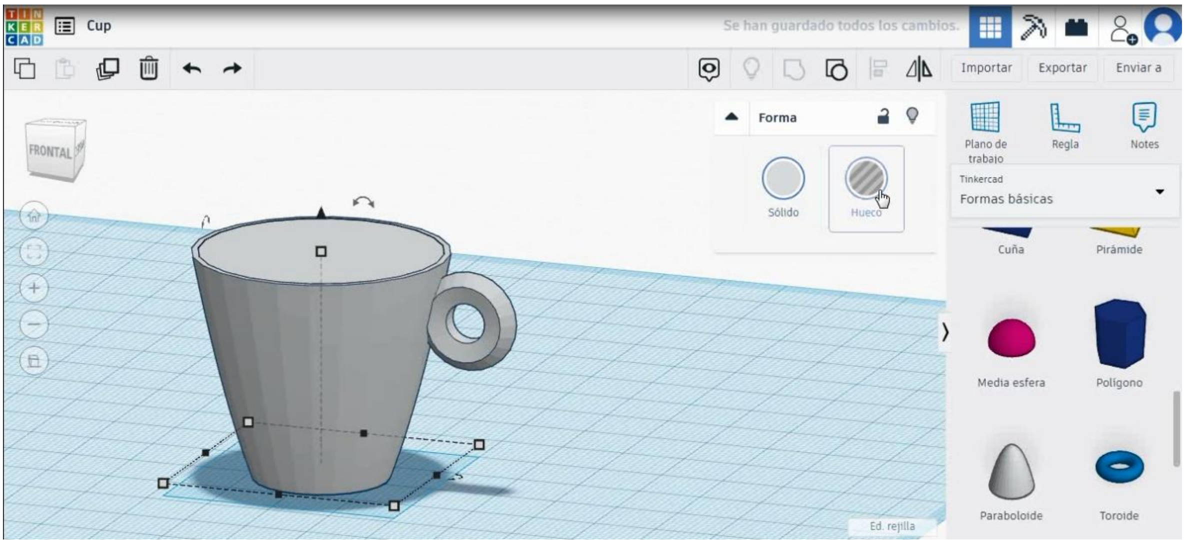
8. To create the handle, choose the torus in color mode and rotate it to put in the right top side of the cup. Then, pressing shif and selecting the cup and the torus press group to merge both.

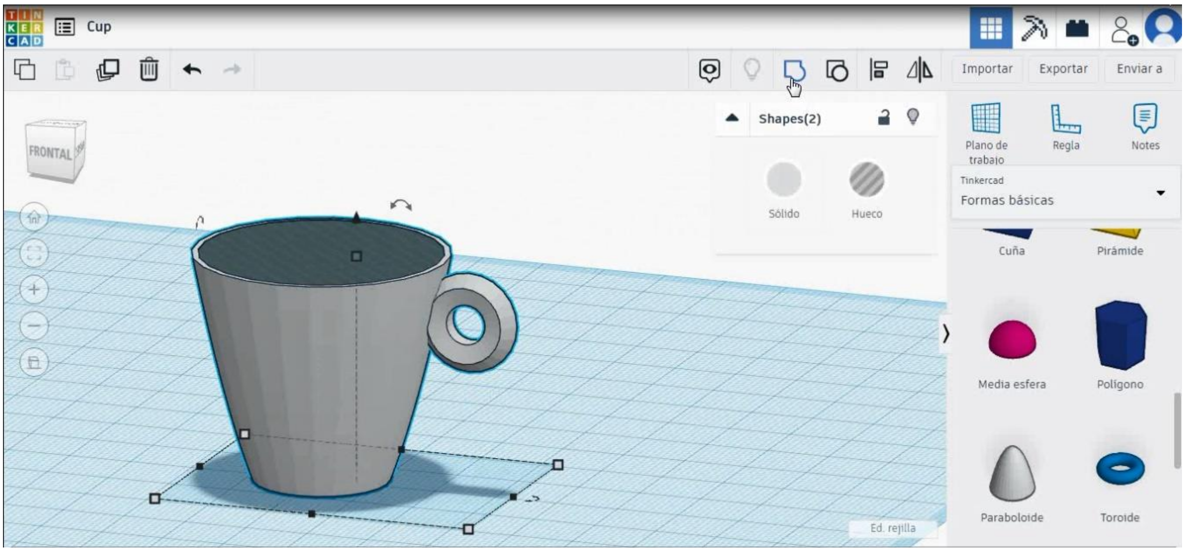




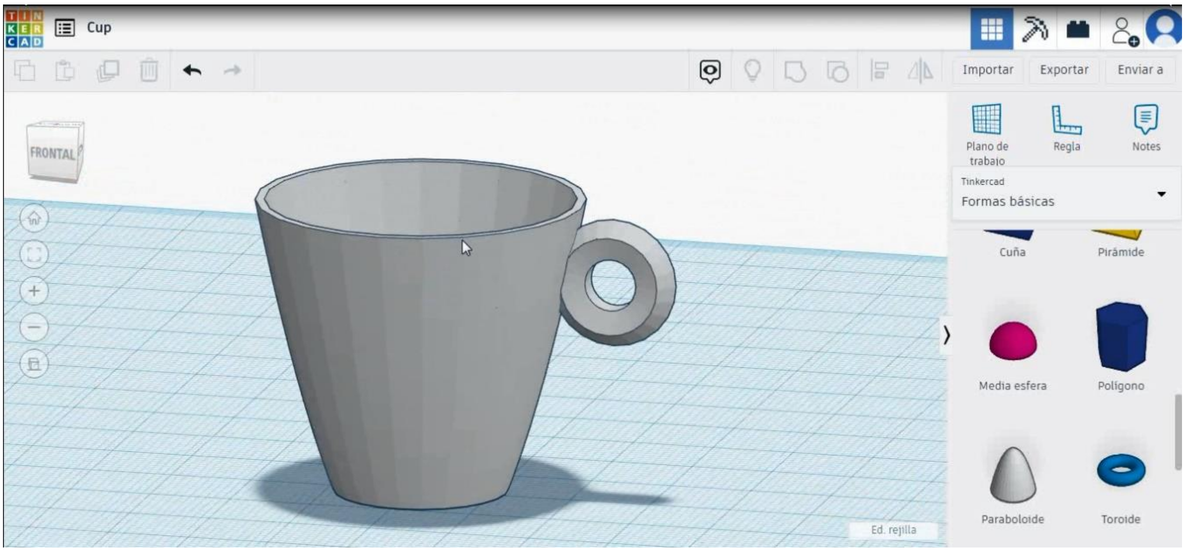


9. Select the smaller paraboloid that we have created before and choose hole mode. Select both entities pressing shift key and then select group to empty the core.





10. Now, the cup is finished.



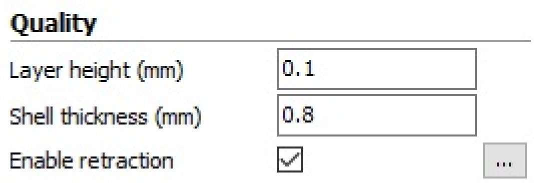
9.3.4.2 Cup 3D printing seetings

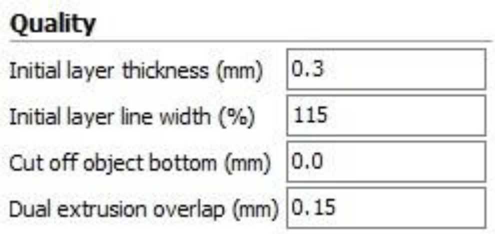
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

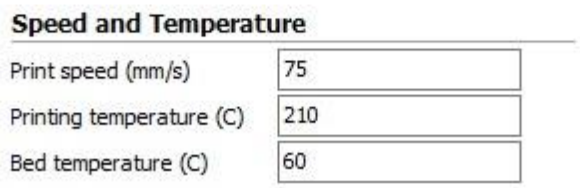
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

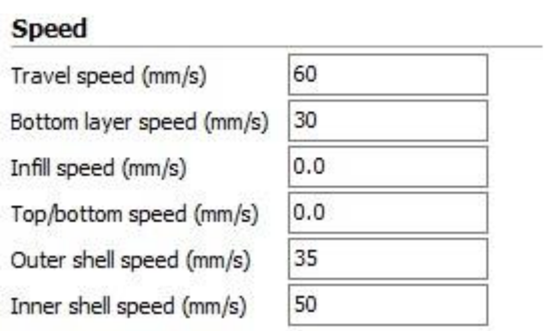




Speed and Temperature

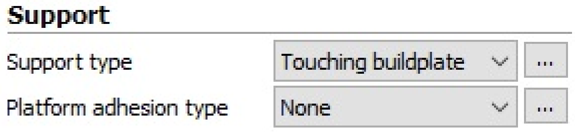
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

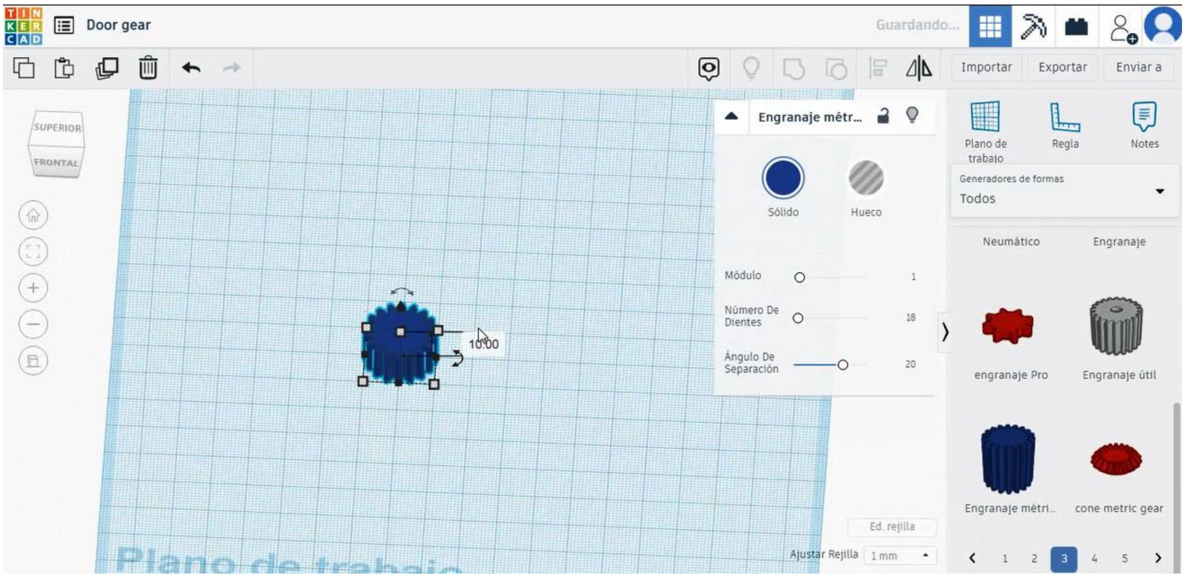
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

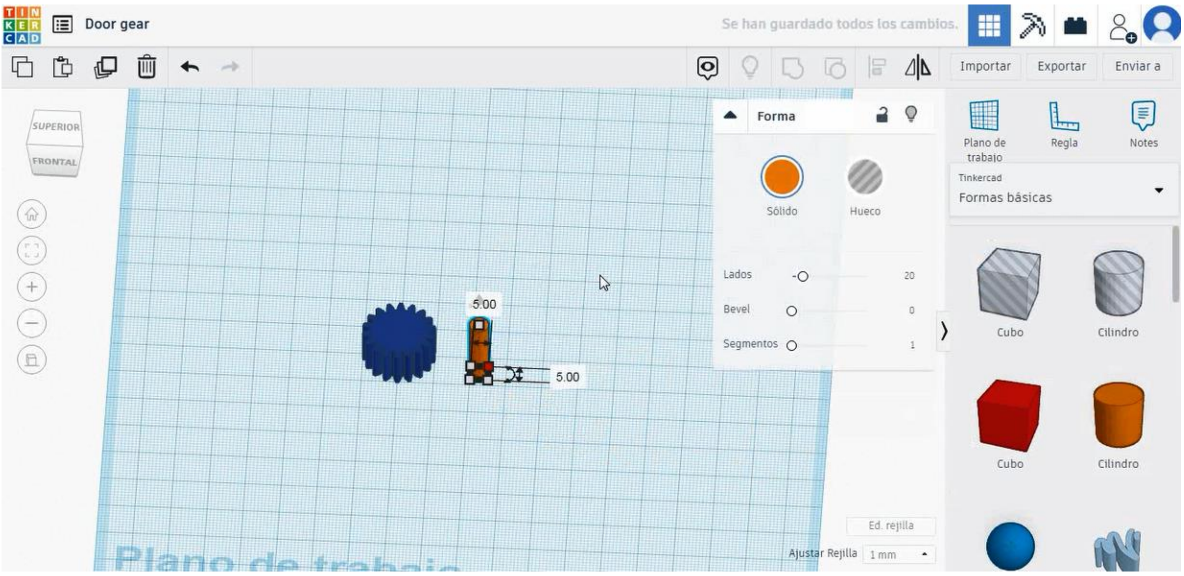


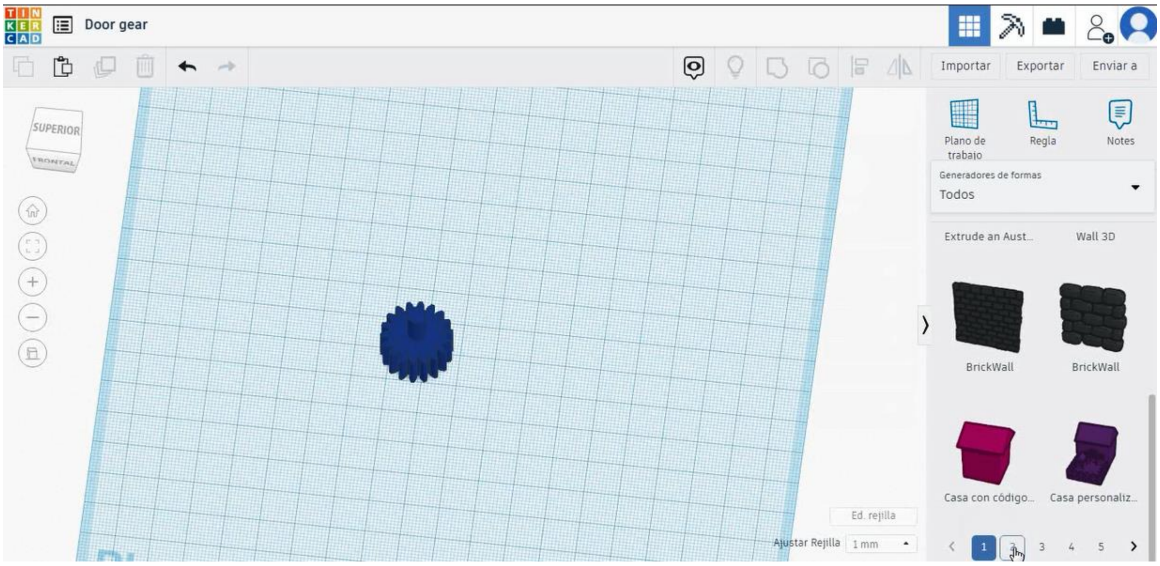
9.3.5 Part 5: Door Gear

9.3.5.1 Door Gear Design

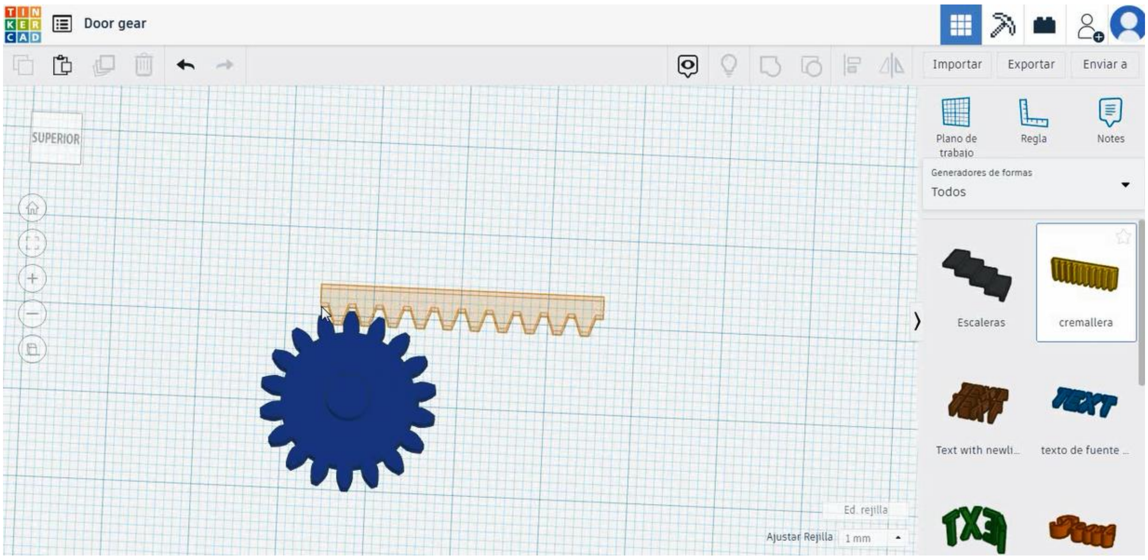
1. Choose the metric gear shape from the 3rd page of all categories and change the height to 10 mm. Now choose the cylinder shape, size it to 5x5x20 mm and align in the center of the gear, select both objects and press group.

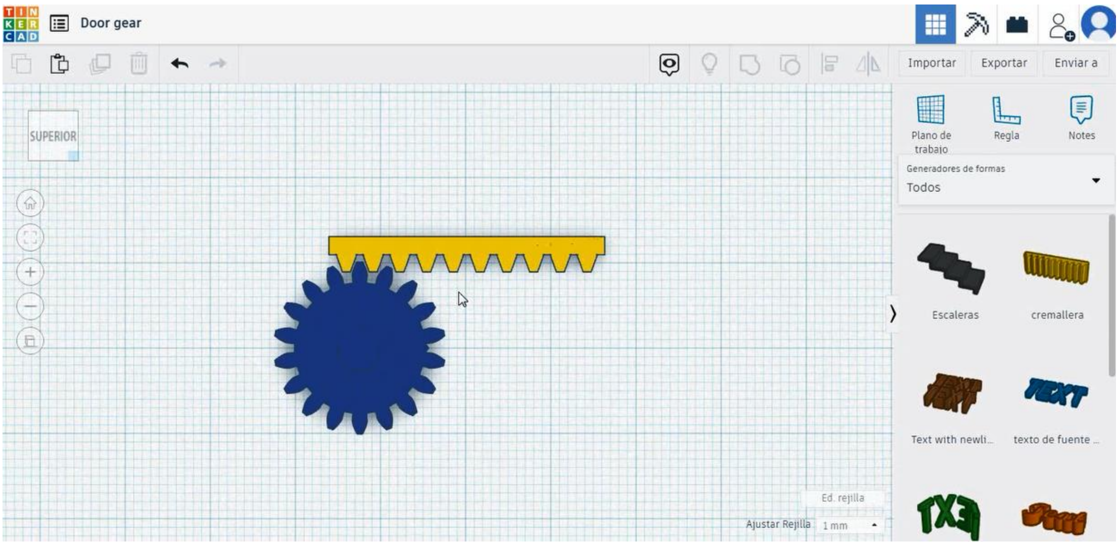




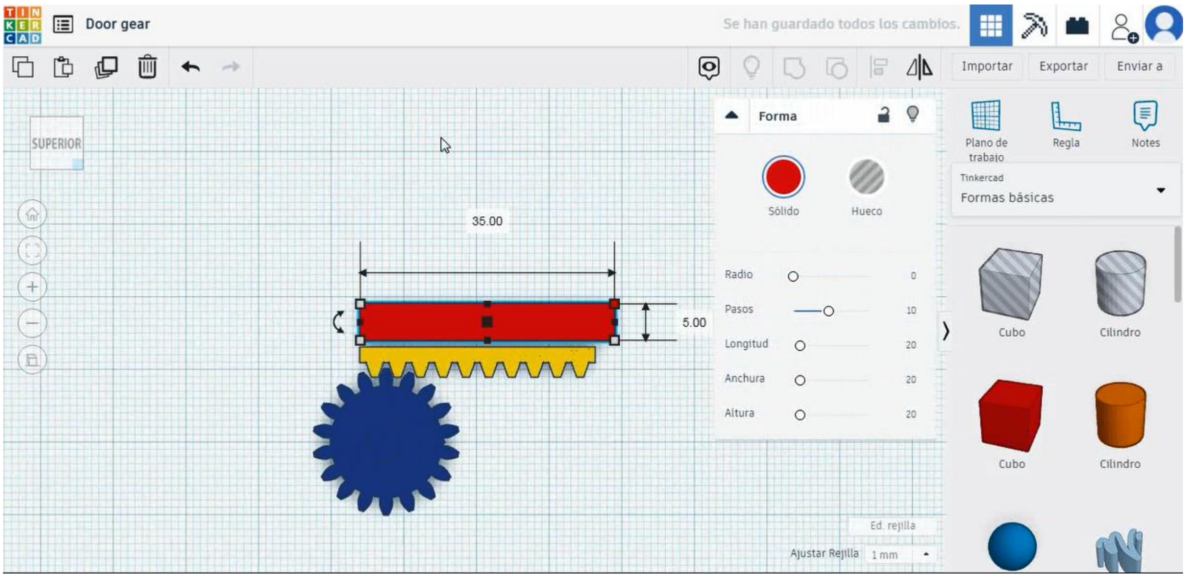


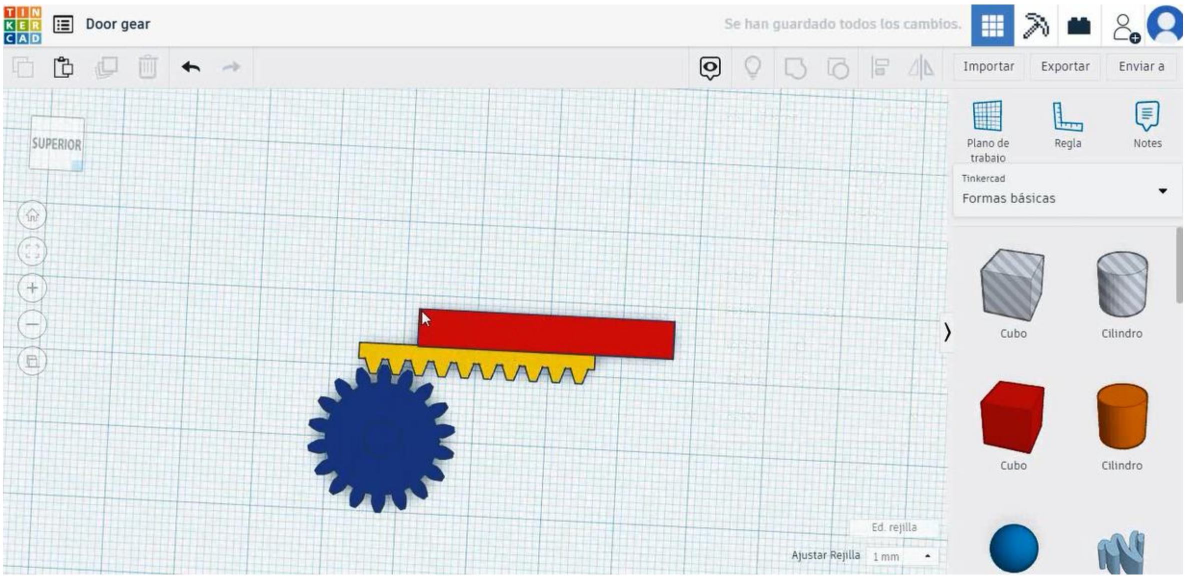
2. Choose the gear rack shape from the 2nd page of all categories and align the gap between the first and the second tooth with the central axis of the gear.



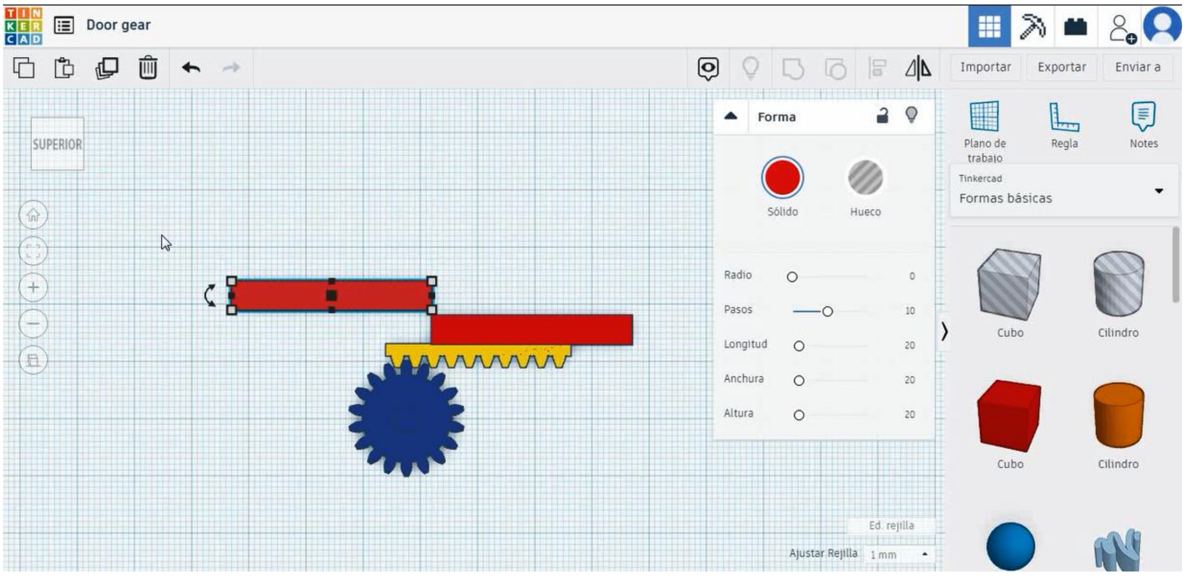


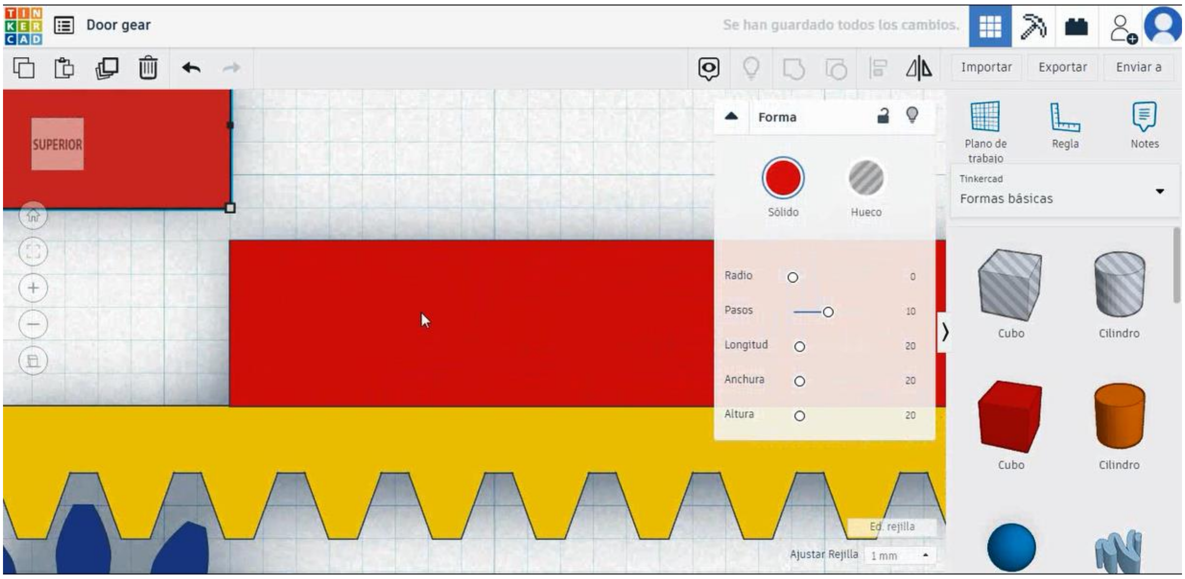
3. Select the cube shape to create the door, size it to 35x5x20 mm and align the left border with it in the third tooth of the gear rack.



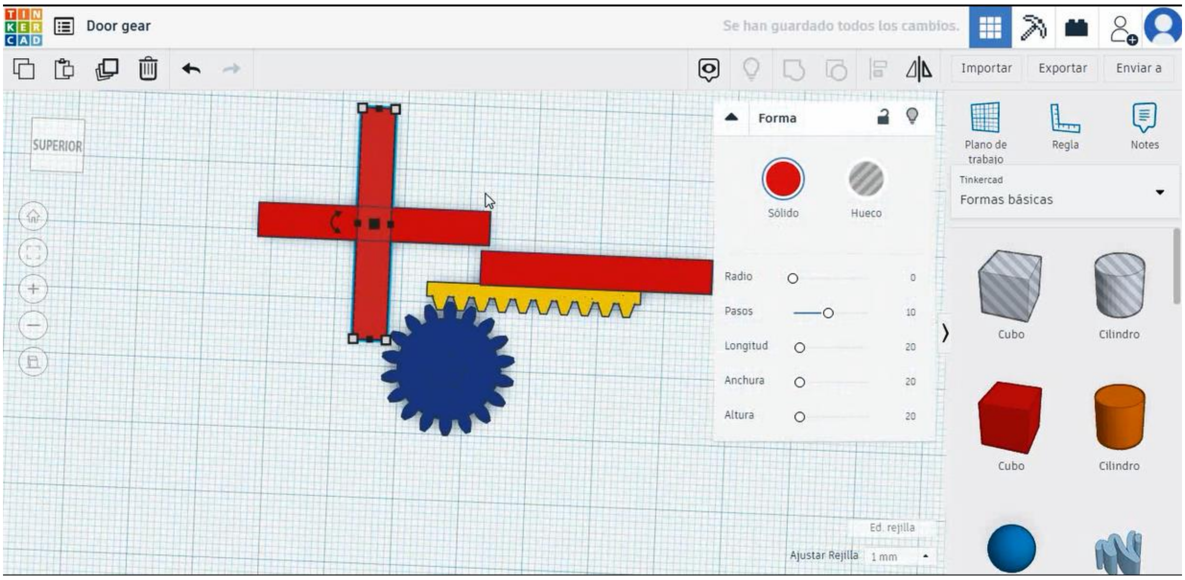


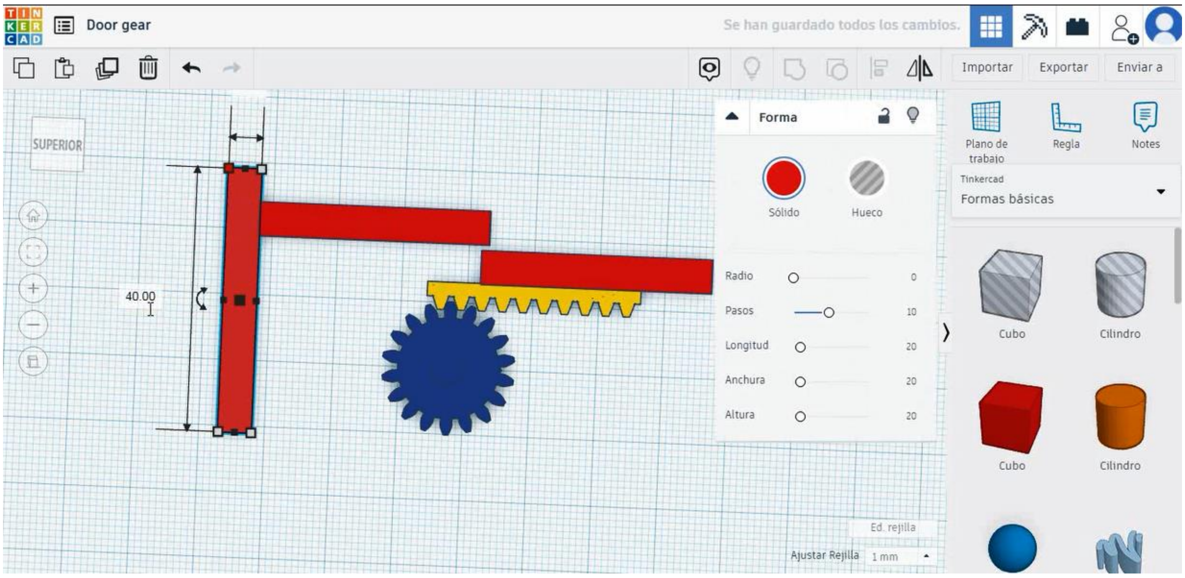
4. Copy the door to start doing the wall and move to align the right side of the new cube with the left side of the door in X axis. We need to leave a gap in Y axis like the image below.

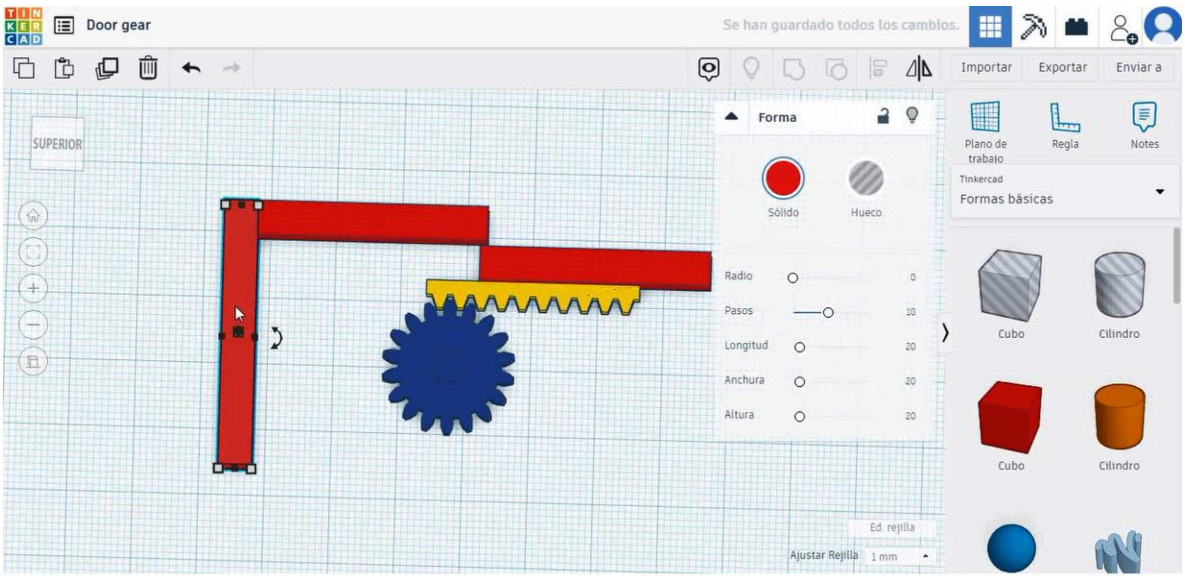




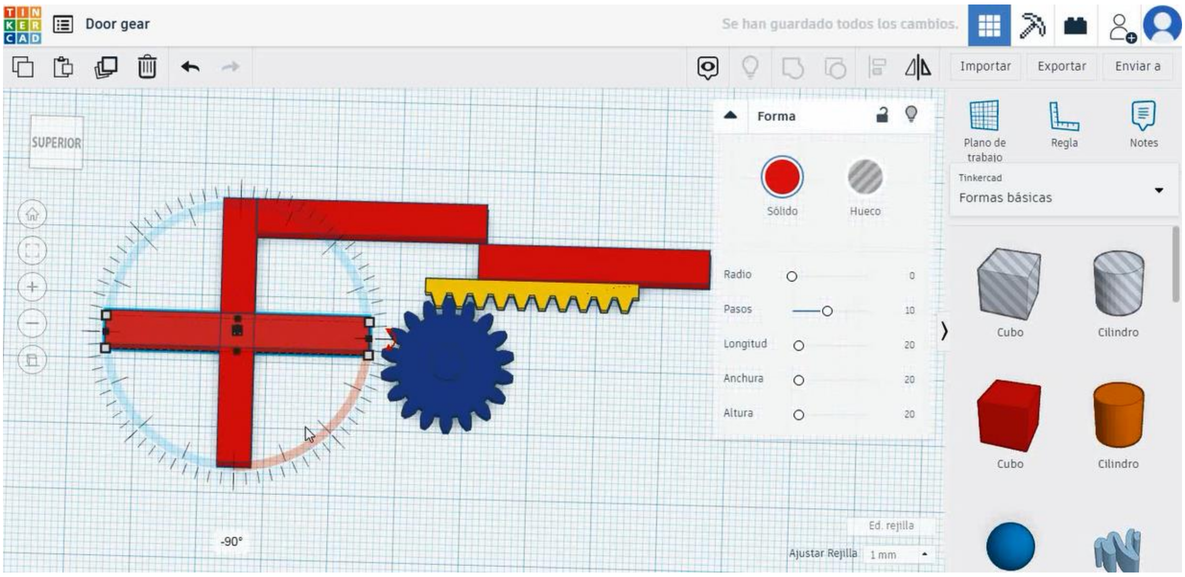
5. Copy the wall cube, turn it 90 degrees , size it to 40x5x20 mm and move it to match with the first wall like the image below.

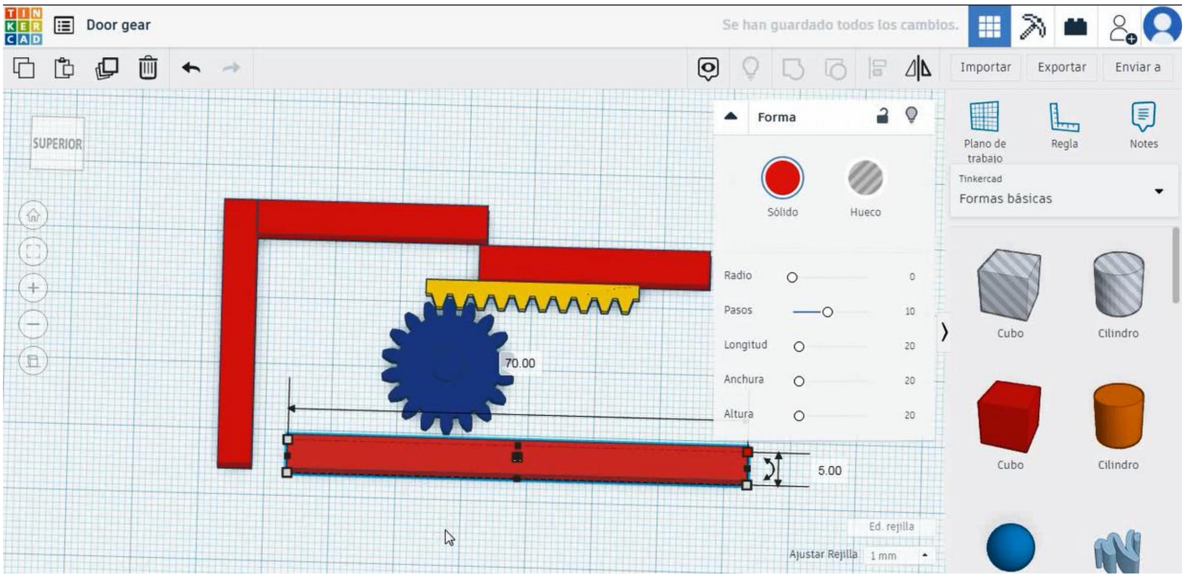

Door gear
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
SUPERIOR
Forma
Sólido
Hueco
Radio
Pasos
Longitud
Anchura
Altura
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Ed. rejilla
Ajustar Rejilla
1 mm

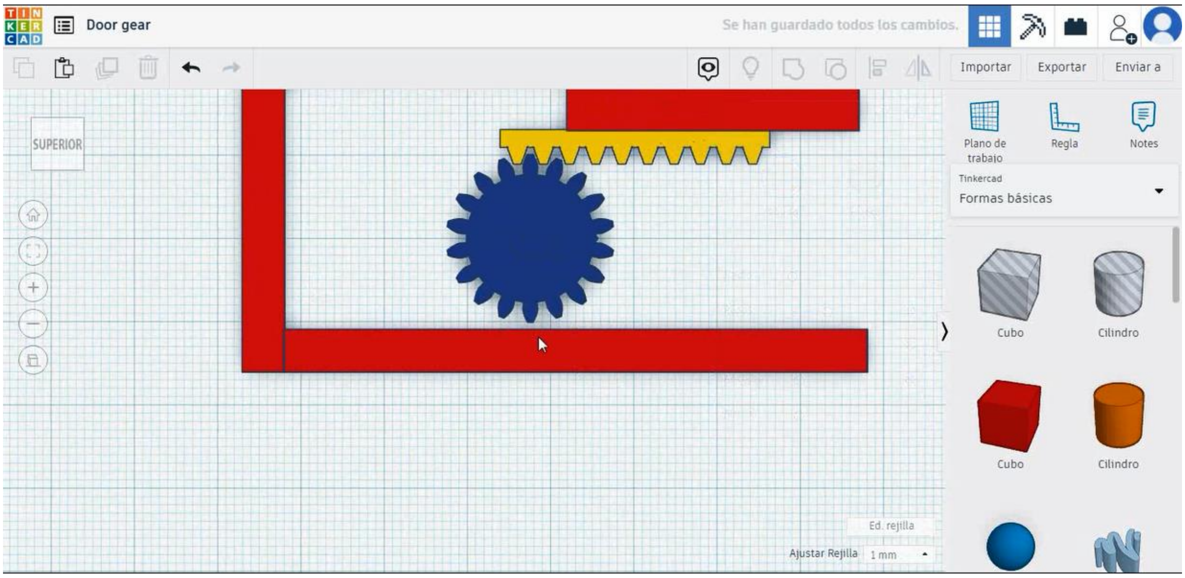

Door gear
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
SUPERIOR
40.00
Forma
Sólido
Hueco
Radio
Pasos
Longitud
Anchura
Altura
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Ed. rejilla
Ajustar Rejilla
1 mm


Door gear
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
SUPERIOR
Forma
Sólido
Hueco
Radio
Pasos
Longitud
Anchura
Altura
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Ed. rejilla
Ajustar Rejilla
1 mm

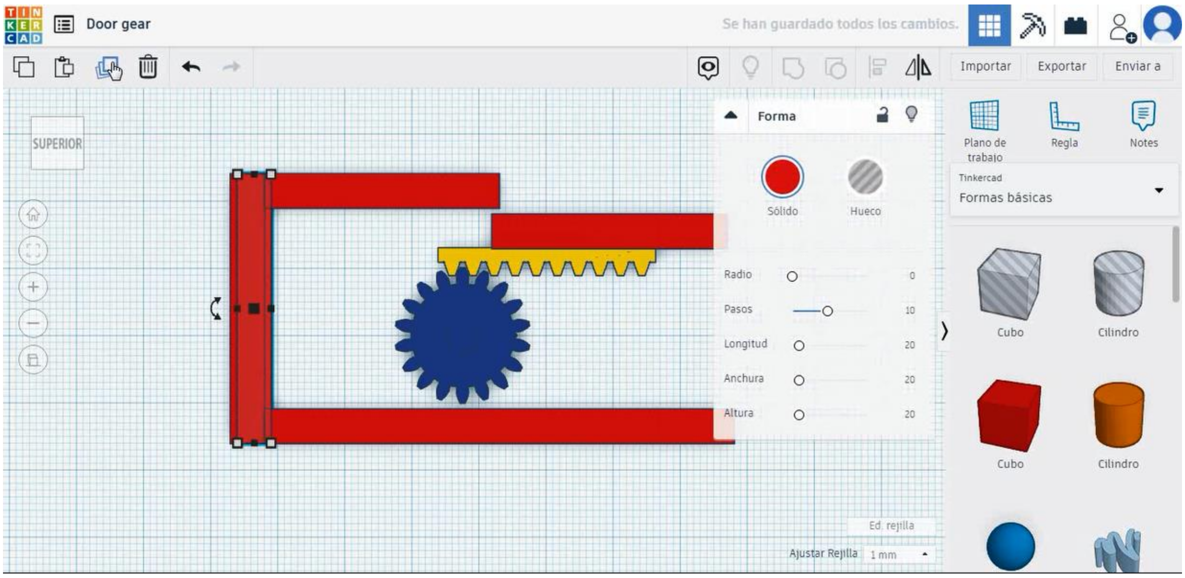
6. Copy the second wall cube, turn it 90 degrees , and size it to 70x5x20 mm and move it to match with the second wall cube like the image below.

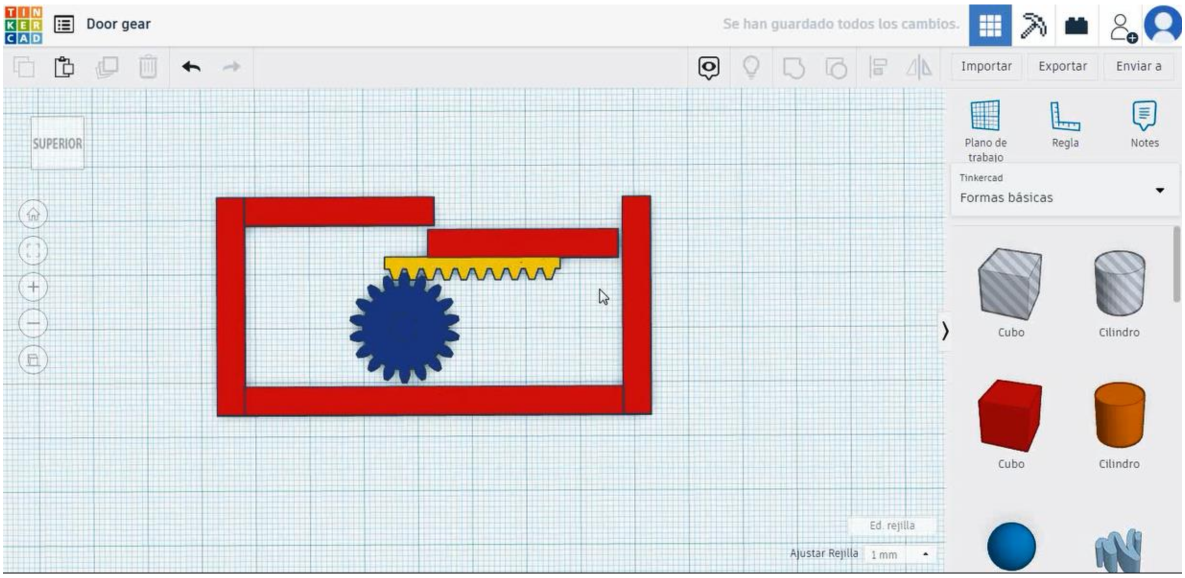


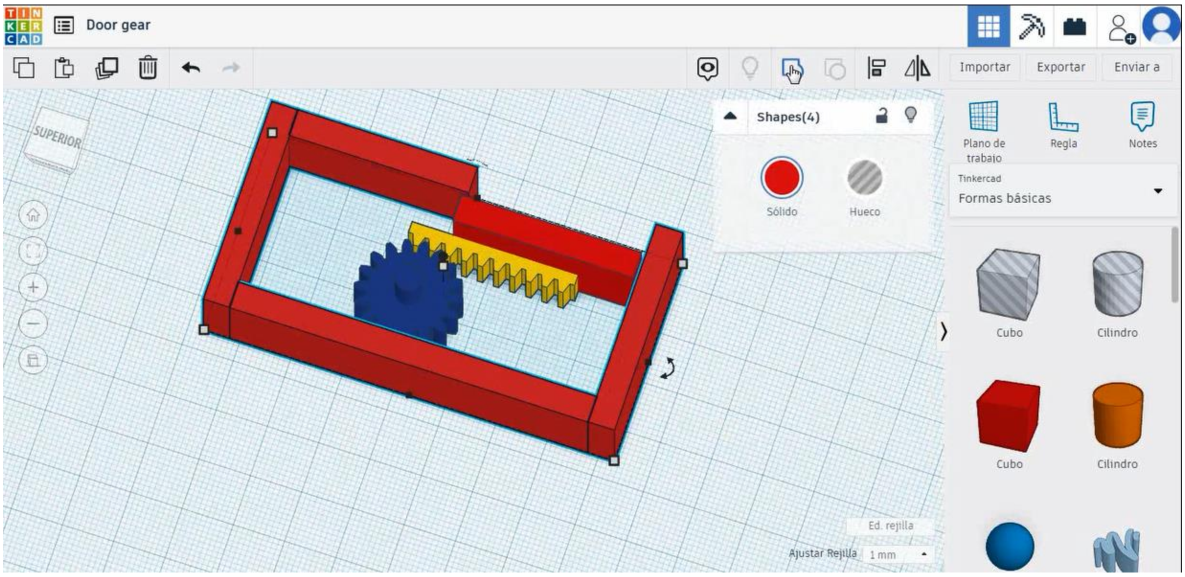


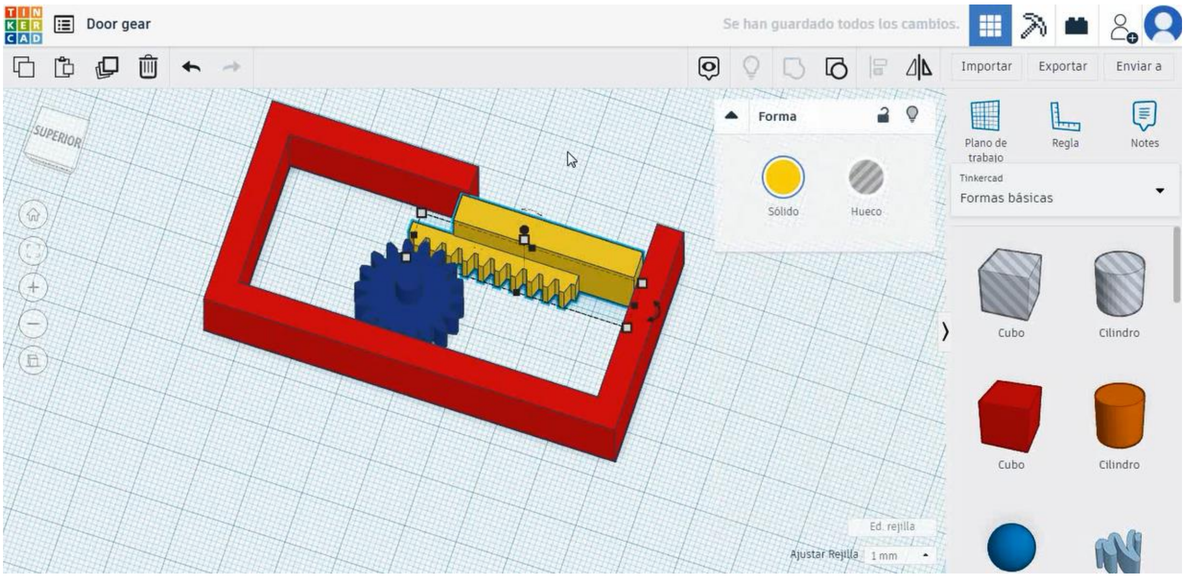


7. Copy the second wall cube and move it to match with the third wall cube like the image below. Select the four wall cubes and press align. Then select the door and the rack gear and press group.

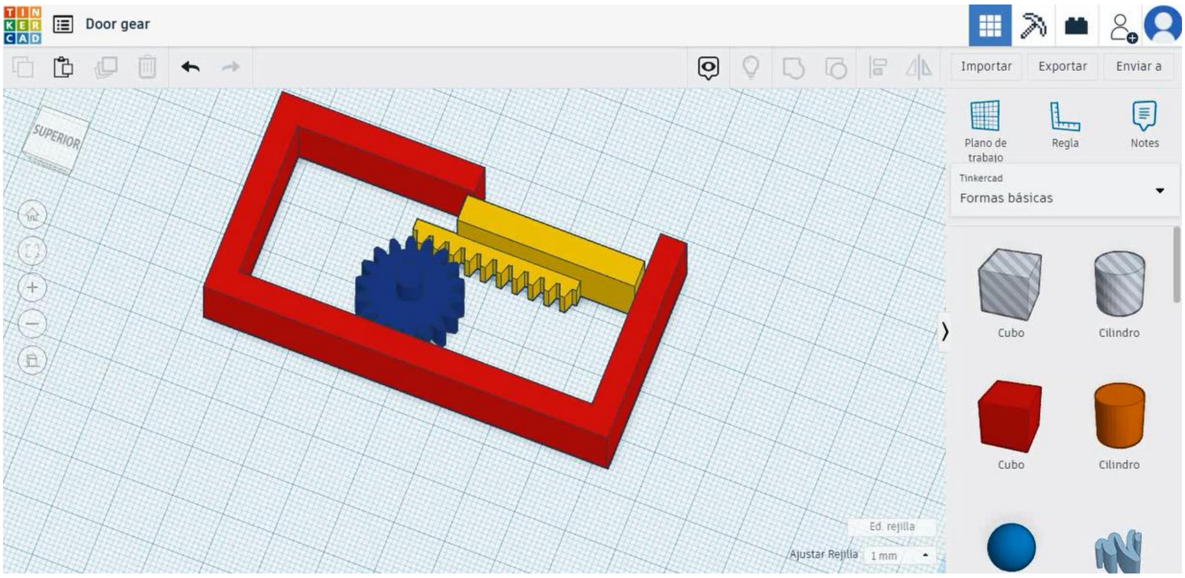



Door gear
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Cubo
Cilindro
SUPERIOR
Ed. rejilla
Ajustar Rejilla
1 mm


Door gear
Importar
Exportar
Enviar a
Shapes(4)
Sólido
Hueco
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Cubo
Cilindro
SUPERIOR
Ed. rejilla
Ajustar Rejilla
1 mm


Door gear
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Cubo
Cilindro
SUPERIOR
Ed. rejilla
Ajustar Rejilla
1 mm

8. Now the door gear is finished.



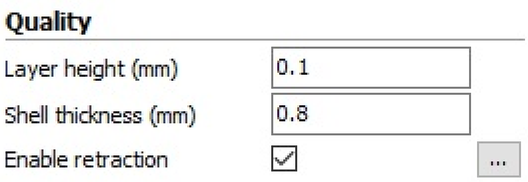
9.3.5.2 Door Gear 3D printing seetings

<u>Filament</u>

PLA
Diameter - 1.75 (mm)
Flow - 100%

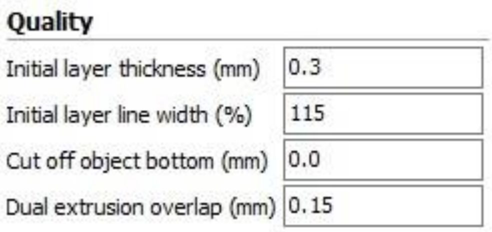
<u>Quality</u>

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)



Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑



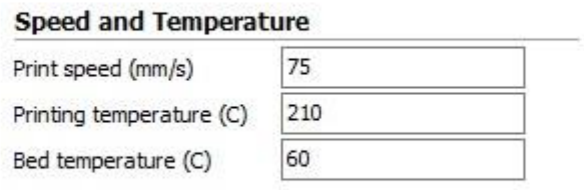
Quality

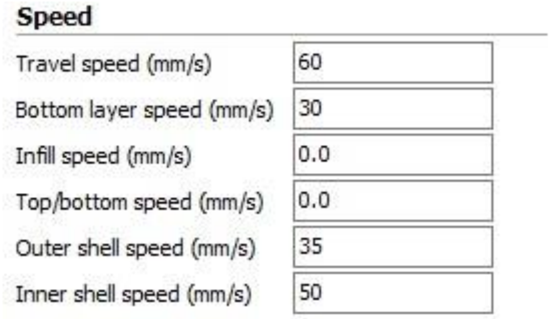
Initial layer thickness (mm)	0.3
Initial layer line width (%)	115
Cut off object bottom (mm)	0.0
Dual extrusion overlap (mm)	0.15

Co-funded by the Erasmus+ Programme of the European Union

Speed and Temperature

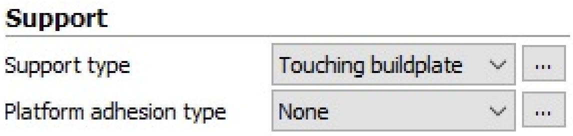
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

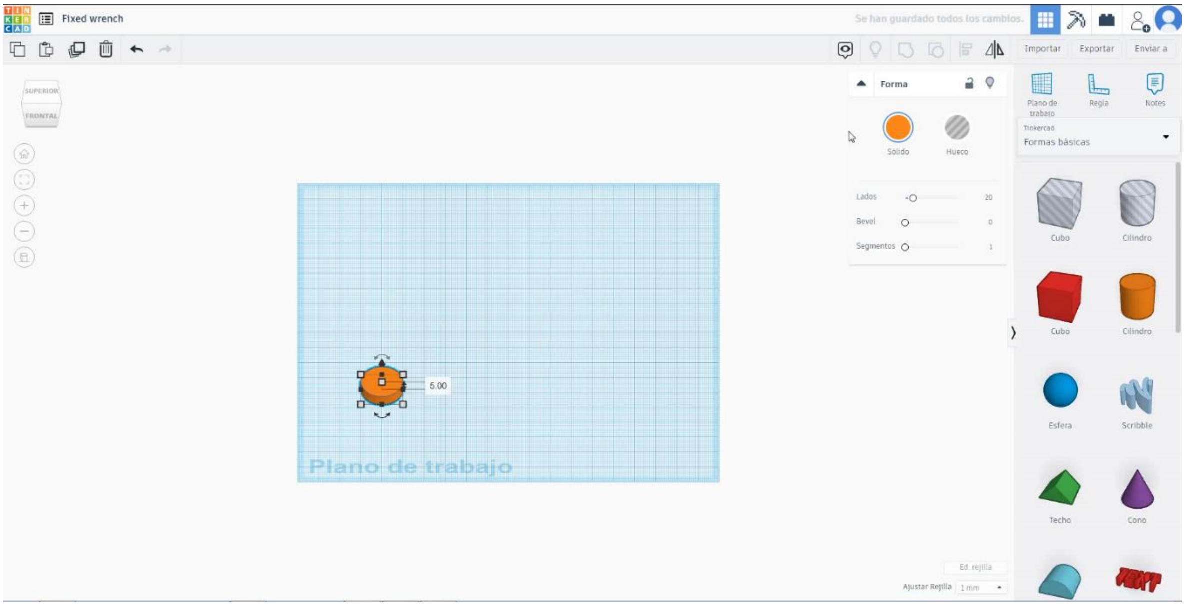
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



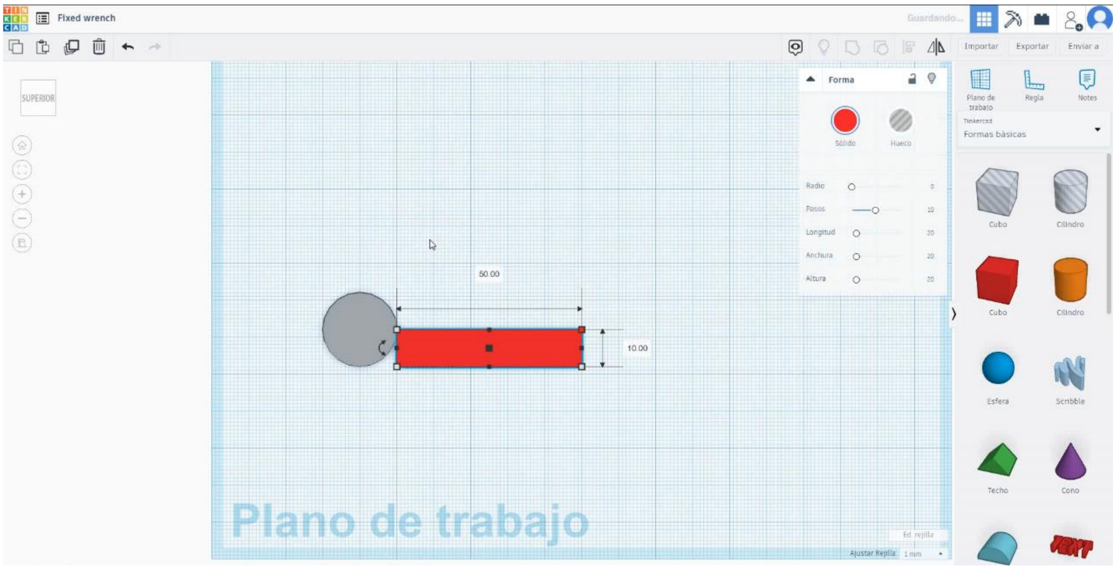
9.3.6 Part 6: Fixed Wrench

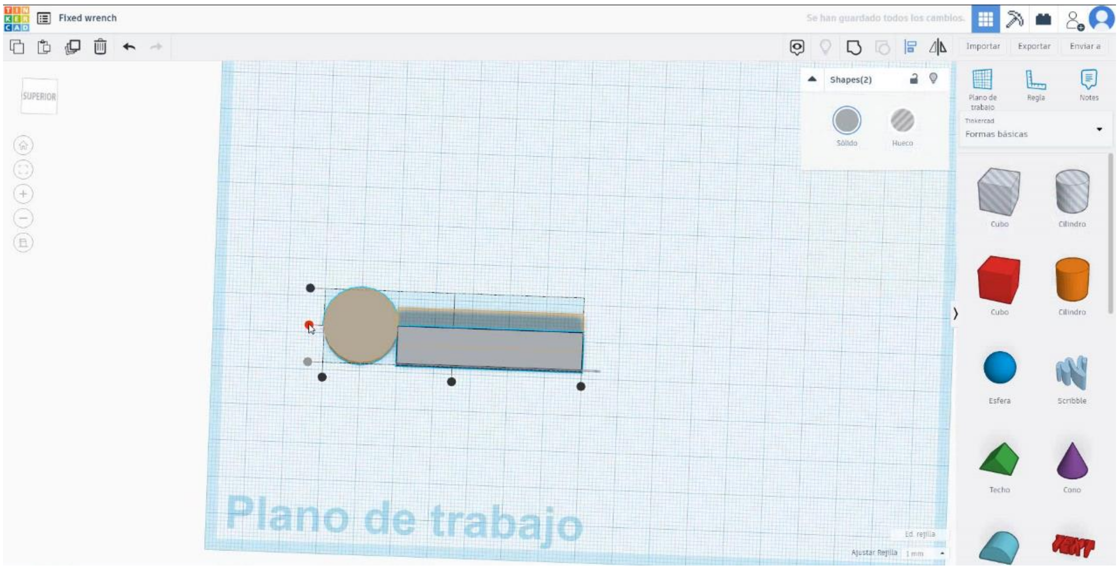
9.3.6.1 Boomeran Design

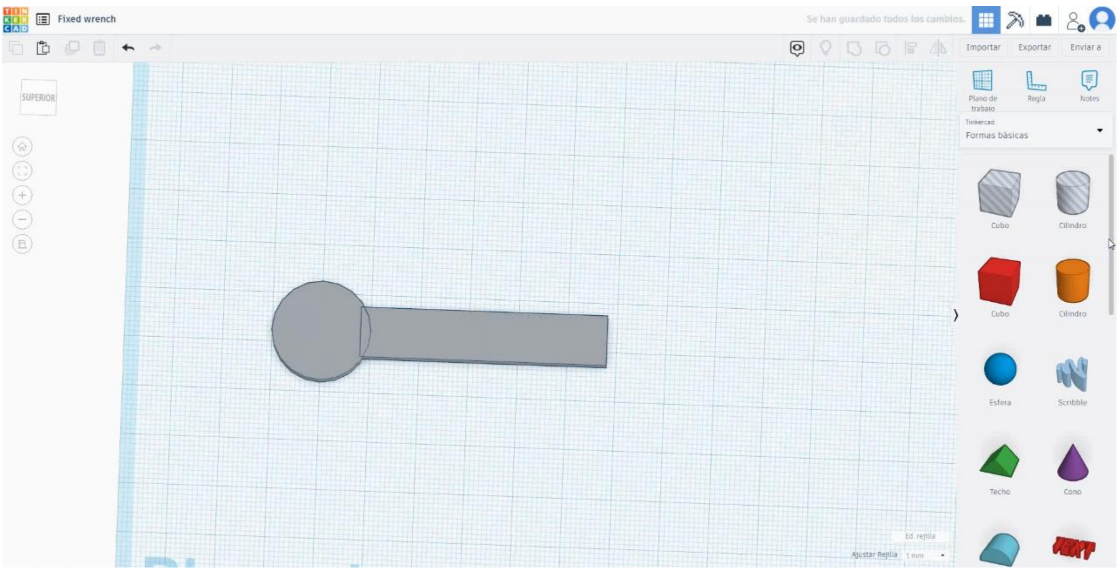
1. Choose the cylinder shape and size it to 20x20x5 mm.



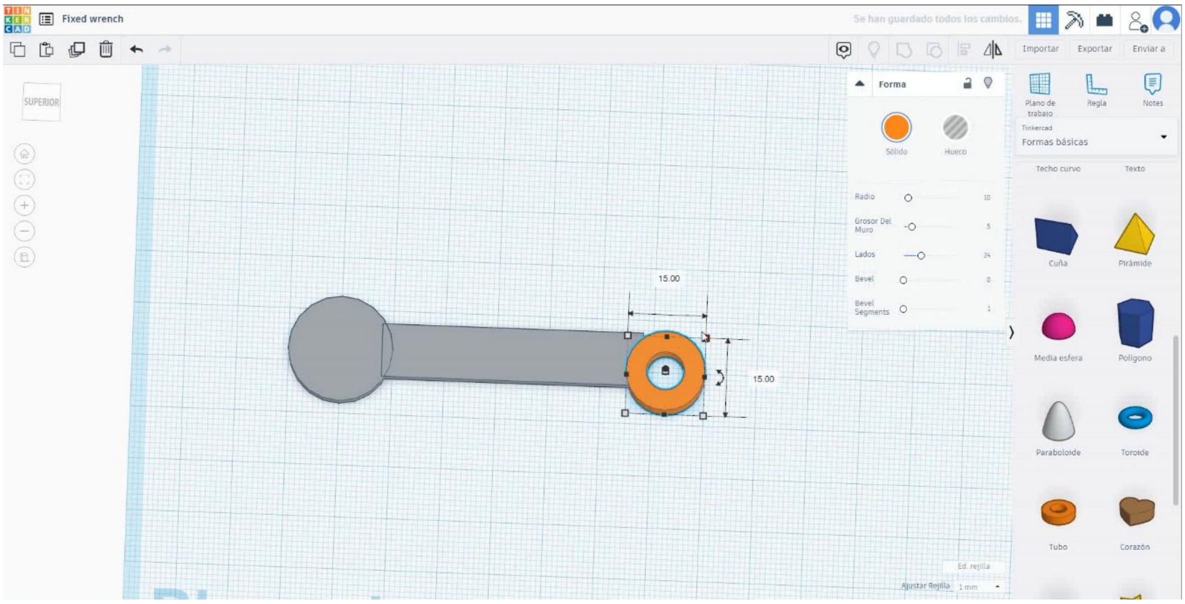
2. Choose the cube shape and size it to 50x10x5 mm and align it with the cylinder in Y axis. Make sure both objects touch each other.

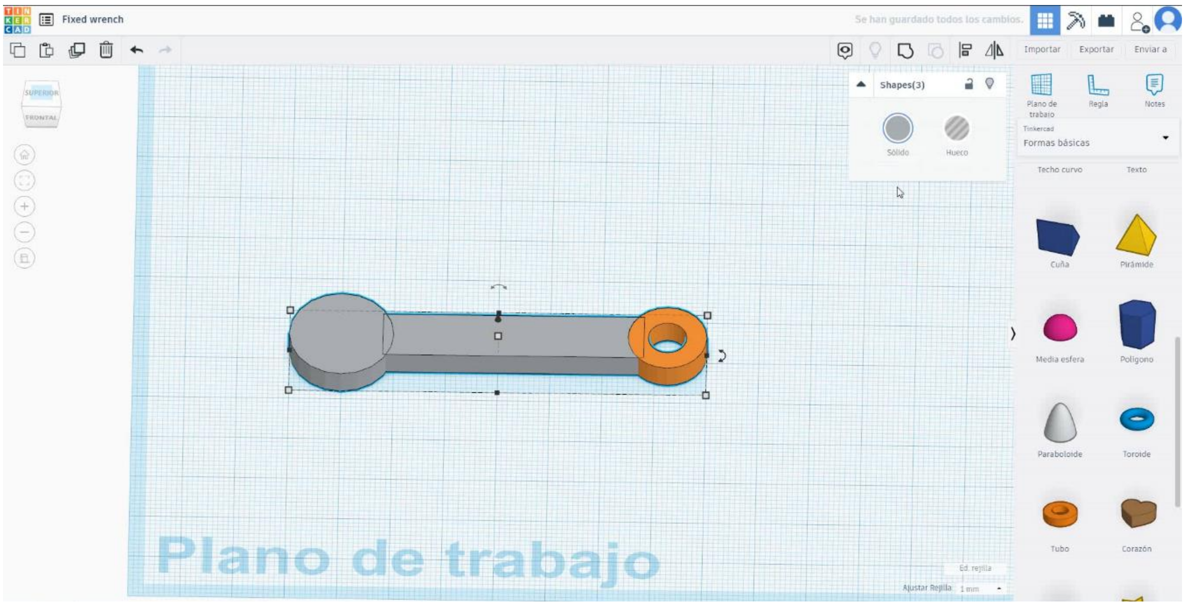

Fixed wrench
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
50.00
10.00
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
Plano de trabajo

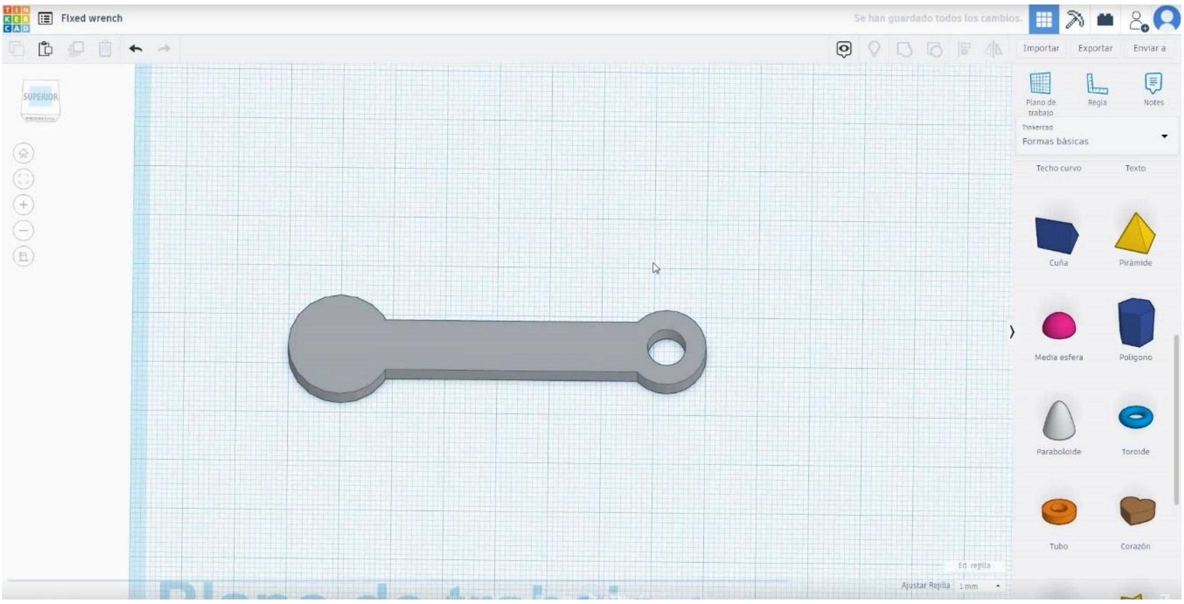

Fixed wrench
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Shapes(2)
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
Plano de trabajo


Fixed wrench
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono

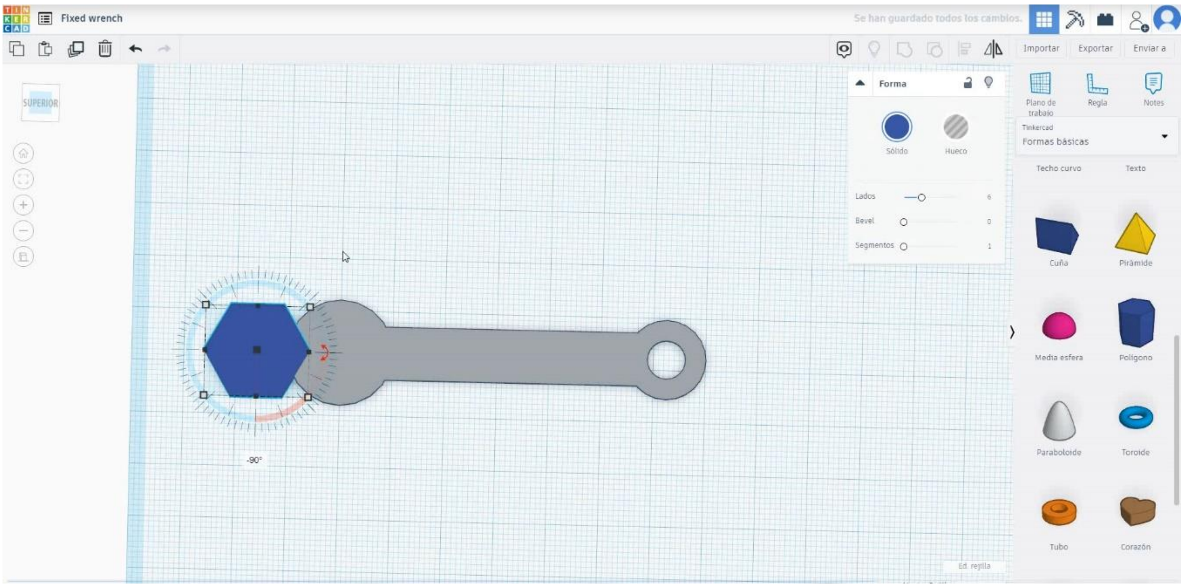
3. Choose the tube shape and size it to 15x15x5 mm with 5 mm wall thickness and align it with the cylinder in Y axis. Make sure both objects touch each other. Select the three objects and press group.

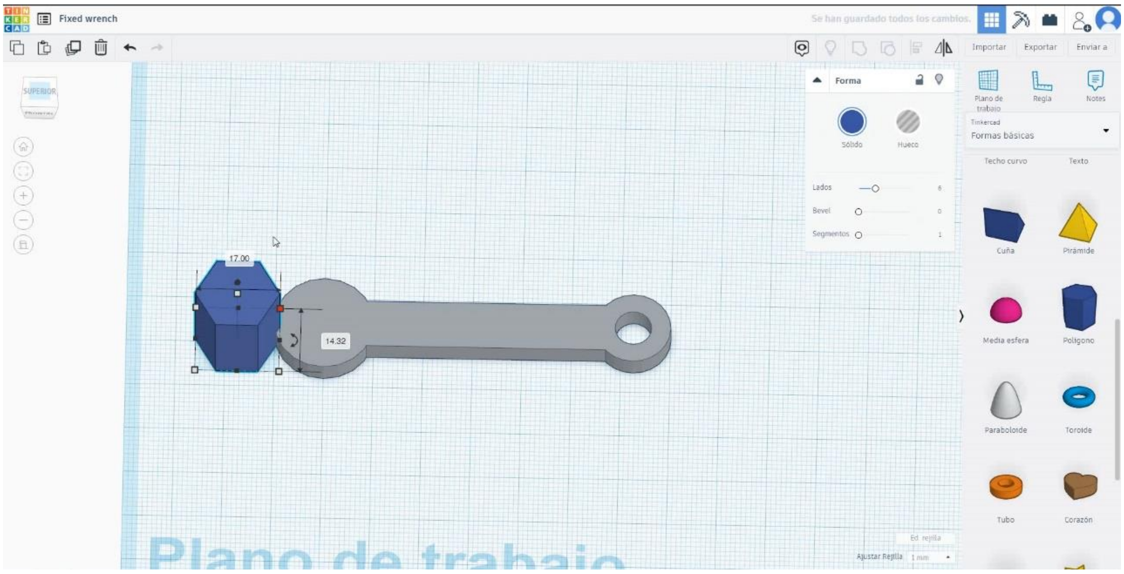




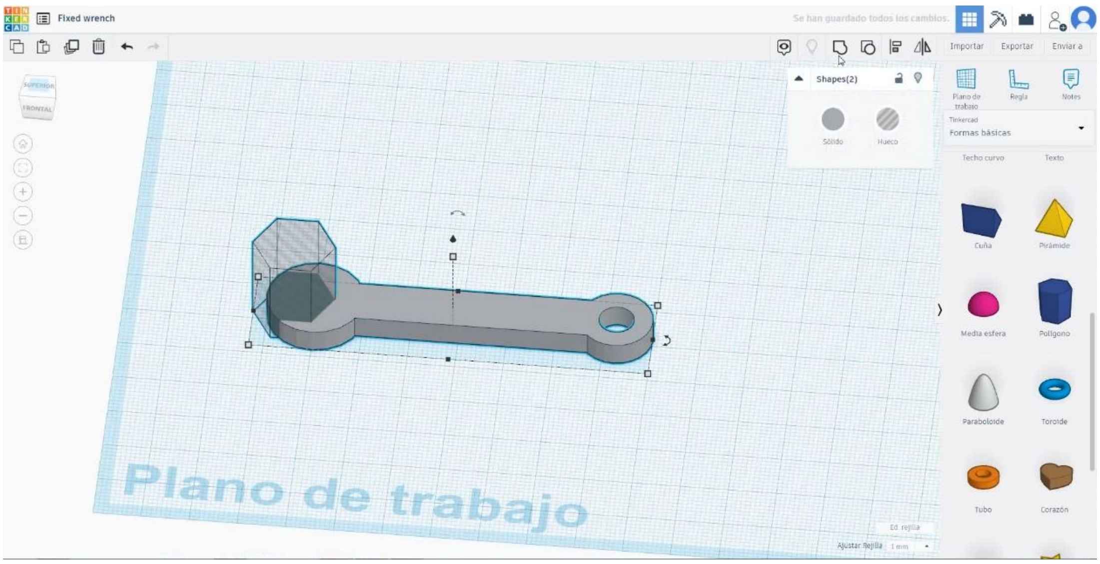


4. Choose the polygon shape and size it to 17x14.32x5 mm in hole mode, turn it 90 degrees and align it with the wrench in Y axis. Move it 14 mm from left first touch each other.

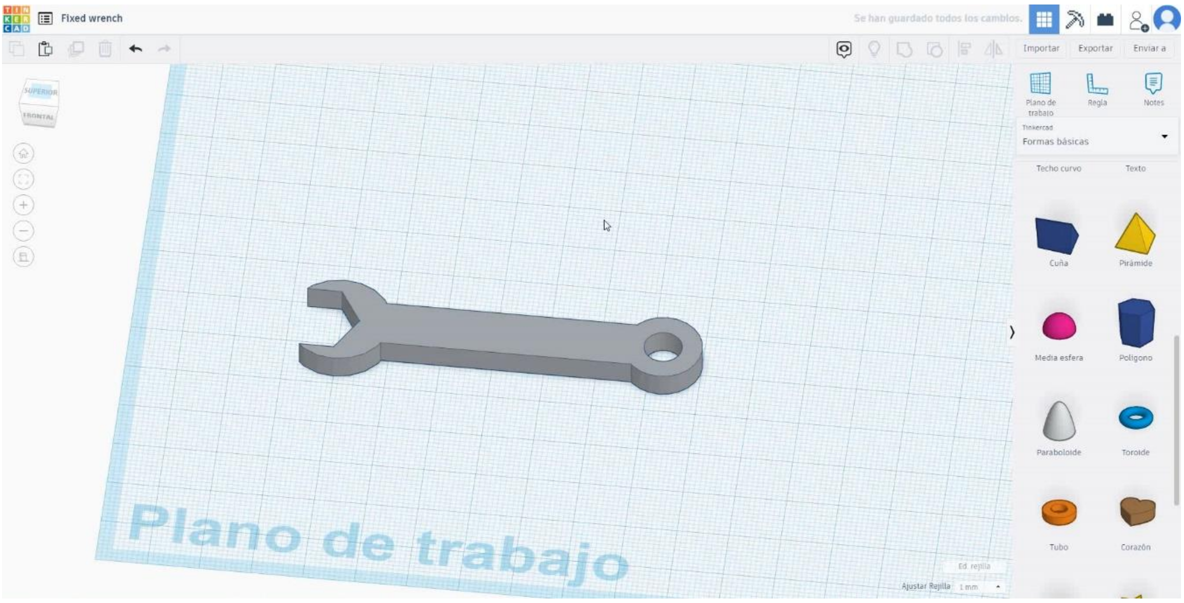




5. Select both objects and press group.



6. Now, the fixed wrench is finished.



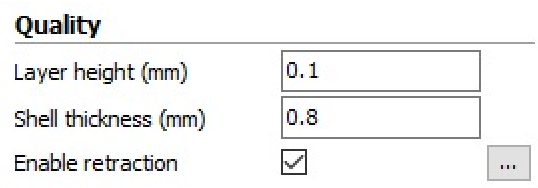
9.3.6.2 Fixed Wrench 3D printing seetings

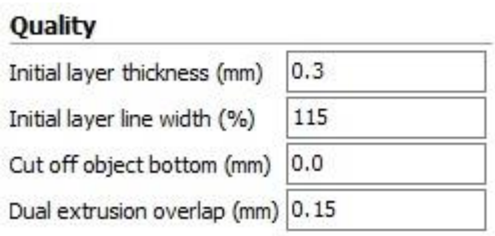
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

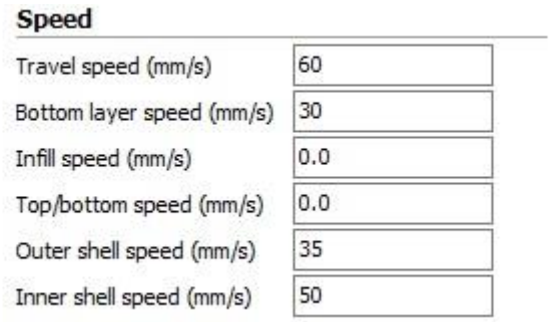
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

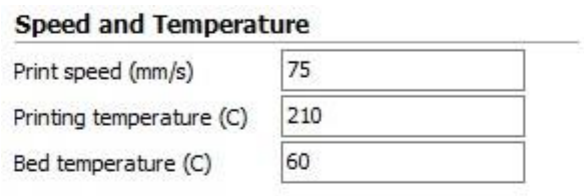




Speed and Temperature

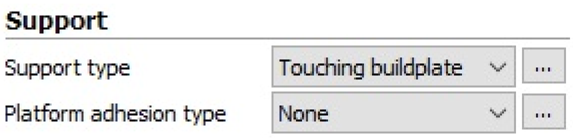
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





<u>Support Type</u>

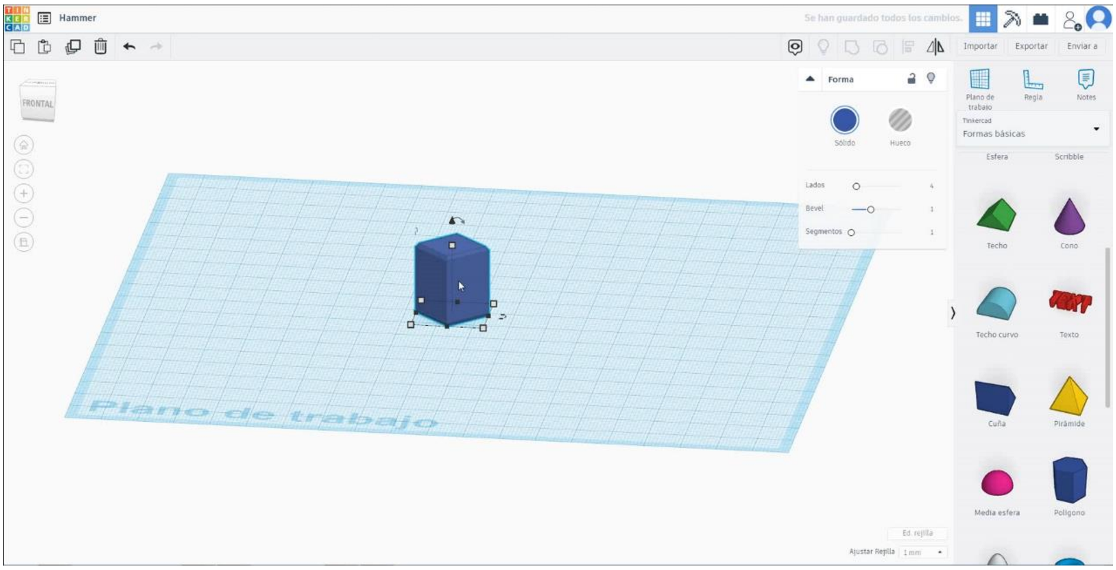
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

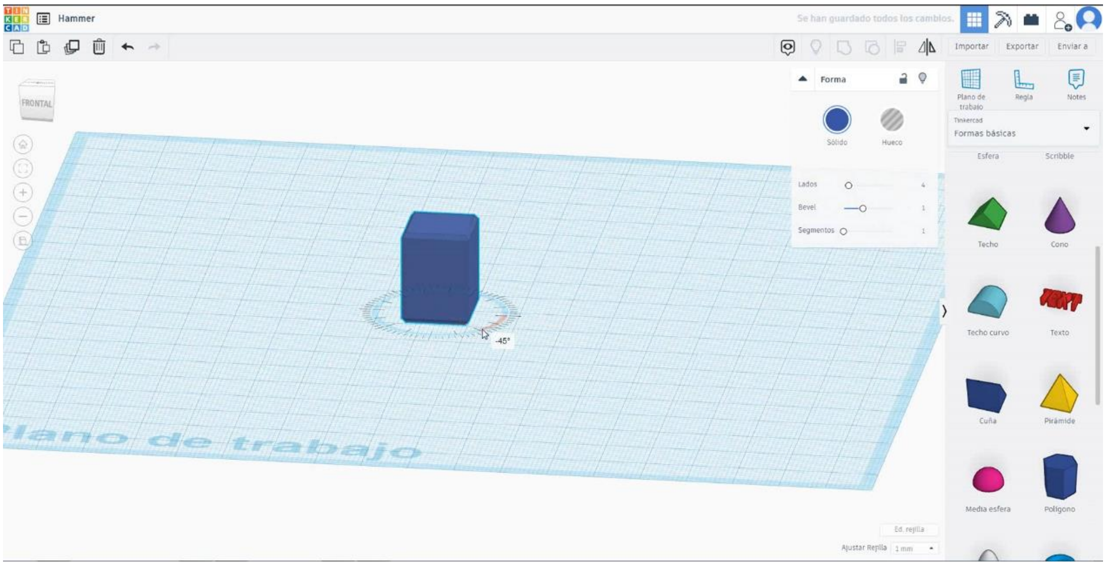


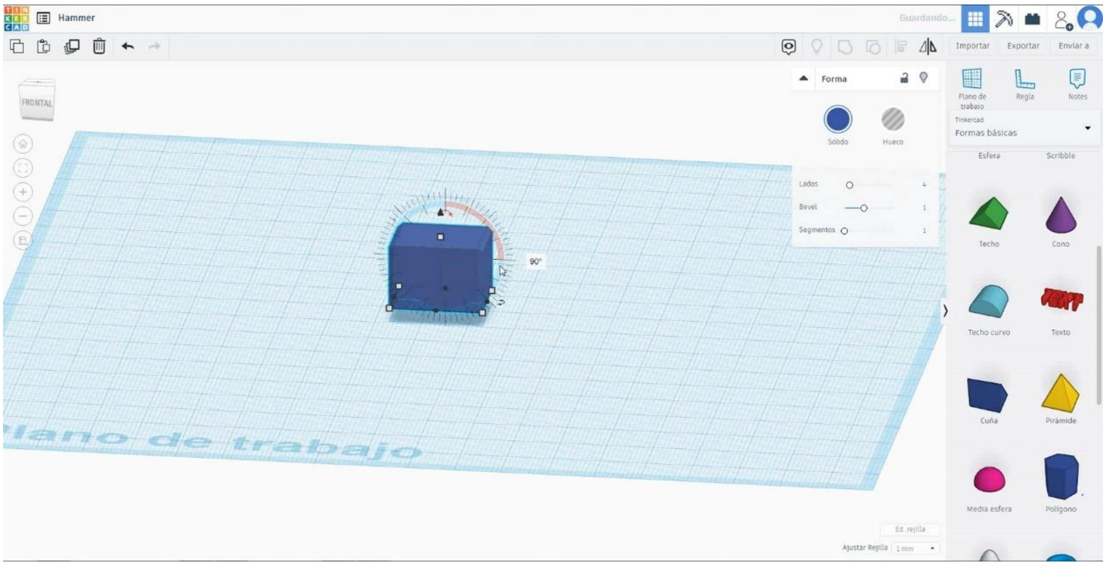
9.3.7 Part 7: Hammer

9.3.7.1 Hammer Design

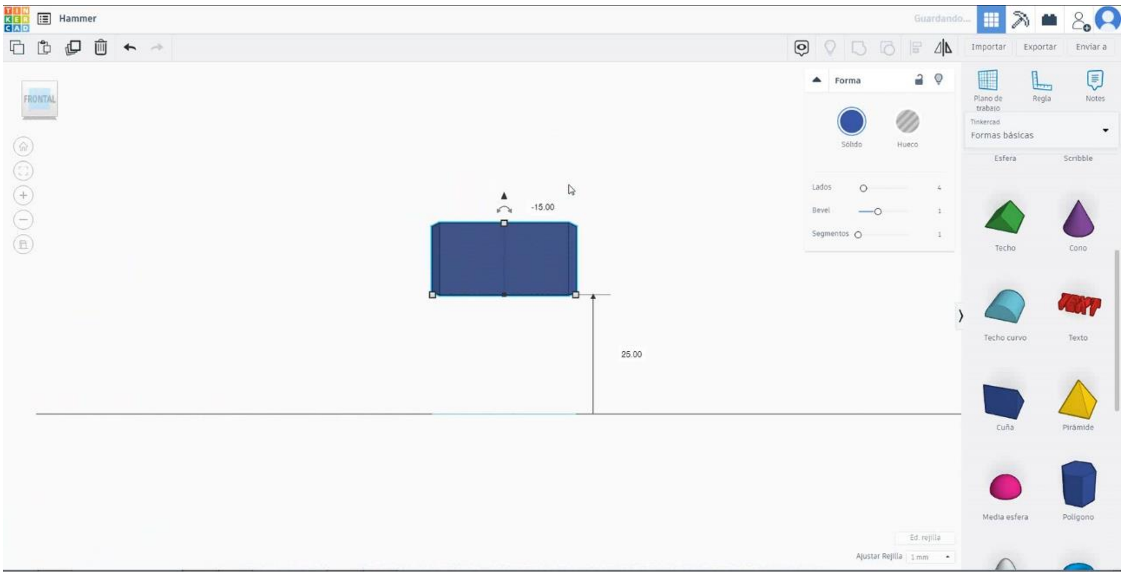
1. Select the polygon shape and change to 4 sides and 1.5mm brevel. Rotate it 45 degrees first and 90 degrees later in the other way.

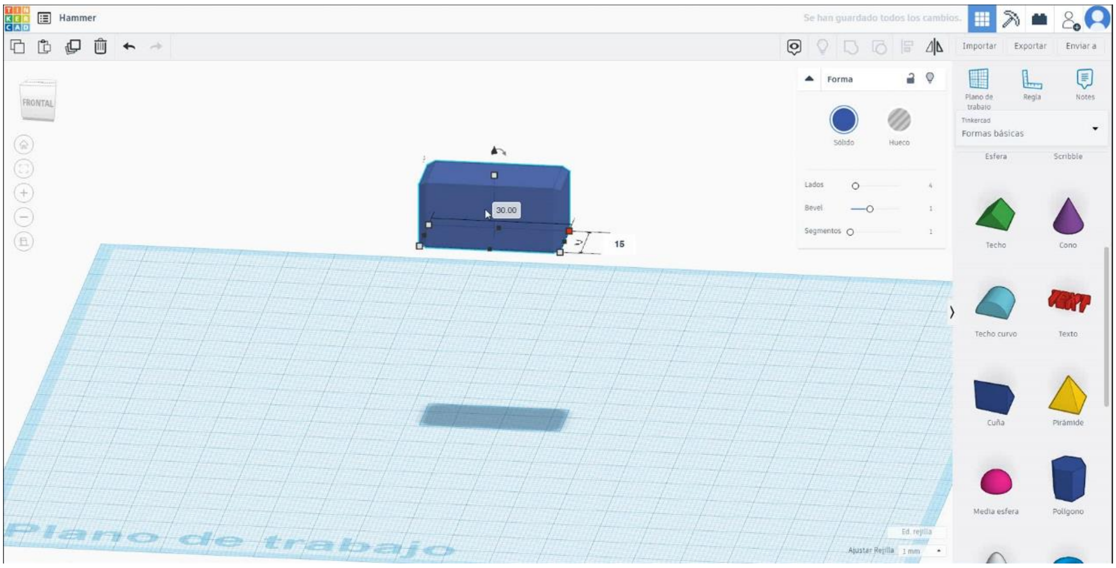




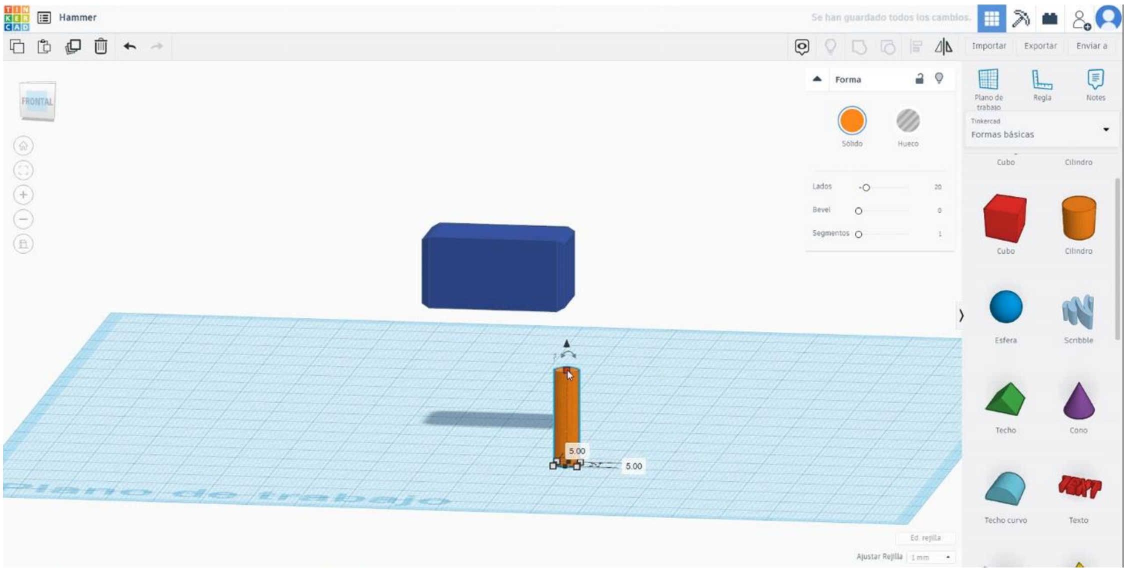


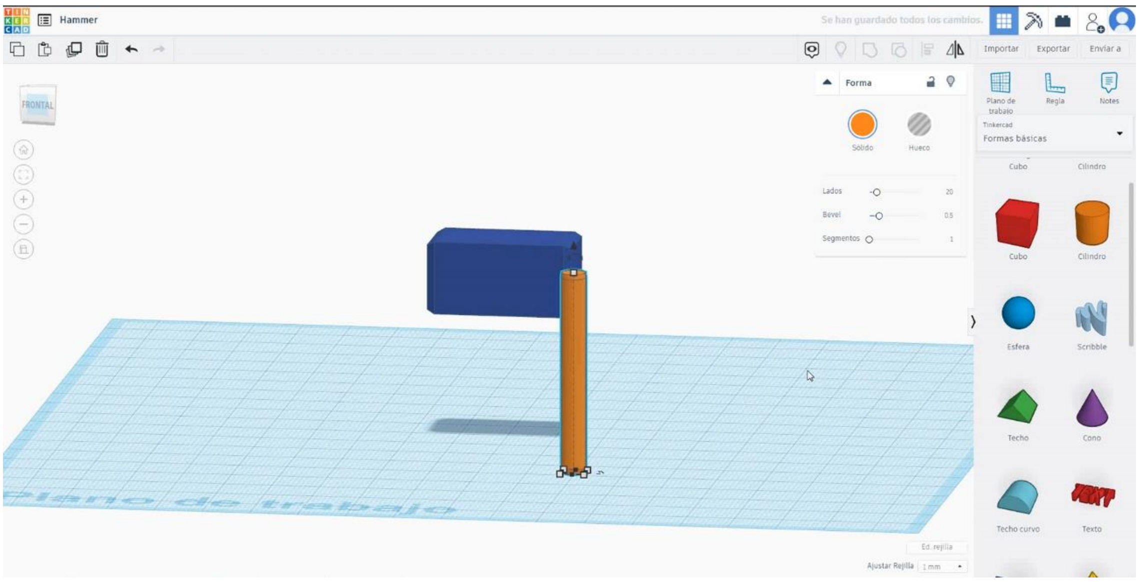
2. Move it to the height of 25 mm. Size the polygon to 30x15x15 mm.





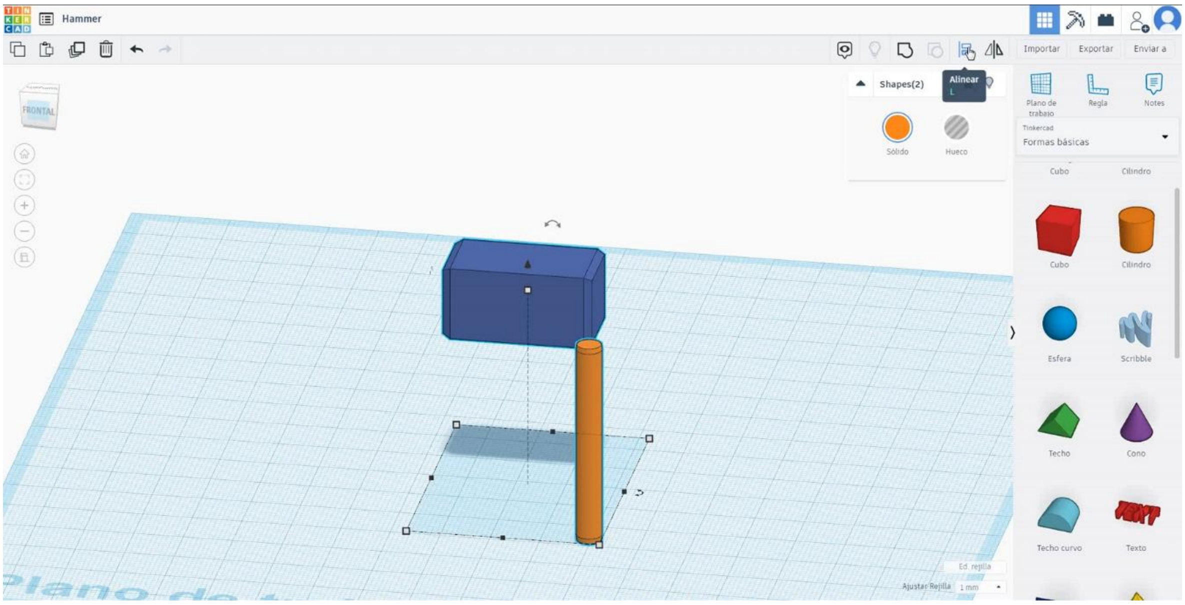
3. To create the handle, select a cylinder and size it to 5x5x42 mm with 0.5 mm brevel

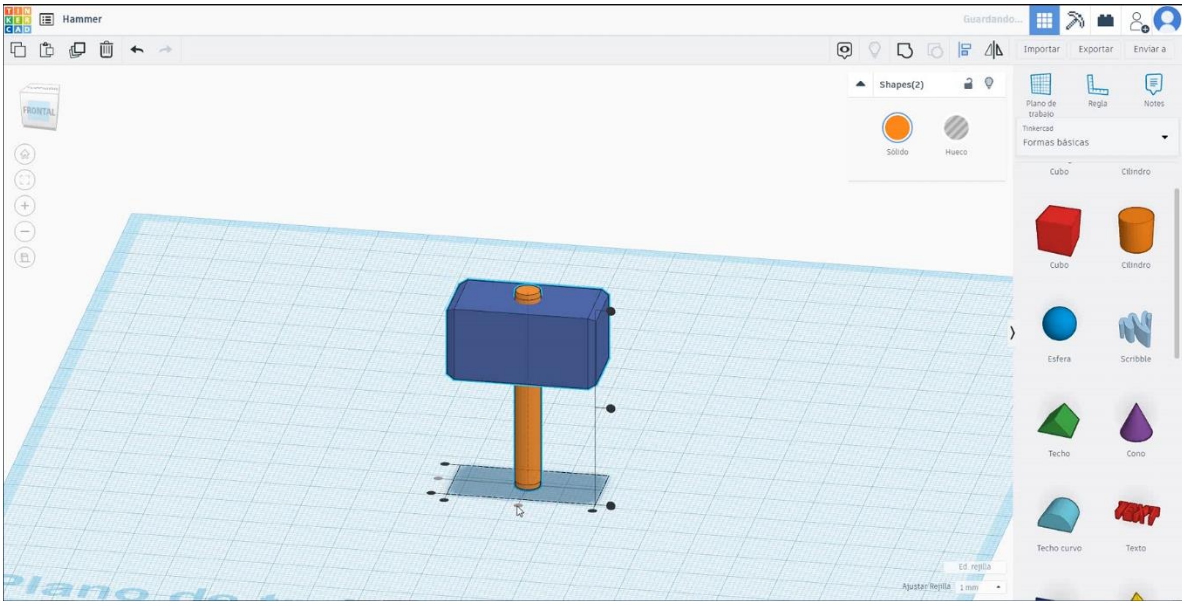




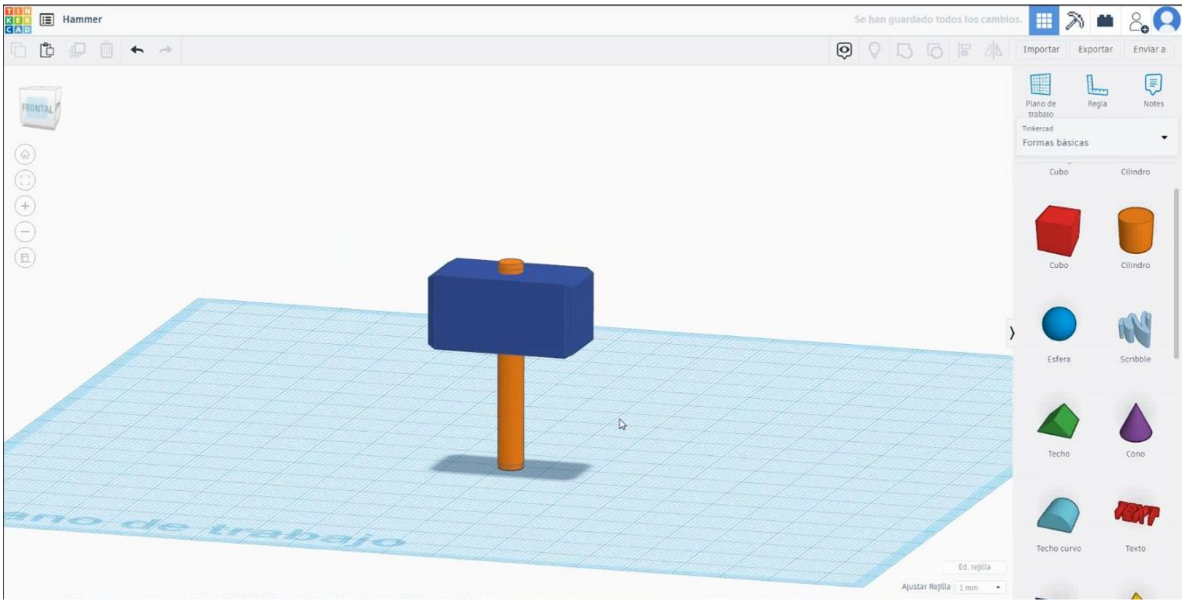
Co-funded by the Erasmus+ Programme of the European Union

4. Select both objects and press the option align and select the central dots in the two directions of work plane.





5. Now the hammer is finished



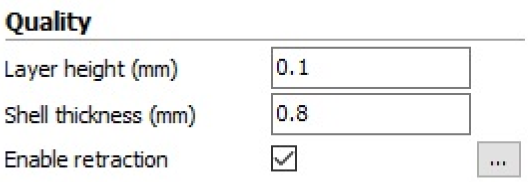
9.3.7.2 Hammer 3D printing seetings

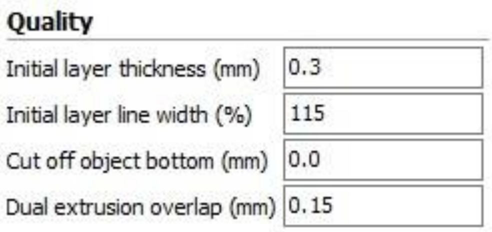
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

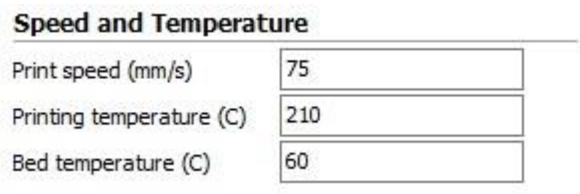


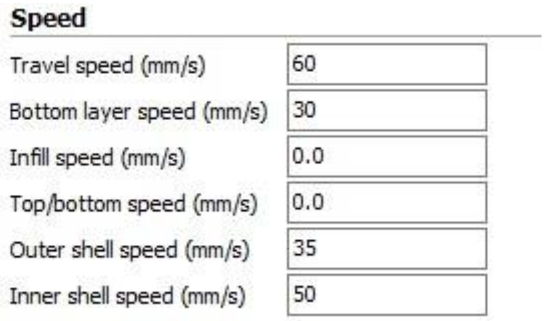


Speed and Temperature

Print Speed - 50 (mm/s)

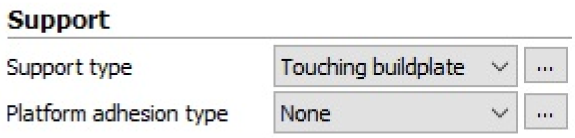
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





<u>Support Type</u>

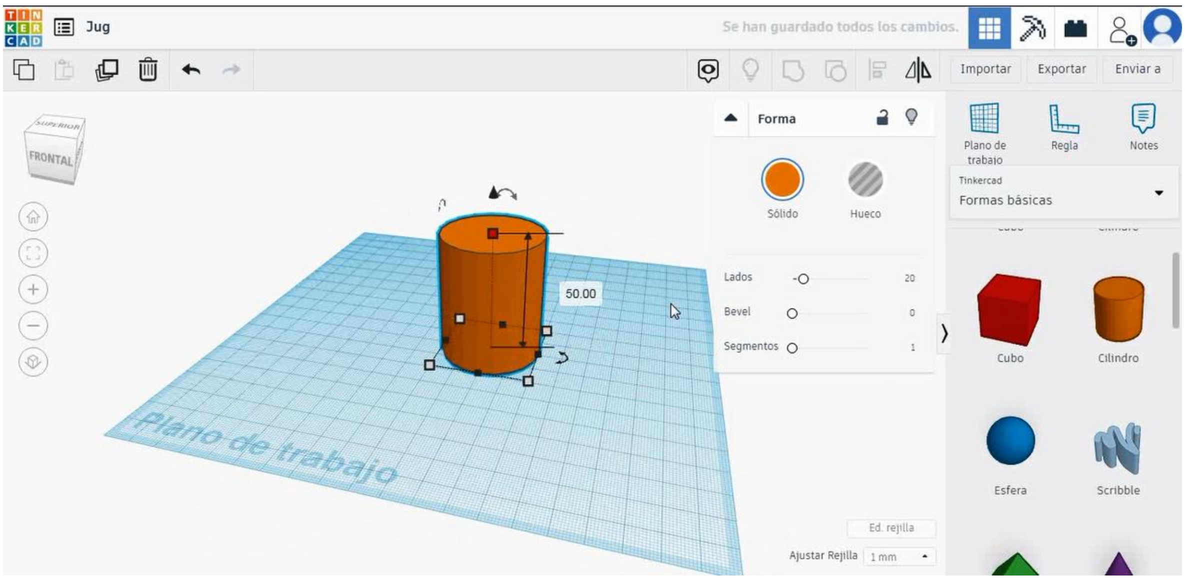
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



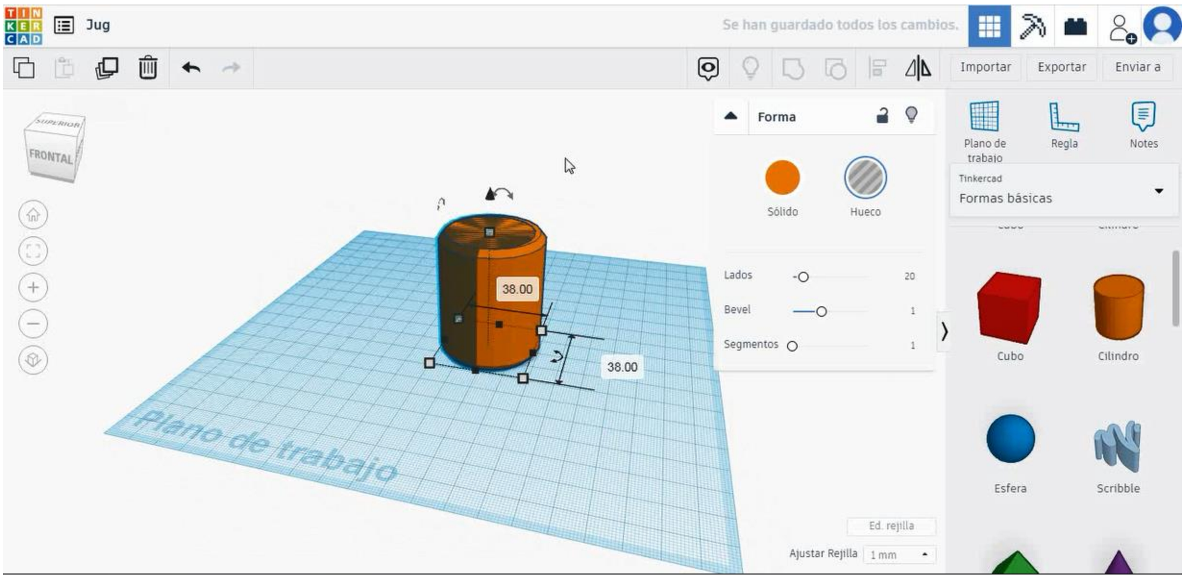
9.3.8 Part 8: Jug

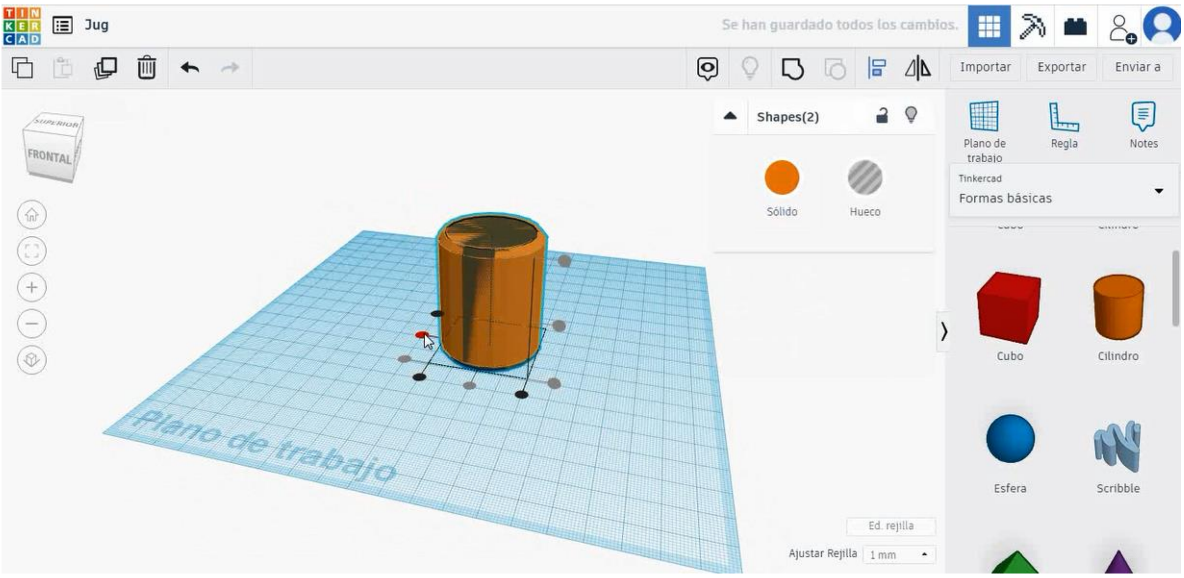
9.3.8.1 Jug Design

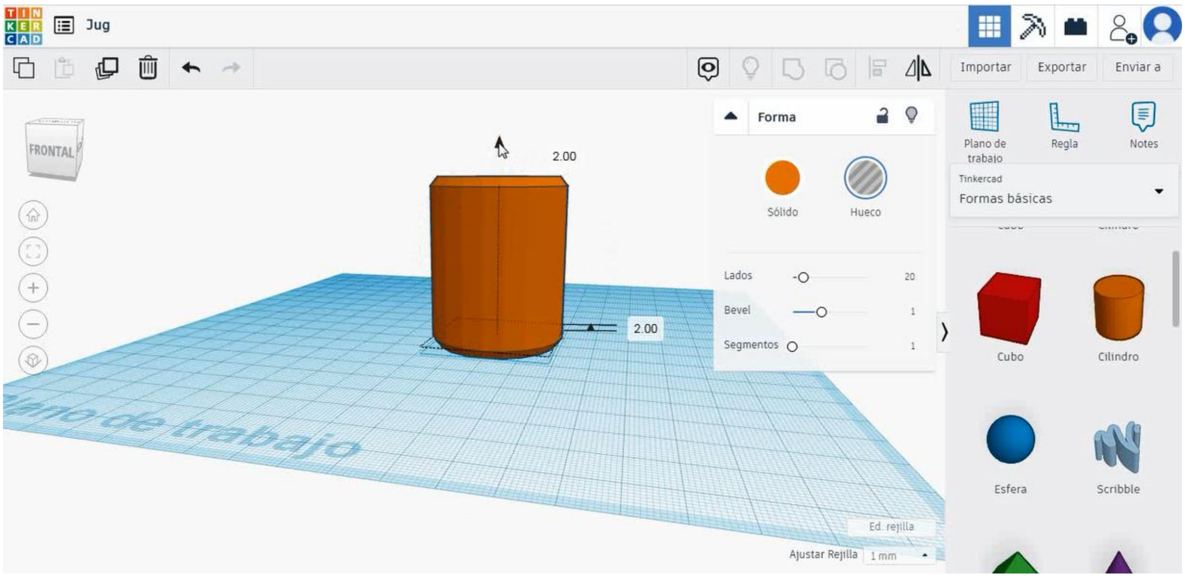
1. Choose the cylinder shape and size it to 40x40x50 mm with 1mm brevel.



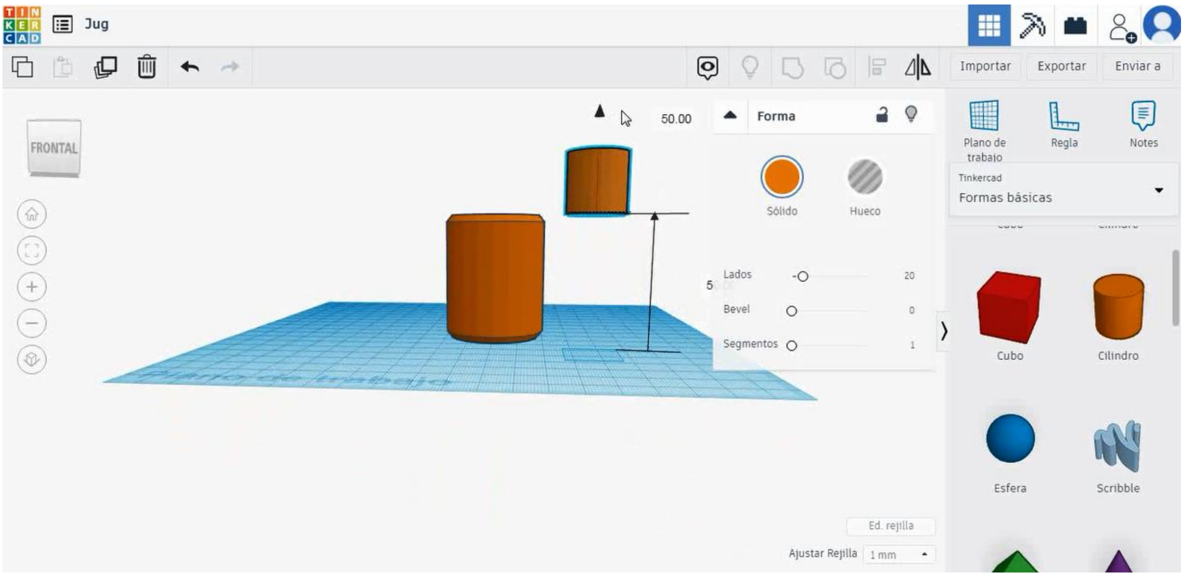
2. Duplicate the cylinder and select hole mode. Size it to 38x38x46 mm, align in the center of the first cylinder and move to the height of

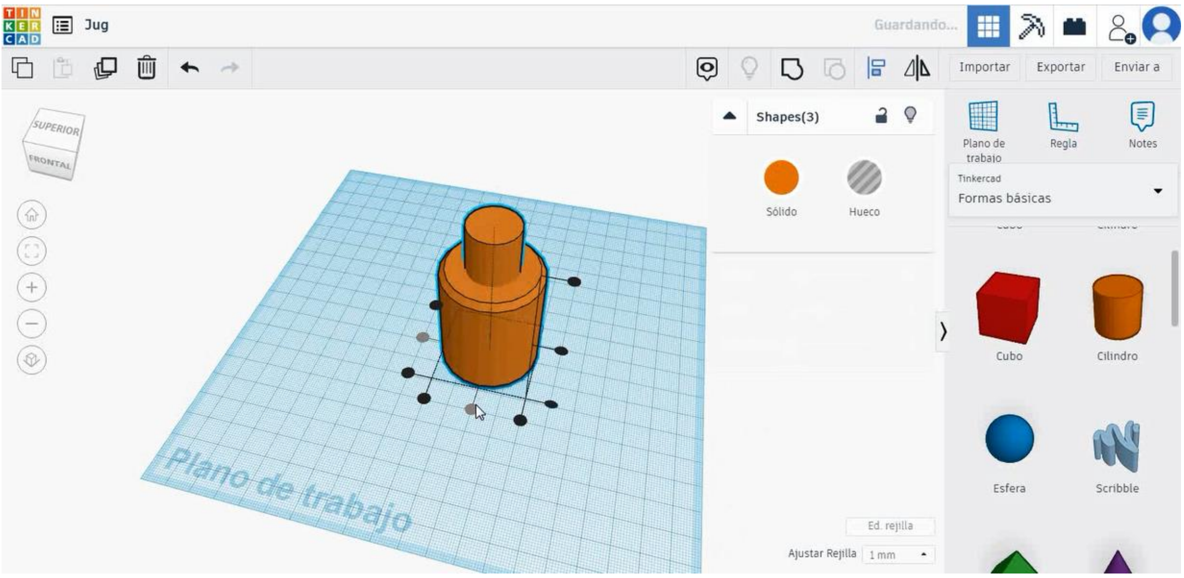

Jug
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
38.00
38.00
Plano de trabajo
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Ed. rejilla
Ajustar Rejilla
1 mm


Jug
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Shapes(2)
Sólido
Hueco
Plano de trabajo
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Ed. rejilla
Ajustar Rejilla
1 mm

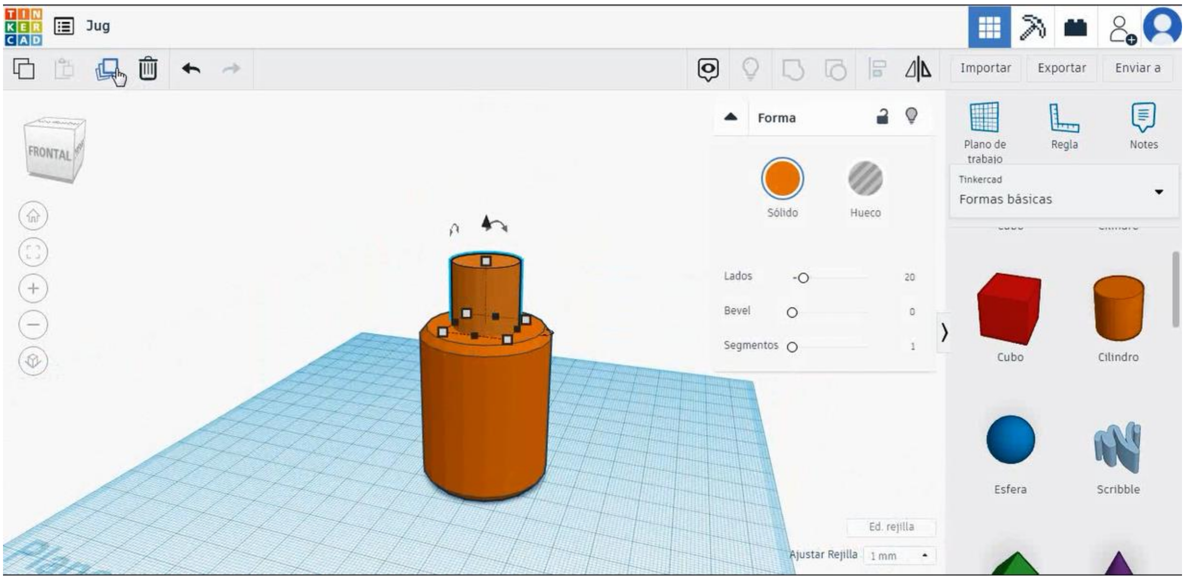

Jug
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
2.00
2.00
Plano de trabajo
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Ed. rejilla
Ajustar Rejilla
1 mm

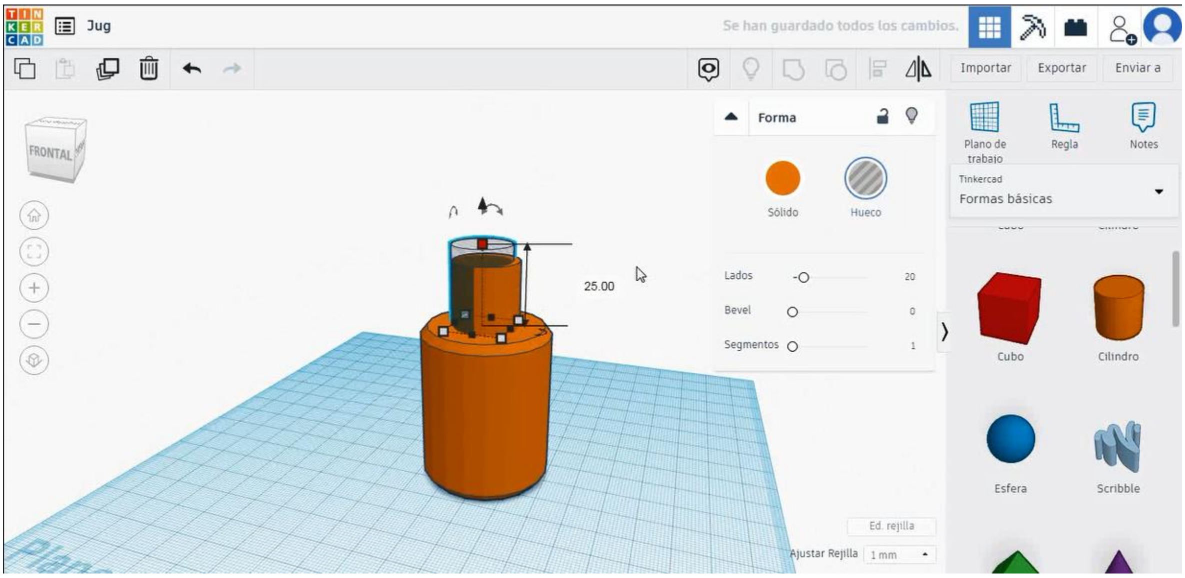
3. Select a new cylinder and move it to the height of 50 mm. Align in the center of the first cylinder.

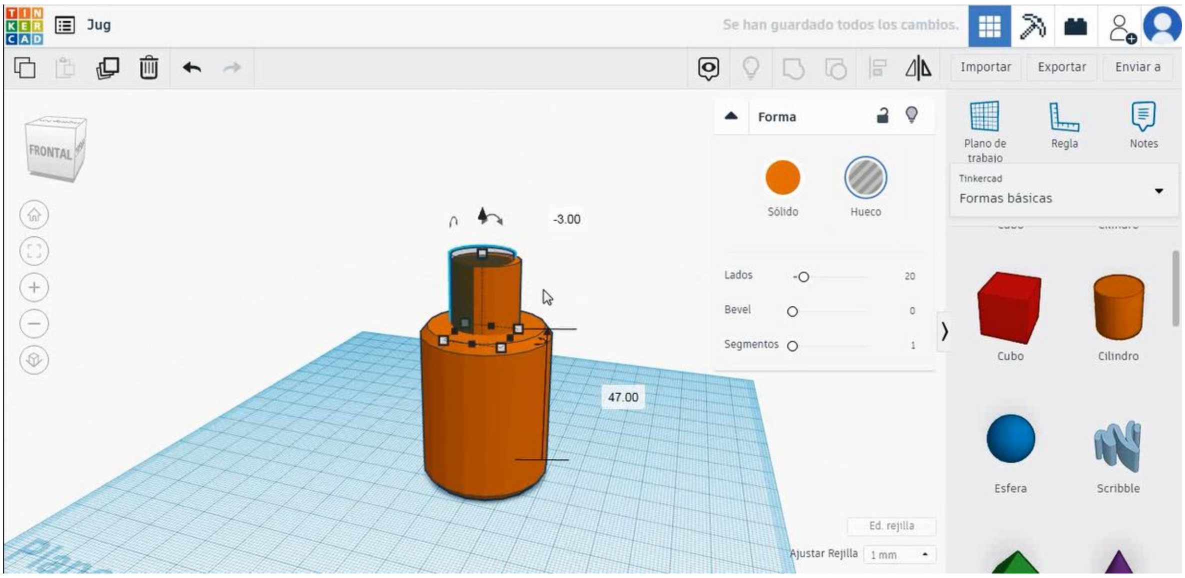




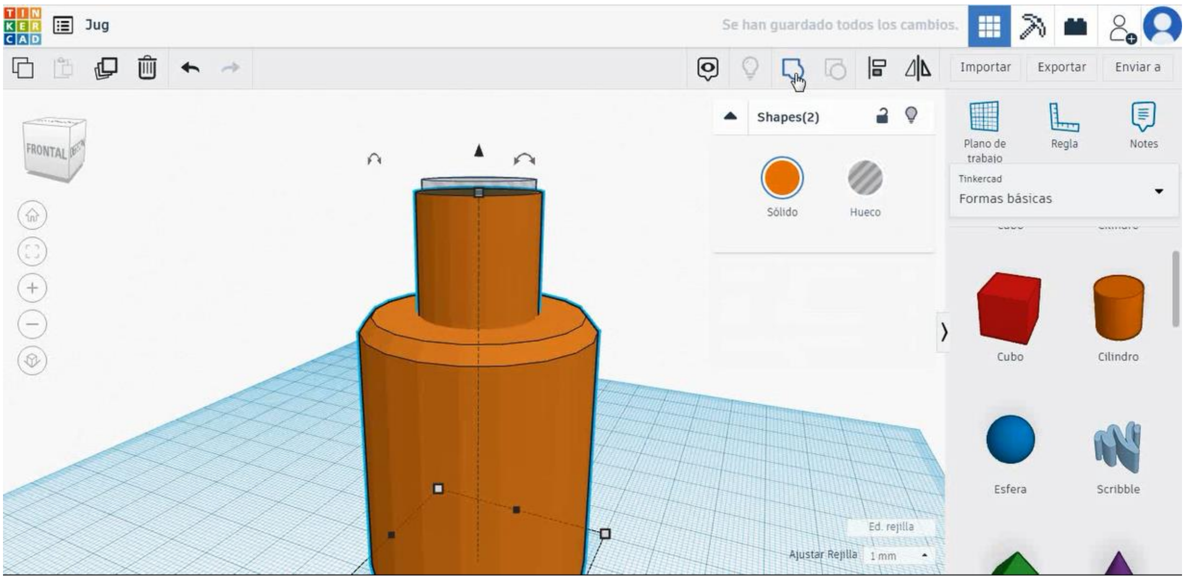
4. Duplicate the second cylinder, select hole mode and size it to 18x18x25 mm. Then move it to the height of 47 mm and align in the center.

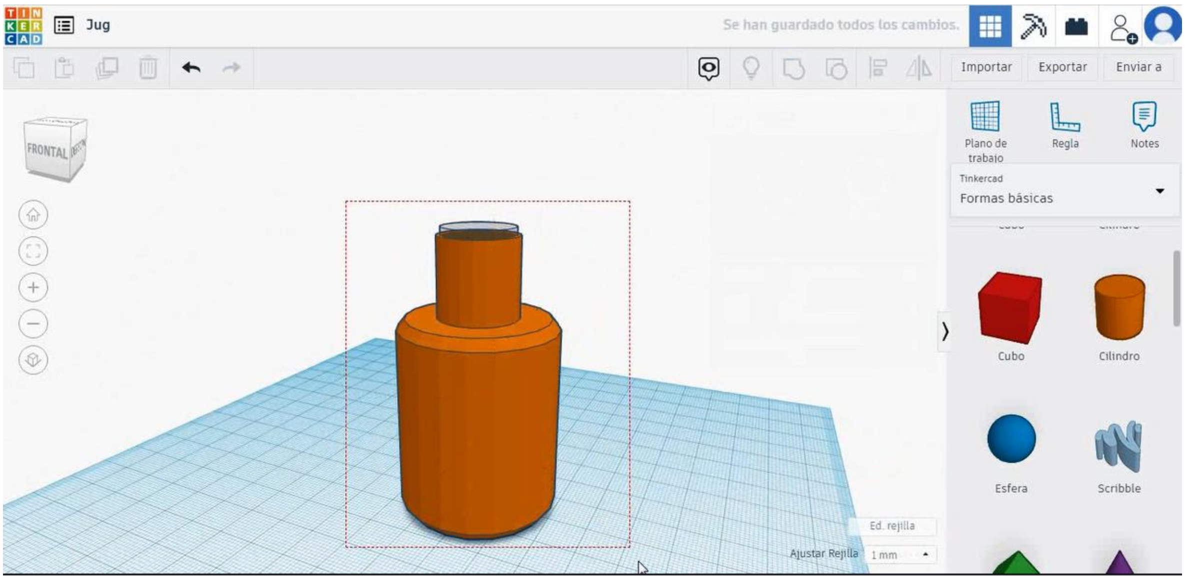

Jug
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Ed. rejilla
Ajustar Rejilla
1 mm


Jug
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
25.00
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Ed. rejilla
Ajustar Rejilla
1 mm

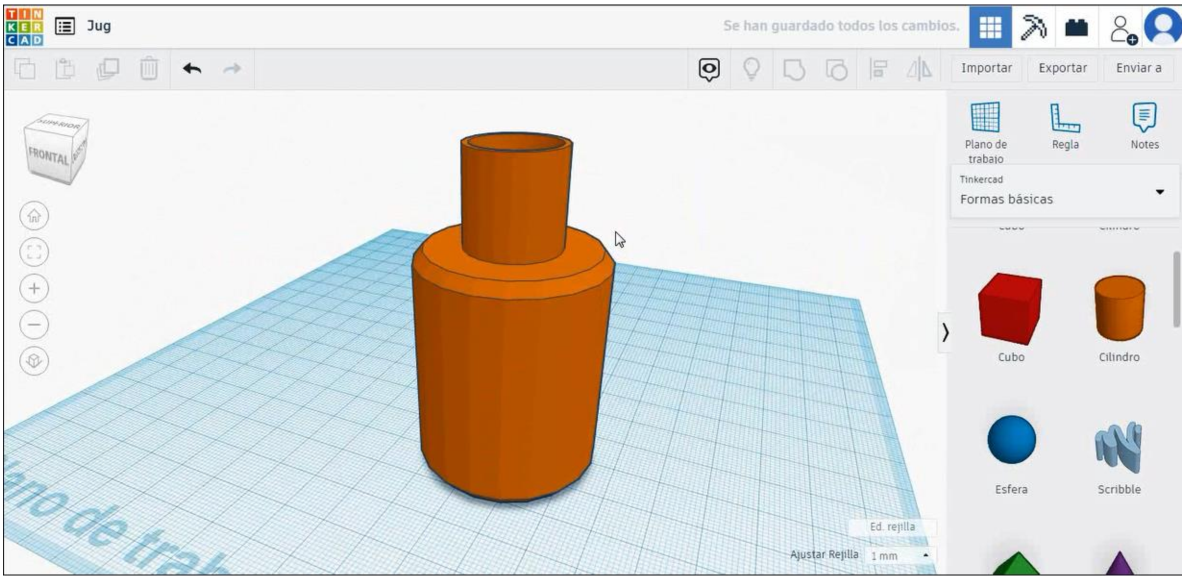

Jug
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
-3.00
47.00
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Ed. rejilla
Ajustar Rejilla
1 mm

5. Select the two cylinders in solid mode and press group, then select all the objects and press group again.





6. Now, the jug is finished.



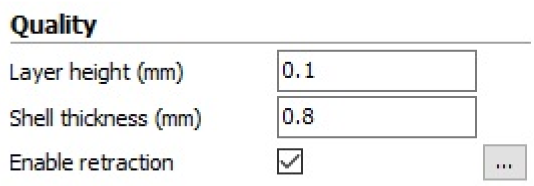
9.3.8.2 Jug 3D printing seetings

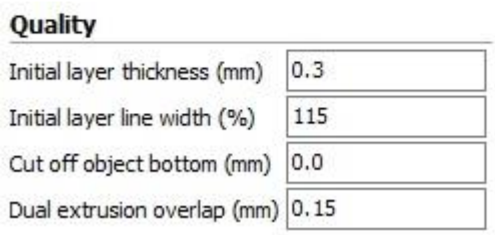
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

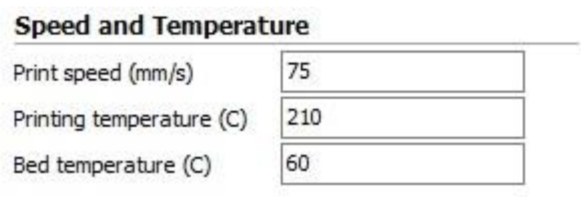


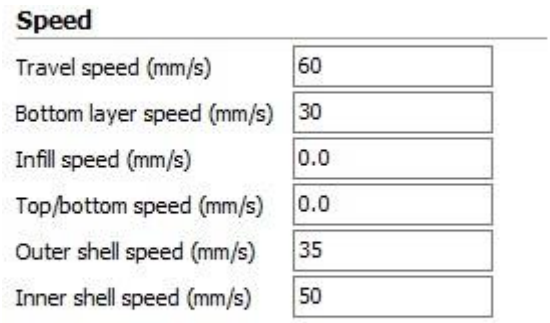


Speed and Temperature

Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)

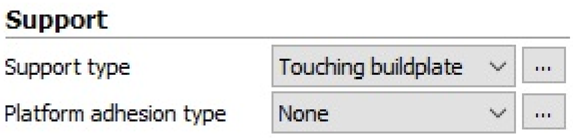
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





<u>Support Type</u>

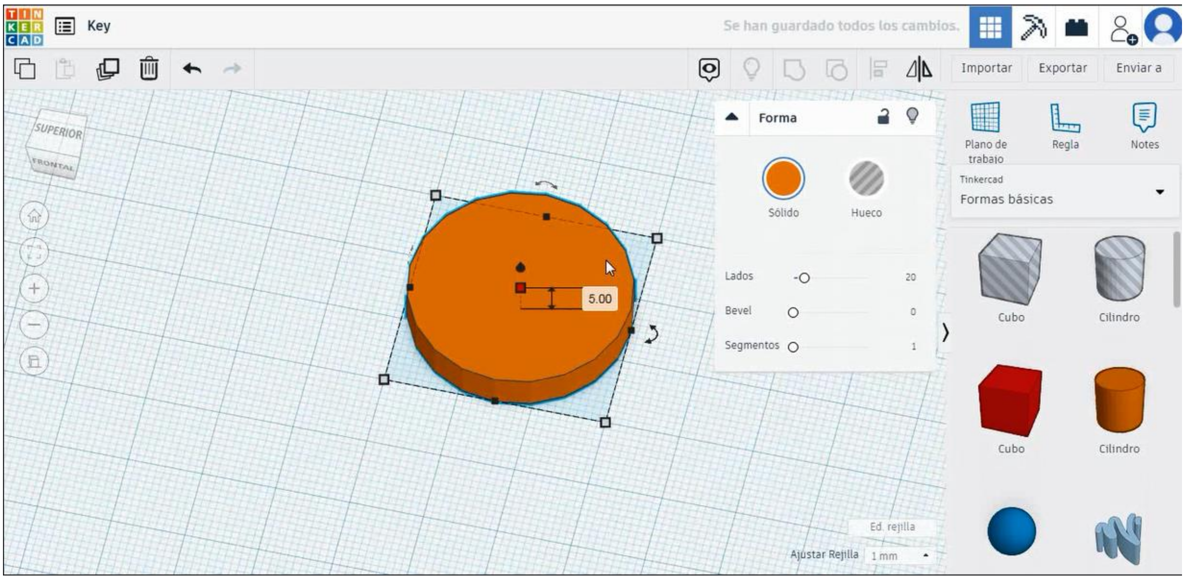
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



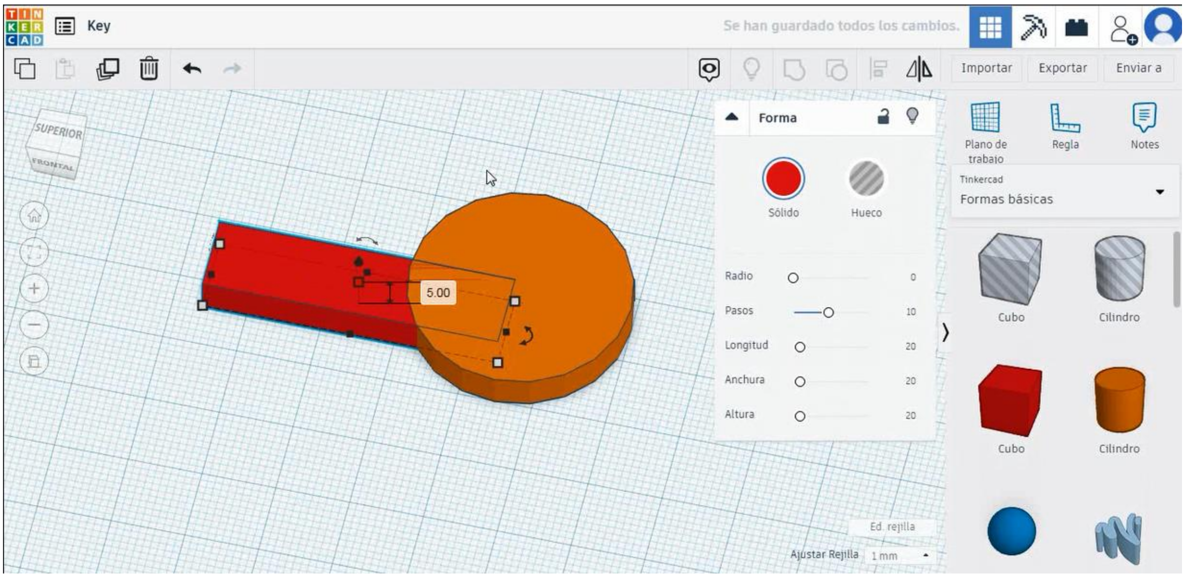
9.3.9 Part 9: Key

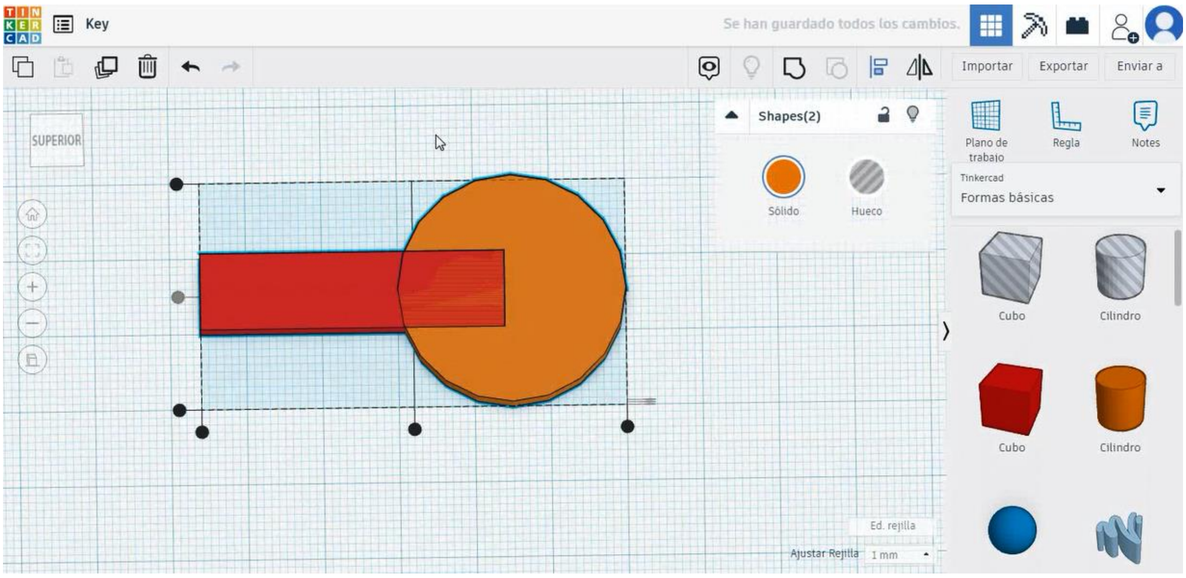
9.3.9.1 Key Design

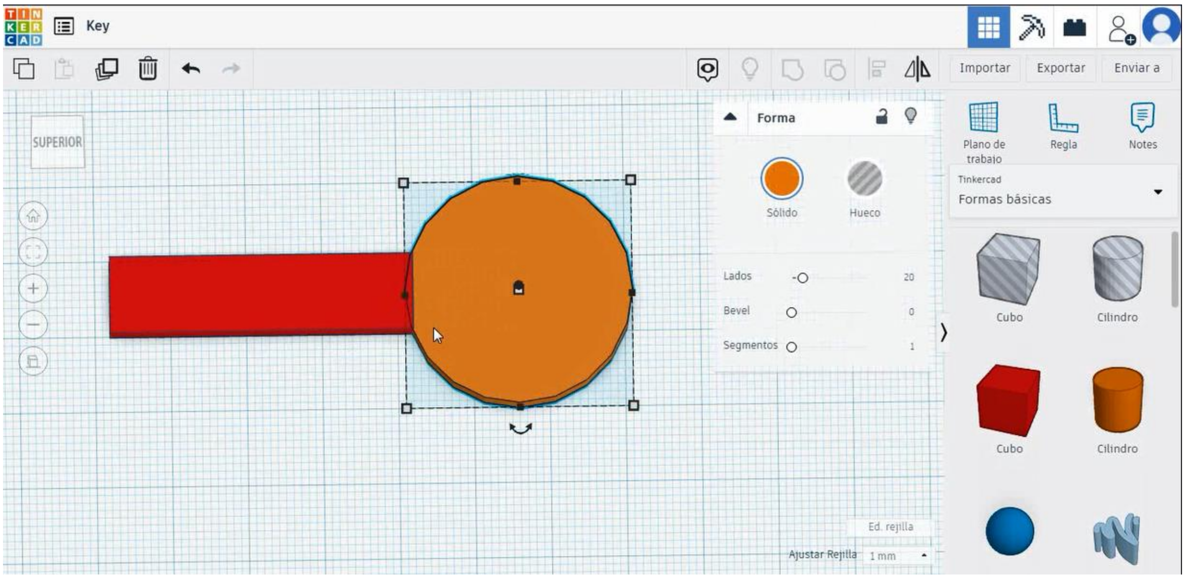
1. Choose the cylinder shape and size it to 30x30x5 mm.



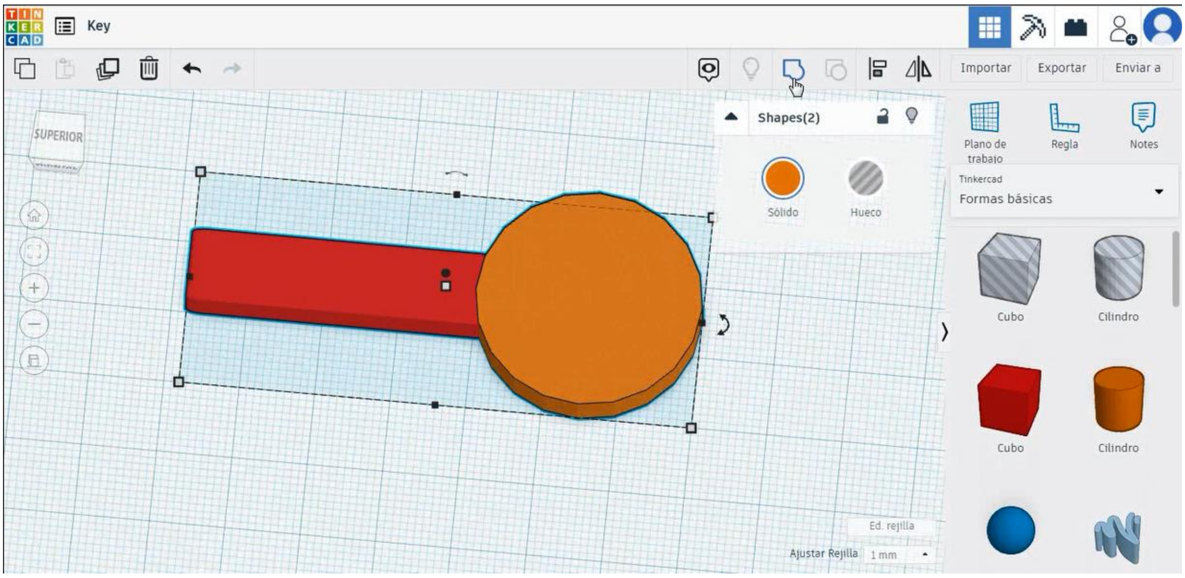
2. Choose the cube shape and size it to 40x10x5 mm with radius 1 mm and align it on Y axis with the cylinder. Then move the cube to put in the same position of the picture below.



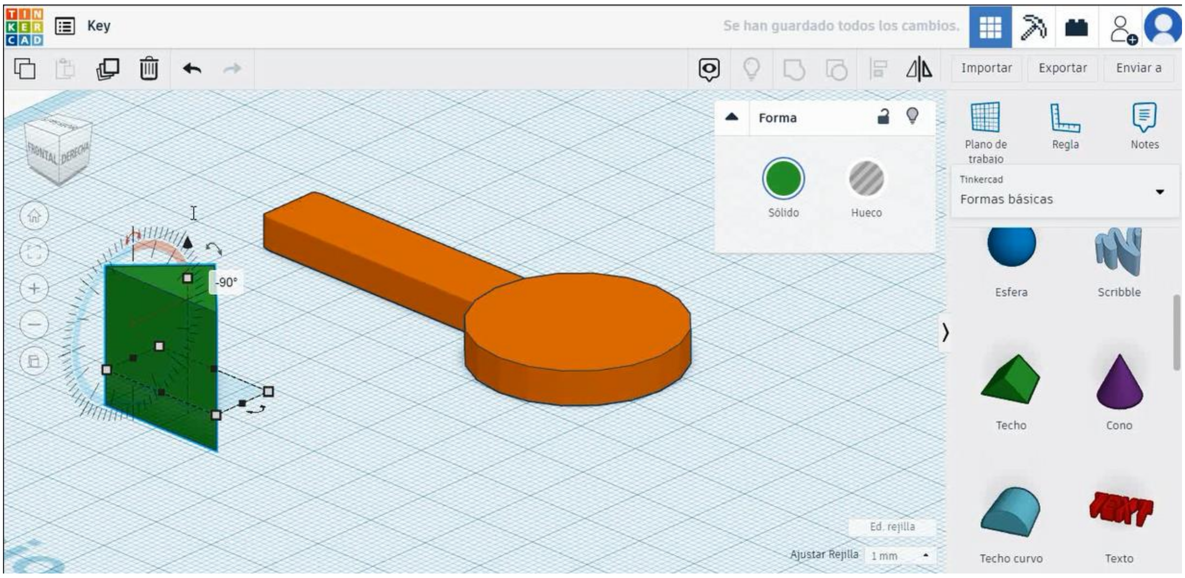


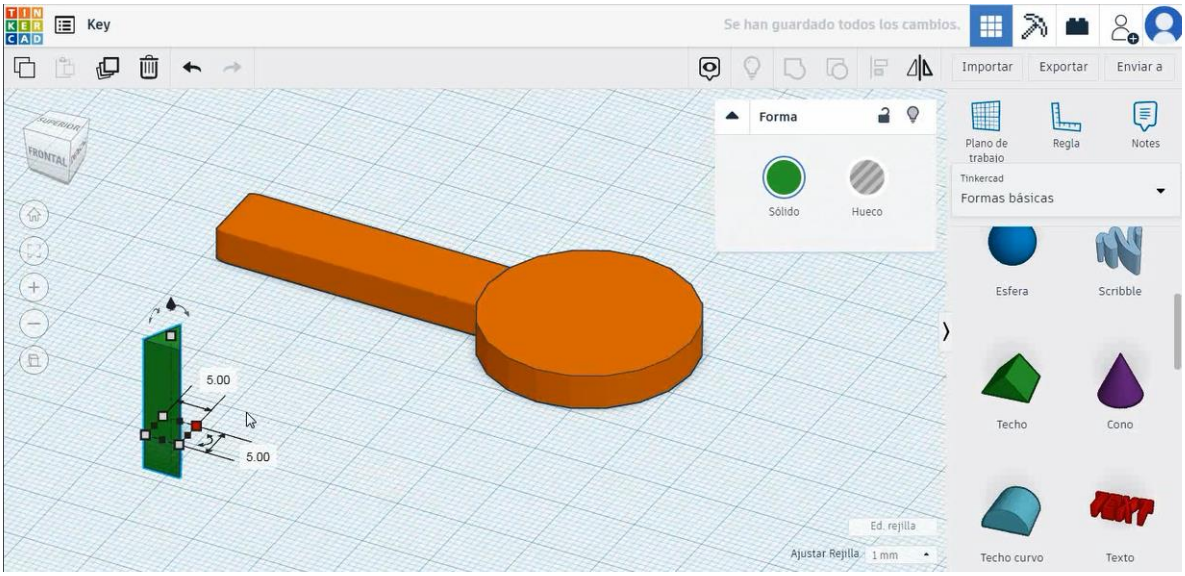


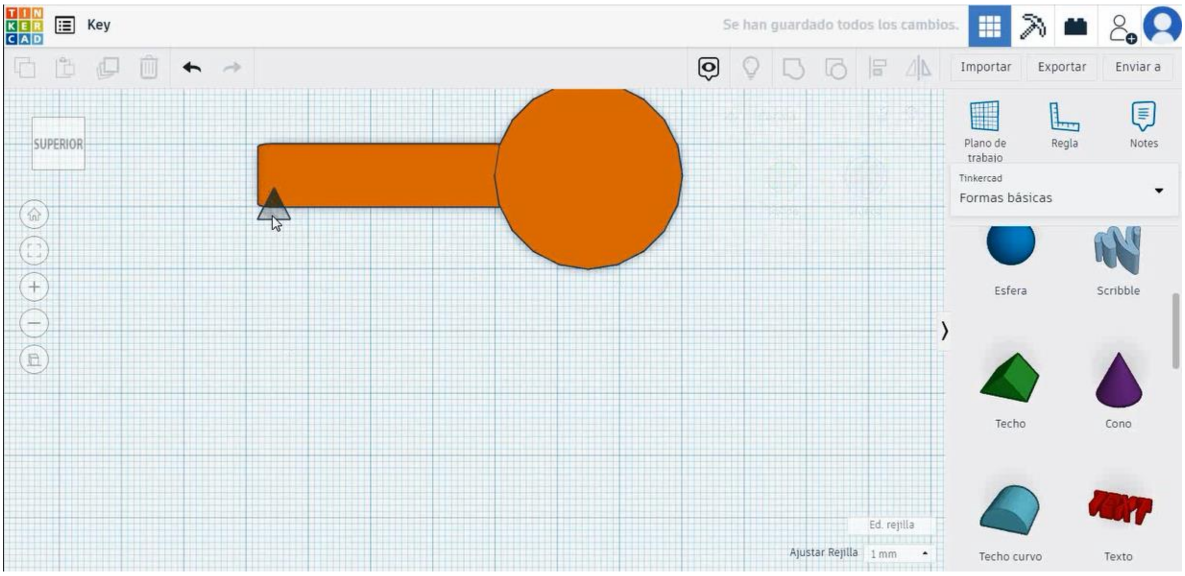
3. Select both objects and press group to merge them.

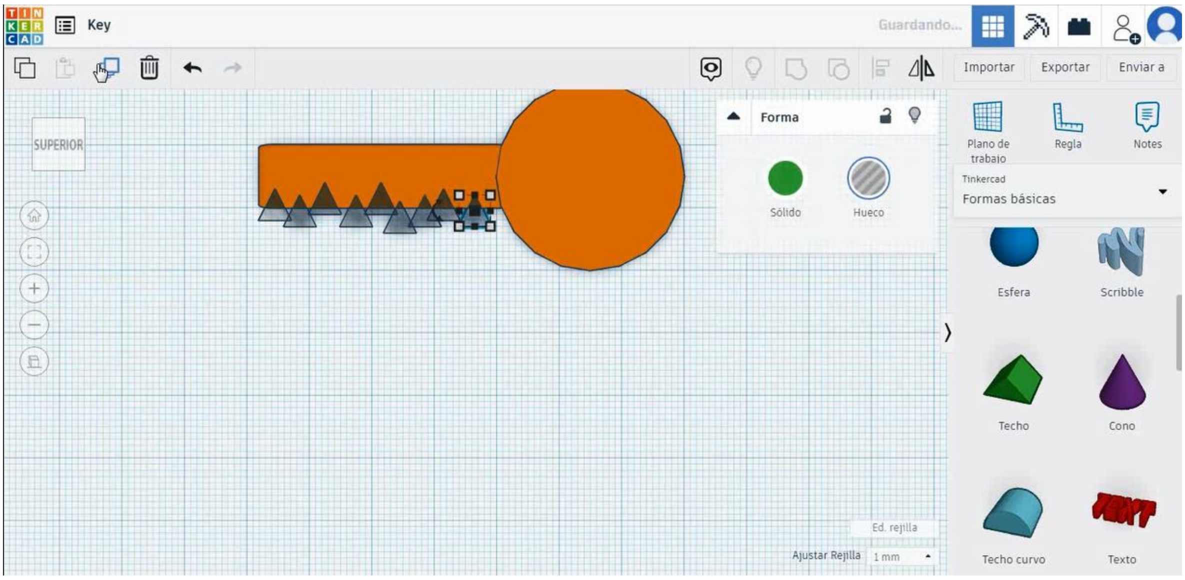


4. Choose the roof shape in hole mode size it to 5x5x20 mm and move it 90 degrees. Then move it to the left part of the key and duplicate it moving it up and down to create the different clefts.

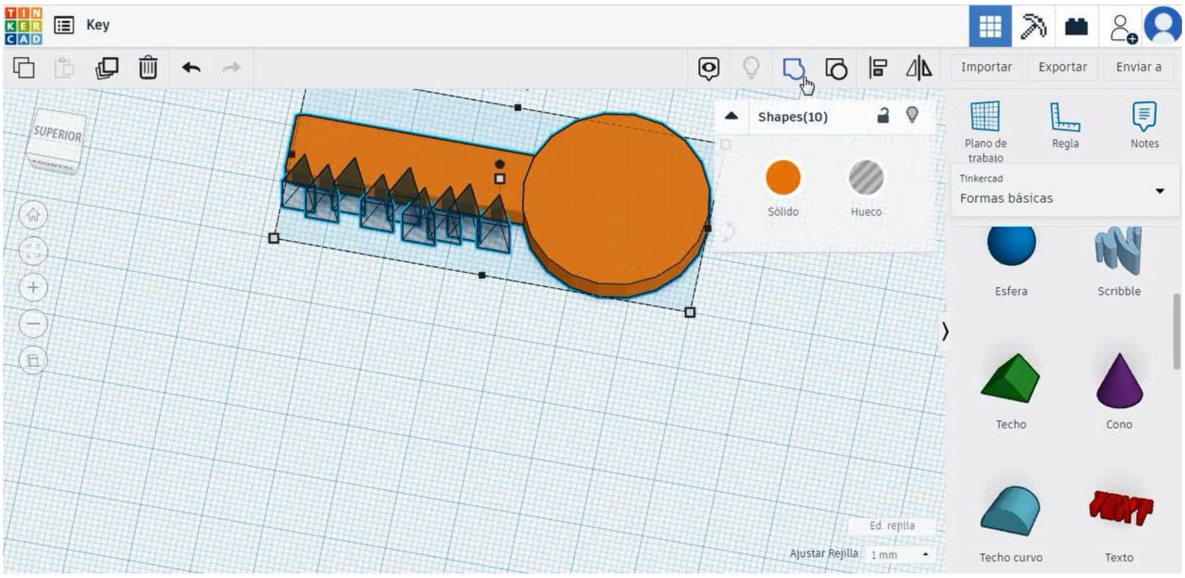


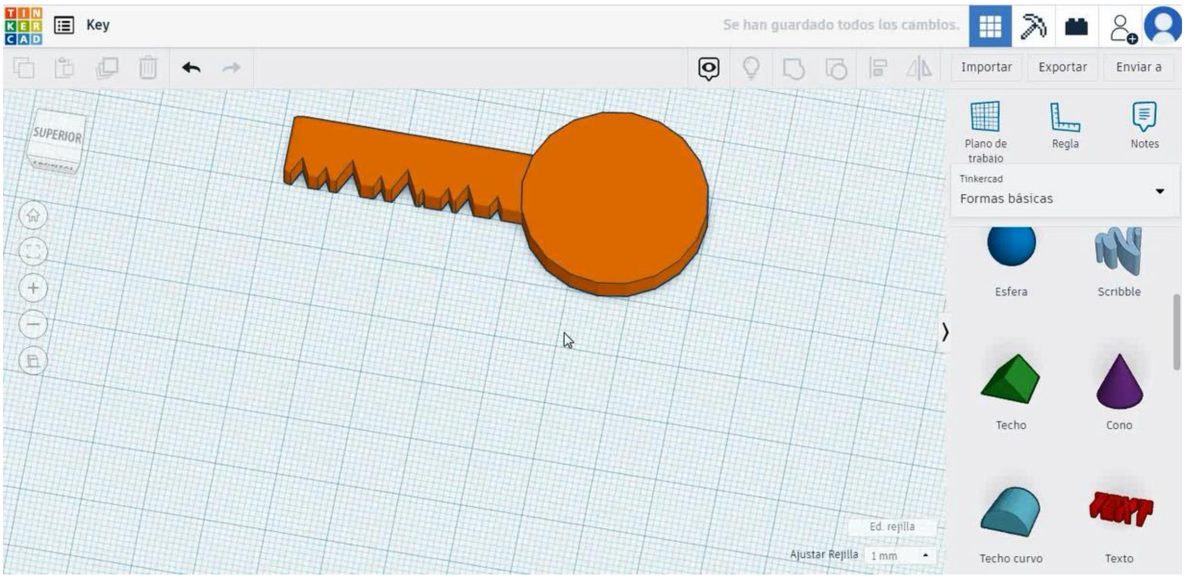

Key
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Esfera
Scribble
Techo
Cono
Techo curvo
Texto
5.00
5.00
Ed. rejilla
Ajustar Rejilla
1 mm


Key
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
SUPERIOR
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Esfera
Scribble
Techo
Cono
Techo curvo
Texto
Ed. rejilla
Ajustar Rejilla
1 mm

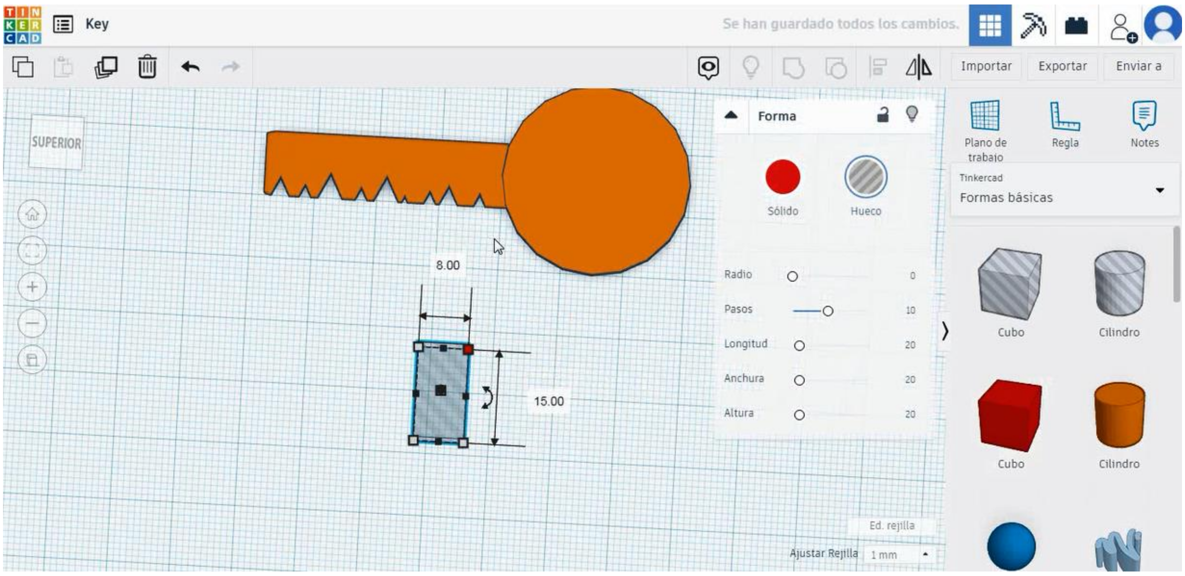

Key
Guardando...
Importar
Exportar
Enviar a
SUPERIOR
Forma
Sólido
Hueco
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Esfera
Scribble
Techo
Cono
Techo curvo
Texto
Ed. rejilla
Ajustar Rejilla
1 mm

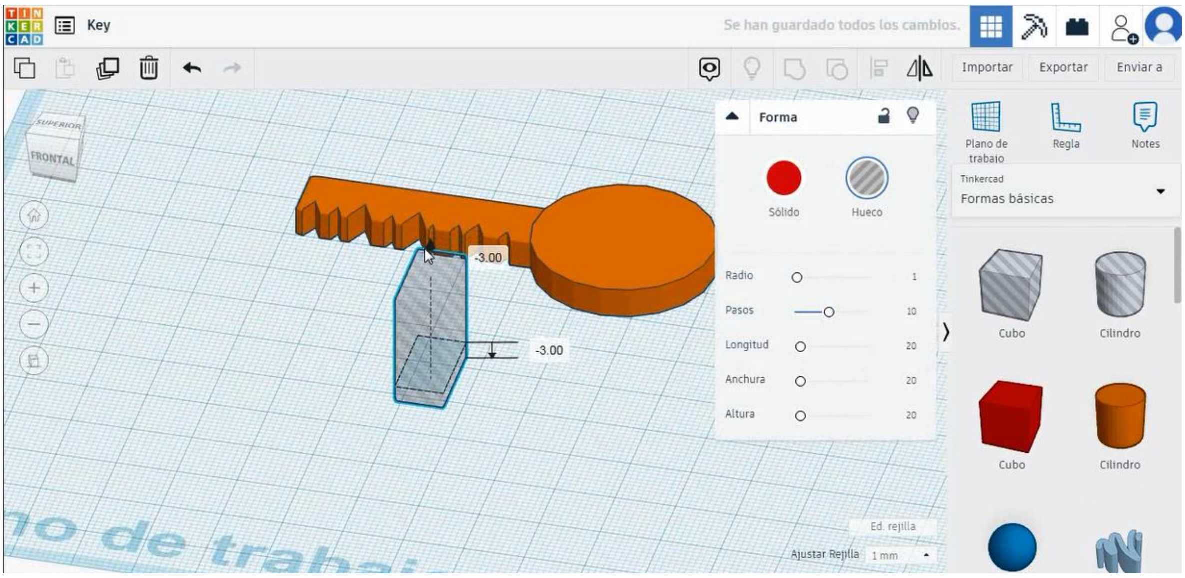
5. Select all the objects and press align to remove the roof shape.

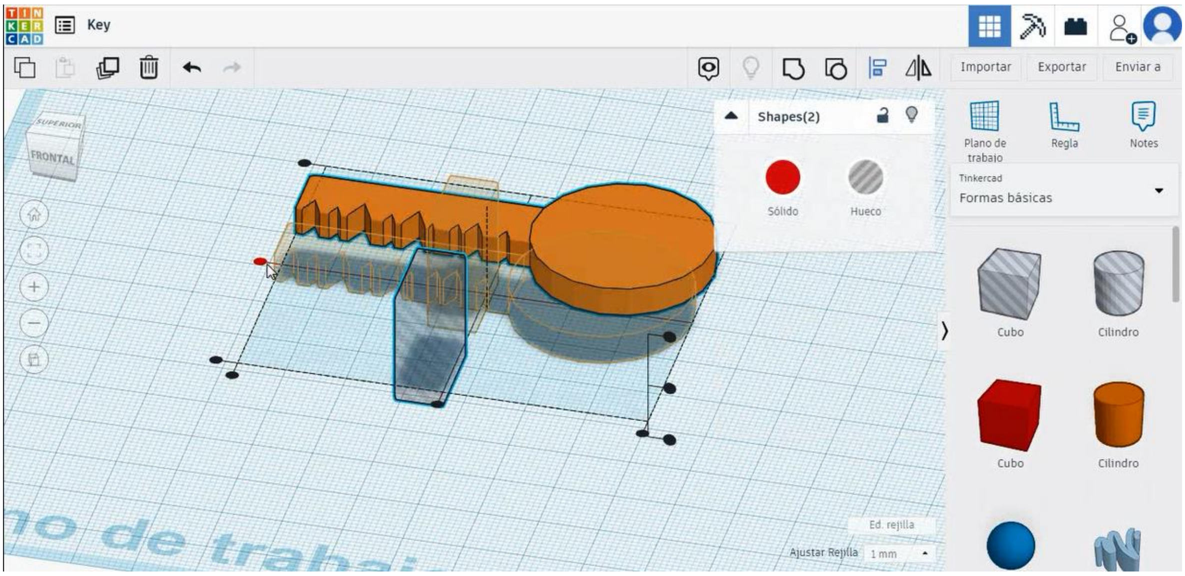


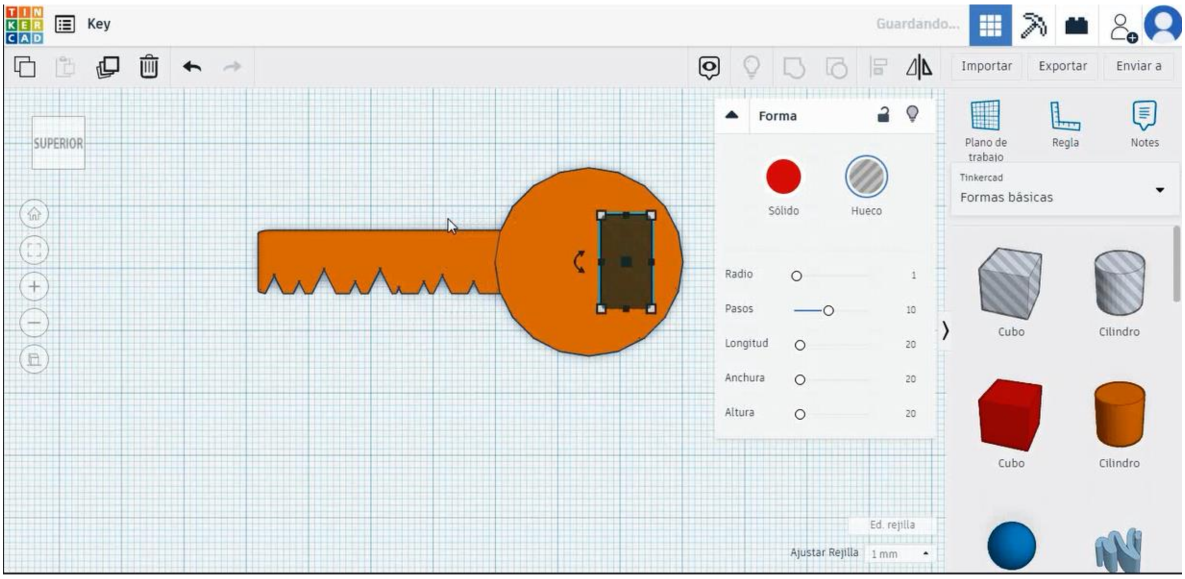


6. Choose the cube shape in hole mode and size it to 8x15x5 mm with radius 1 mm and move it to the height of -3 mm. Then align in the Y axis and move it to the cylindrical part of the key. Select both objects and press group.

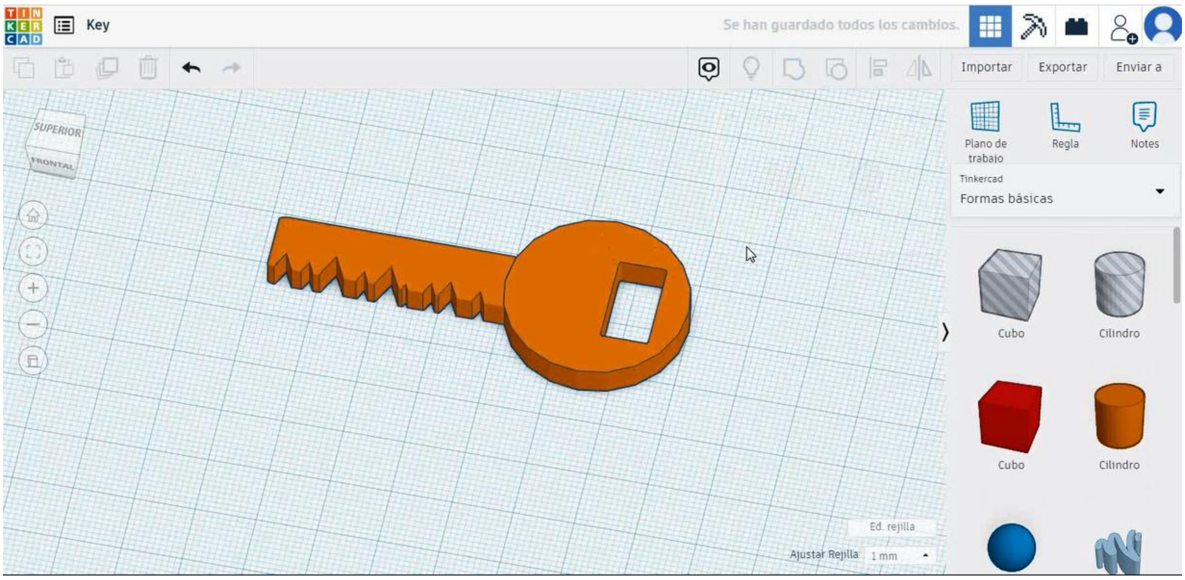

Key
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
SUPERIOR
8.00
15.00
Forma
Sólido
Hueco
Radio
Pasos
Longitud
Anchura
Altura
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Ed. rejilla
Ajustar Rejilla
1 mm


Key
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
SUPERIOR
FRONTAL
-3.00
-3.00
Forma
Sólido
Hueco
Radio
Pasos
Longitud
Anchura
Altura
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Ed. rejilla
Ajustar Rejilla
1 mm


Key
Importar
Exportar
Enviar a
SUPERIOR
FRONTAL
Shapes(2)
Sólido
Hueco
Plano de trabajo
Regla
Notes
Tinkercad
Formas básicas
Cubo
Cilindro
Ed. rejilla
Ajustar Rejilla
1 mm



7. Now, the key is finished.



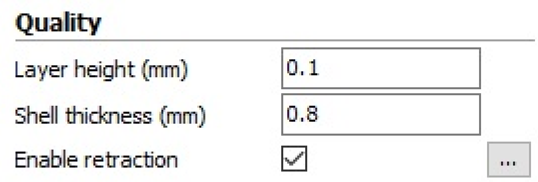
9.3.9.2 Key 3D printing seetings

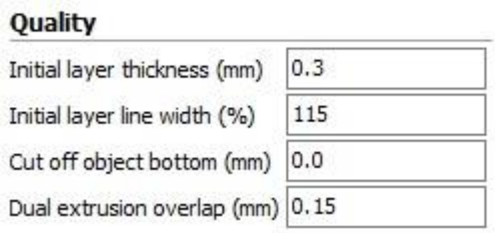
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

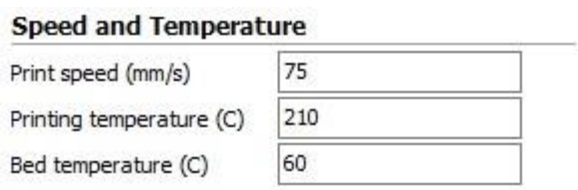
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

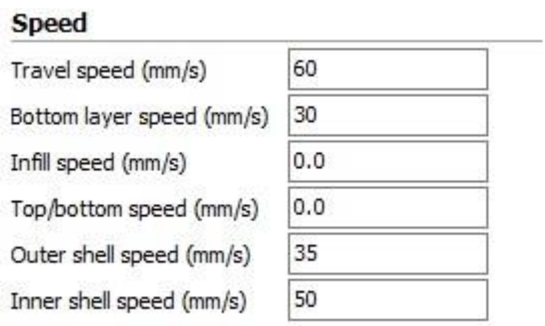




Speed and Temperature

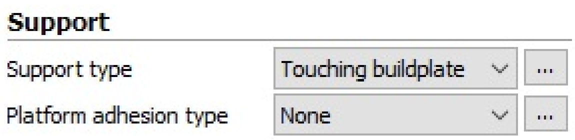
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

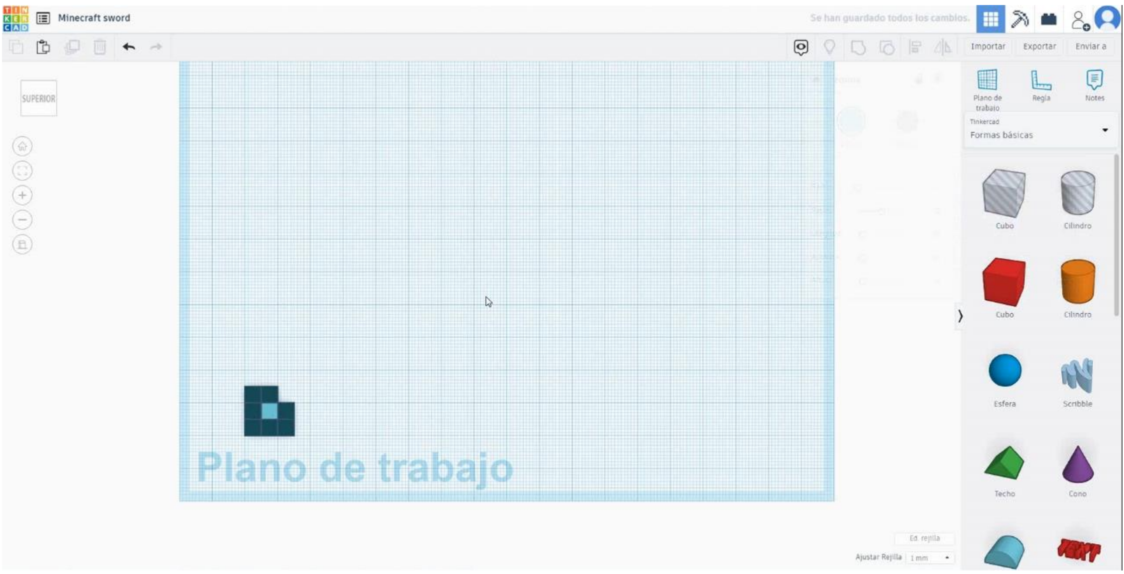


9.3.10 Part 10: Miniecraft sword

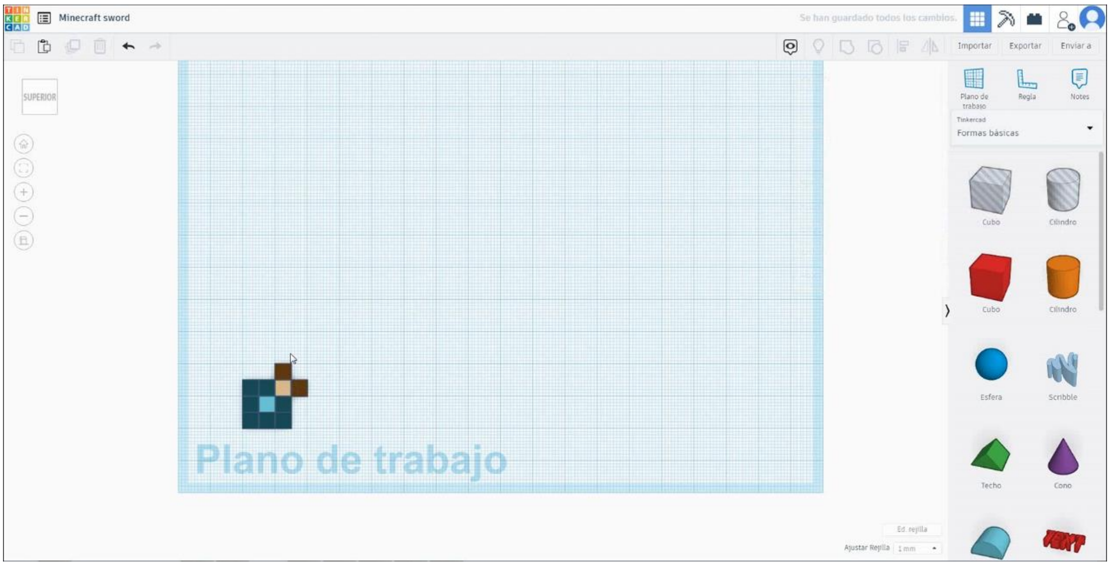
9.3.10.1 Minecraft sword Design

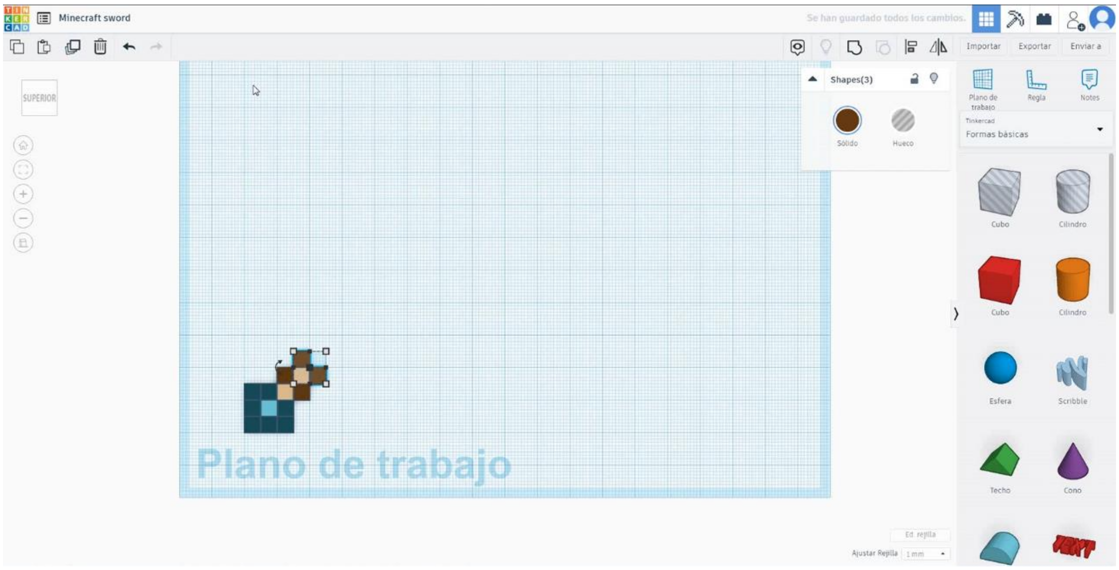
1. Choose the cube shape and size it to 5x5x10 mm, select green color and move it to out reference position.

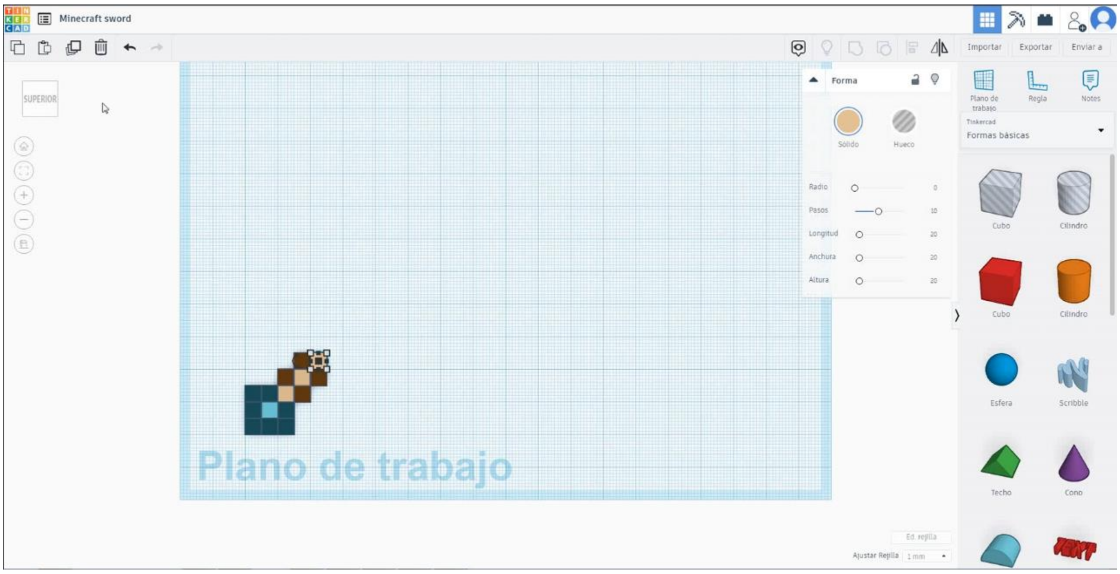
2. Duplicate it 6 times with in the same color to make a square. Then copy one of them to the center and change the color to blue.



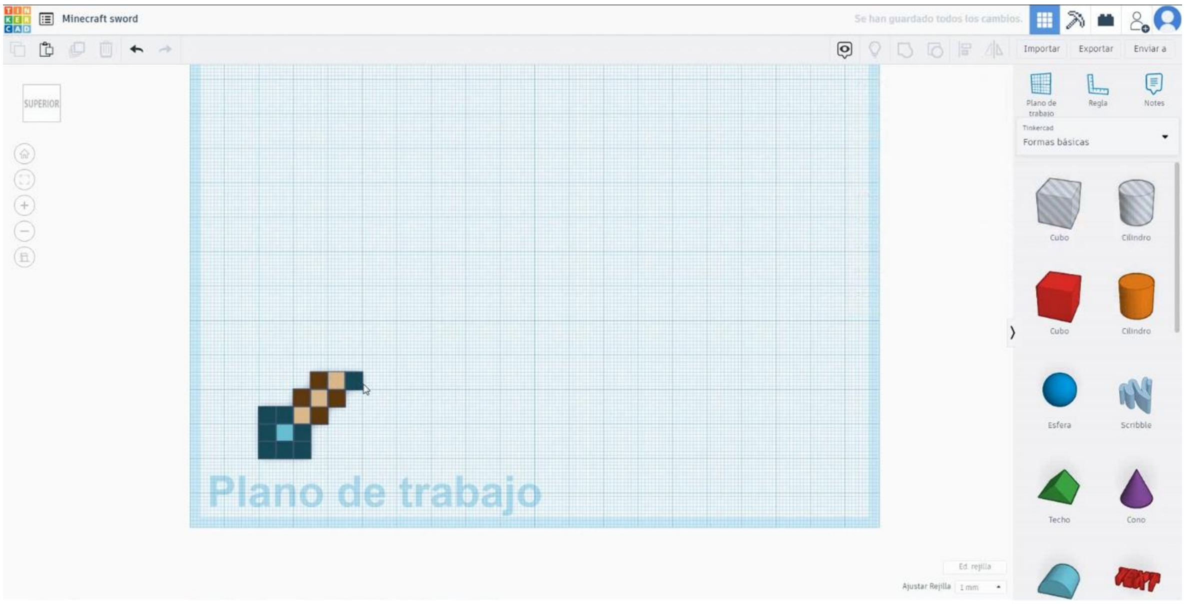
3. Duplicate the cube three times changing to a light and dark brown color, select this three cubes and duplicate them one time. Select the light brown cube and copy one time

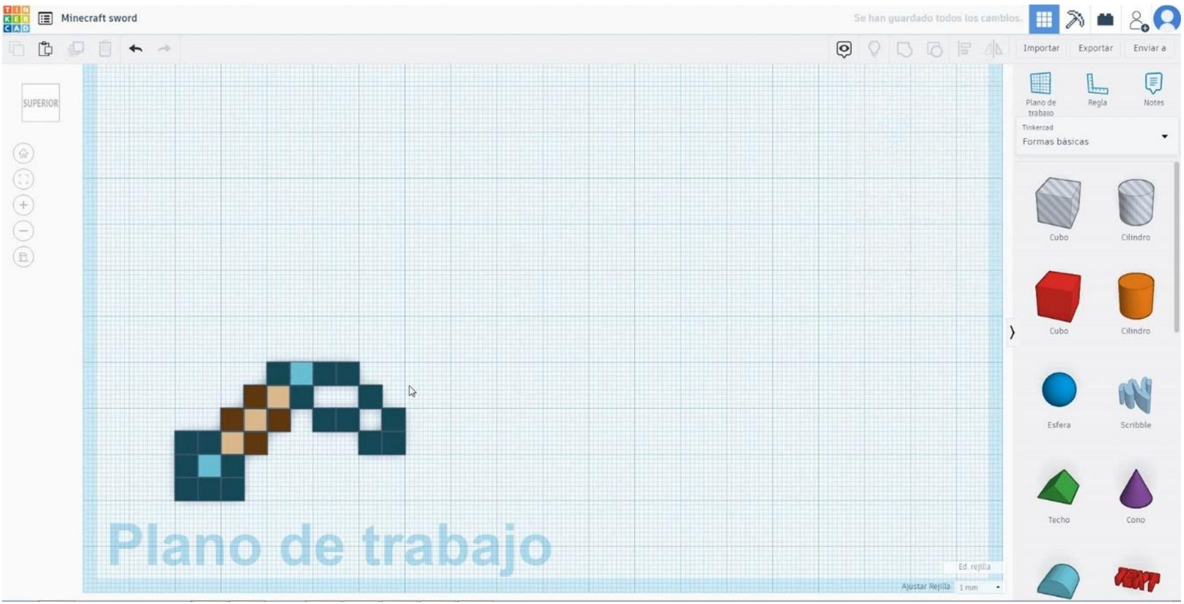


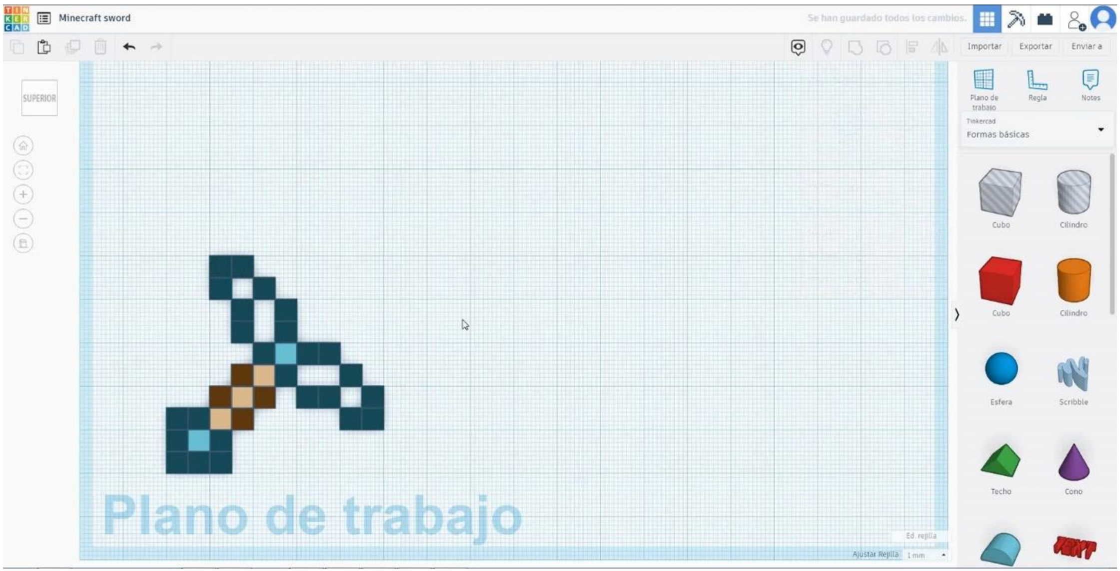


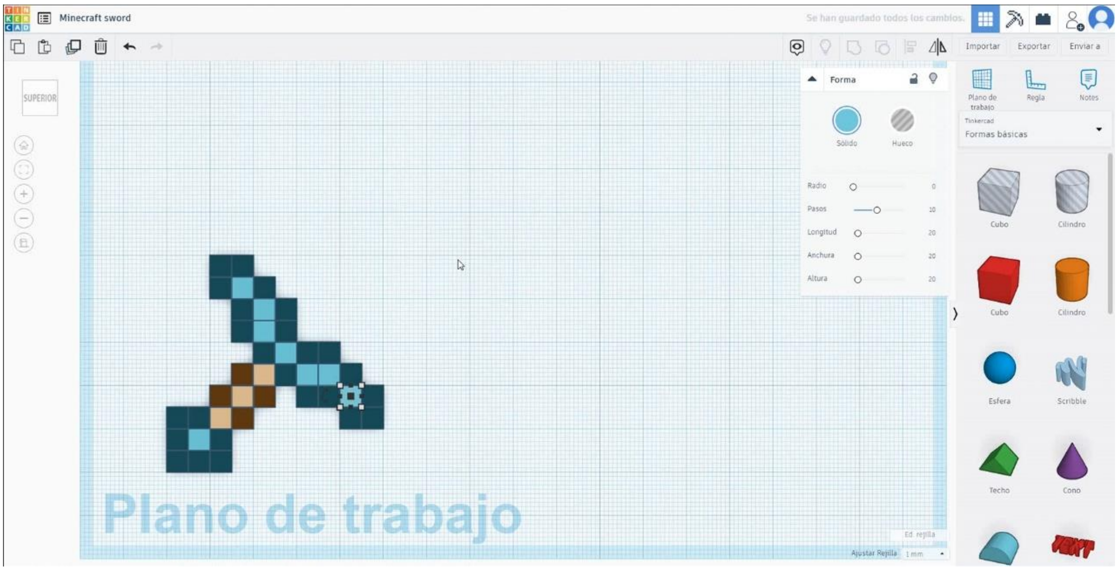


4. Duplicate one cube and change the color to green, then duplicate nine times. Duplicate one cube and change the color to blue and put in the center as a reference. Then copy the green cube eight times to create the shape below. Copy the blue cube into the gaps.









5. Select the blue central cube and duplicate it three times. Select the green cube and duplicate two times

Co-funded by the Erasmus+ Programme of the European Union

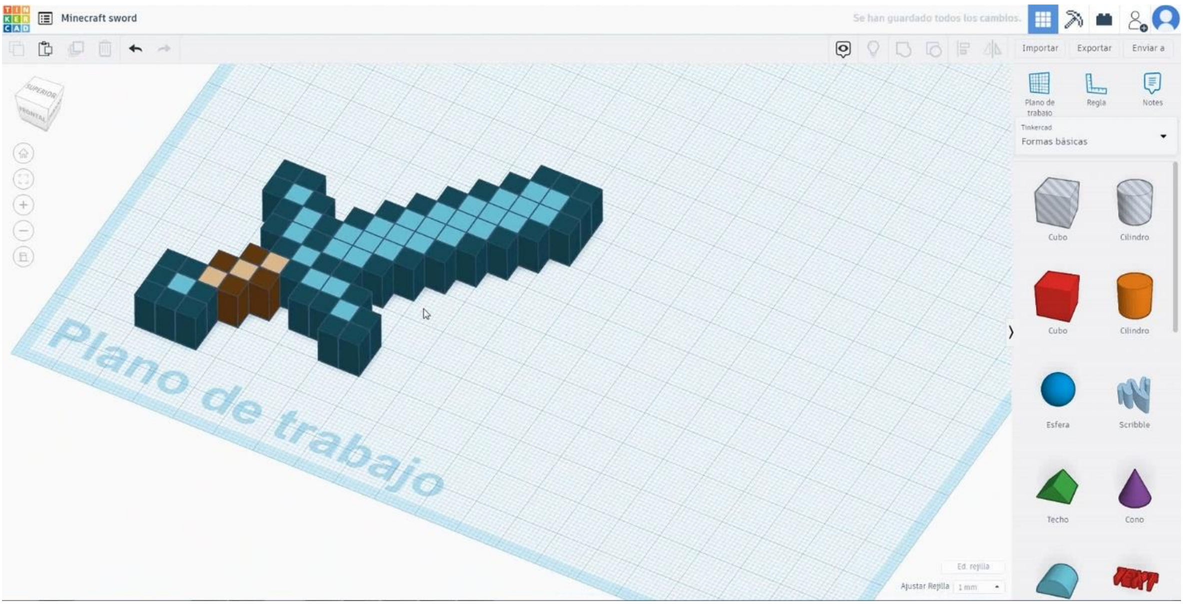
6. Select the three blue cubes and the two green cubes pressing shift and duplicate and move them up and right six times. Then duplicate one blue cube in the center and three green cubes to complete the shape.


Minecraft sword
Importar
Exportar
Enviar a
Shapes(5)
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
Plano de trabajo


Minecraft sword
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
Plano de trabajo


Minecraft sword
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
Plano de trabajo

7. Now, the minecraft sword is finished.



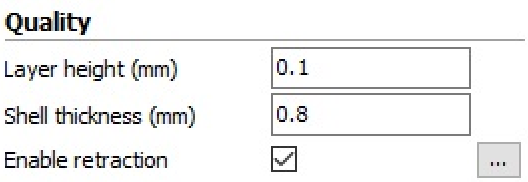
9.3.10.2 Minecraft sword 3D printing seetings

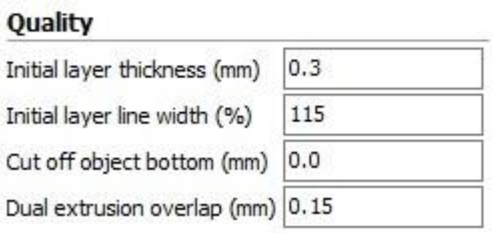
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

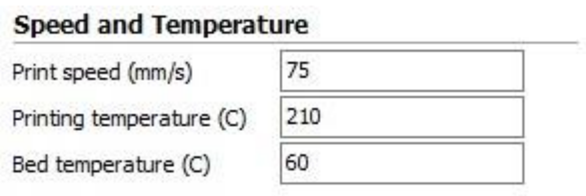
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

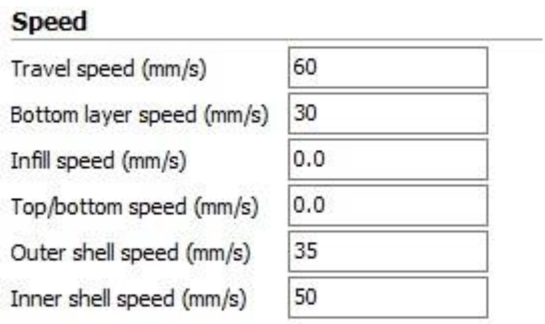




Speed and Temperature

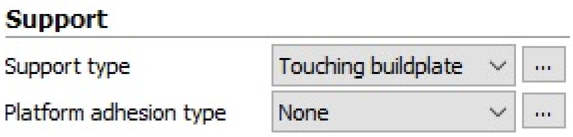
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

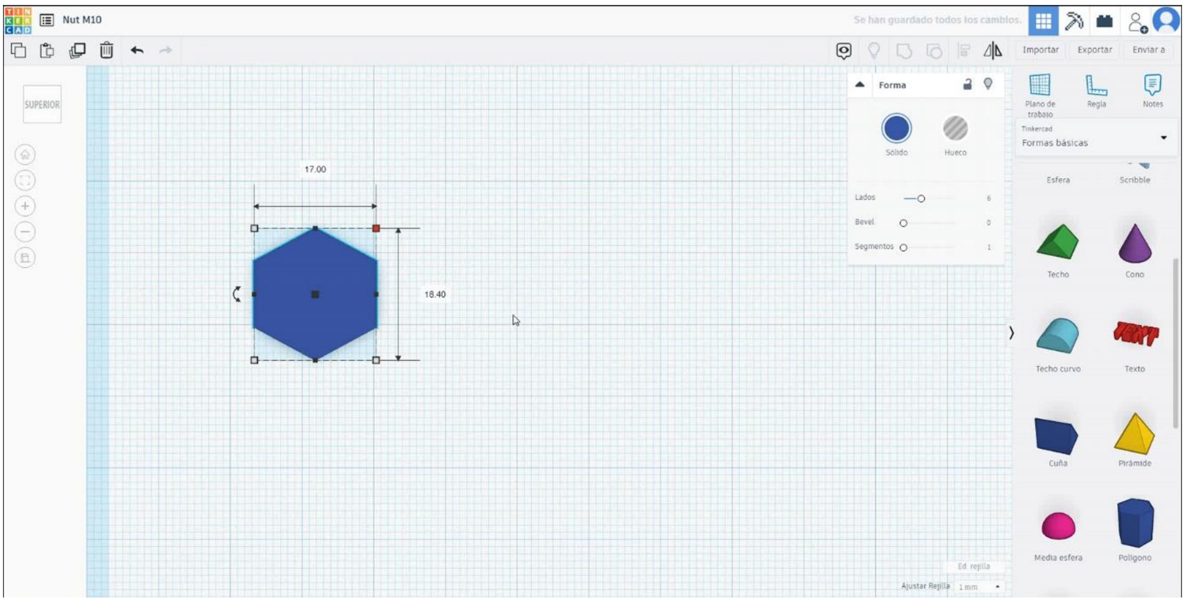
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



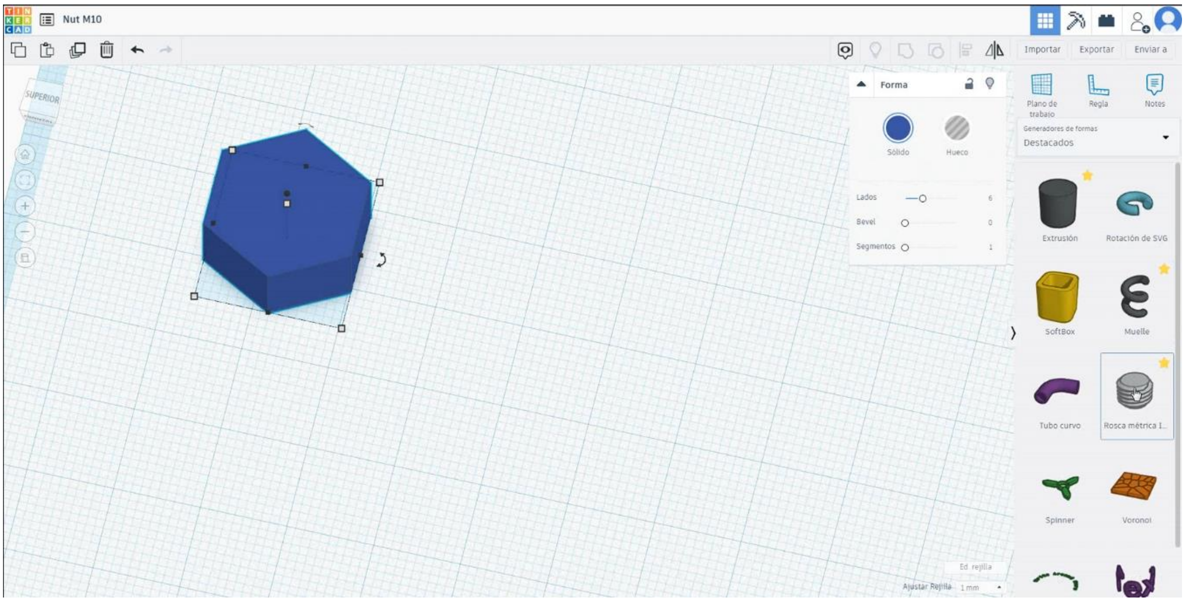
9.3.11 Part 11: Nut M10

9.3.11.1 Nut M10 Design

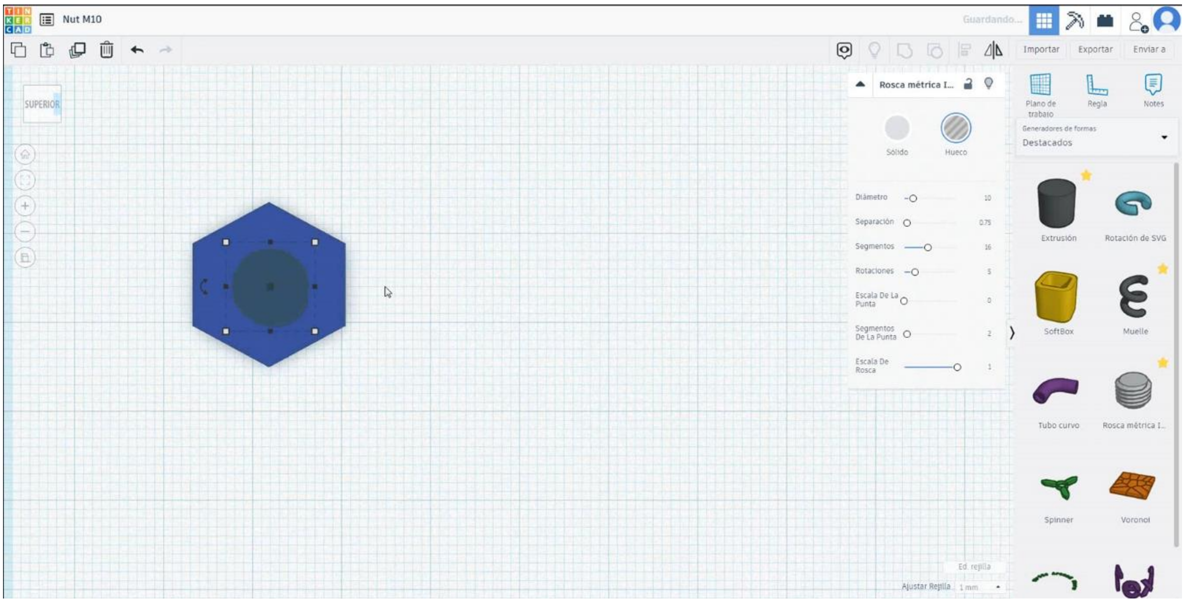
1. Choose the polygon shape and size it to 17x18.4x10 mm with 6 sides.

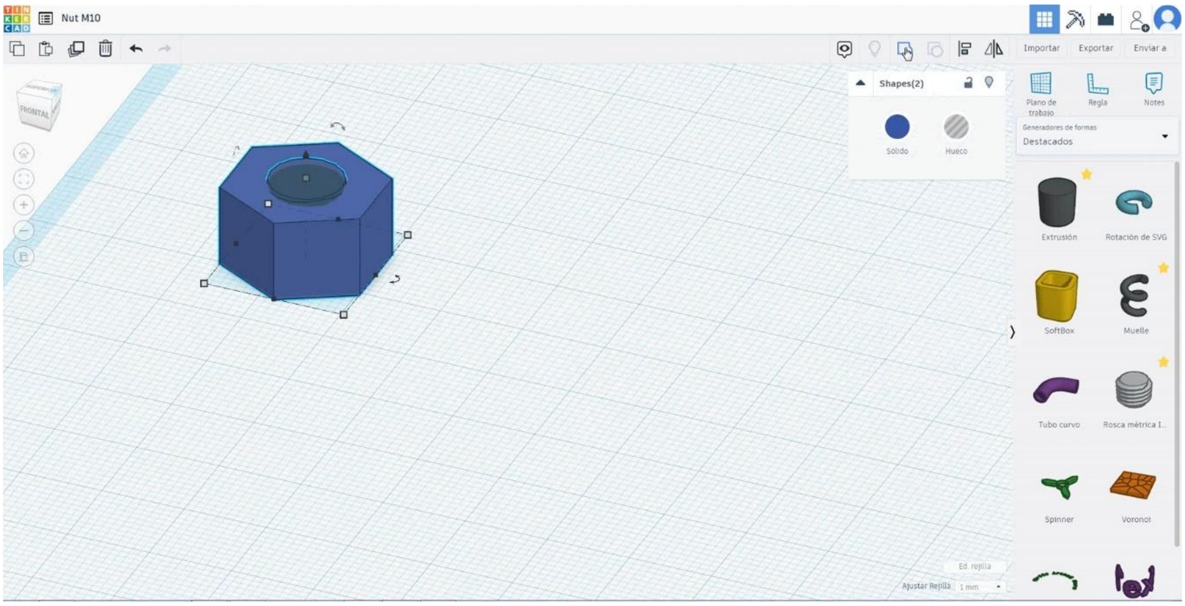


2. Then choose the shape iso metric thread from featured shape generators. Size it to 10 mm diameter and 0.75 mm pitch and 11 mm height and put in hole mode.

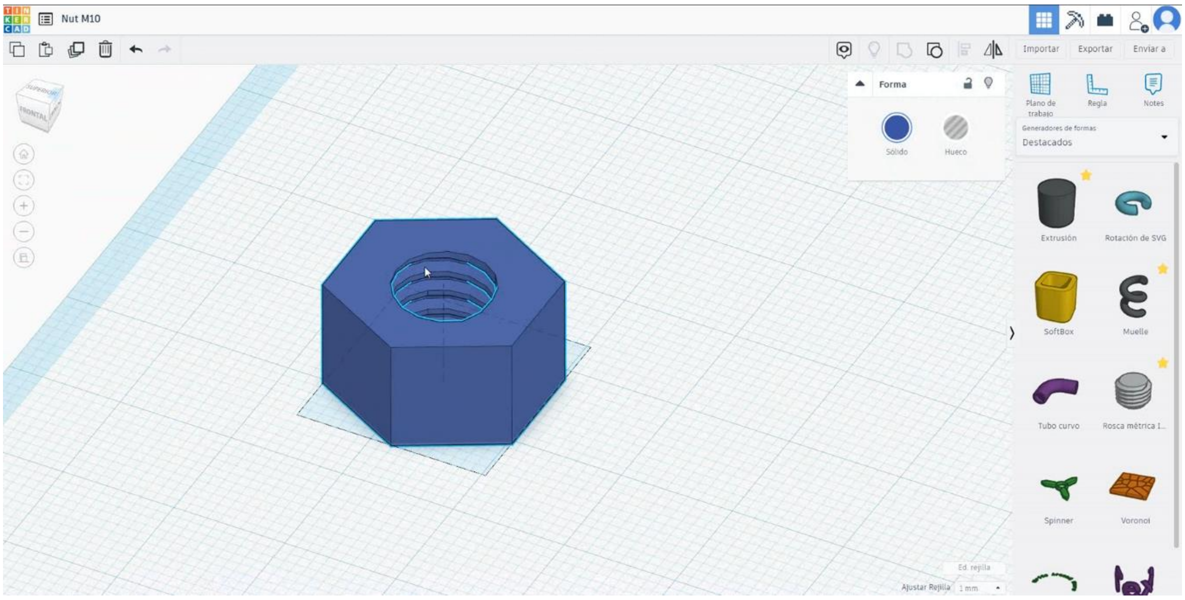


3. Center the metric thread into the polygon. Select both shapes pressing shift and select group.





4. Now, the nut is finished.



9.3.11.2 Nut M10 3D printing seetings

Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑ ...

Quality

Initial layer thickness (mm)	0.3
Initial layer line width (%)	115
Cut off object bottom (mm)	0.0
Dual extrusion overlap (mm)	0.15

Speed and Temperature

Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)

Speed and Temperature

Print speed (mm/s)	75
Printing temperature (C)	210
Bed temperature (C)	60

Speed

Travel speed (mm/s)	60
Bottom layer speed (mm/s)	30
Infill speed (mm/s)	0.0
Top/bottom speed (mm/s)	0.0
Outer shell speed (mm/s)	35
Inner shell speed (mm/s)	50

Su
pport Type

None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

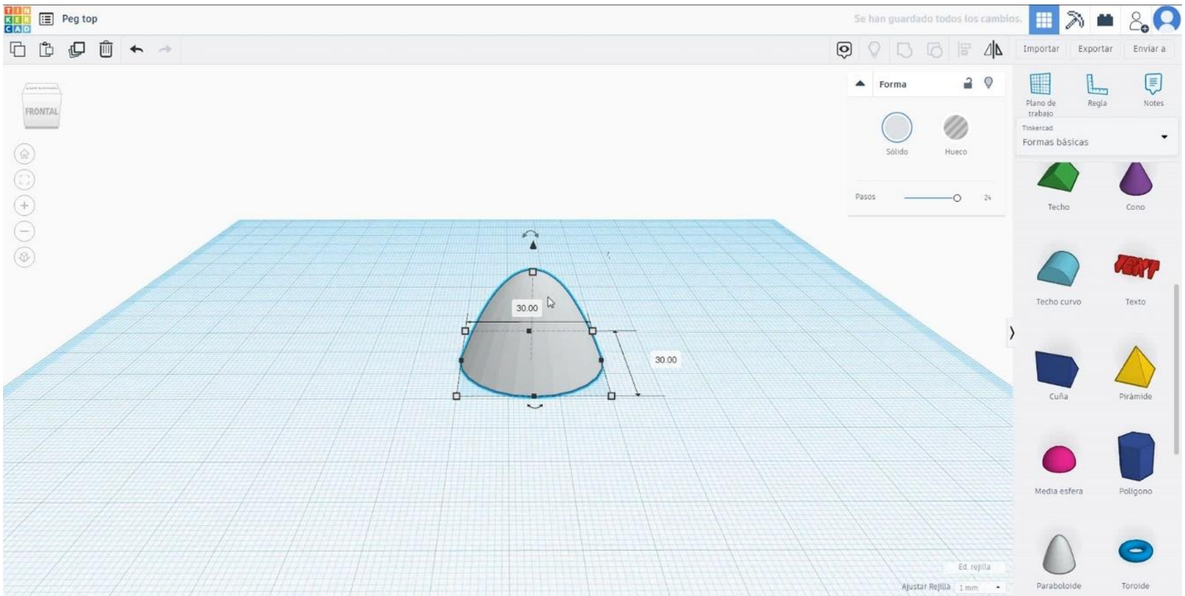
Support

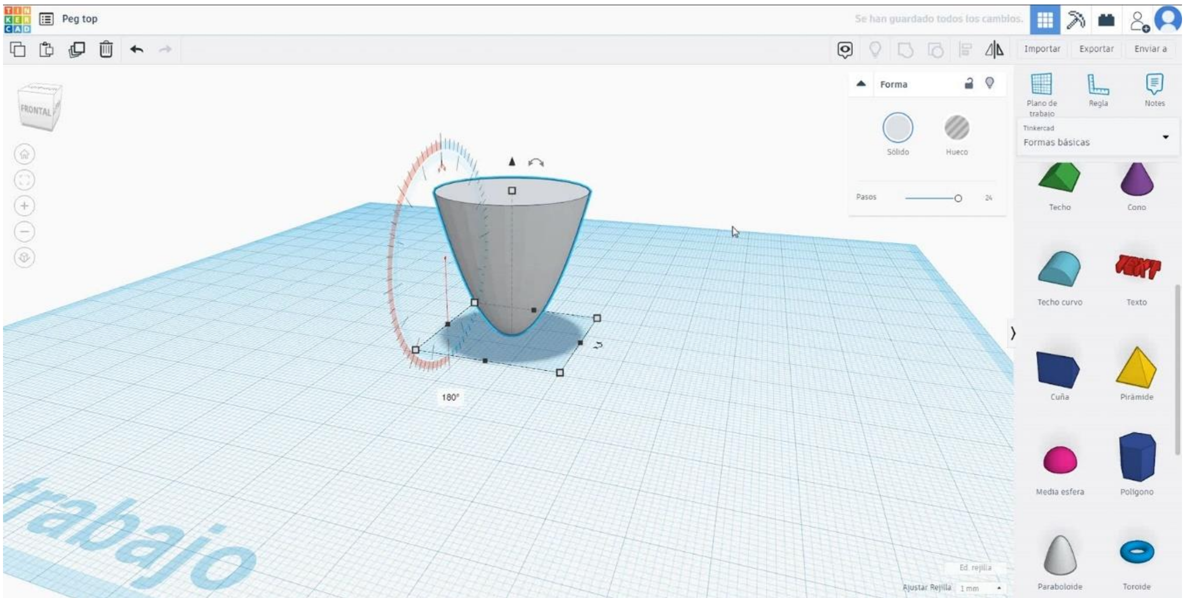
Support type	Touching buildplate ...
Platform adhesion type	None ...

9.3.12 Part 12: Peg Top

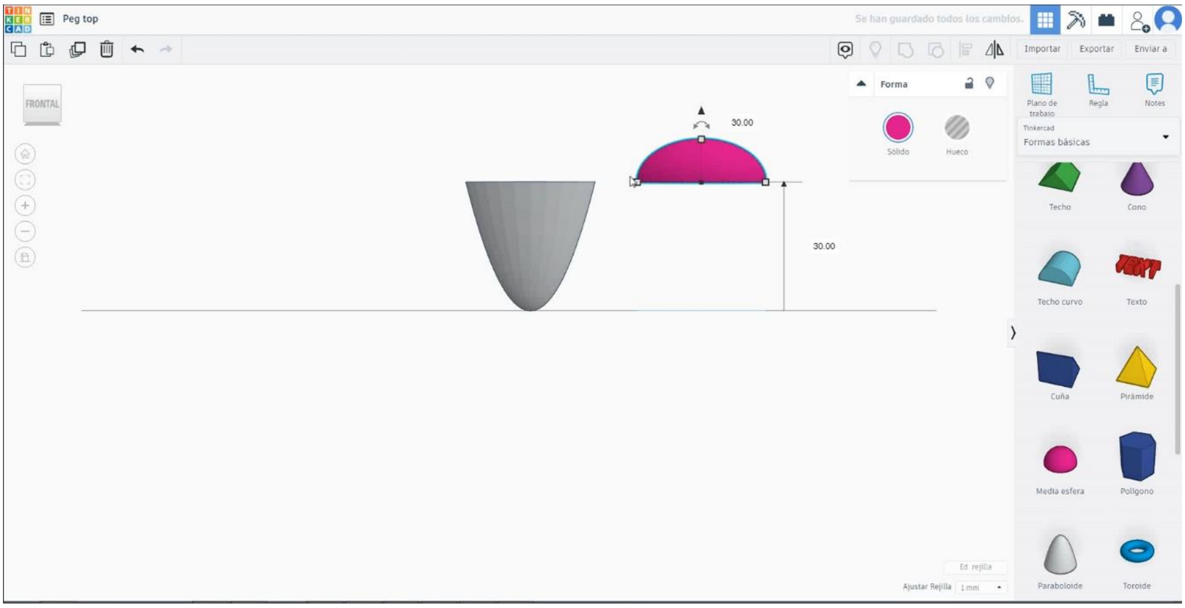
9.3.12.1 Peg Top Design

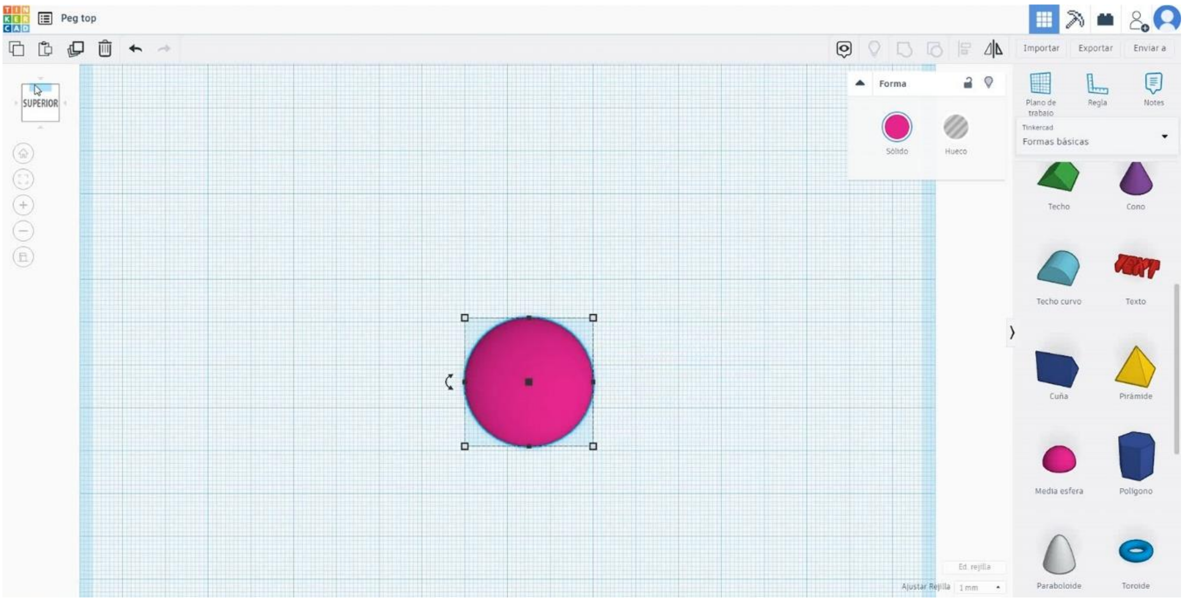
1. Select the paraboloid and size it to 30x30x30 mm. Rotate it 180 degrees



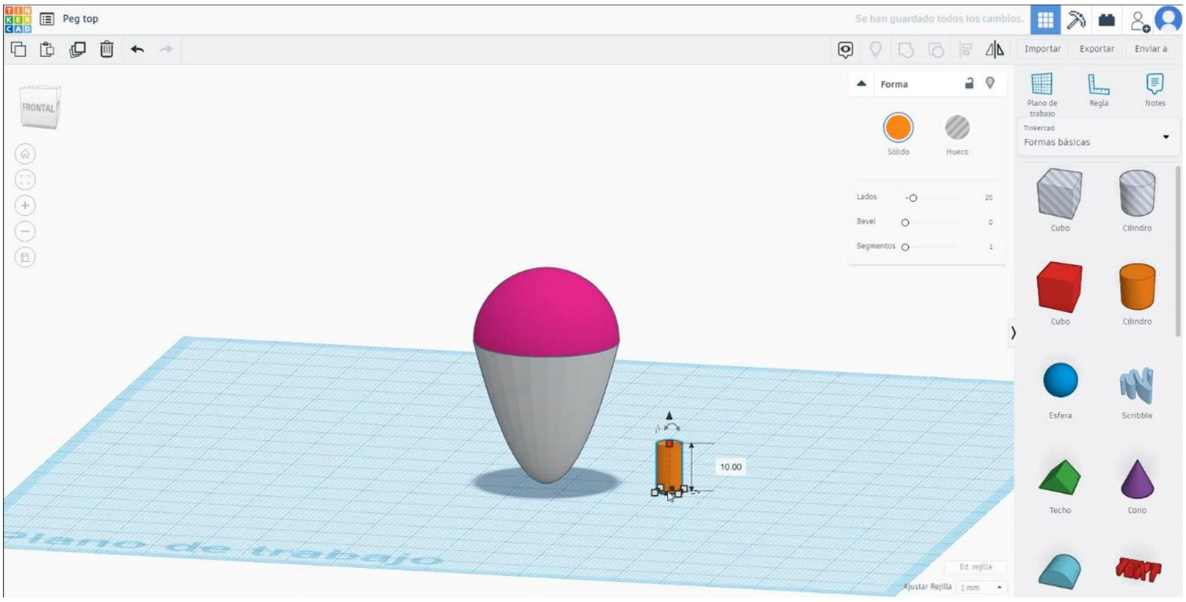


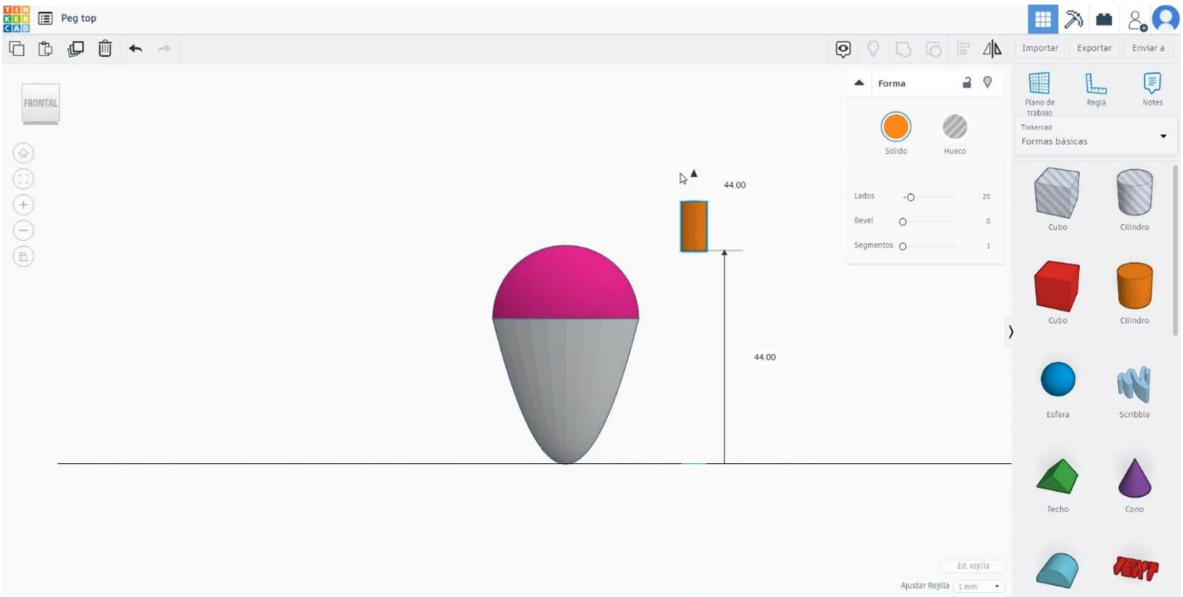
2. Select the half sphere and size it to 30x30x15 mm and move it to the height of 30 mm. Then centrate with the parabolid.

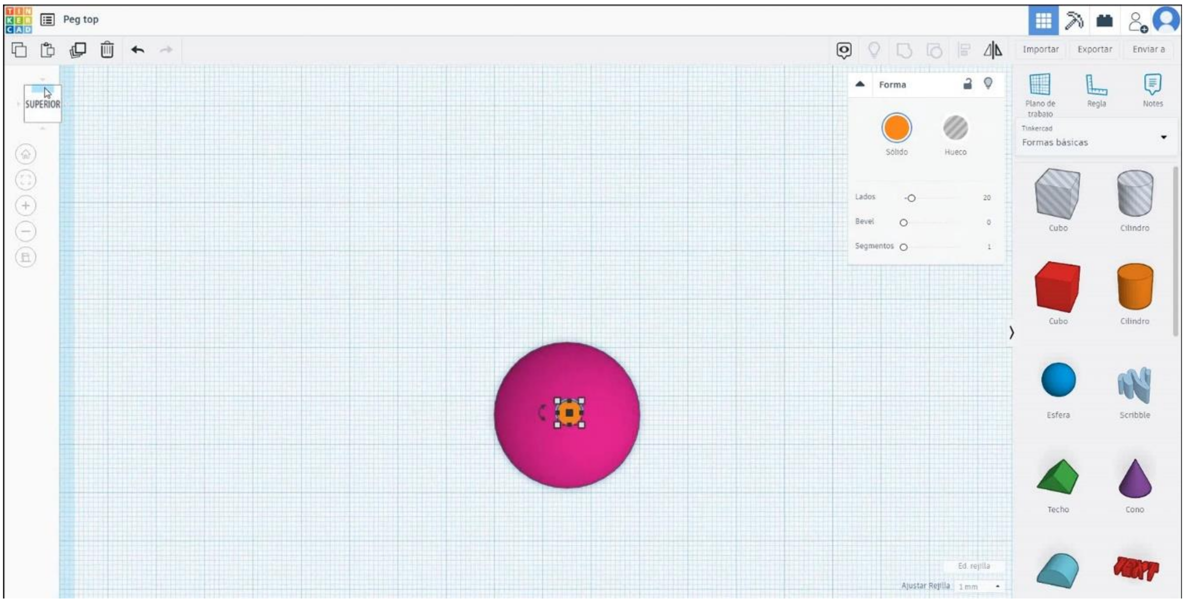




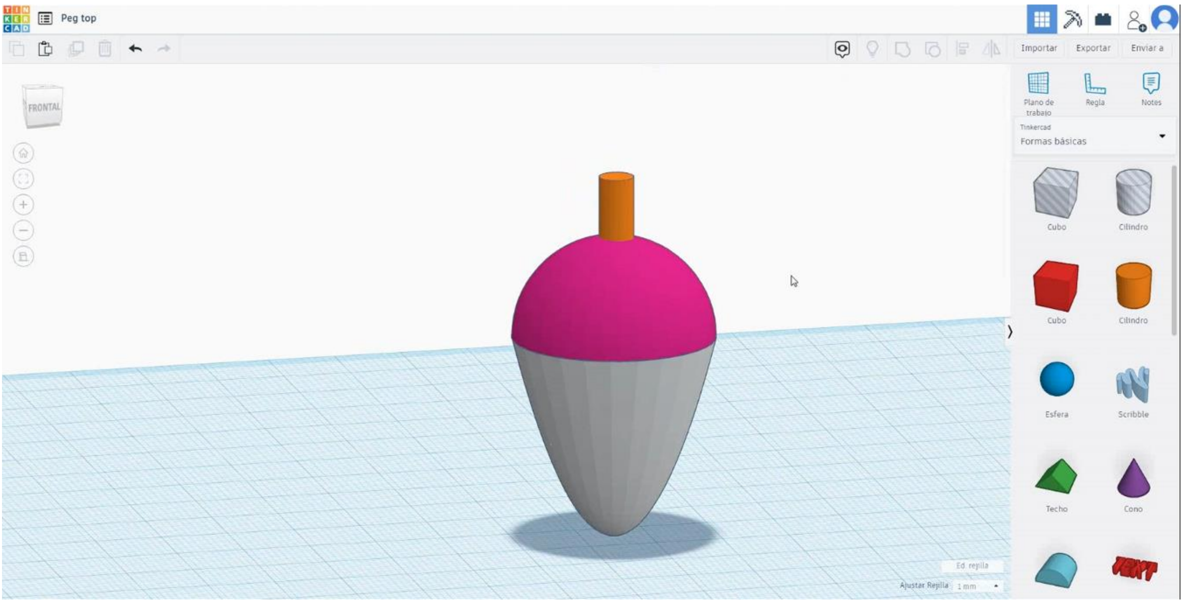
3. Select the cylinder and size it to 5x5x10 mm and move it to the height of 44 mm and centrate it with the half sphere.


Peg top
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Plano de trabajo
Regla
Notes
Formas básicas
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
10.00
Plano de trabajo
Ed. rejilla
Ajustar Rejilla
1 mm


Peg top
Importar
Exportar
Enviar a
Plano de trabajo
Regla
Notes
Formas básicas
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
FRONTAL
44.00
44.00
Ed. rejilla
Ajustar Rejilla
1 mm


Peg top
Importar
Exportar
Enviar a
Plano de trabajo
Regla
Notes
Formas básicas
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
SUPERIOR
Ed. rejilla
Ajustar Rejilla
1 mm

4. Now the peg top is finished



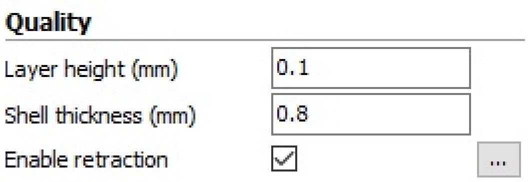
9.3.12.2 Peg Top 3D printing seetings

Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

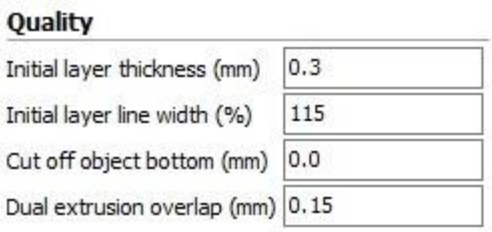
Quality

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)



Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑ ...



Quality

Initial layer thickness (mm)	0.3
Initial layer line width (%)	115
Cut off object bottom (mm)	0.0
Dual extrusion overlap (mm)	0.15

Speed and Temperature

Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)

Speed and Temperature	
Print speed (mm/s)	75
Printing temperature (C)	210
Bed temperature (C)	60

Speed	
Travel speed (mm/s)	60
Bottom layer speed (mm/s)	30
Infill speed (mm/s)	0.0
Top/bottom speed (mm/s)	0.0
Outer shell speed (mm/s)	35
Inner shell speed (mm/s)	50

Support Type

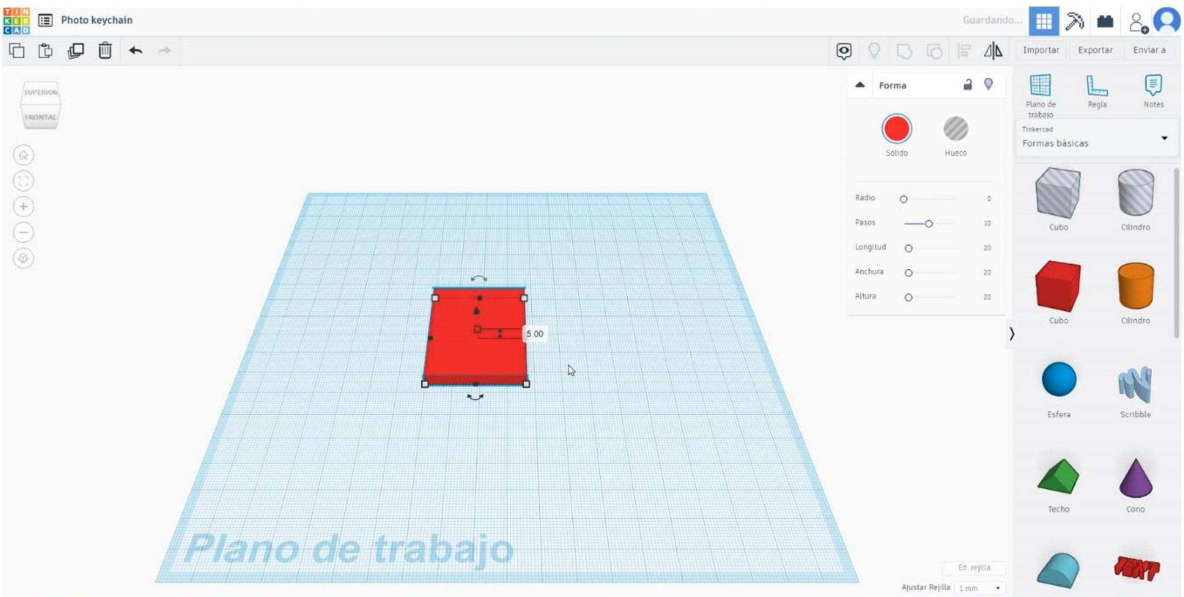
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

Support	
Support type	Touching buildplate
Platform adhesion type	None

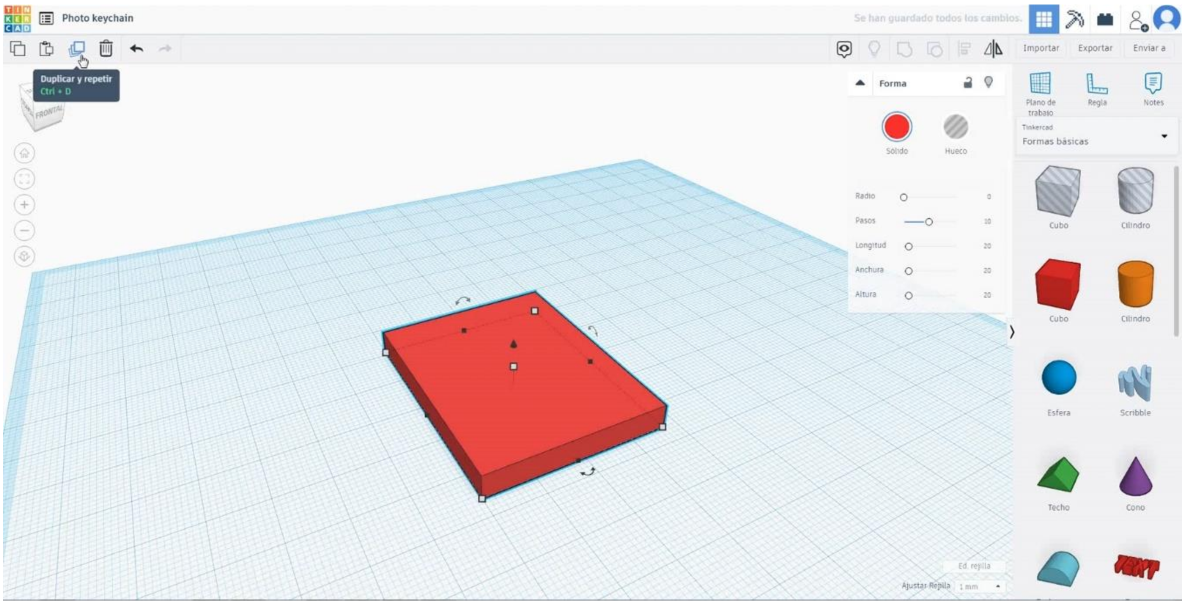
9.3.13 Part 13: Keychain

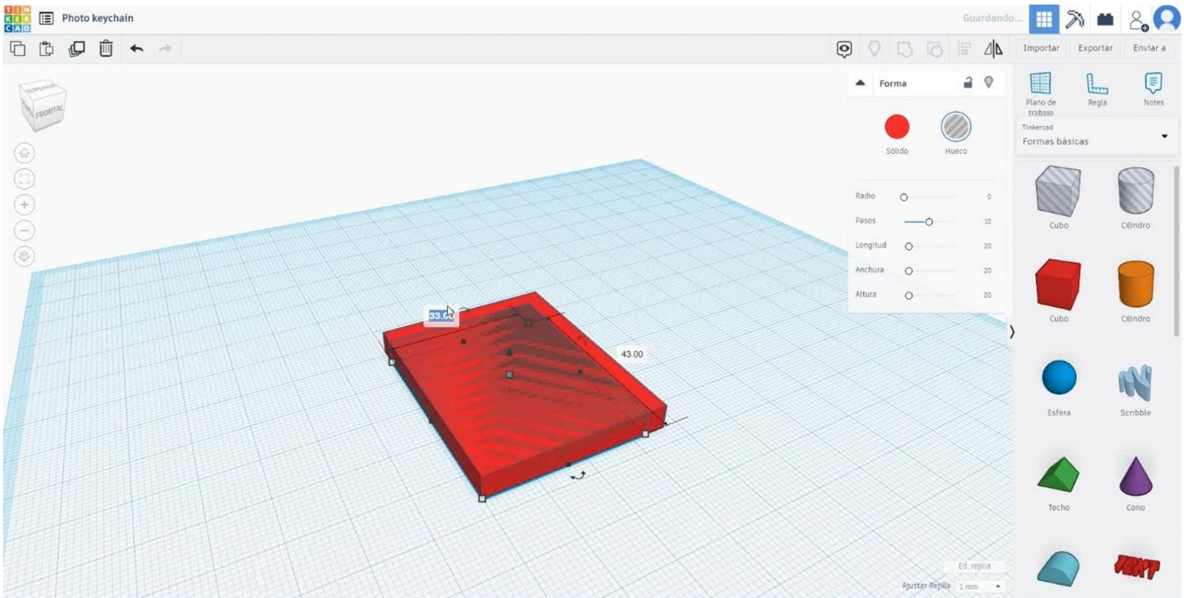
9.3.13.1 Keychain Design

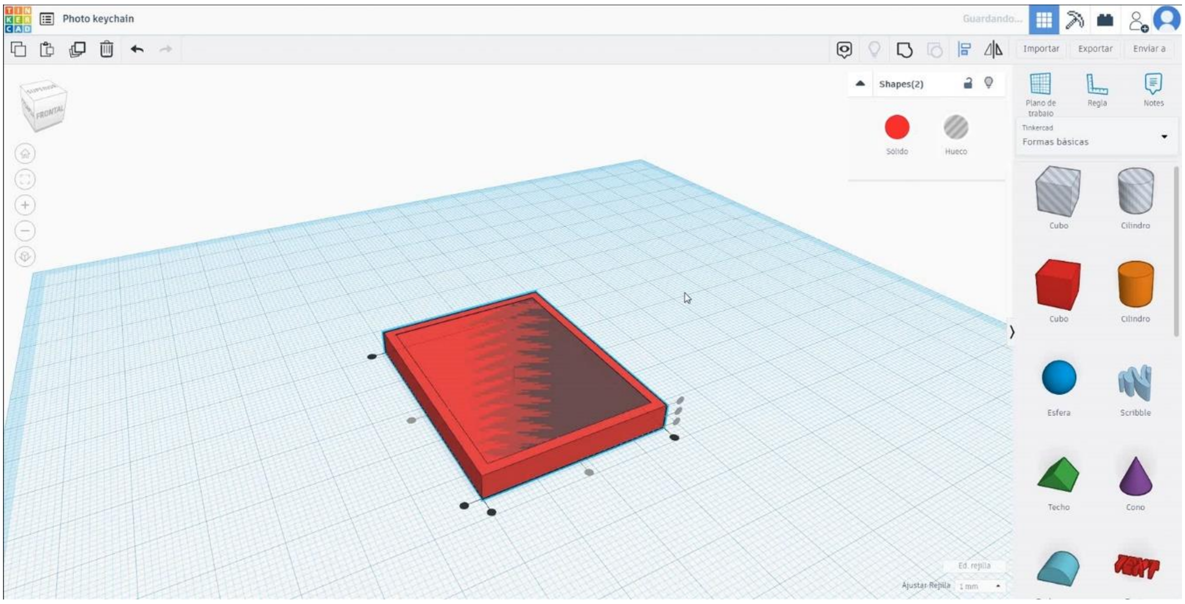
1. Choose the cube shape and size it to 37x47x5 mm.

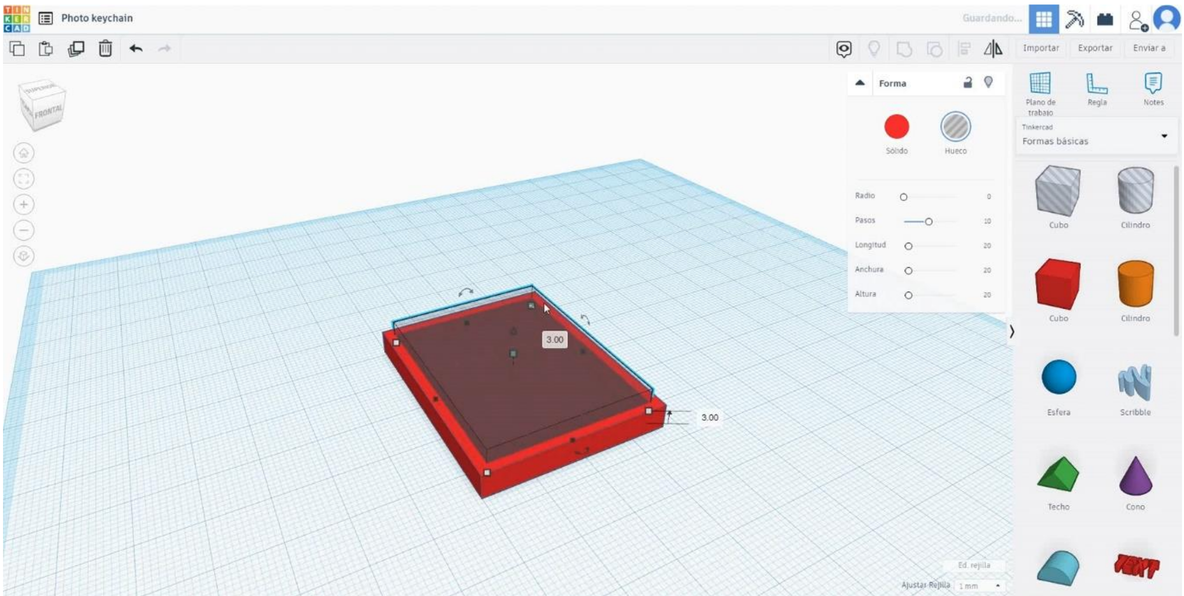


2. Duplicate the cube shape in hole mode and size it to 33x43x5 mm, align it with the first cube and move it to the height of 3 mm

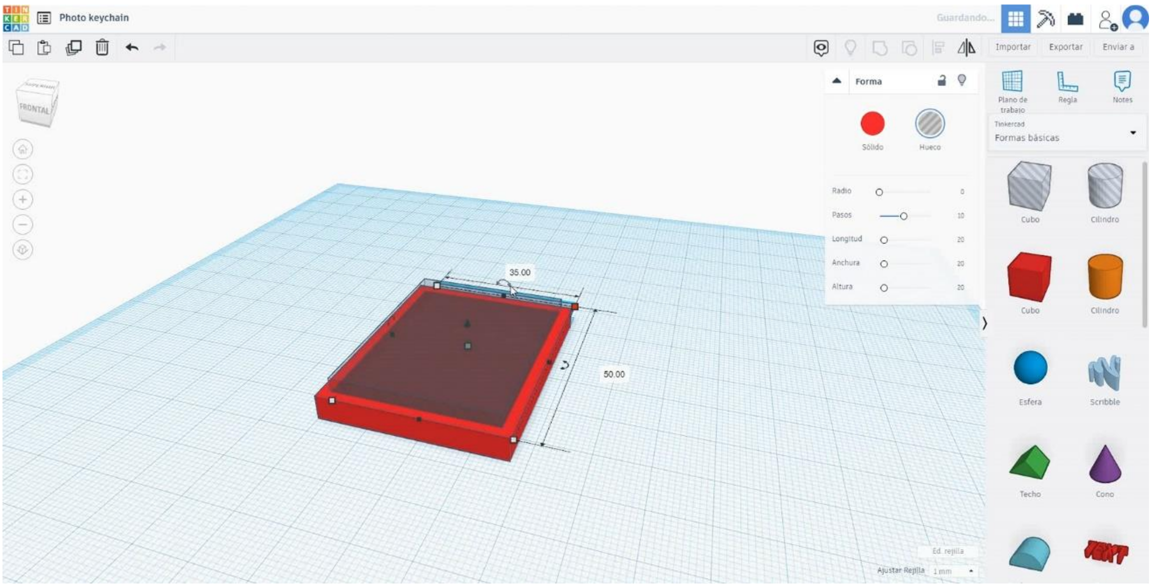


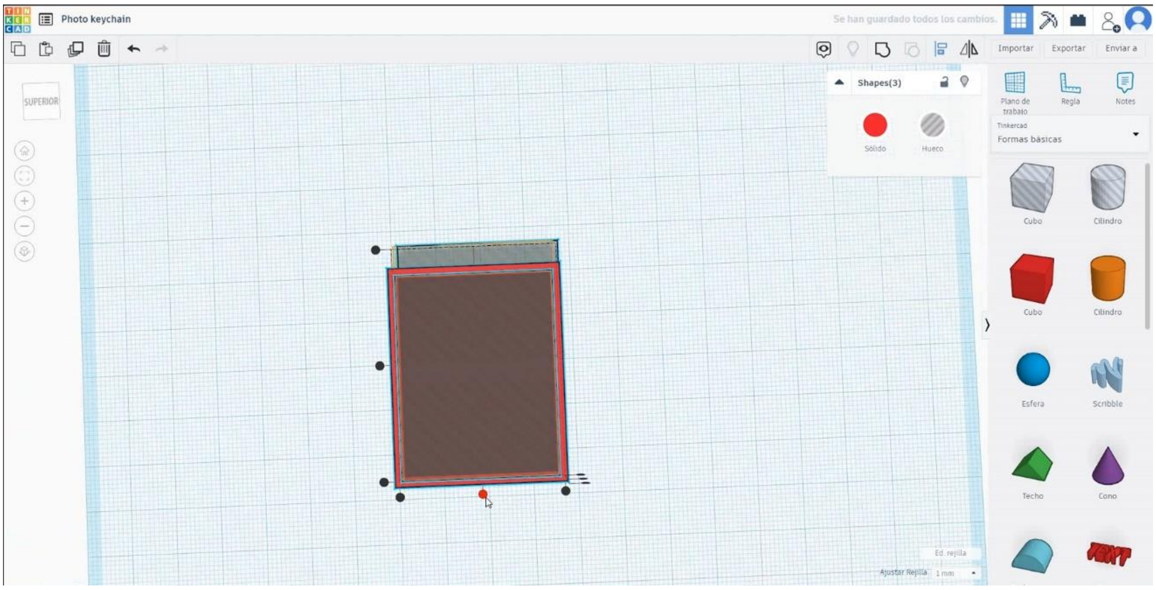


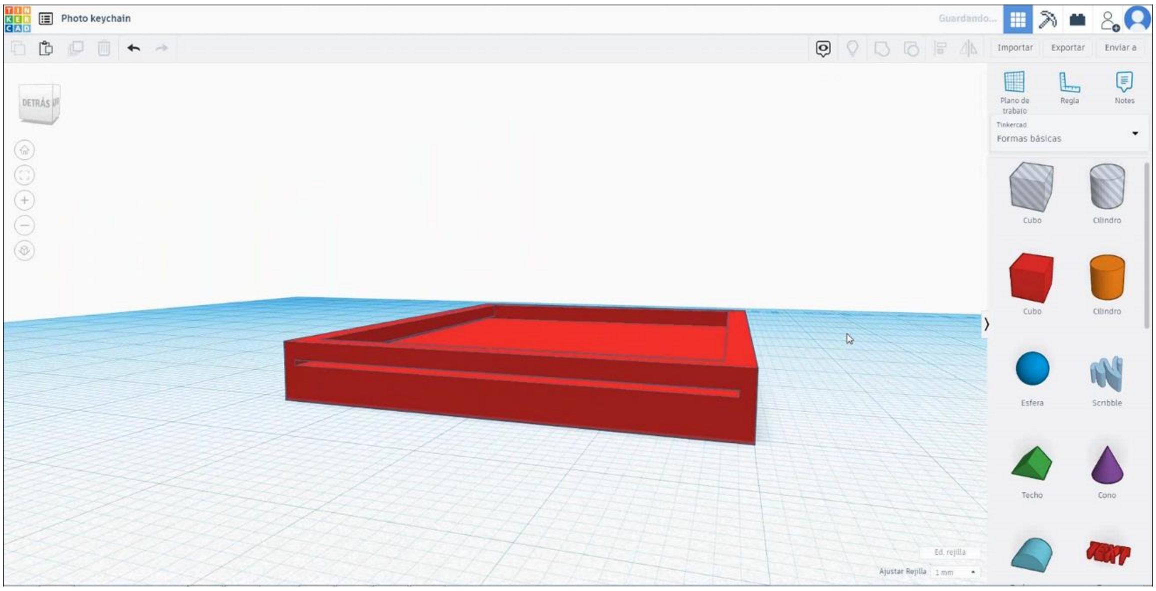




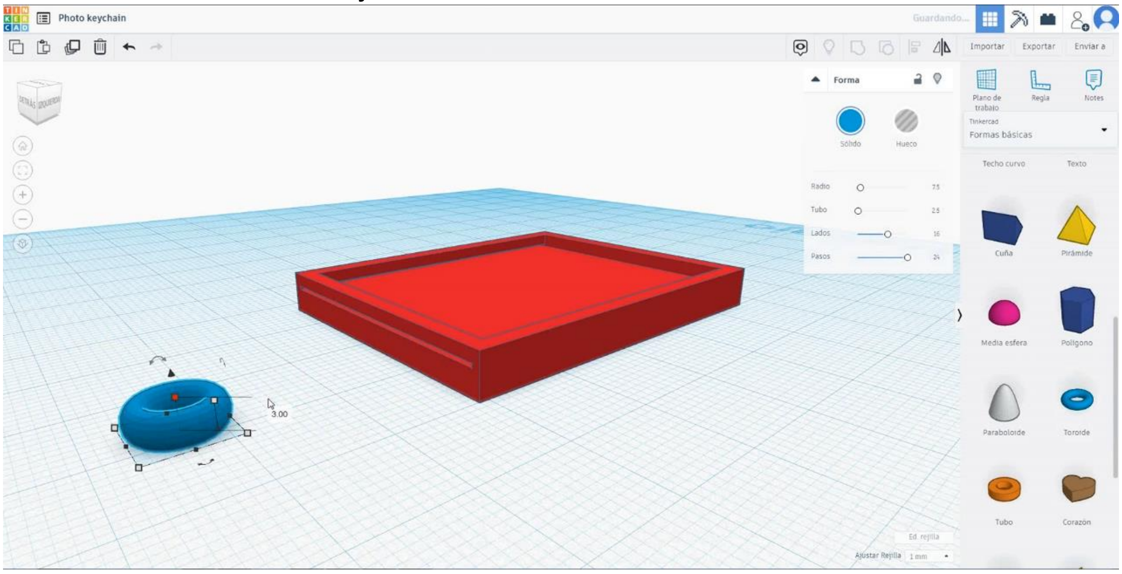
3. Duplicate the second cube in hole mode and size it to 35x50x0.5 mm, align it only in X axis. Make sure the 2 cubes in hole mode are in 3 mm height. Select the 3 objects and press group.

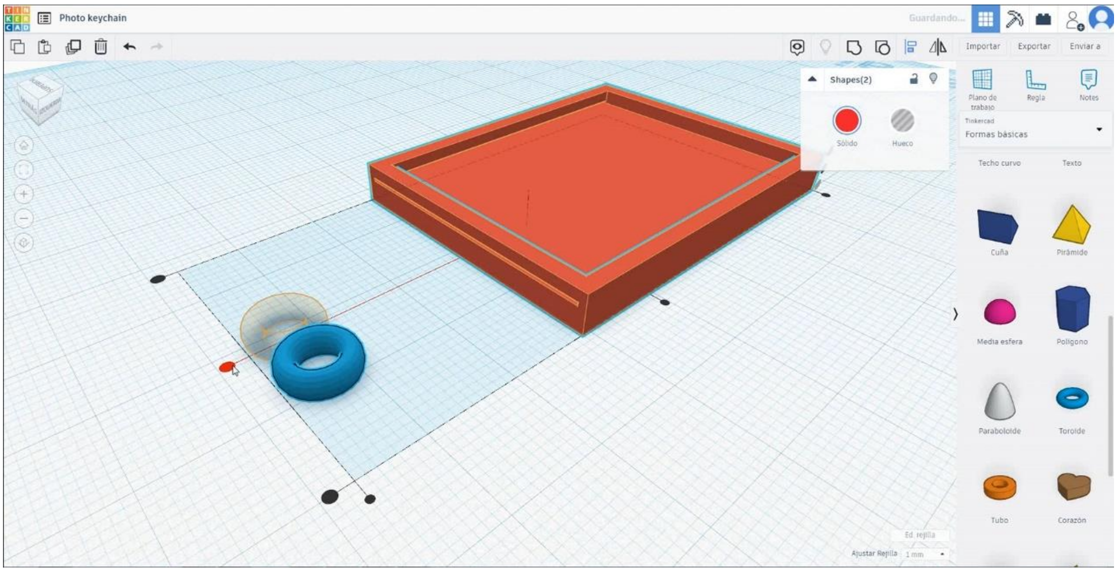


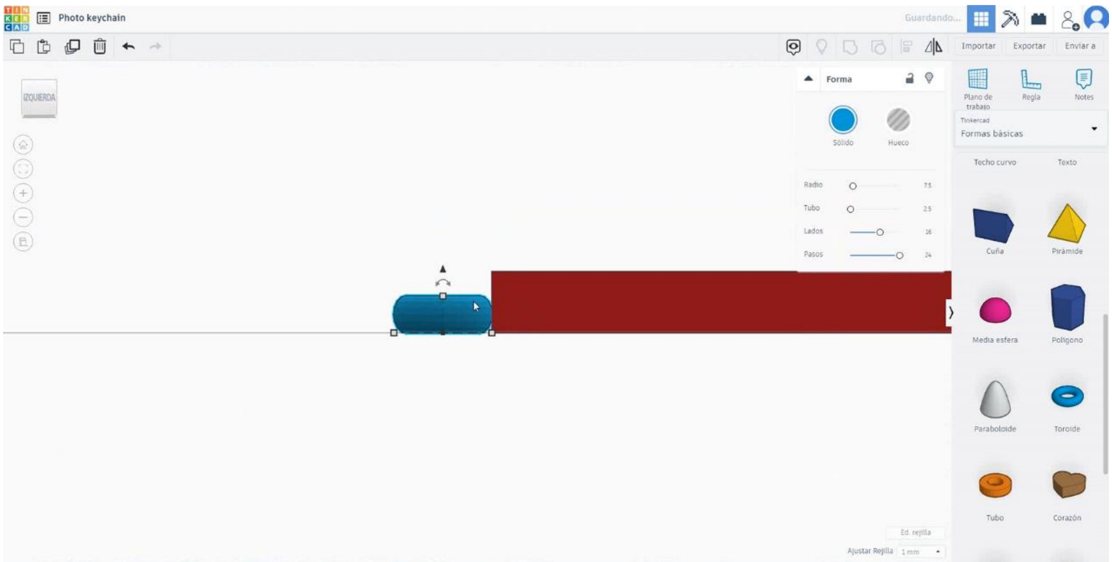




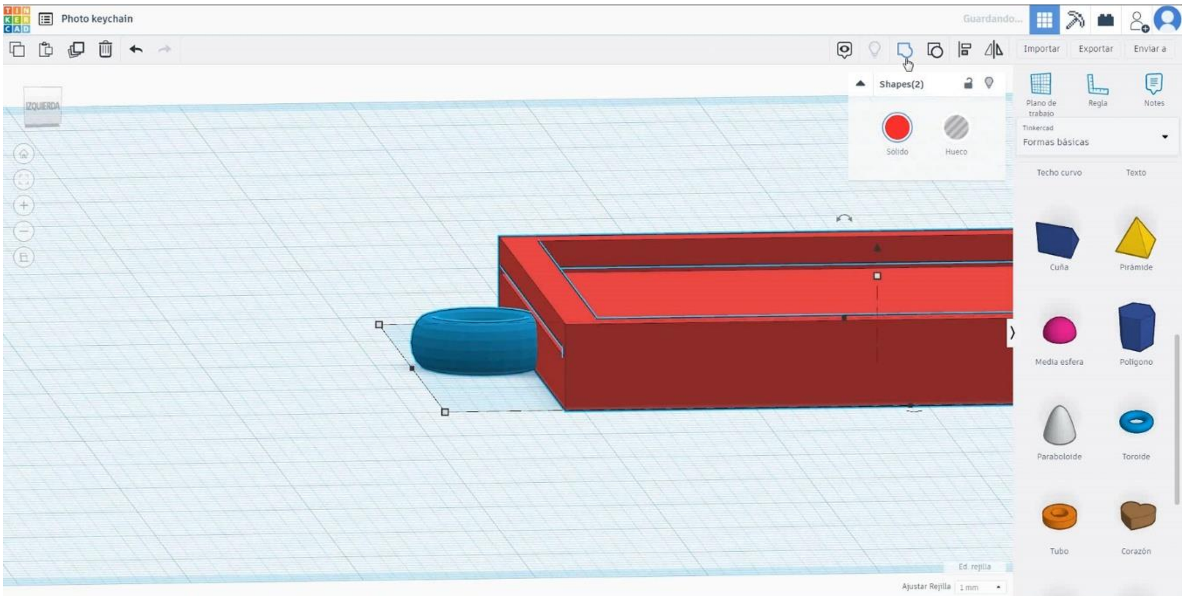
4. Choose the torus shape and size it to 8x8x3 mm and align it with the cube in X axis. Move it in Y axis to make sure both objects touch each other.



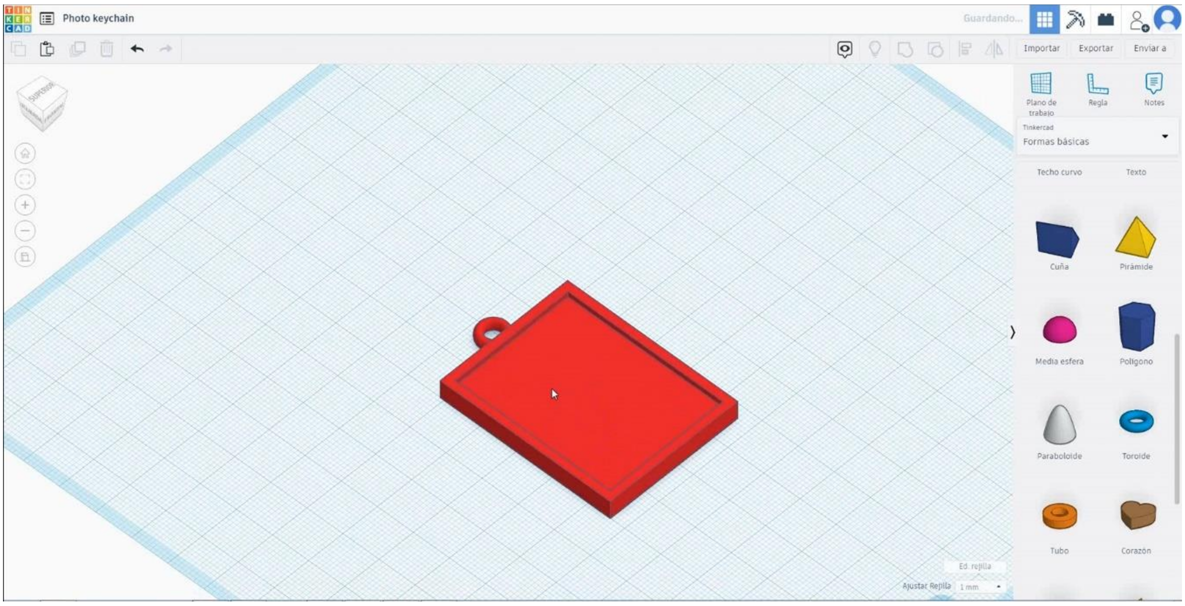




5. Select the torus and the cube and press group.



6. Now, the photo keychain is finished.



Co-funded by the Erasmus+ Programme of the European Union

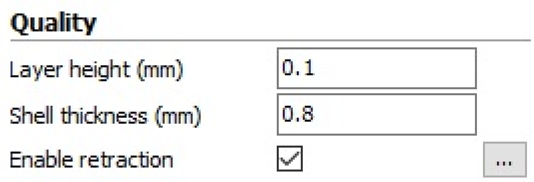
9.3.13.2 Keychain 3D printing seetings

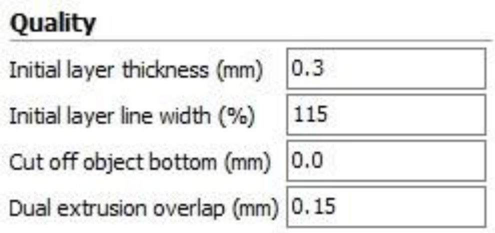
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

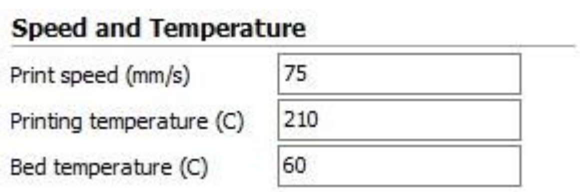
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

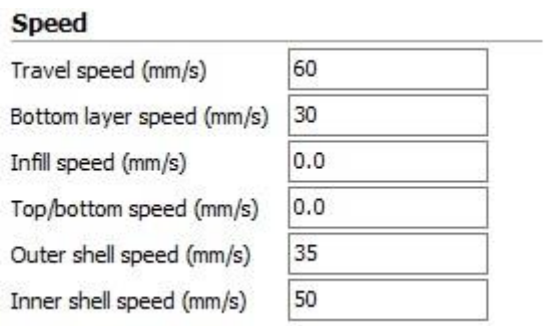




Speed and Temperature

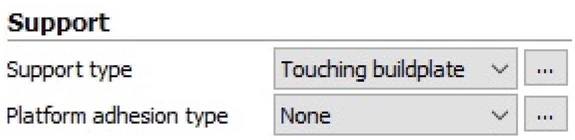
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

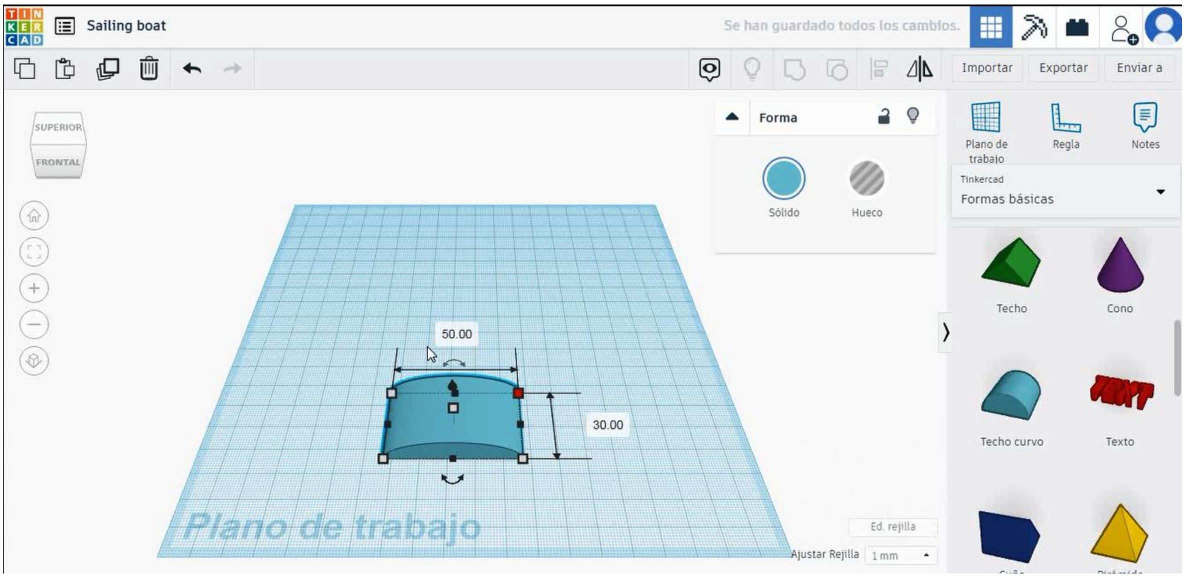
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

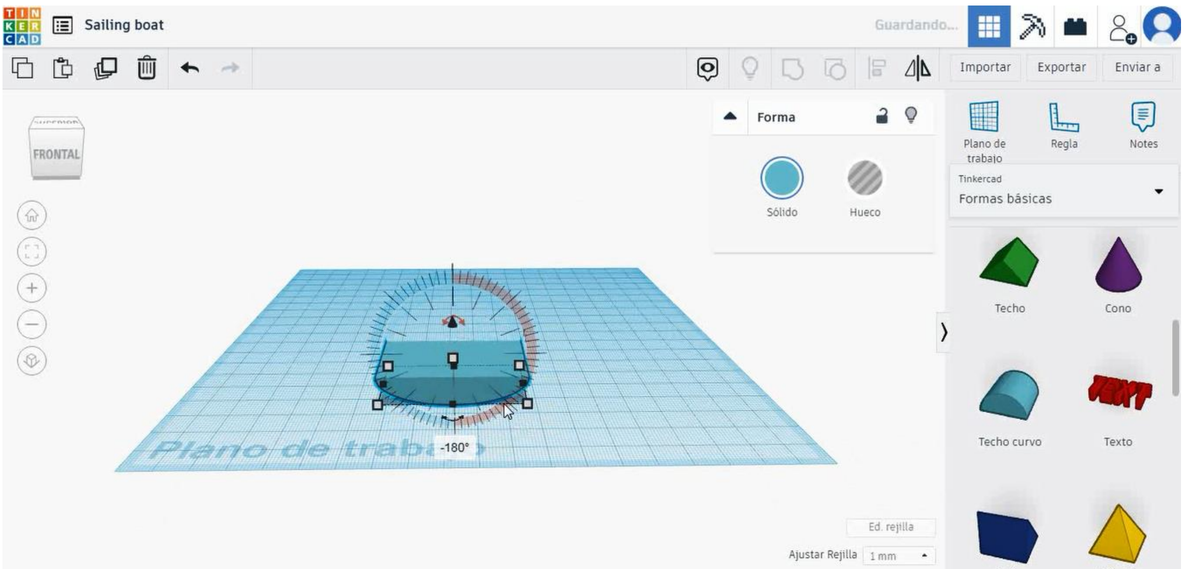


9.3.14 Part 14: Sailing Boat

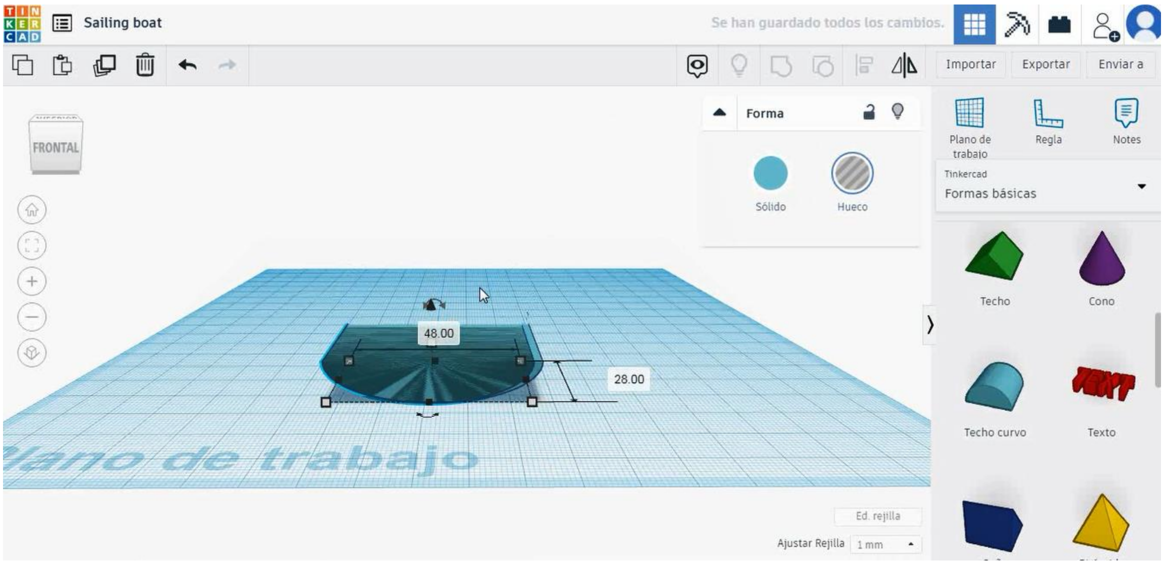
9.3.14.1 Sailing Boat Design

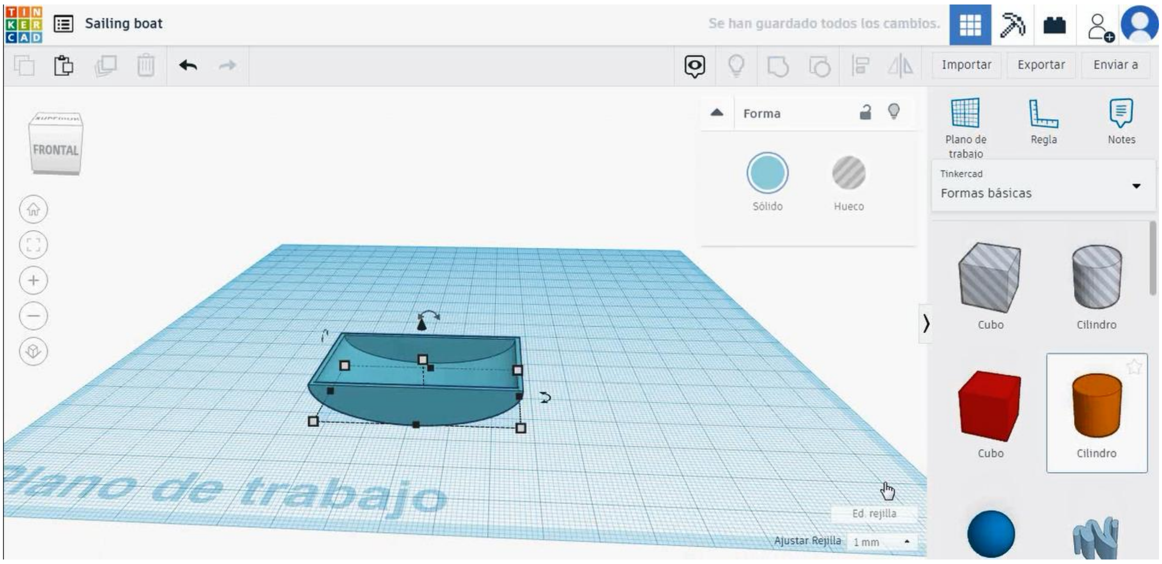
1. Choose the round roof shape, size it to 50x30x10 mm and turn it 180 degrees.



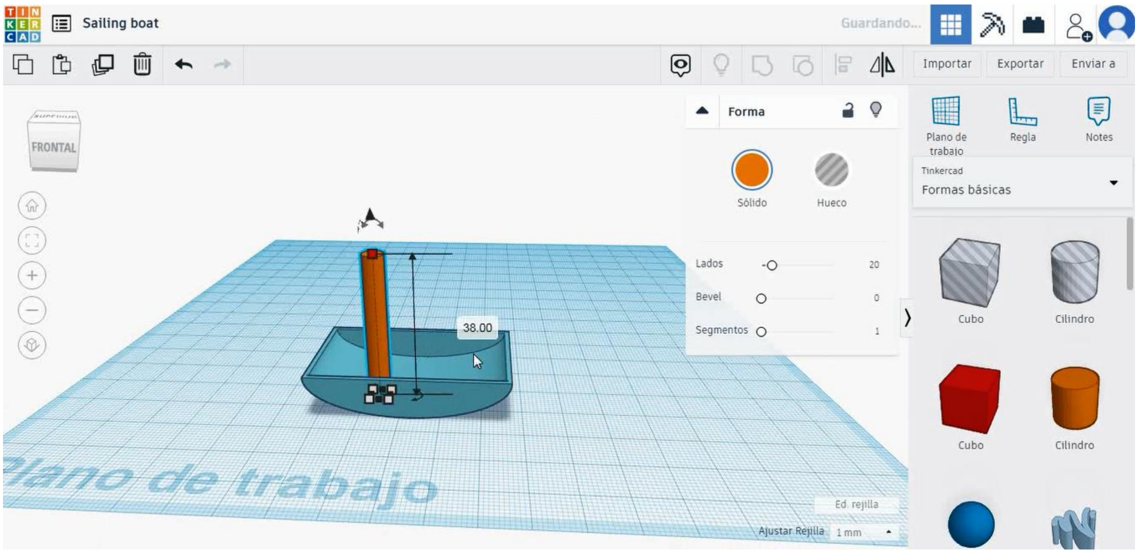


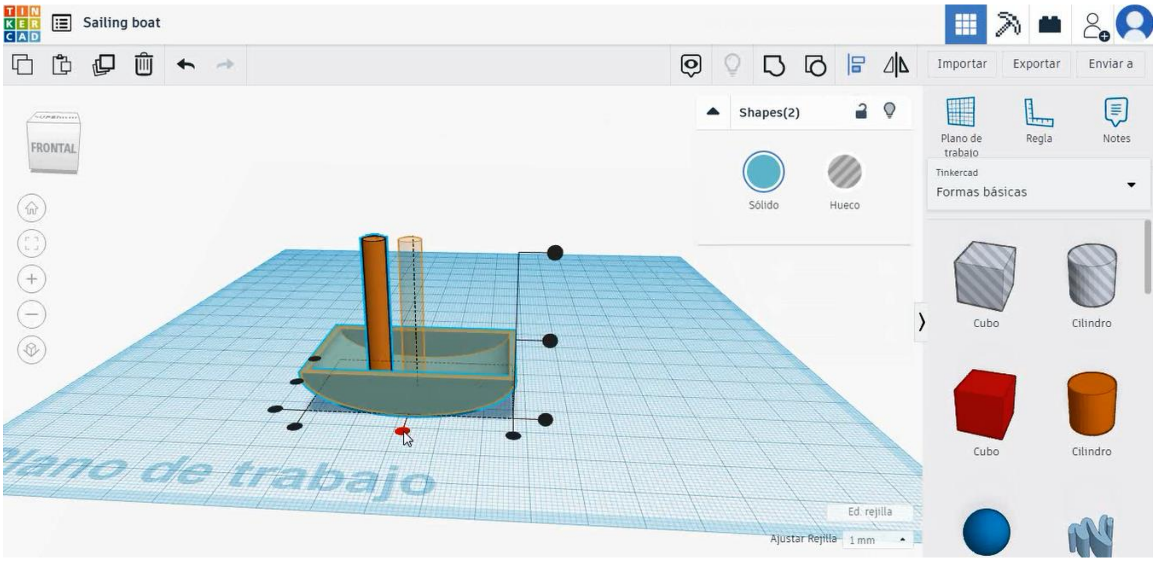
2. Duplicate it in hole mode, size it to 48x28x8 mm, move it to the height of 2 mm and align in the center of the first round roof. Select both objects and press group.



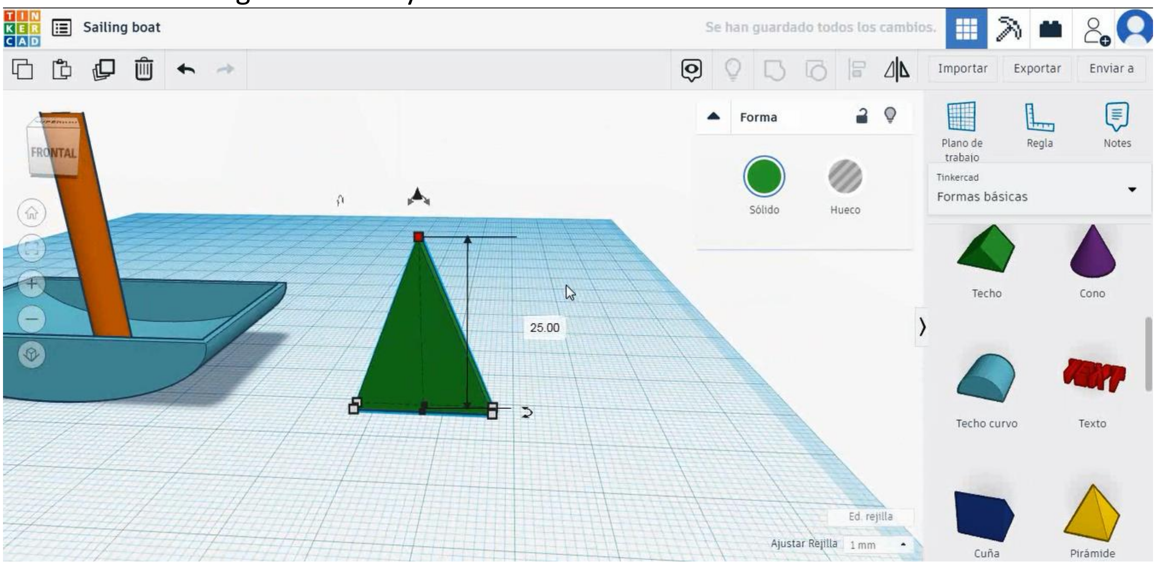


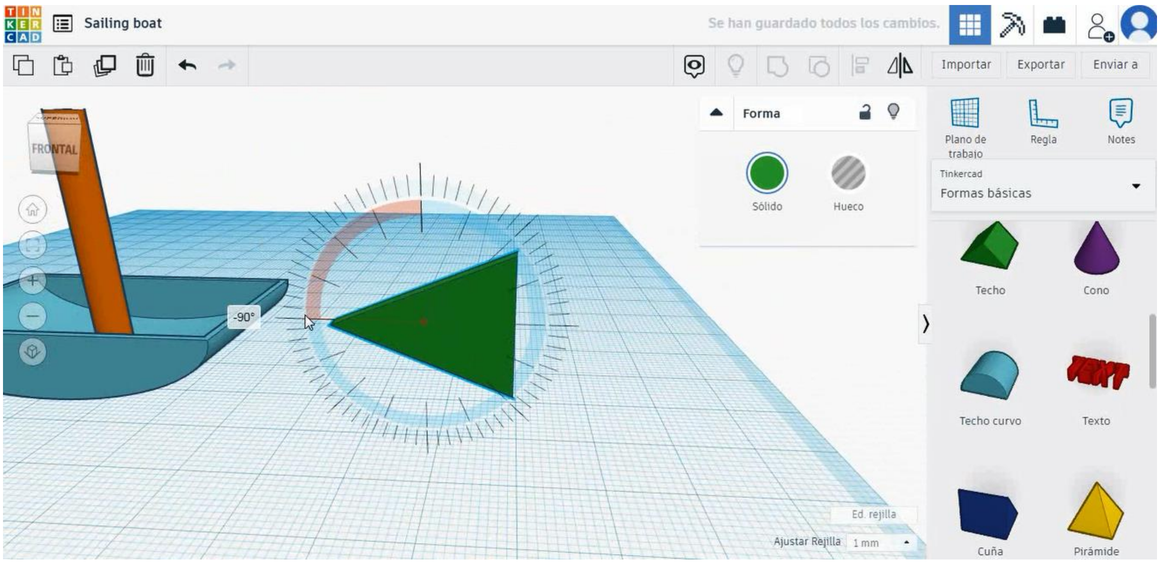
3. Select the cylinder shape and size it to 5x5x38 mm, move it to the height of 2 mm and align in the center of the round roof.

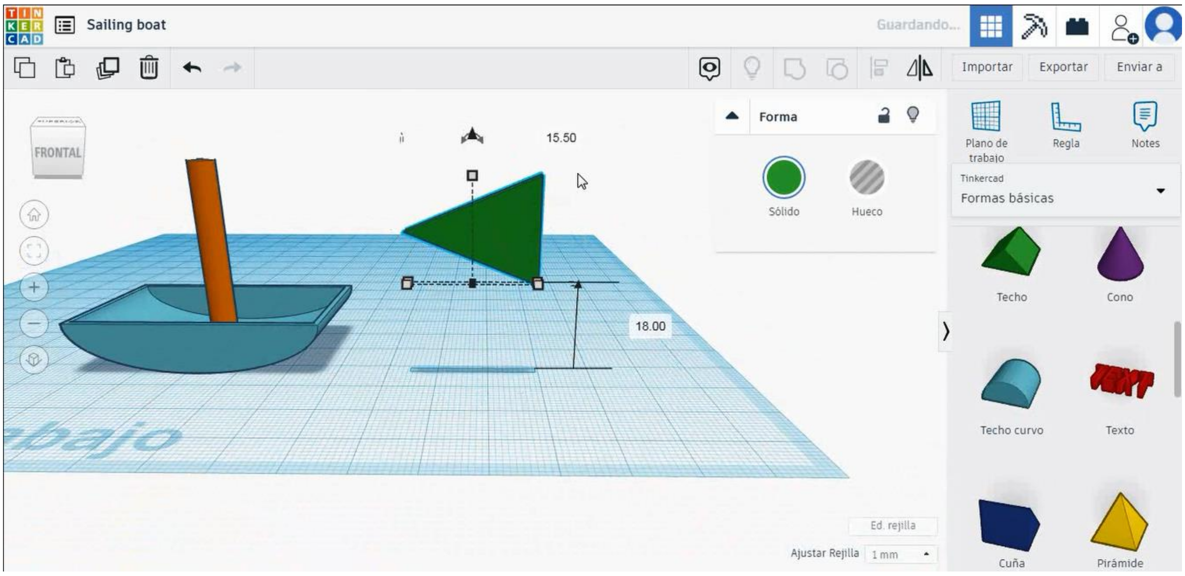


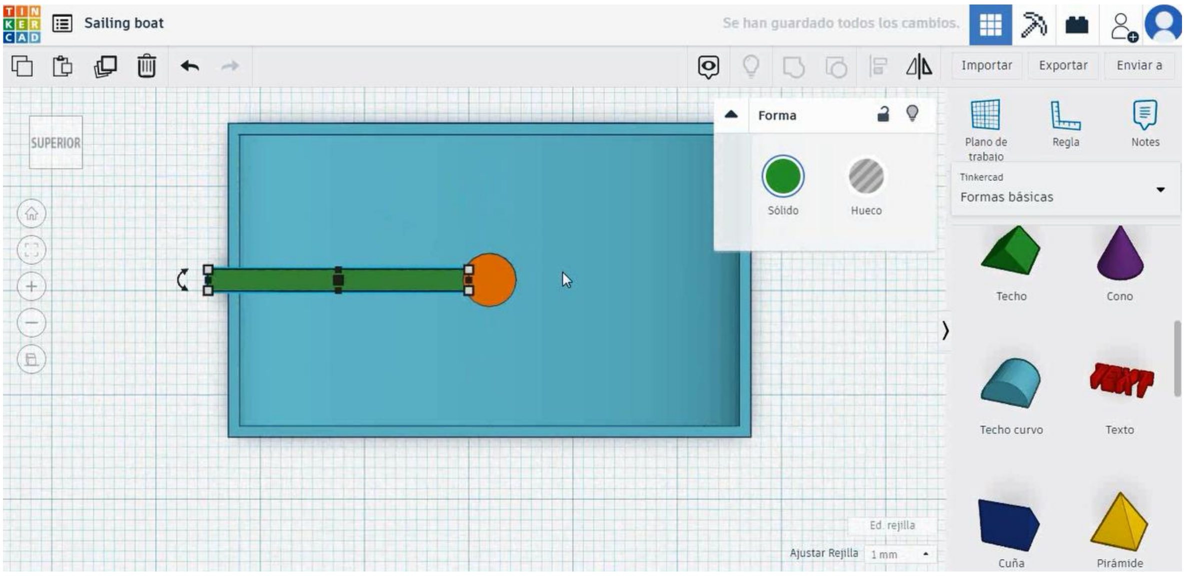


4. Select the roof shape , size it to 20x5x25 mm, turn it 90 degrees and move to the height of 18 mm. Then align with the cylinder

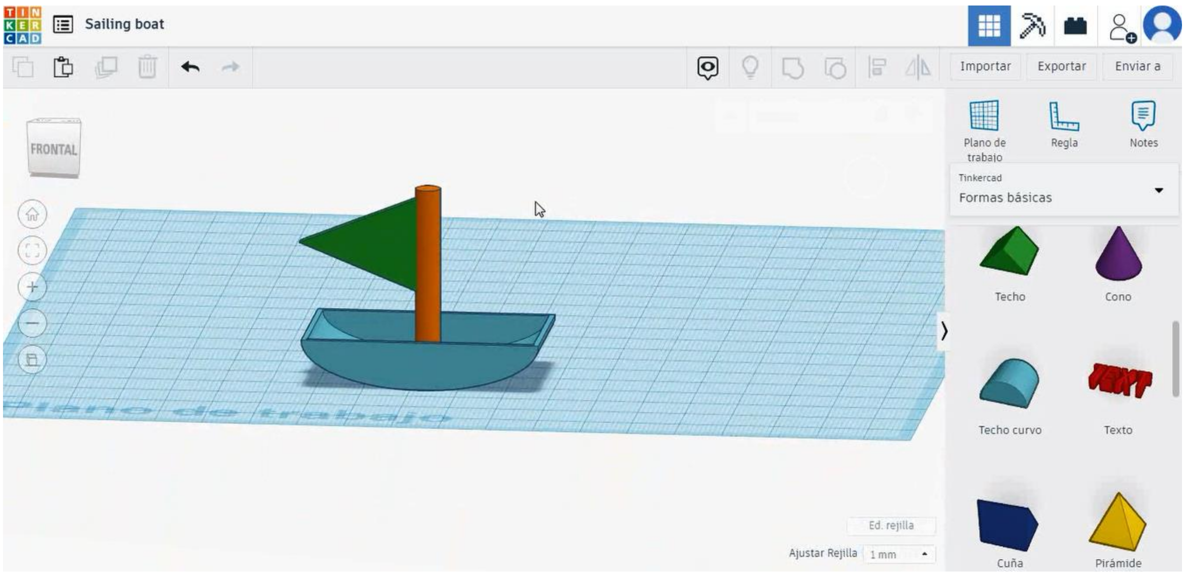








5. Now the sailing boat is finished.



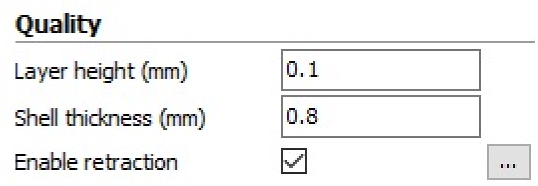
9.3.14.2 Sailing Boat 3D printing seetings

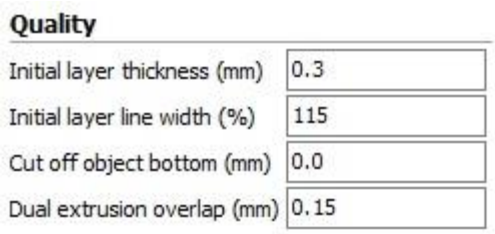
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

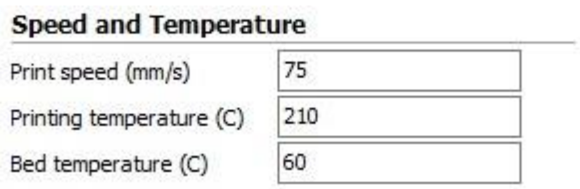
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

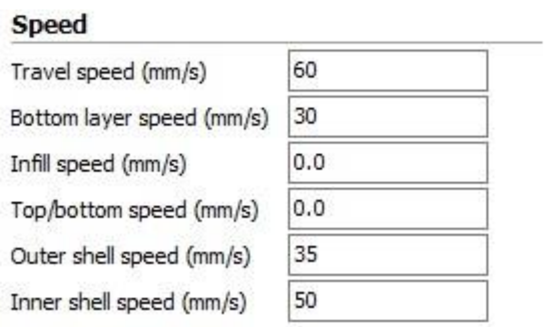




Speed and Temperature

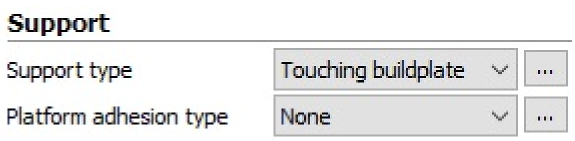
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

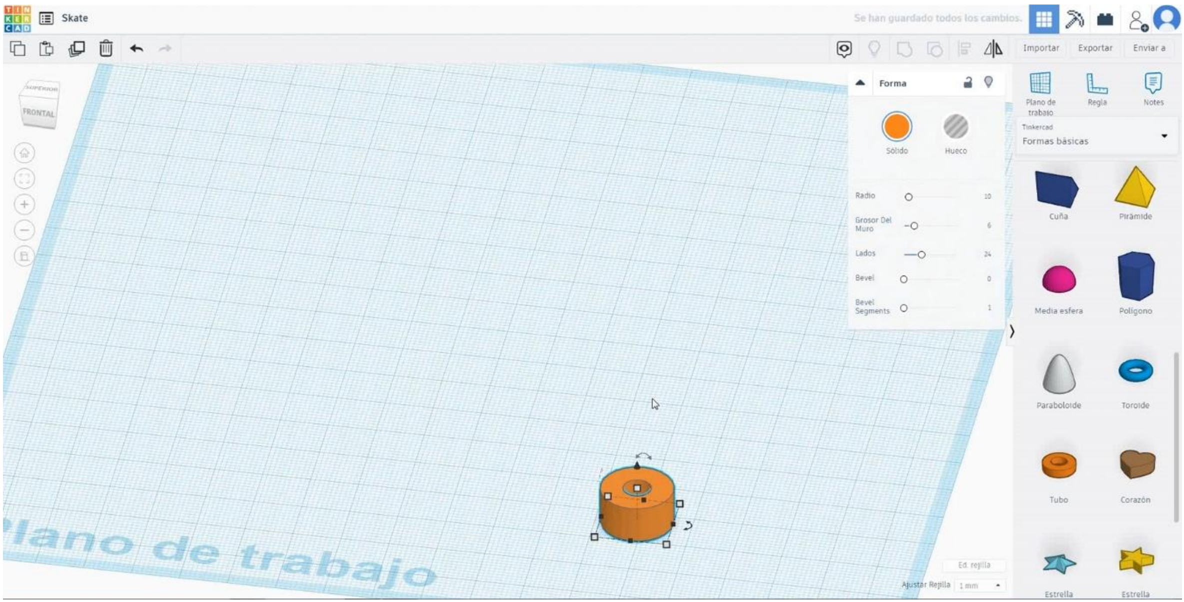
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

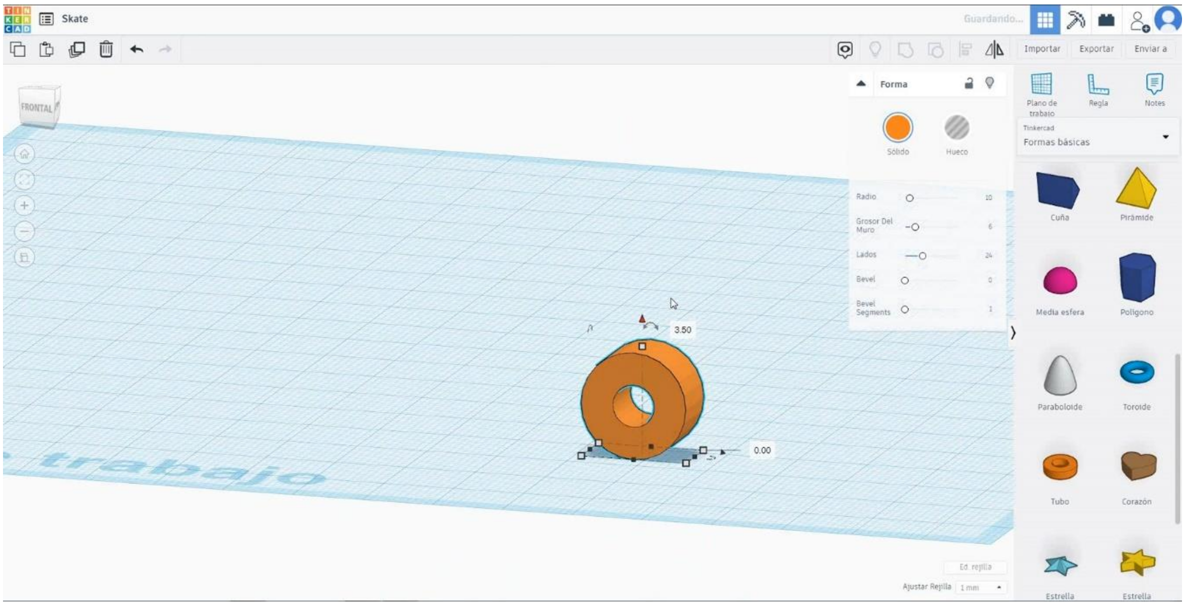


9.3.15 Part 15: Skate

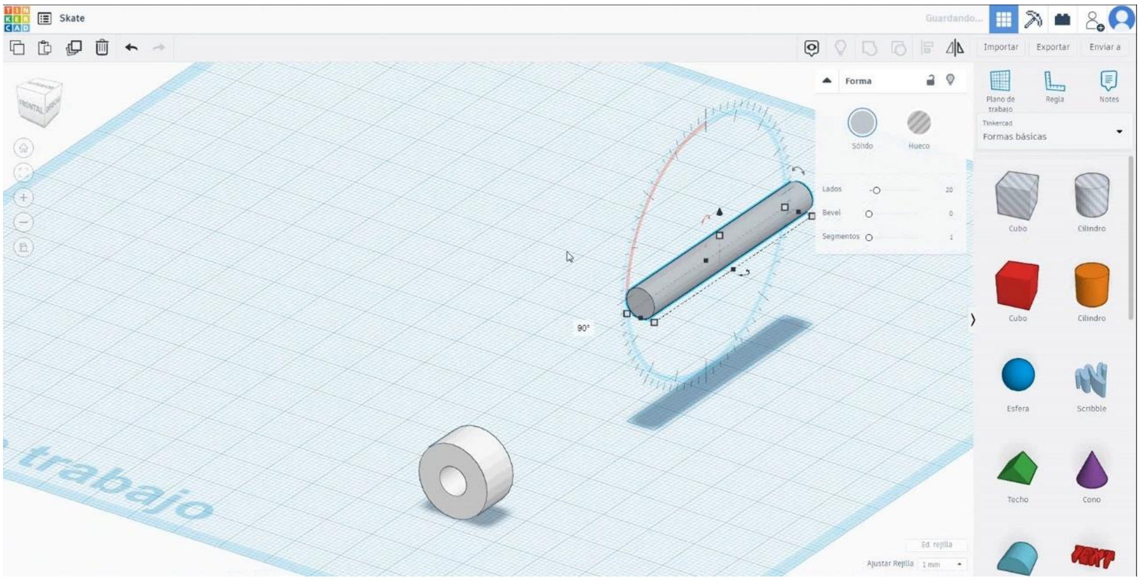
9.3.15.1 Skate Design

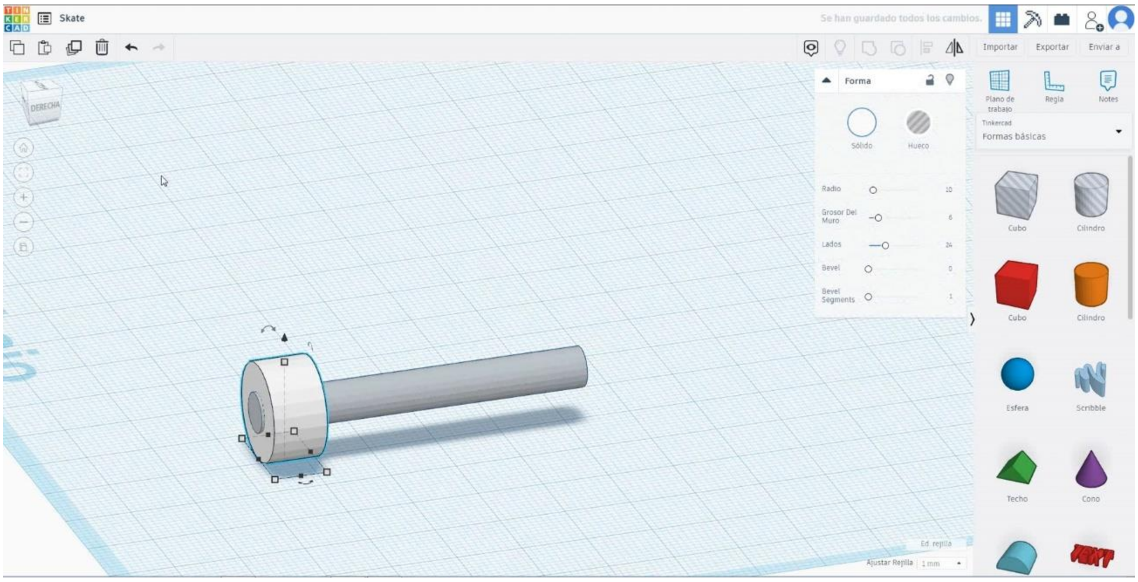
1. Choose the tube shape, size it to 15x15x8 mm with thick wall of 6mm and turn it 90 degrees. Then move it to Z 0 mm.

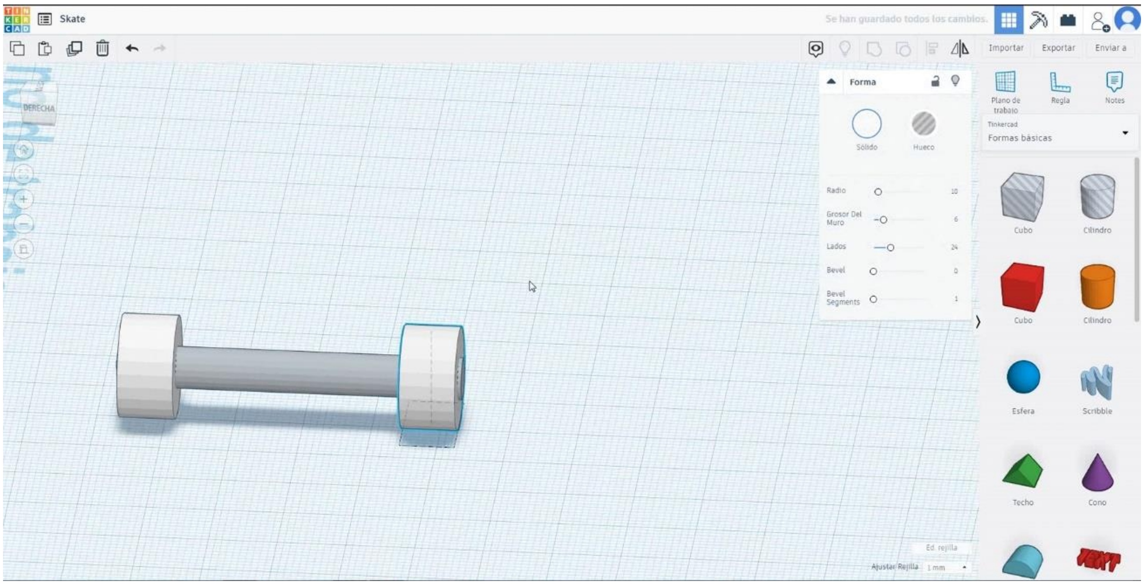


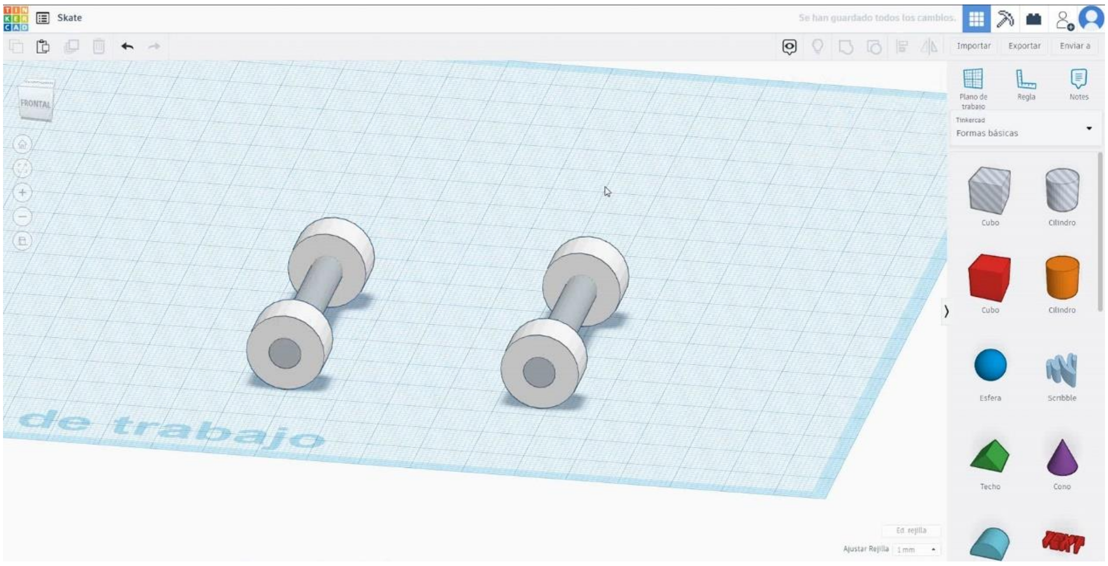


2. Choose the cylinder shape, size it to 6x6x50 mm, turn it 90 degrees and align it with in the center of the tube. Then copy the tube and move it to the opposite side to make the second wheel and copy all the set 50 mm to the left side.

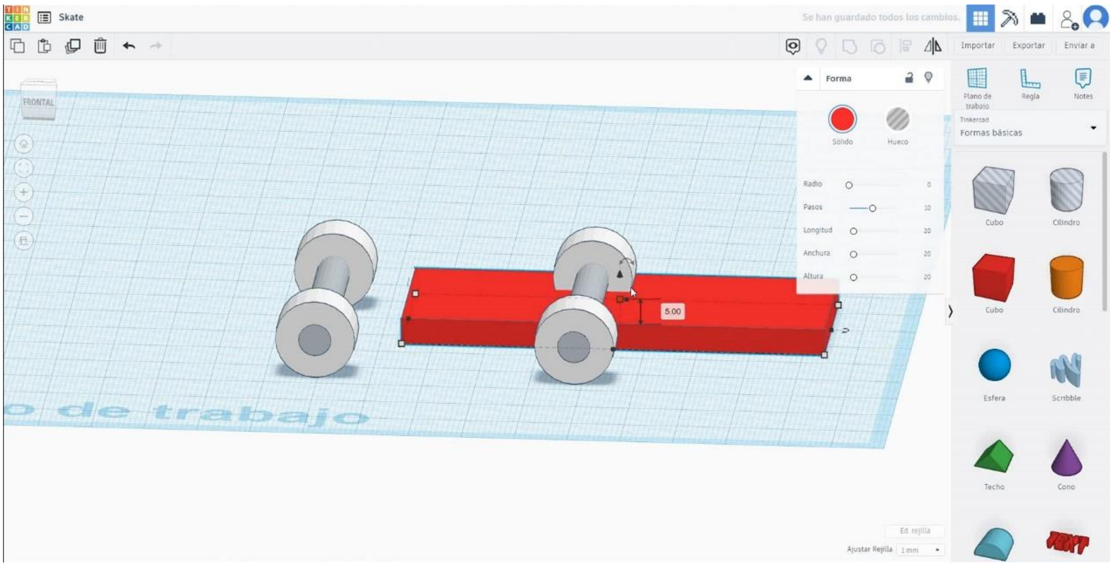

Skate
Guardando...
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
90°
trabajo
Ajustar Rejilla
1 mm

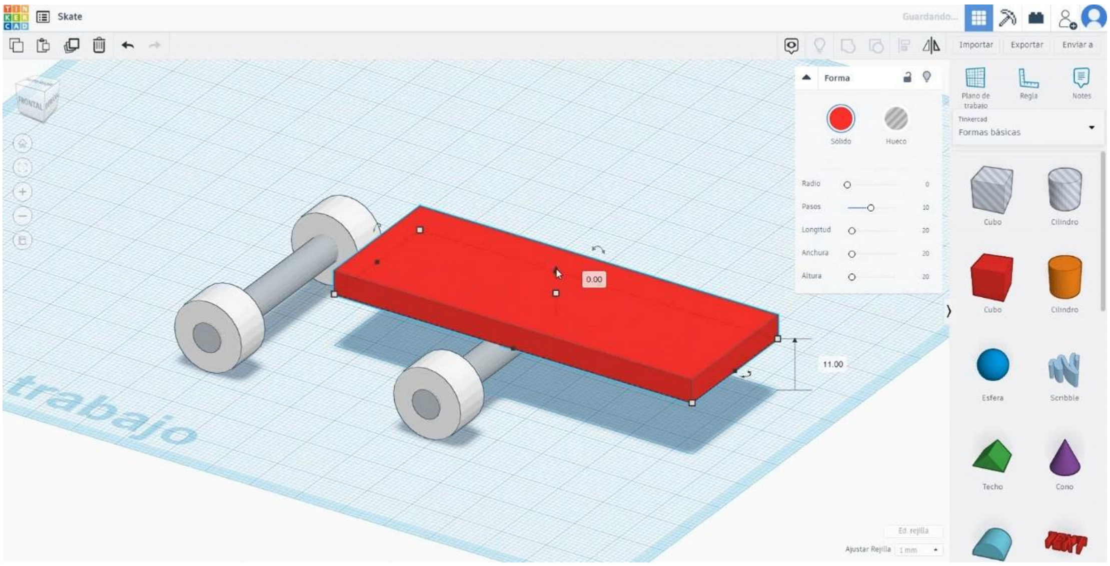

Skate
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Radio
Grosor Del Muro
Lados
Bevel
Bevel Segments
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
Ajustar Rejilla
1 mm

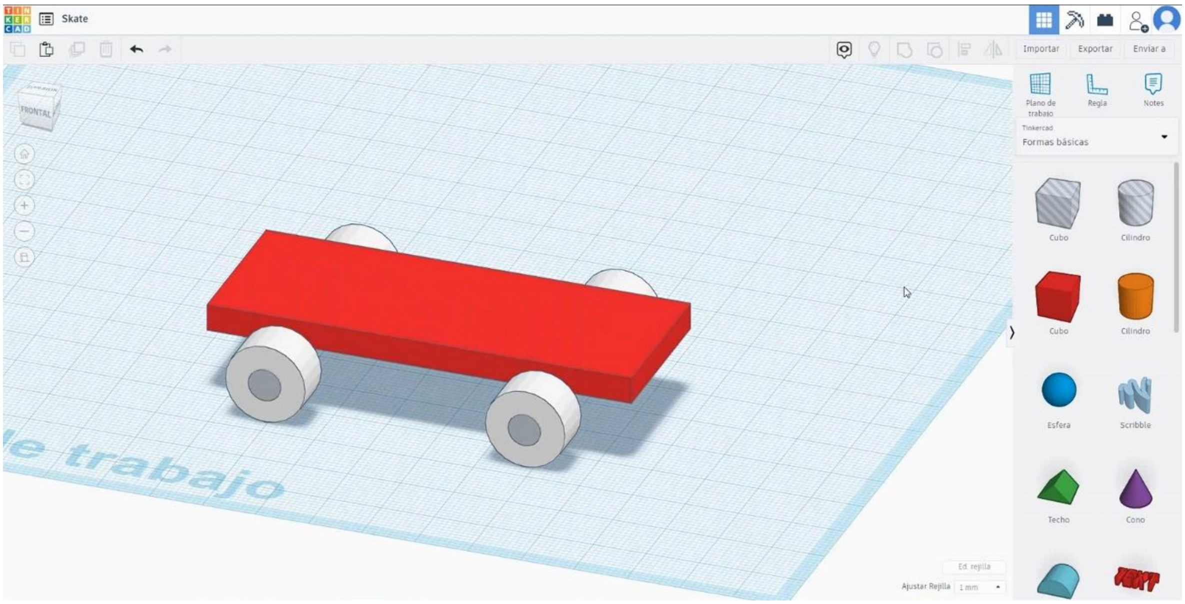

Skate
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Radio
Grosor Del Muro
Lados
Bevel
Bevel Segments
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
Ajustar Rejilla
1 mm



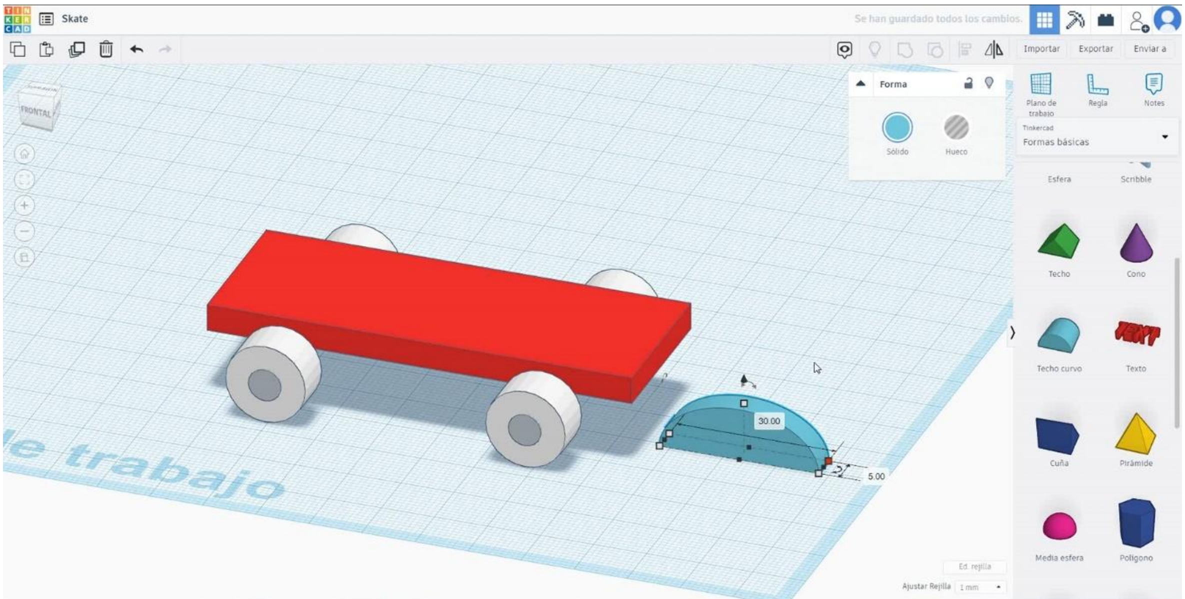
3. Choose the cube shape and size it to 80x30x5 mm and move it to the height of 11 mm. Then align in the center of the set.

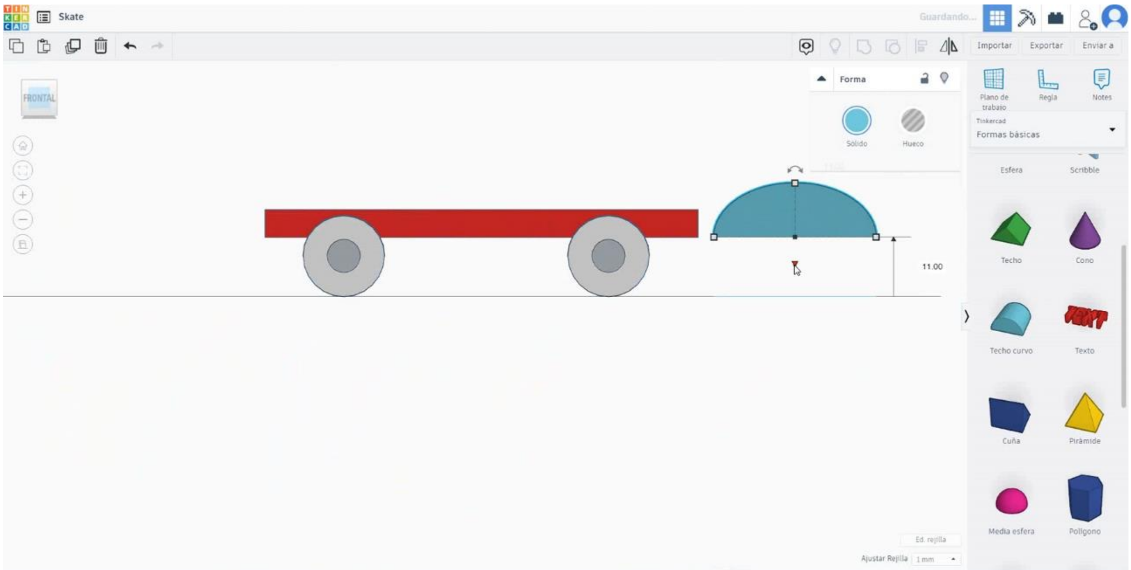


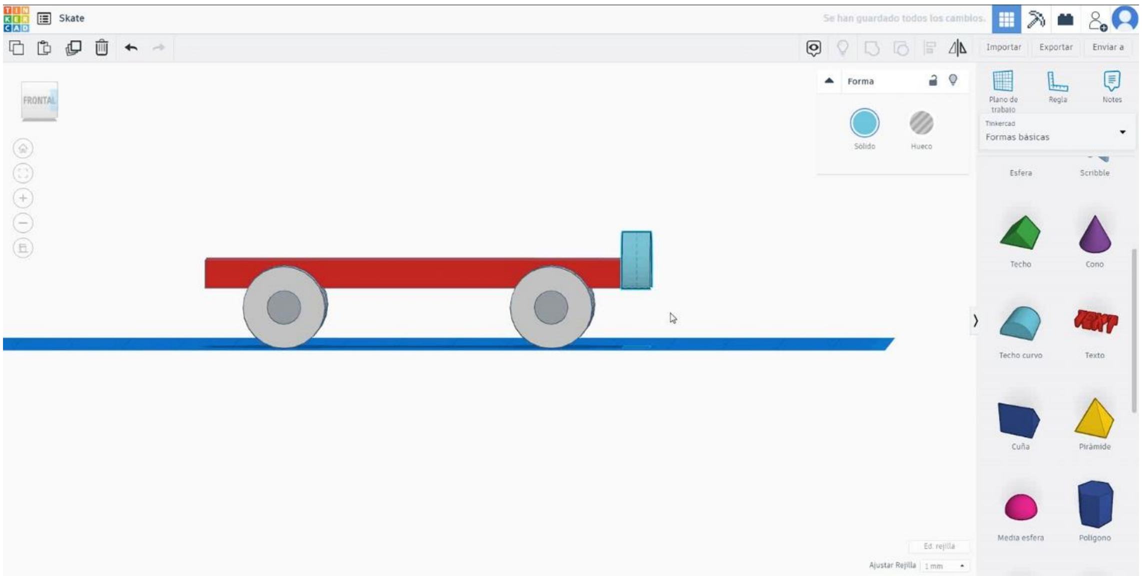


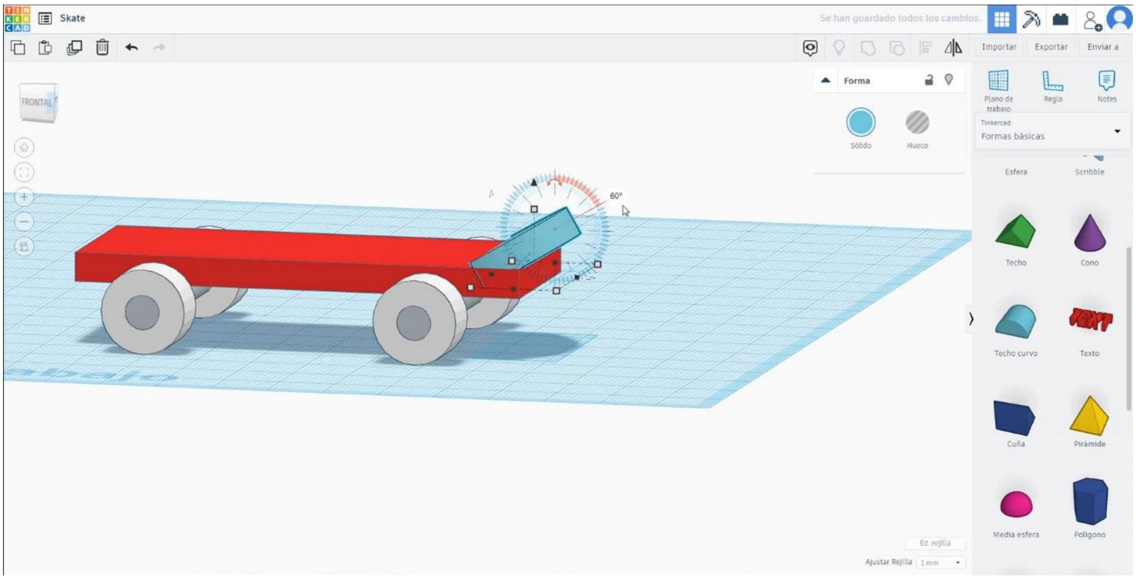


4. Choose the round roof shape and size it to 30x5x15 mm and move it to the height of 11 mm and the right edge of the cube. Then align in the center of the cube. Turn it 60 degrees and align with the right edge of the cube



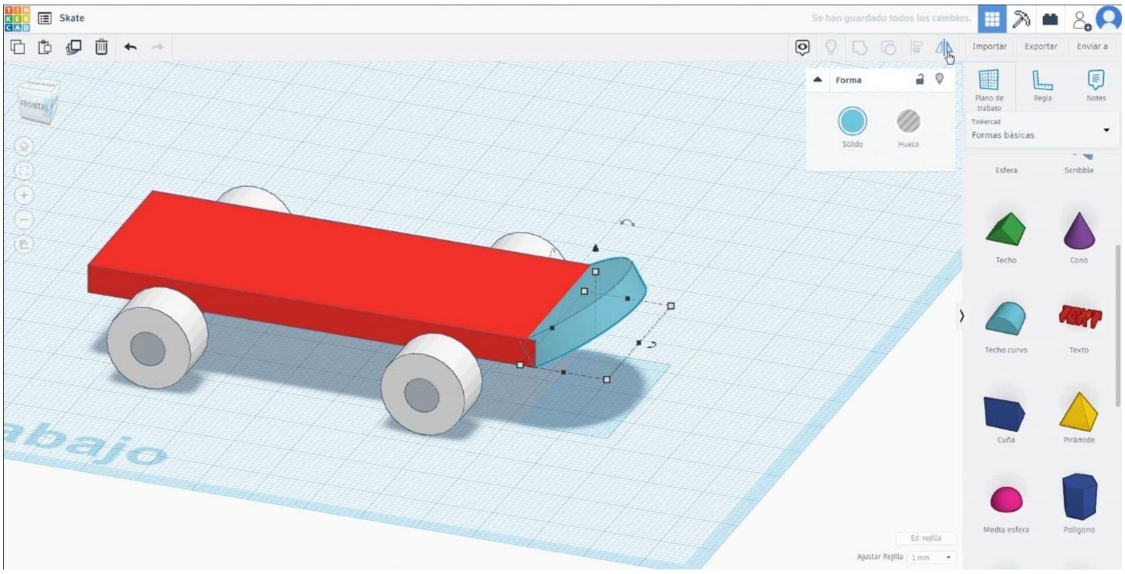


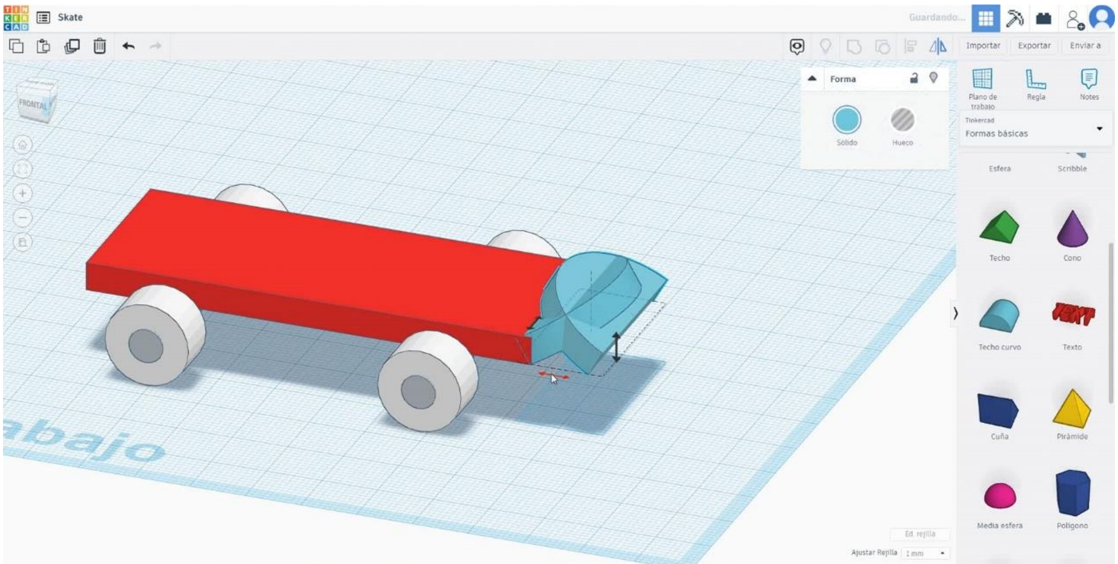


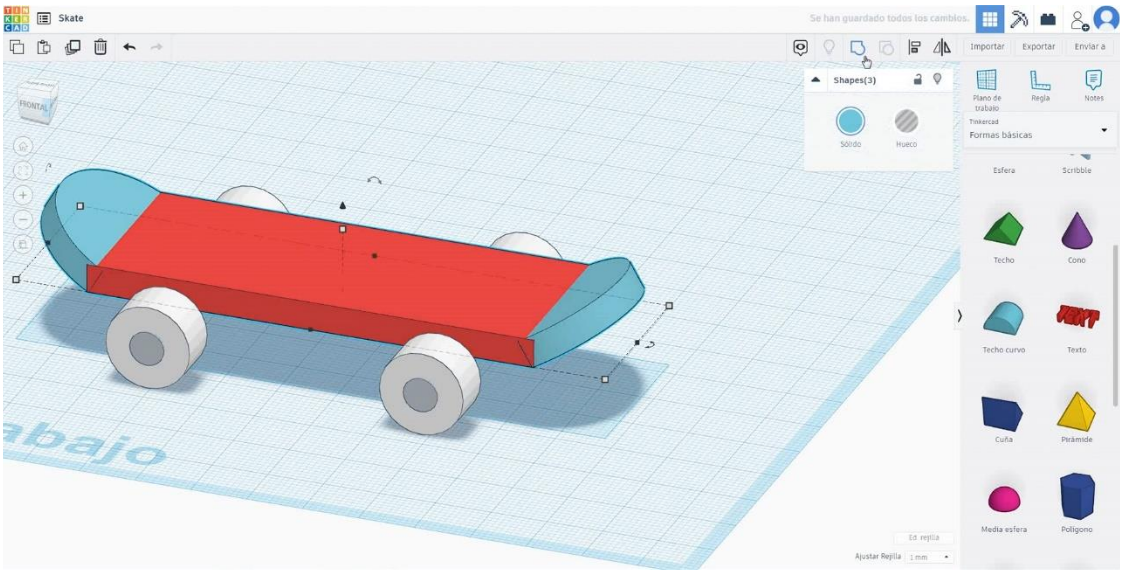


Co-funded by the Erasmus+ Programme of the European Union

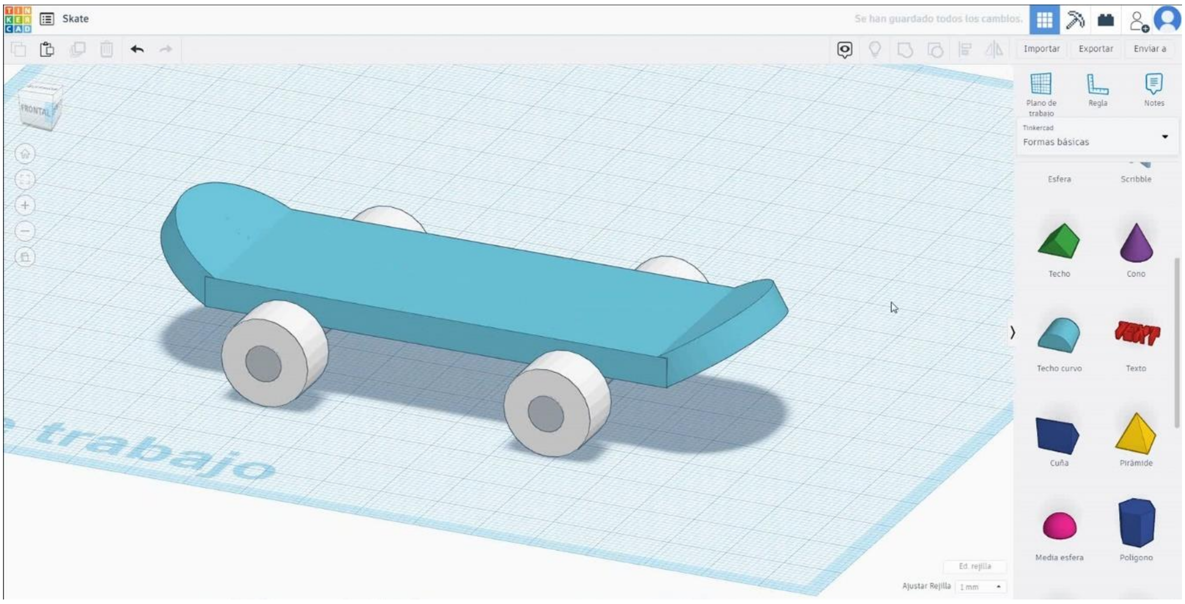
5. Select the round roof shape and press duplicate + mirror and move it to the left edge of the cube. Select the cube and the two round roof and press group.







6. Now, the skate is finished.



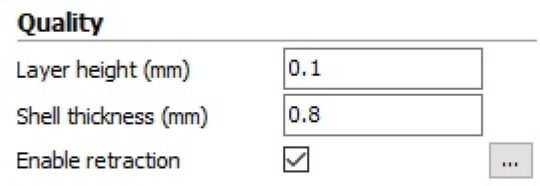
9.3.15.2 Skate 3D printing seetings

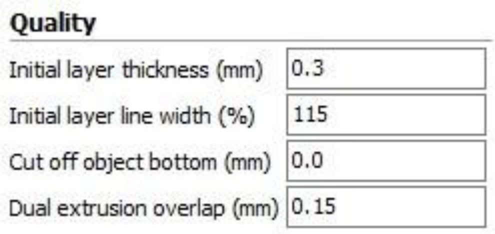
<u>Filament</u>

PLA
Diameter - 1.75 (mm)
Flow - 100%

<u>Quality</u>

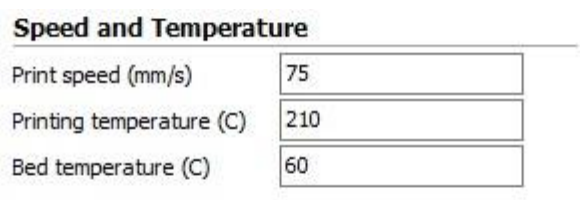
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

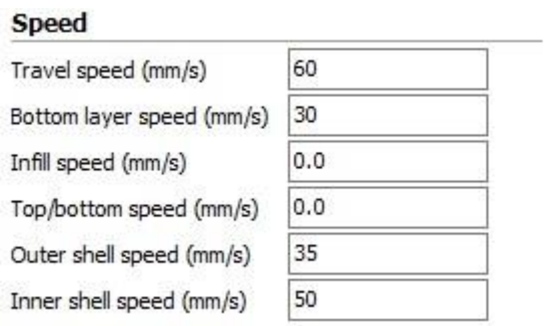




<u>Speed and Temperature</u>

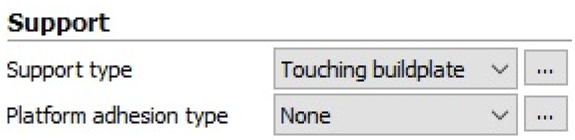
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





<u>Support Type</u>

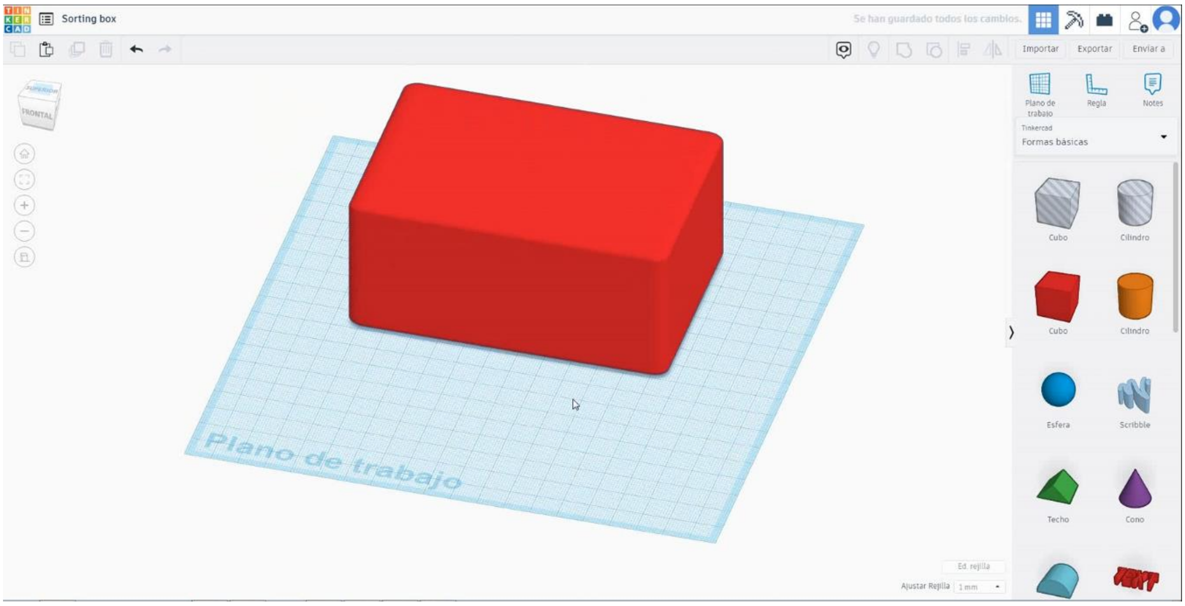
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



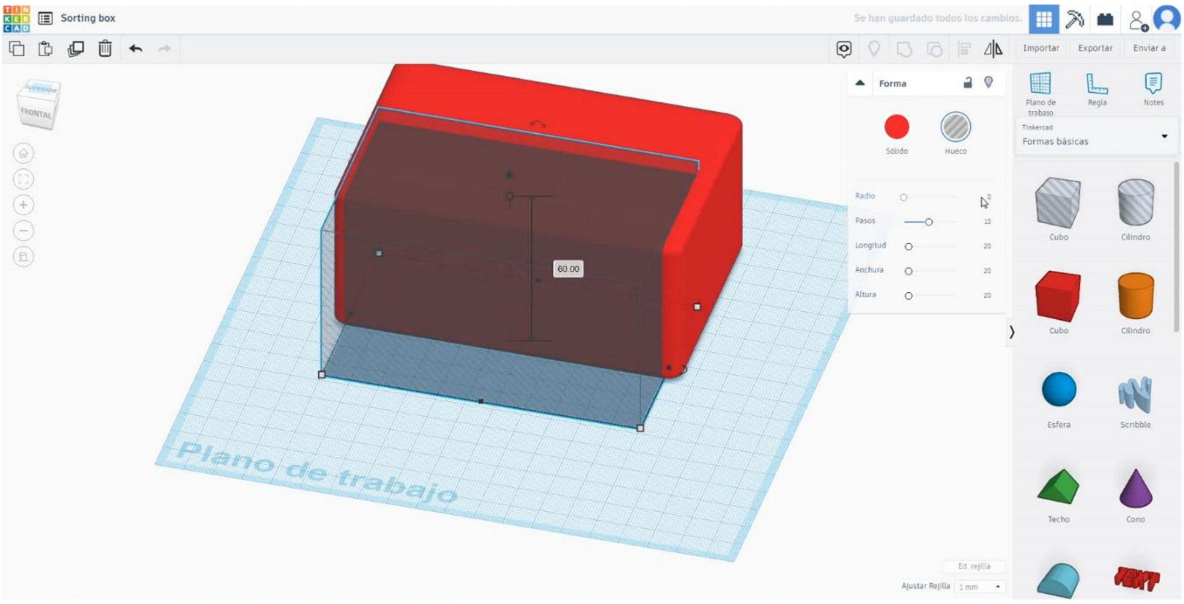
9.3.16 Part 16: Sorting box

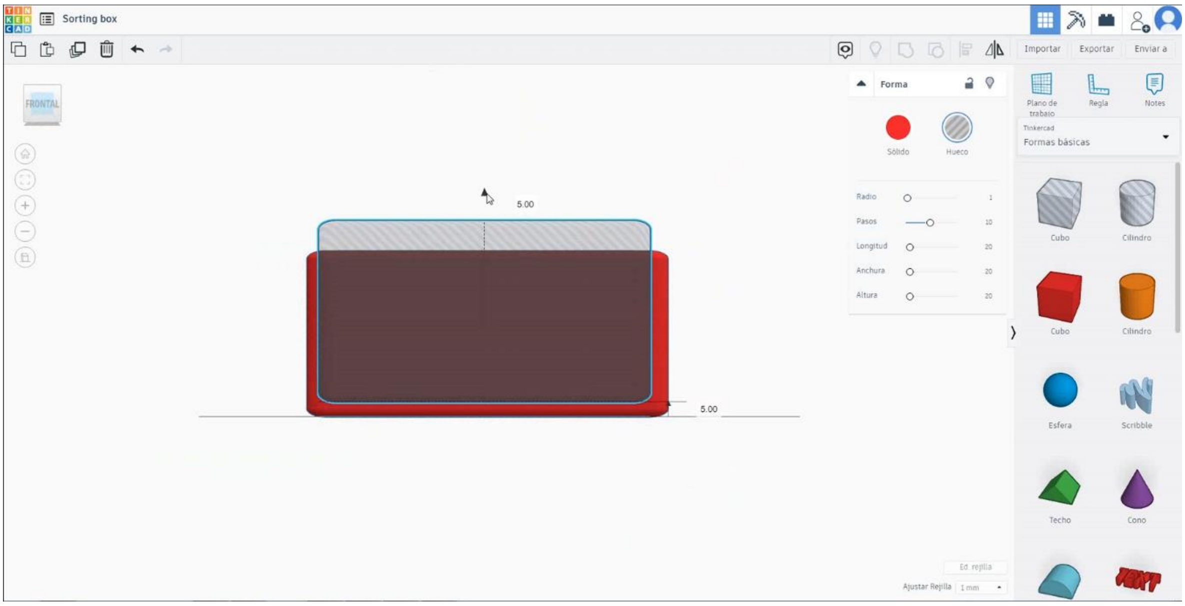
9.3.16.1 Sorting box Design

1. Choose the cube shape and size it to 120x80x55 mm with radius 1 mm and center it in the workplane.

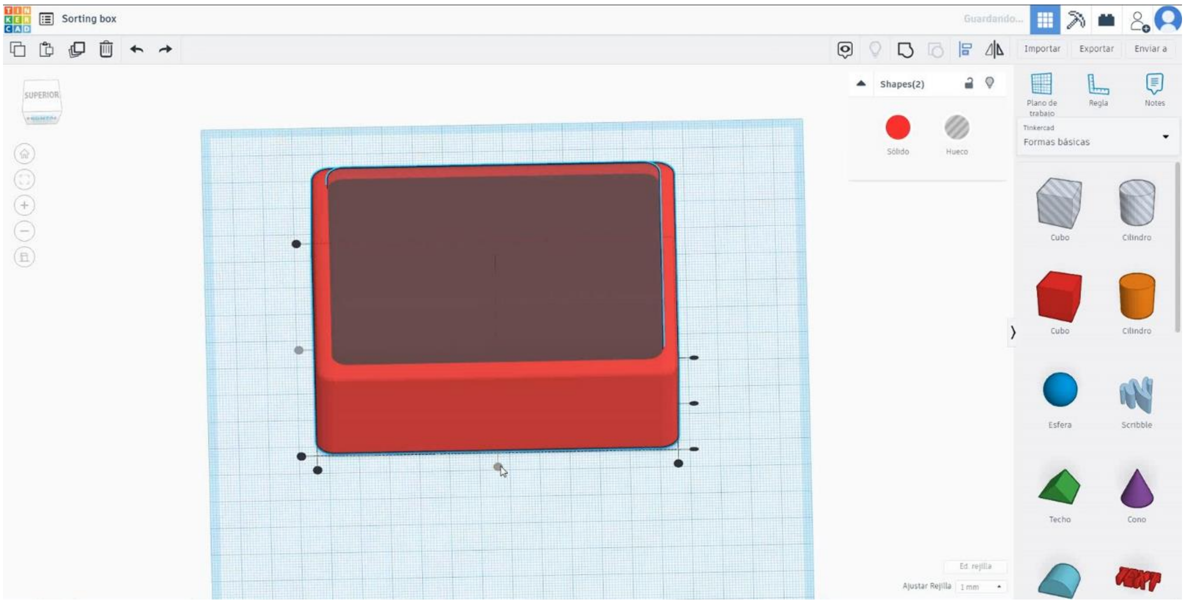


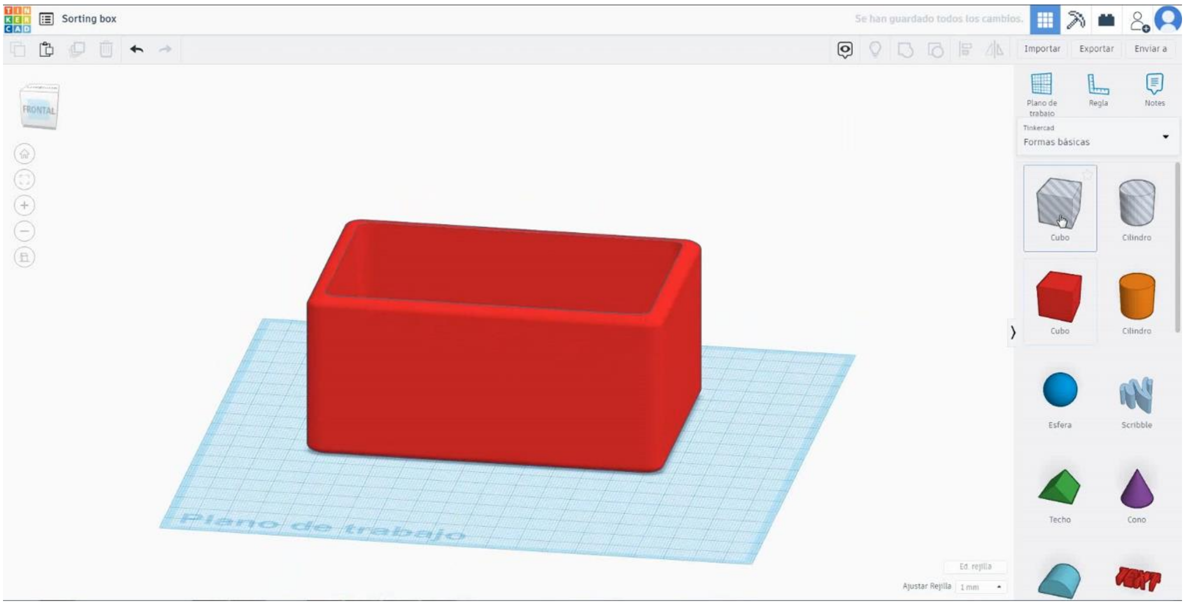
2. Choose the cube shape in hole mode and size it to 110x70x60 mm with radius 1 mm and move it to the height of 5 mm



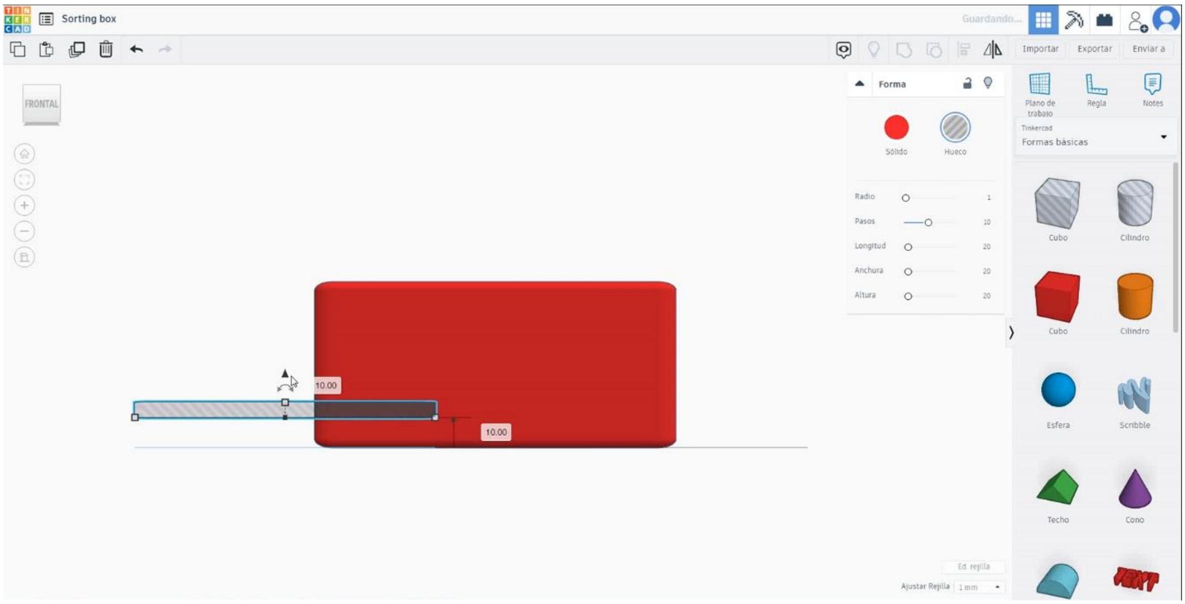


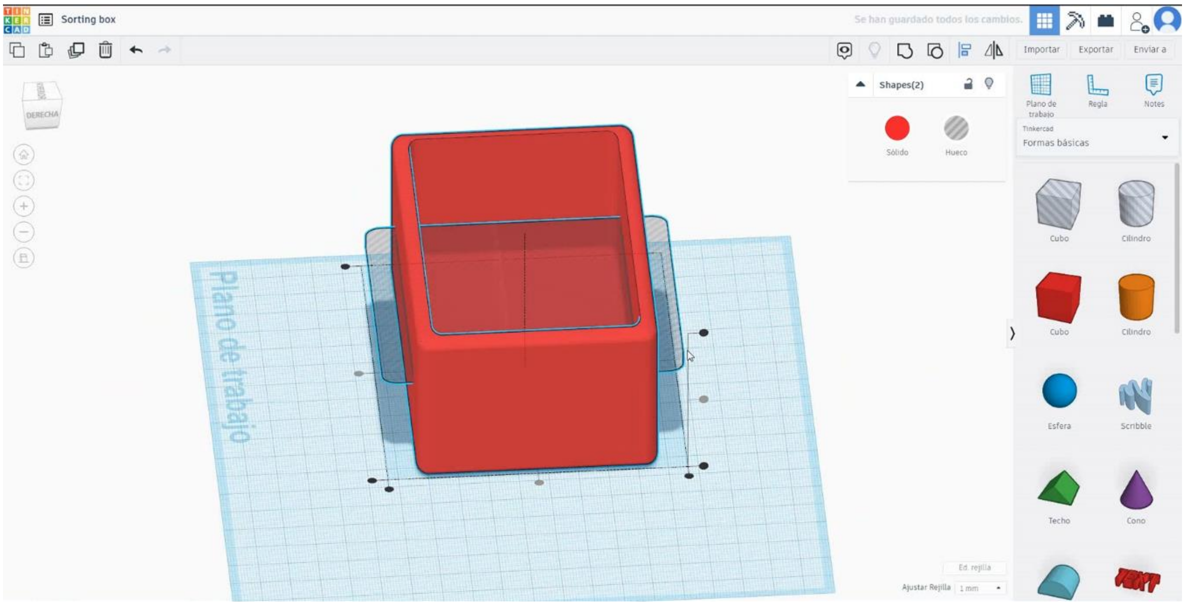
3. Select both objects and press align to center both pieces. Then select both objects and press group to empty the cube.



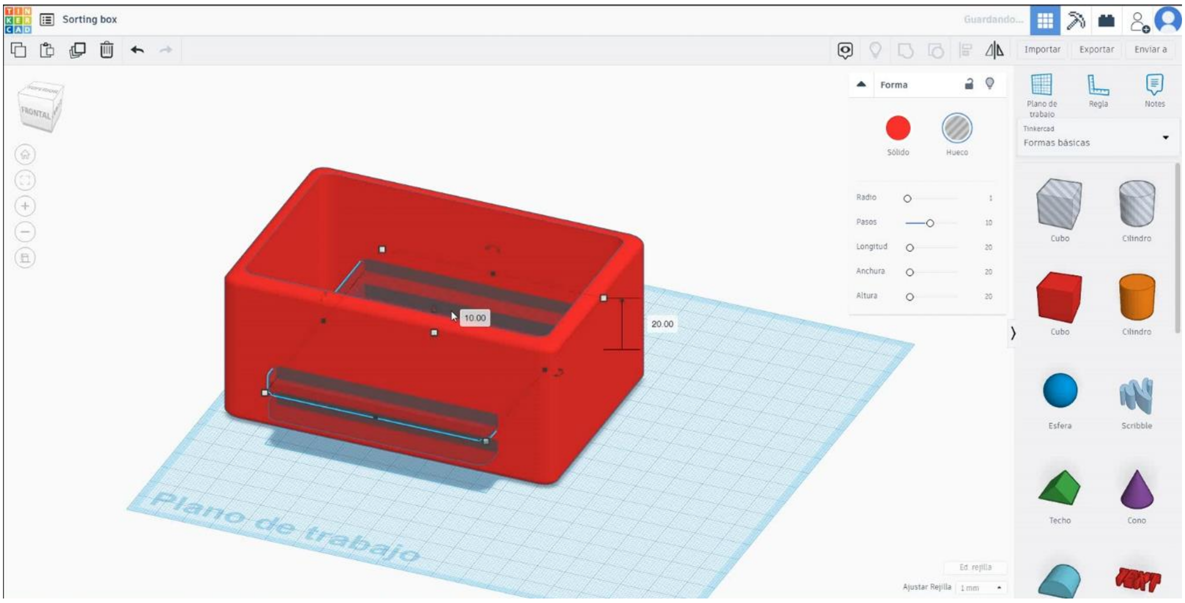


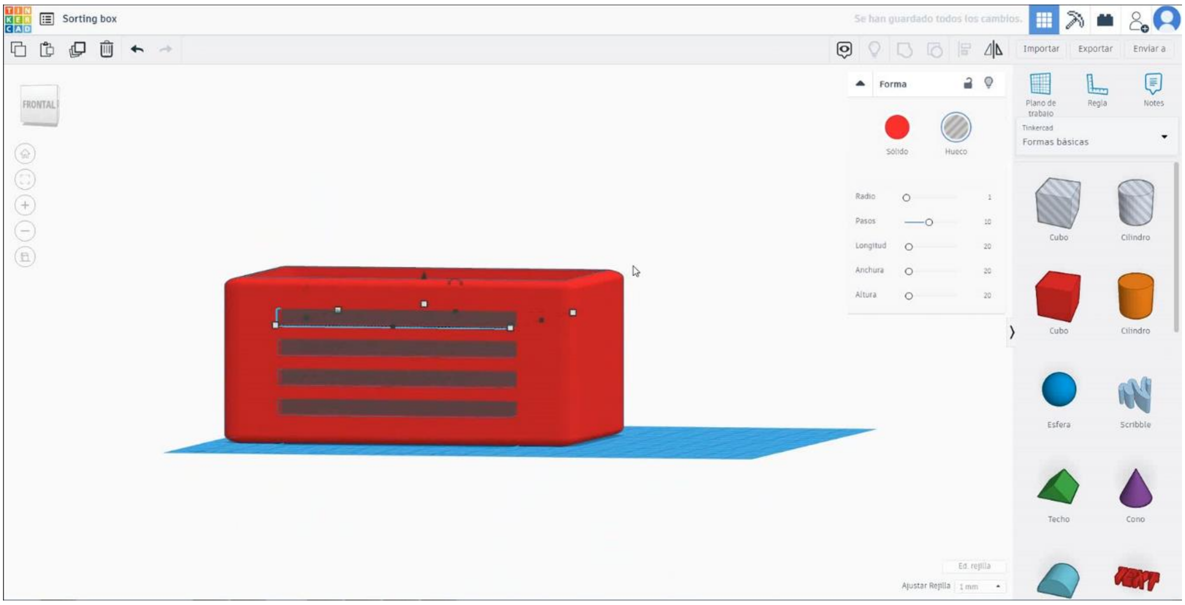
4. Choose the cube shape in hole mode and size it to 80x100x5 mm with radius 1 mm and move it to the height of 10 mm. Then align in the center of the first cube and make sure it appears in both sides.

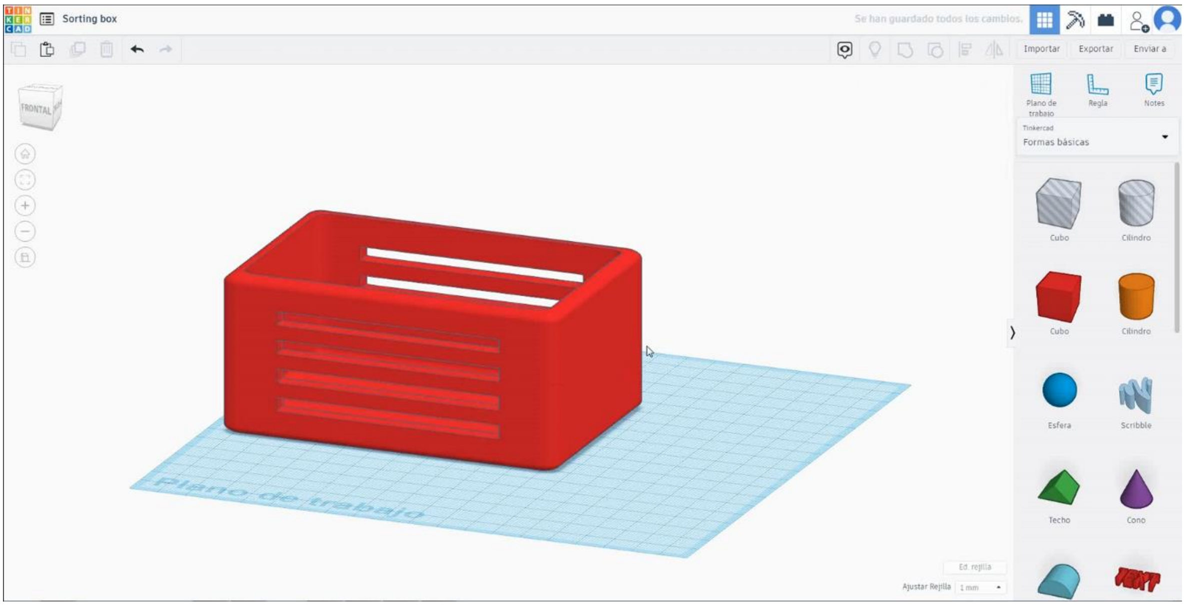




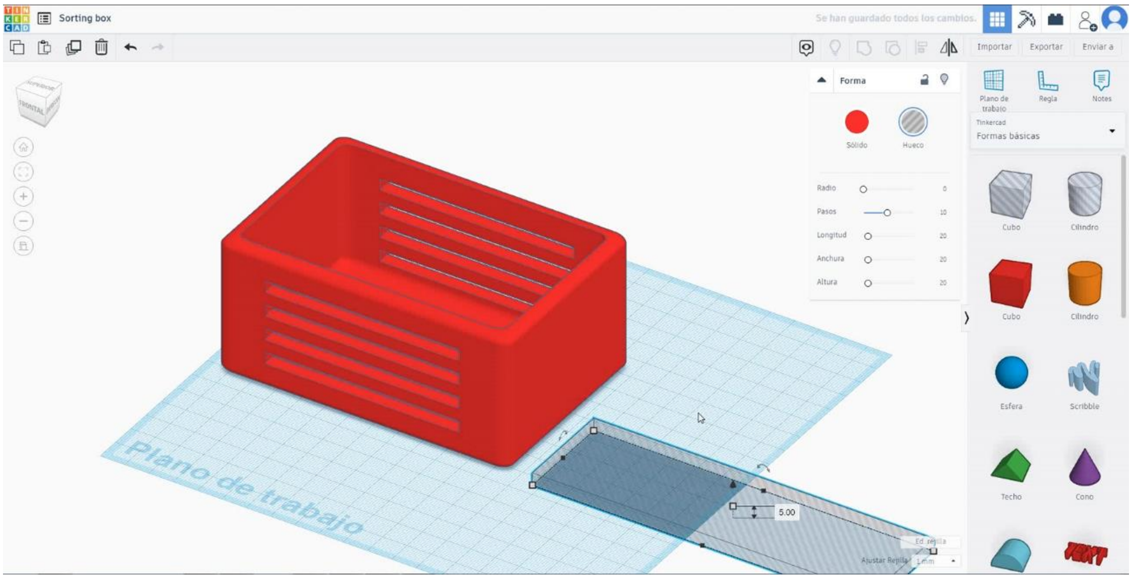
5. Select the cube in hole mode, press duplicate and move it 20 mm in height. Then press two times duplicate to make it automatically. After that, select all the objects and press group.

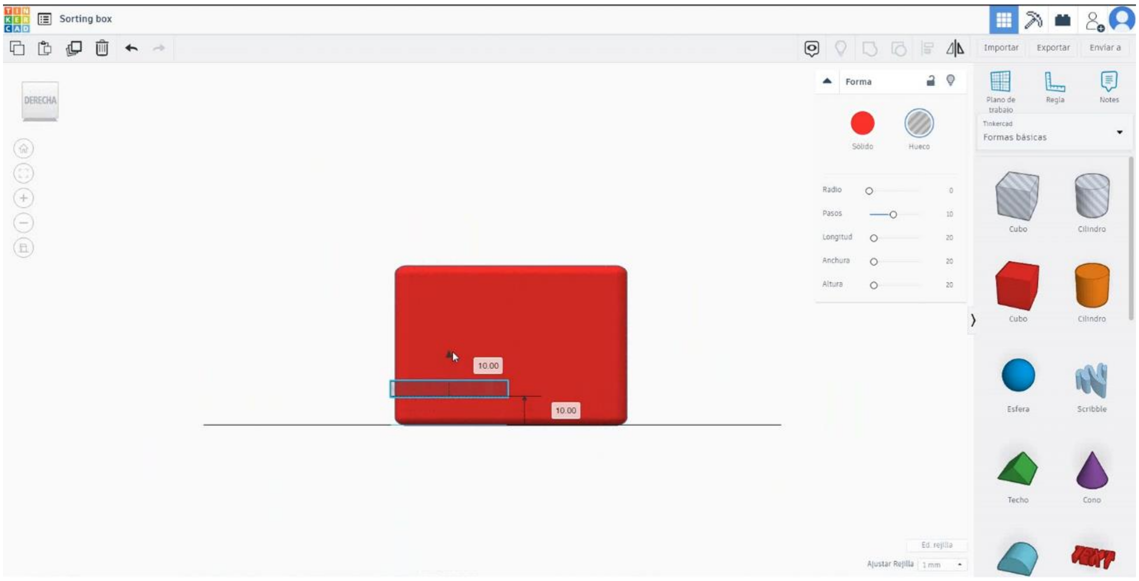


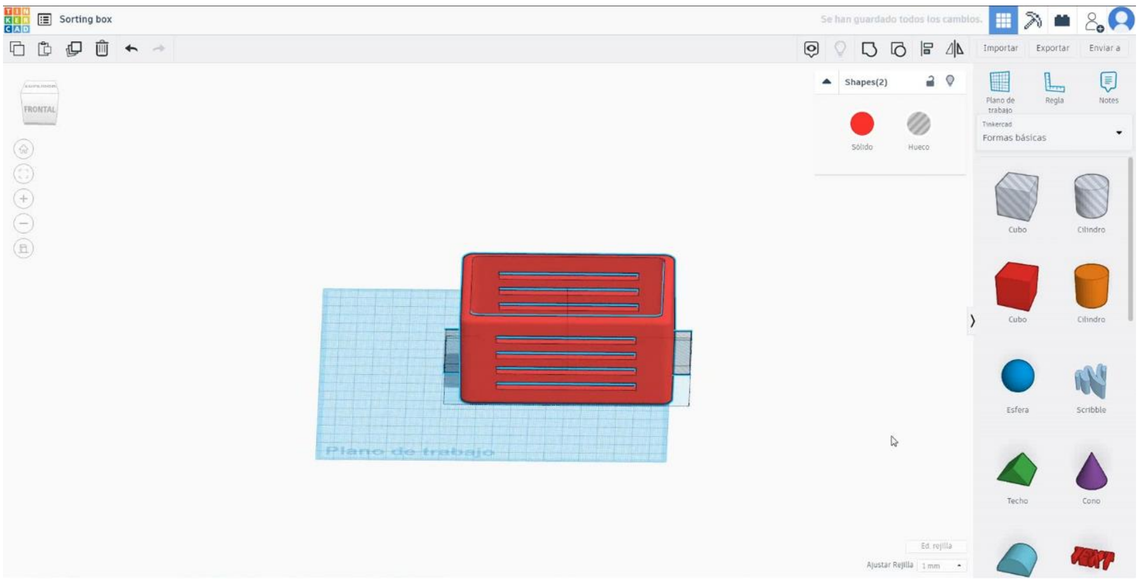




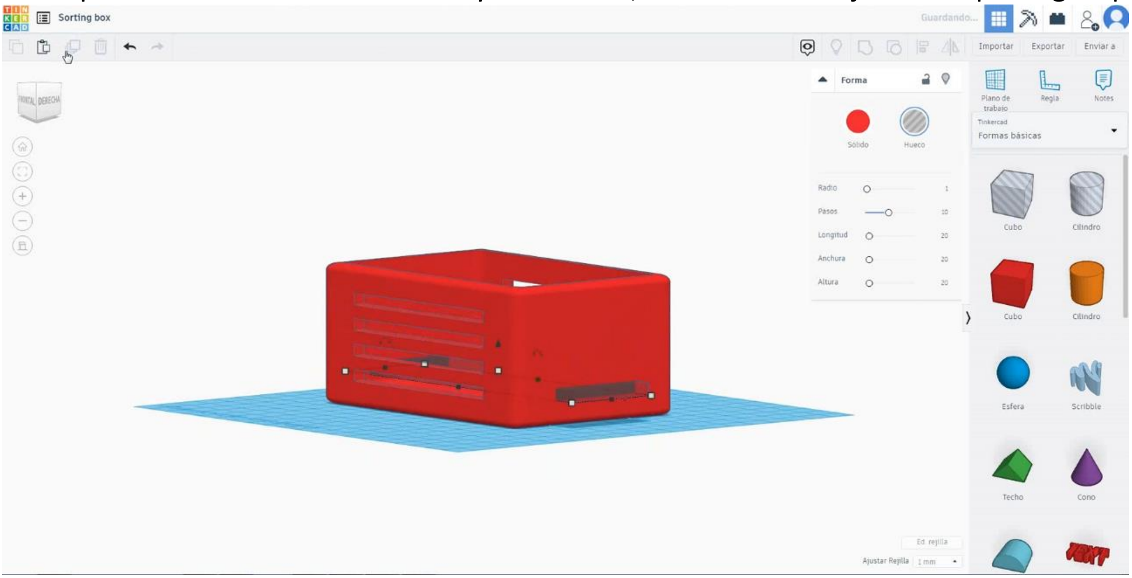
6. Choose the cube shape in hole mode and size it to 140x40x5 mm with radius 1 mm and move it to the height of 10 mm. Then align in the center of the first cube and make sure it appears in both sides.

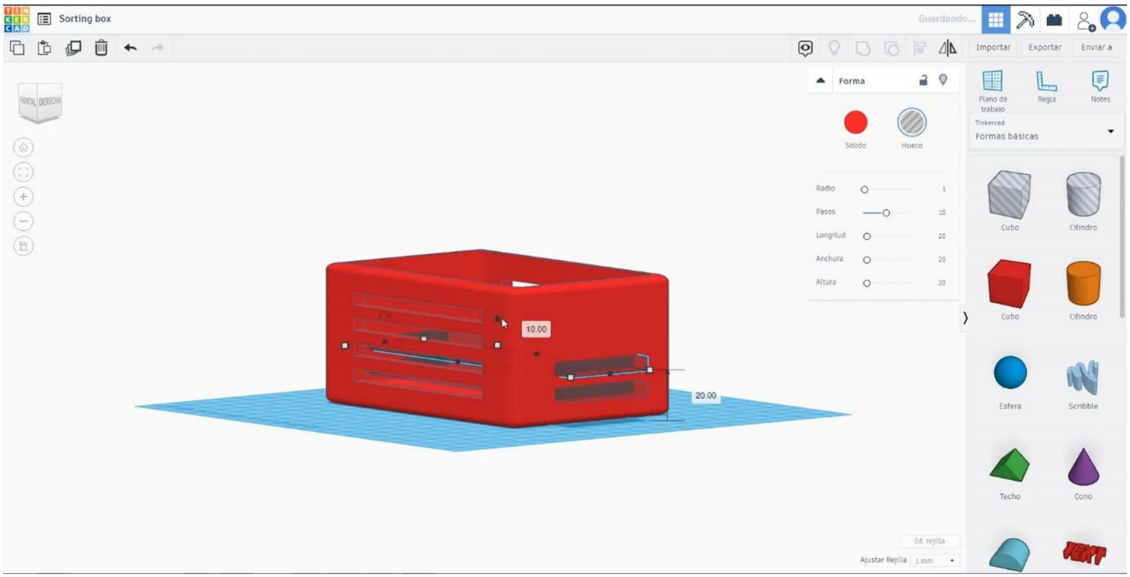


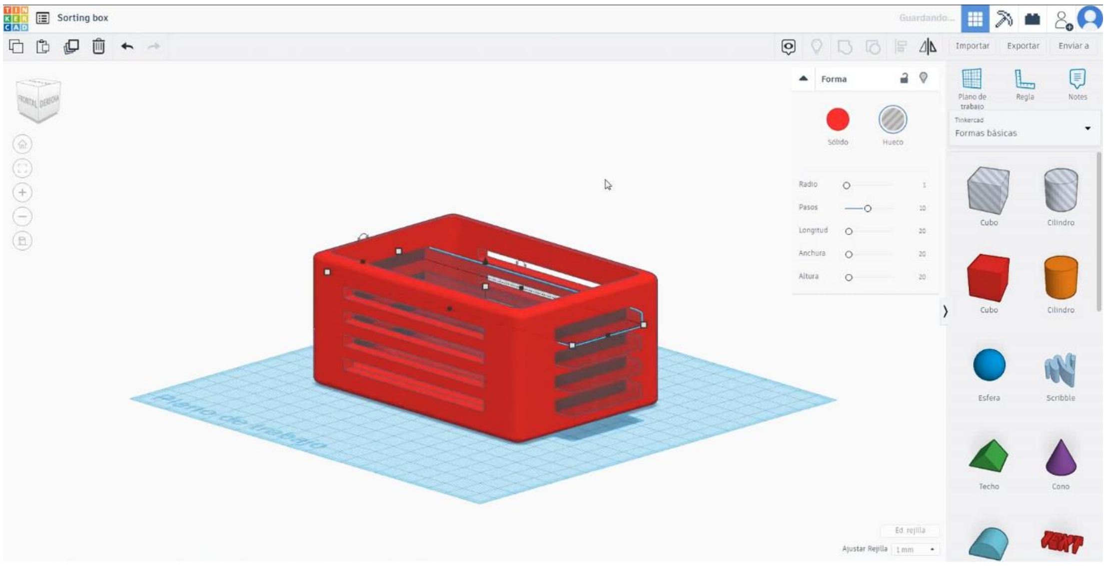




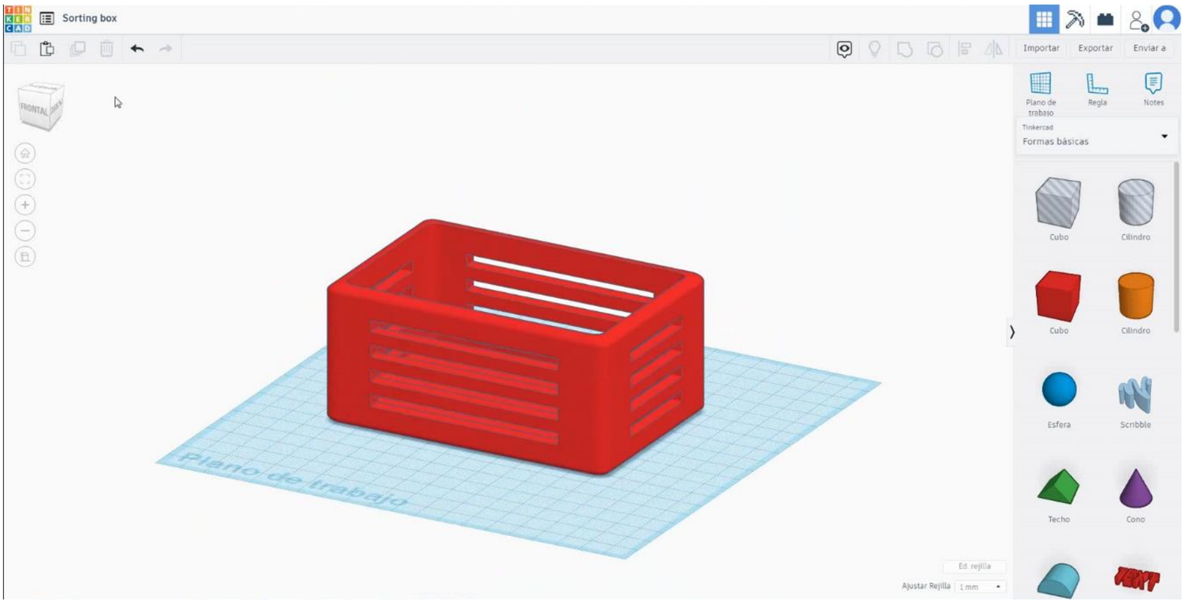
7. Select the cube in hole mode, press duplicate and move it 20 mm in height. Then press two times duplicate to make it automatically. After that, select all the objects and press group.







8. Now, the sorting box is finished.



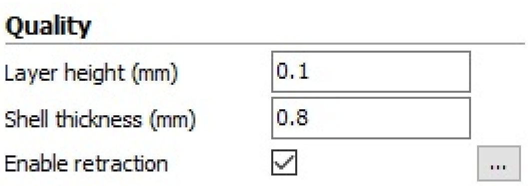
9.3.16.2 Sorting box3D printing seetings

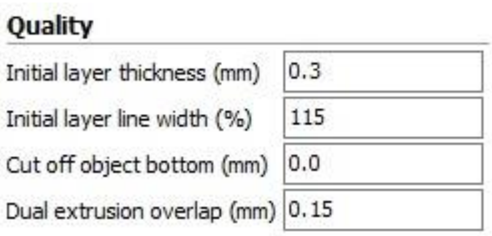
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

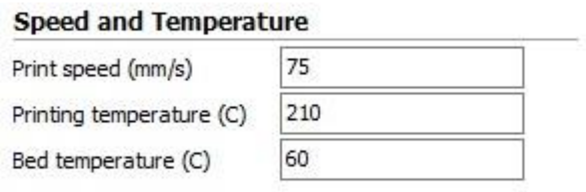
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

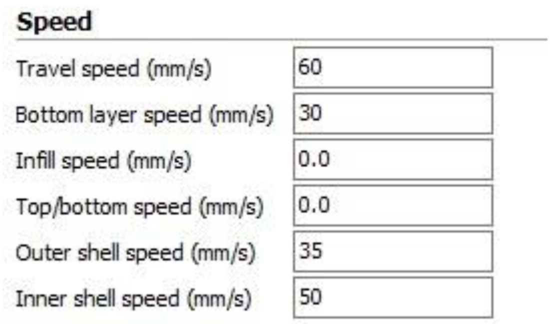




Speed and Temperature

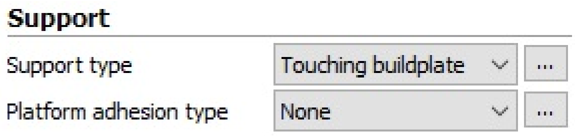
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

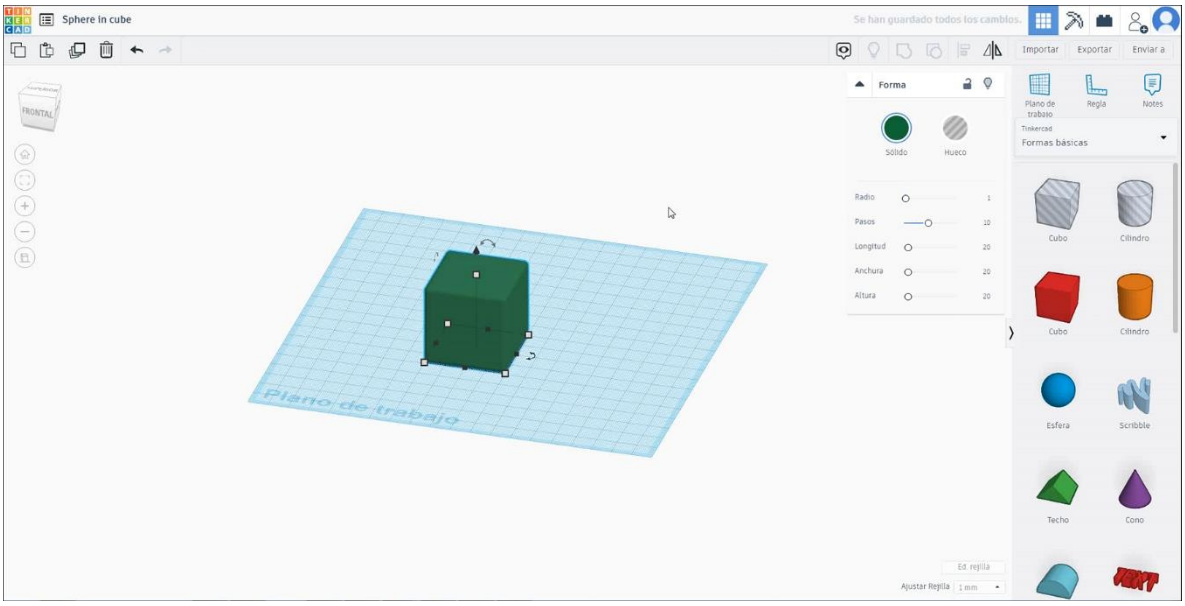
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft



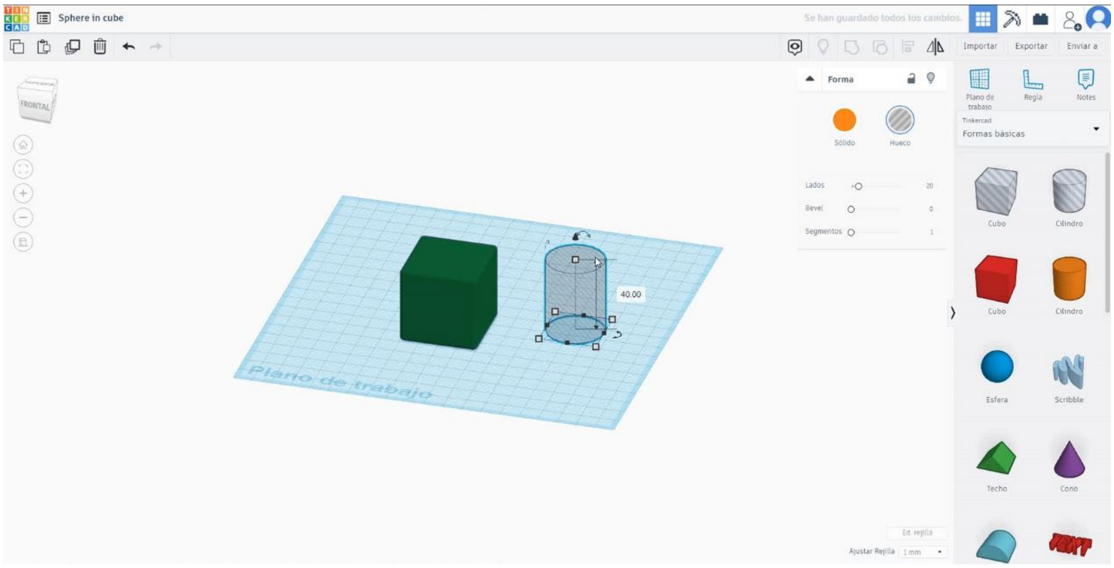
9.3.17 Part 17: Sphere in cube

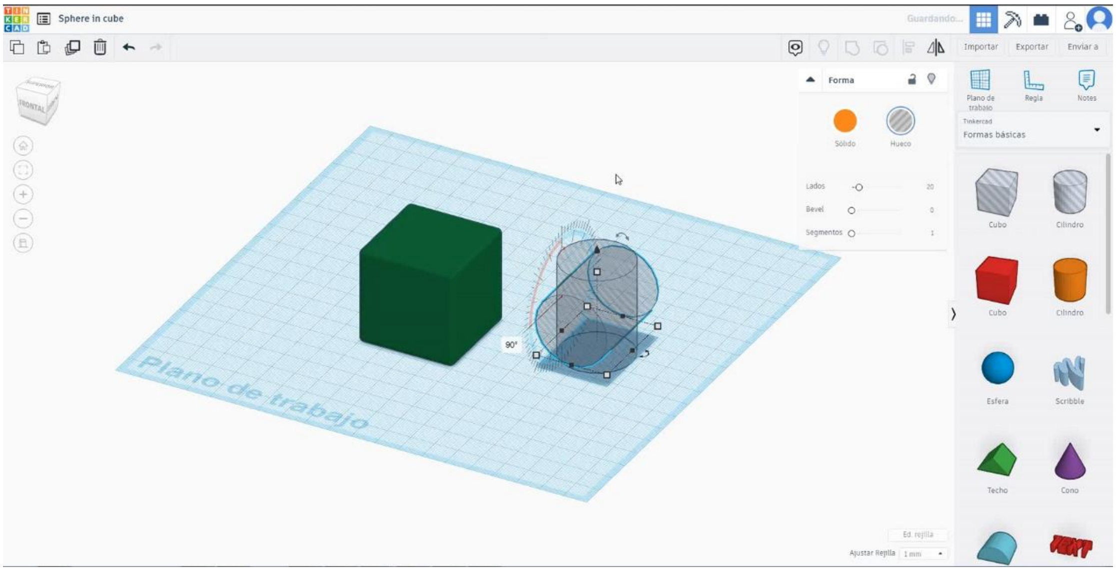
9.3.17.1 Sphere in cube Design

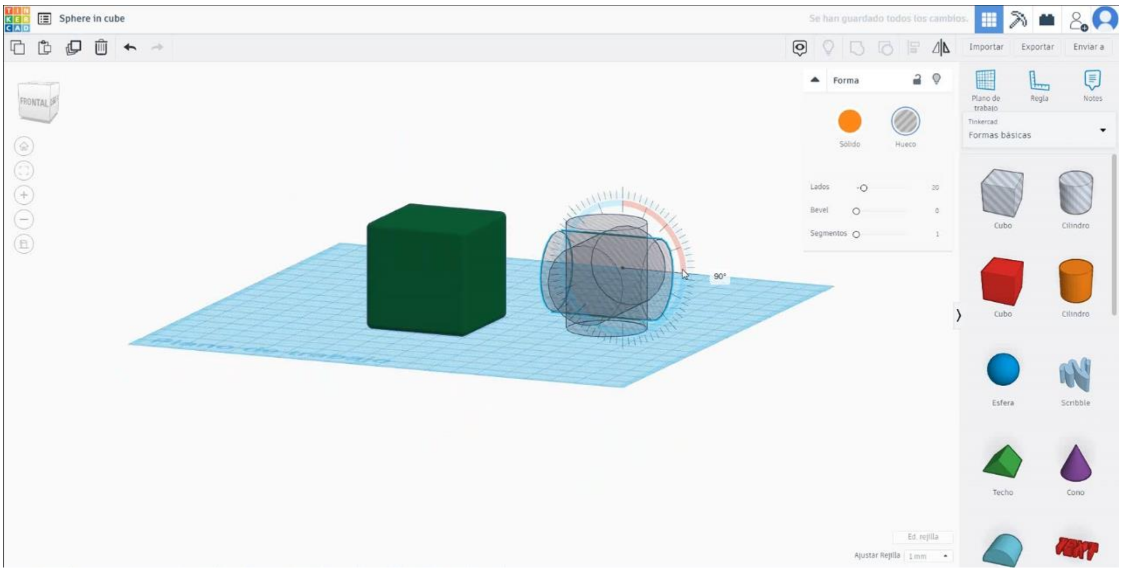
1. Choose the cube shape and size it to 40x40x40 mm with 1 mm radius.

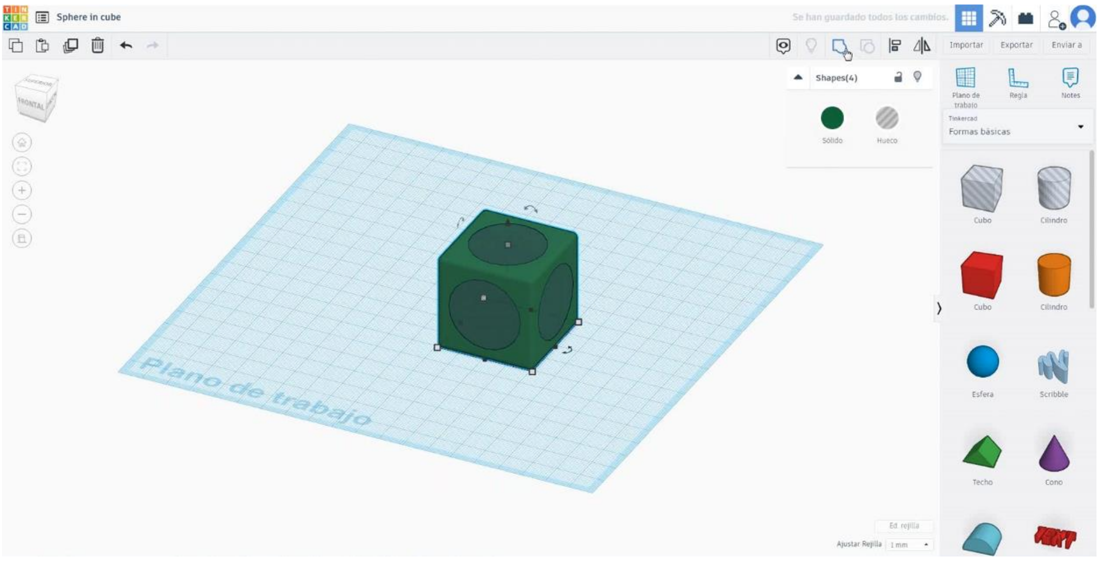


2. Choose the cylinder shape in hole mode and size it to 30x30x40 mm and copy two times 90 degrees. Then align with the cube. Select all the objects and press group.

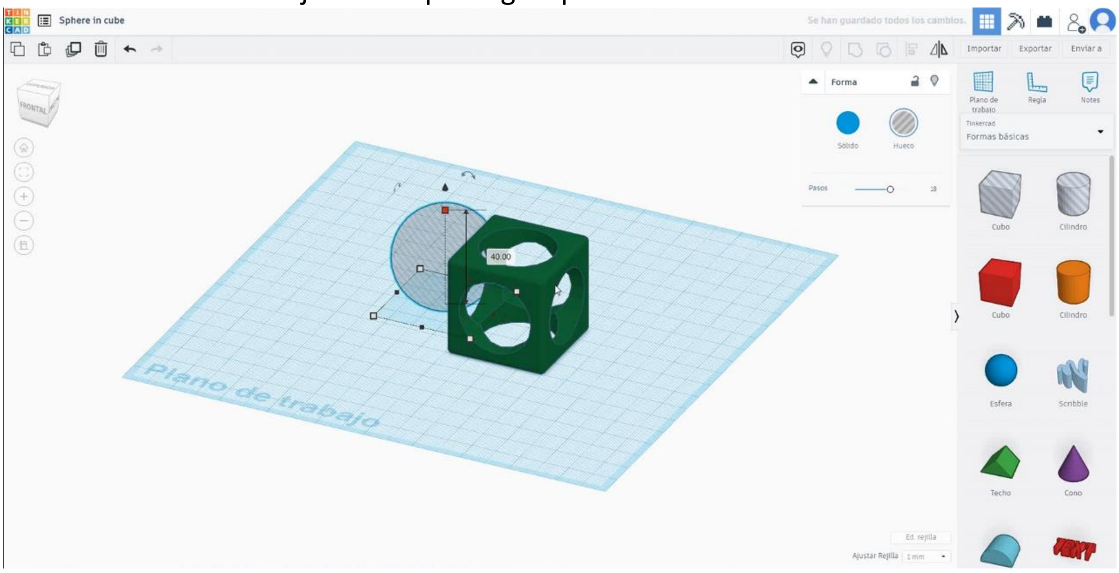

Sphere in cube
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notas
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
40.00
Plano de trabajo
Ed. rejilla
Ajustar Rejilla

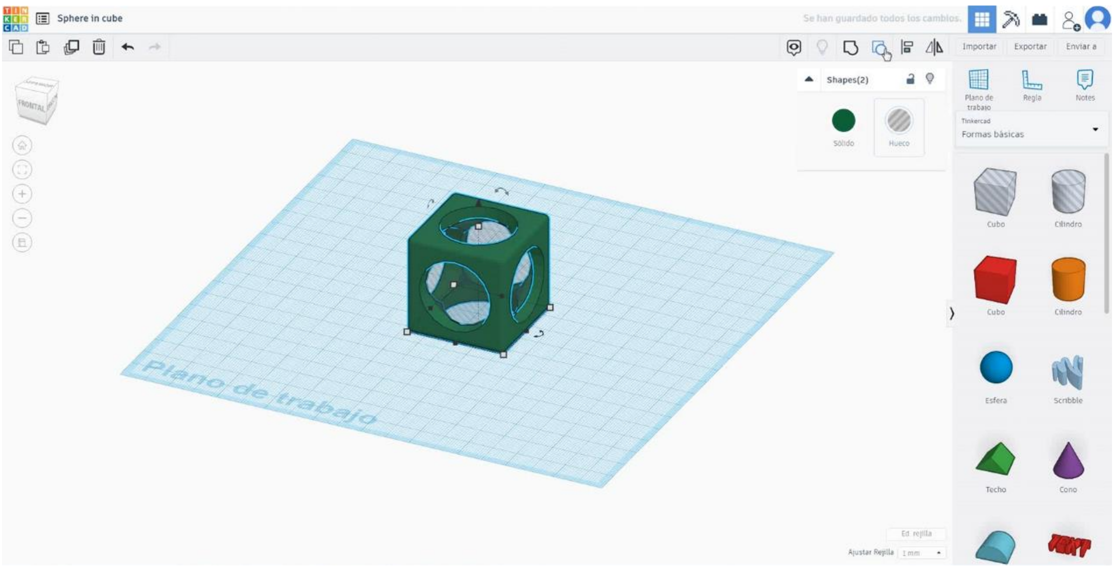

Sphere in cube
Guardando...
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Plano de trabajo
Regla
Notas
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
90°
Plano de trabajo


Sphere in cube
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Plano de trabajo
Regla
Notas
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
90°

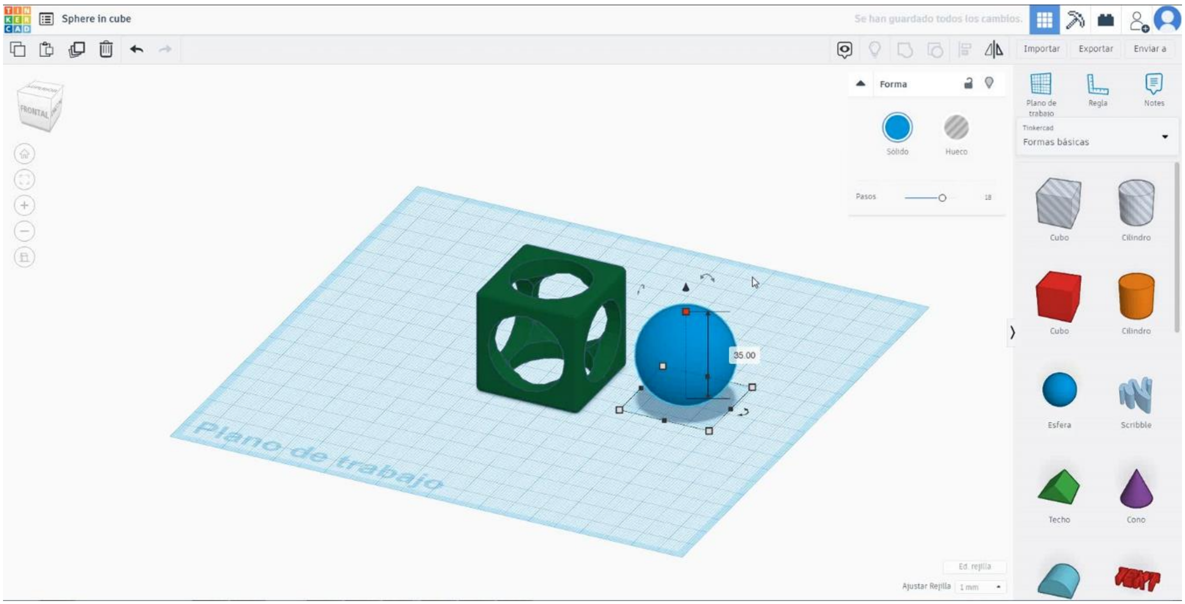


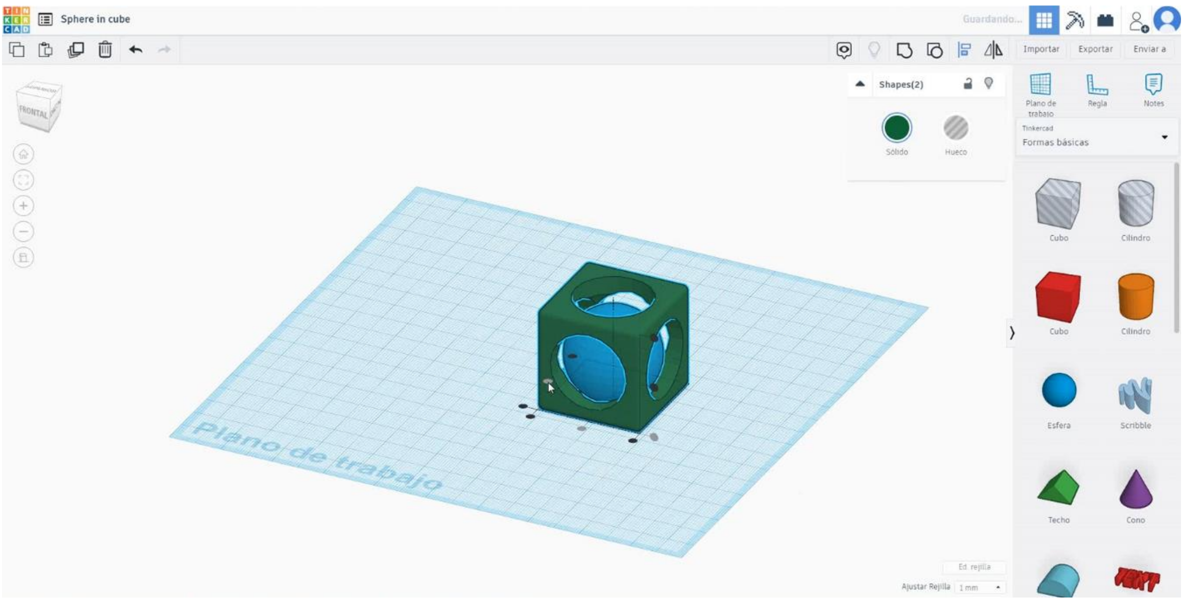
3. Choose the sphere shape and size it to 40x40x40 mm in hole mode and align it with the cube all the axis. Select both objects and press group.



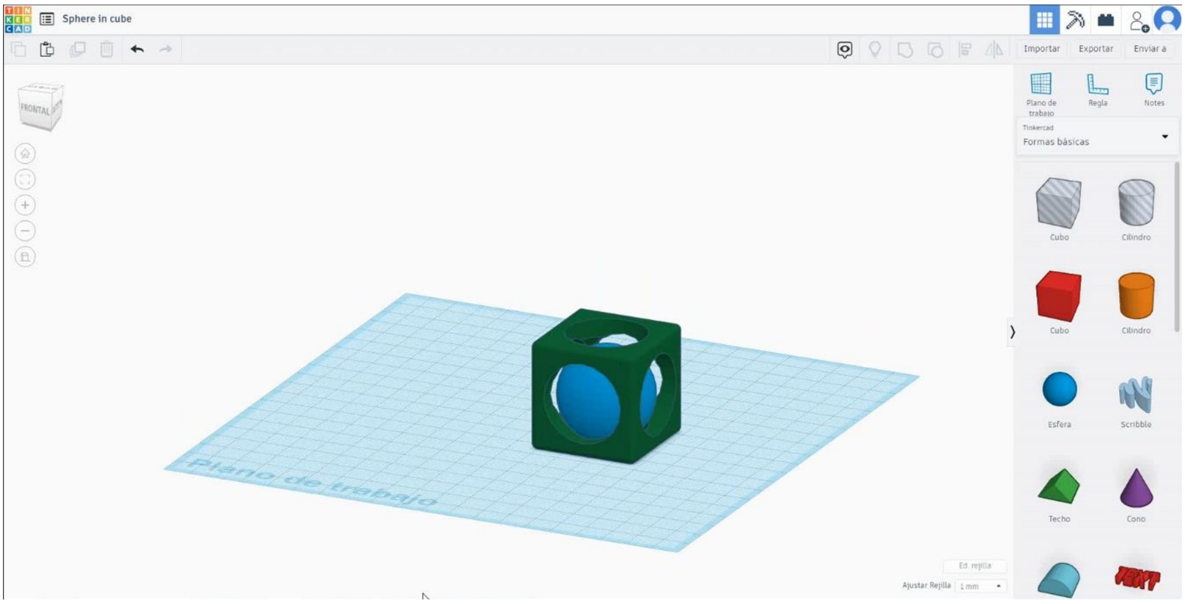


4. Choose the sphere shape and size it to 35x35x35 mm and align in the X and Y axis with the cube.





5. Now, the sphere in cube is finished.



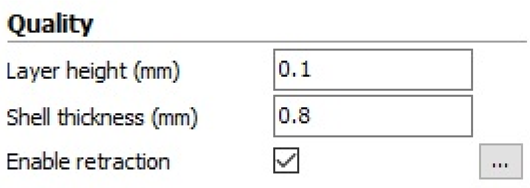
9.3.17.2 Sphere in cube 3D printing seetings

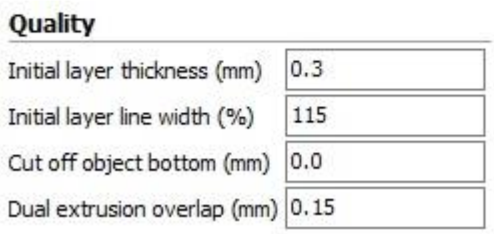
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

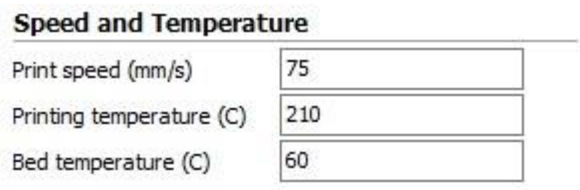


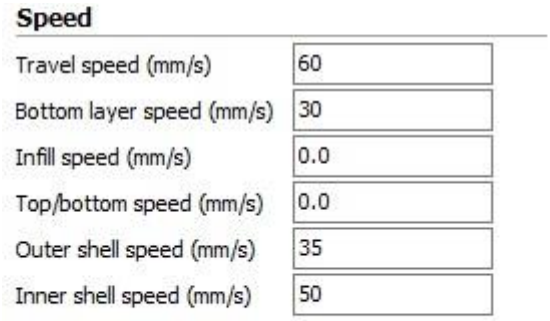


Speed and Temperature

Print Speed - 50 (mm/s)

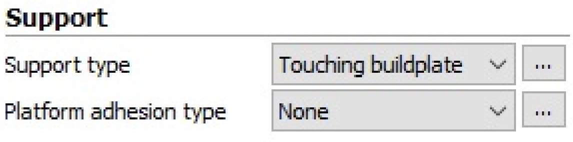
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





<u>Support Type</u>

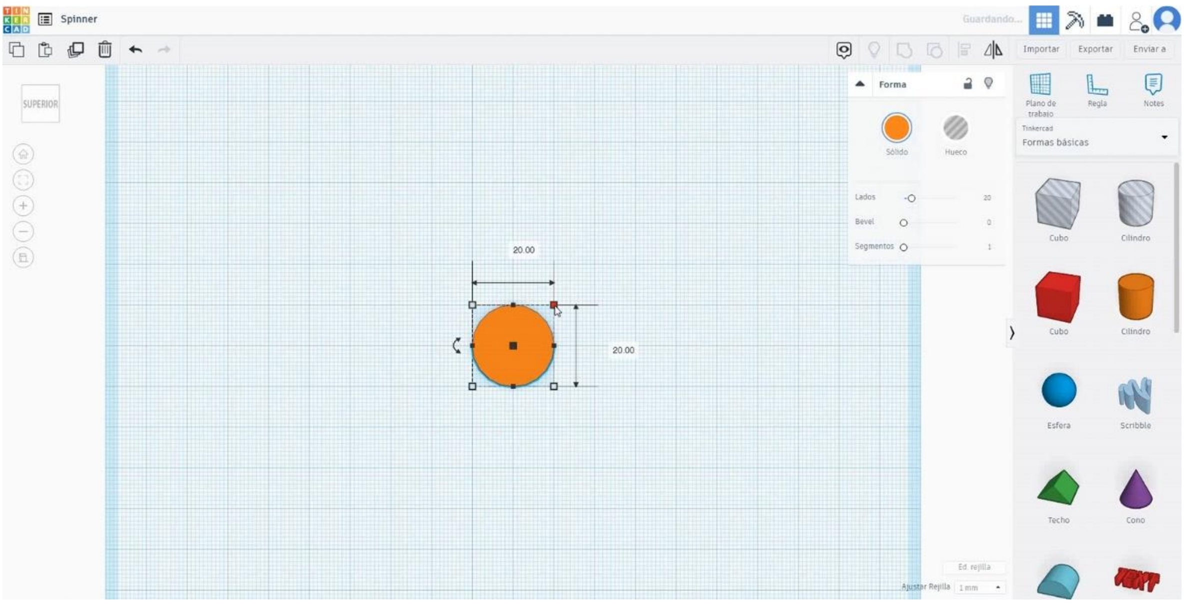
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

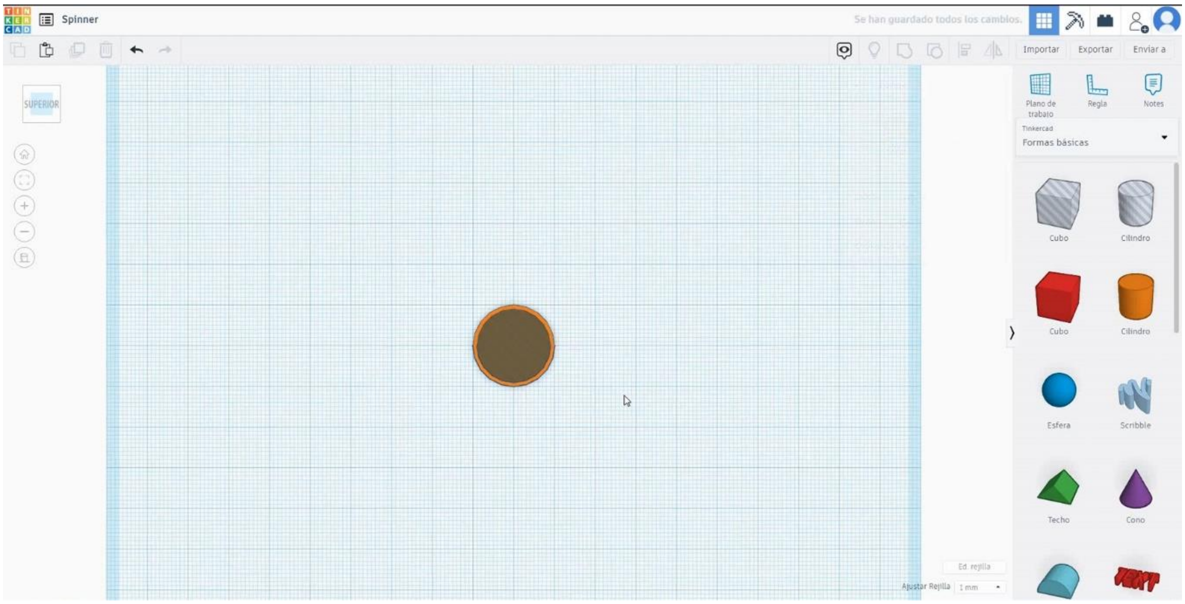


9.3.18 Part 18: Spinner

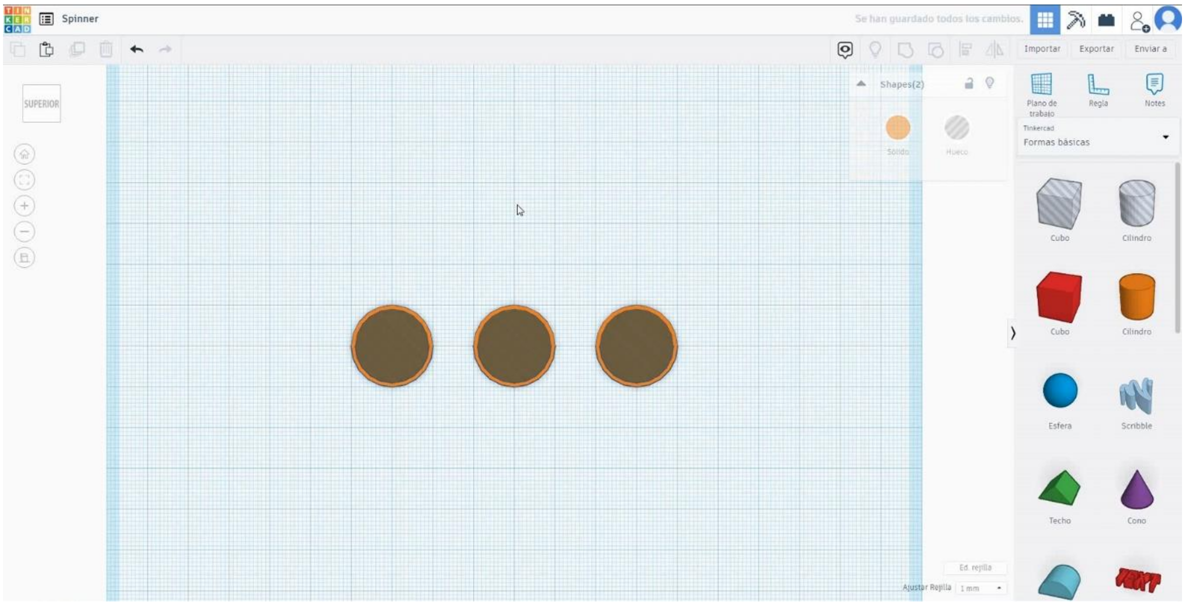
9.3.18.1 Spinner Design

1. Choose the cylinder shape and size it to 20x20x5 mm. Then choose the cylinder shape in hole mode and size it to 18x18x5 mm and align in the center of the first cylinder.

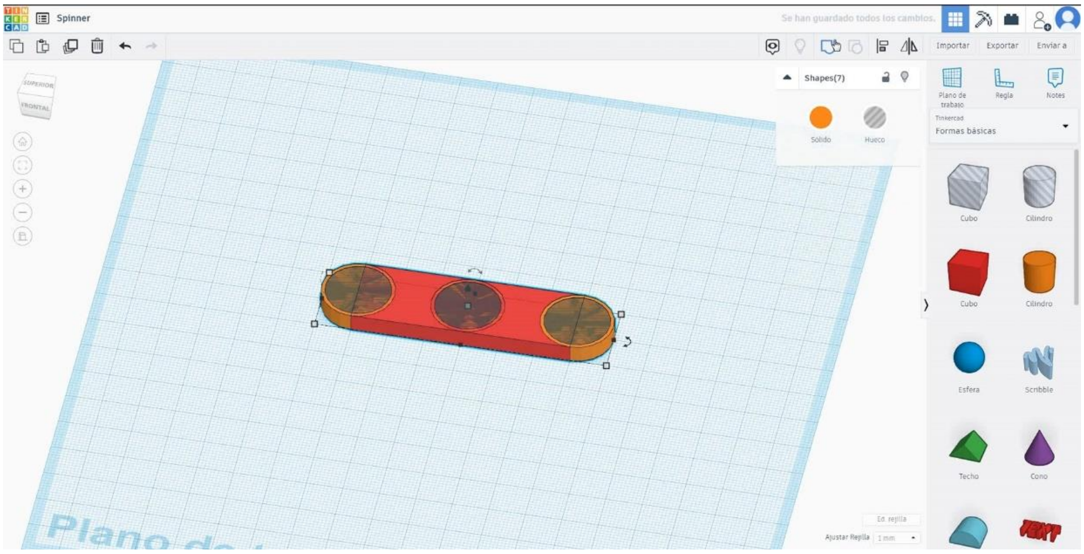


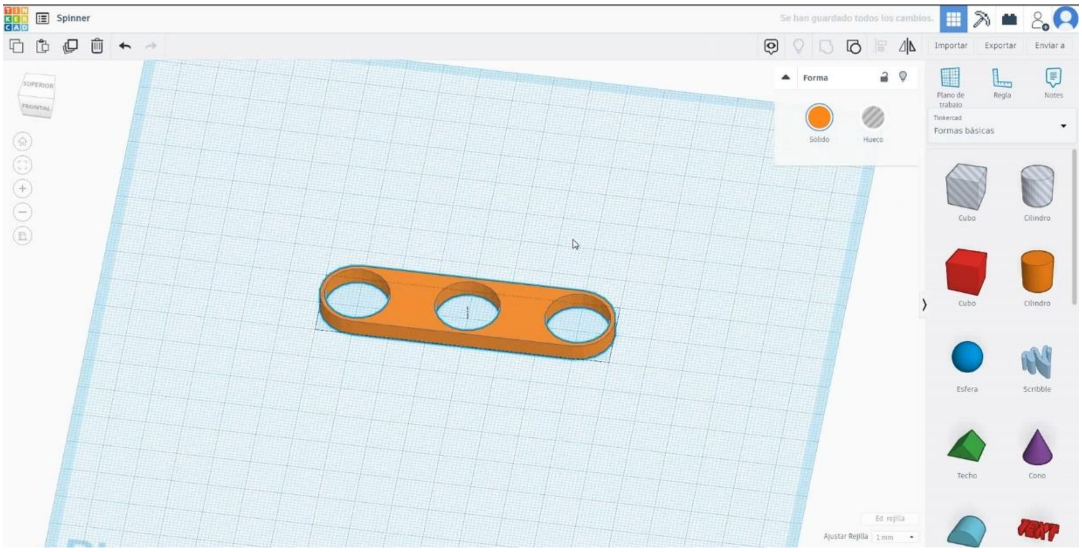


2. Select both shapes and copy two times moving it 30mm to the left and right.

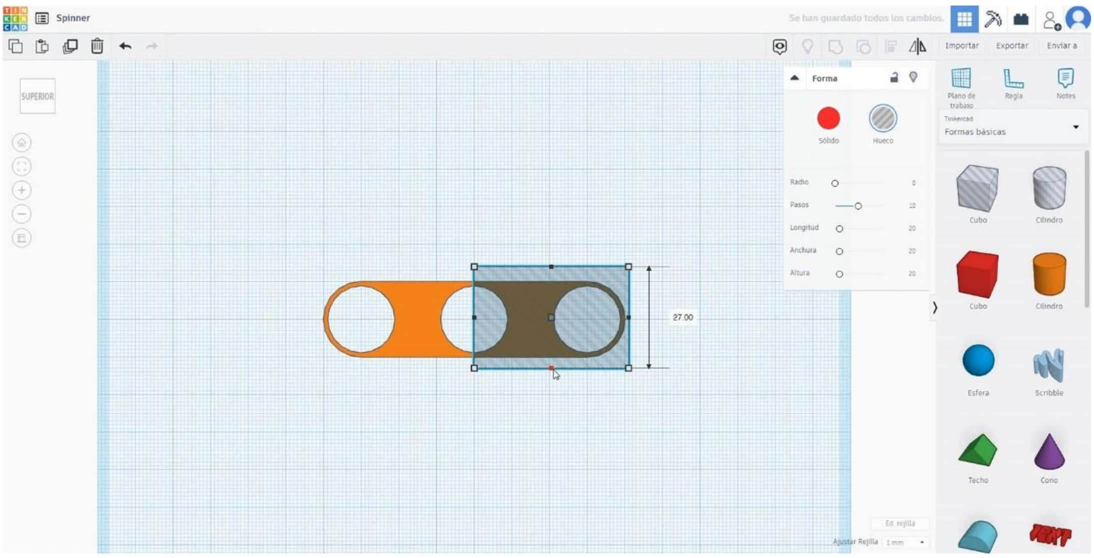


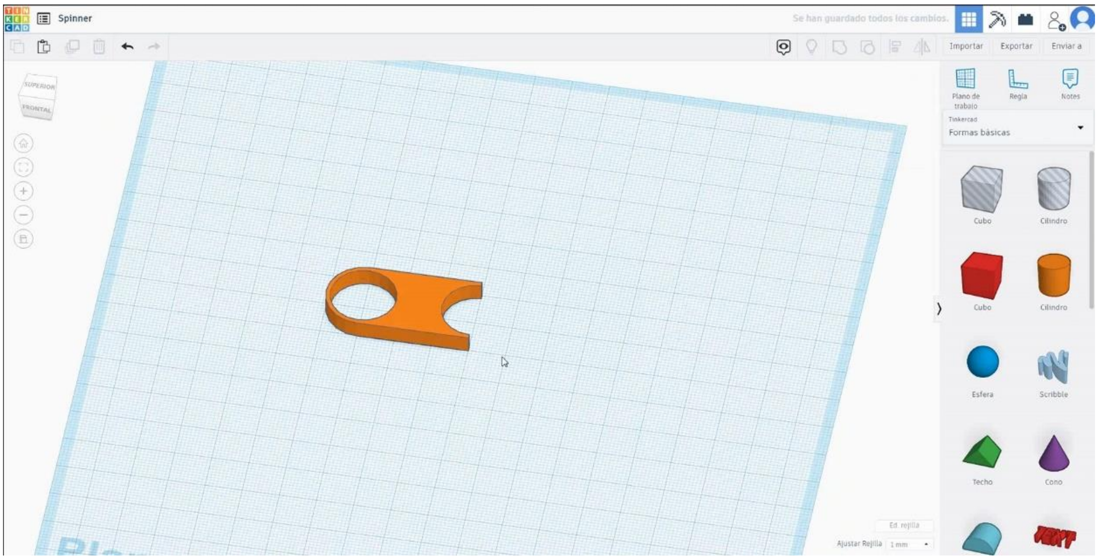
3. Select the cube shape , size it to 60x20x5 mm and put in the center of the three cylinders. Then select all the objects and press group.



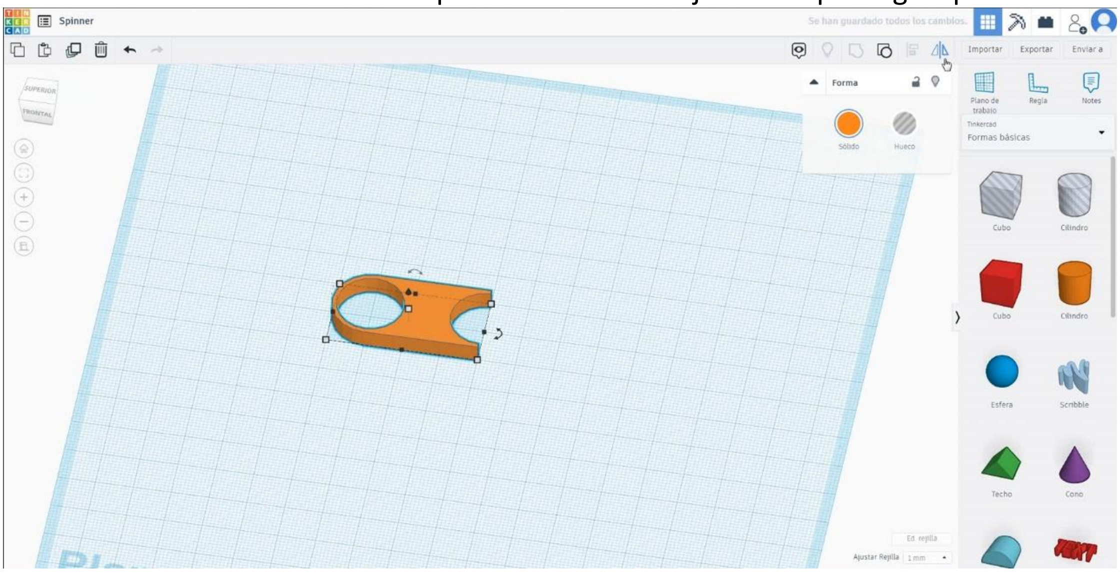


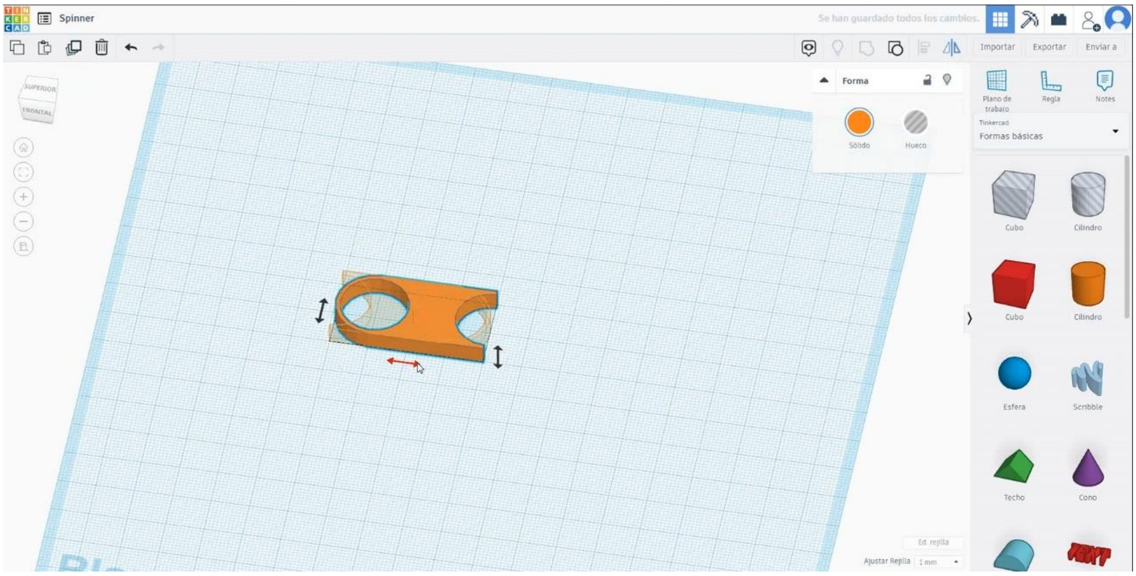
4. Now we have to cut the objet in half. Select cube in hole mode and size it bigger than the half of the object. Then select both objects pressing shift and press group.

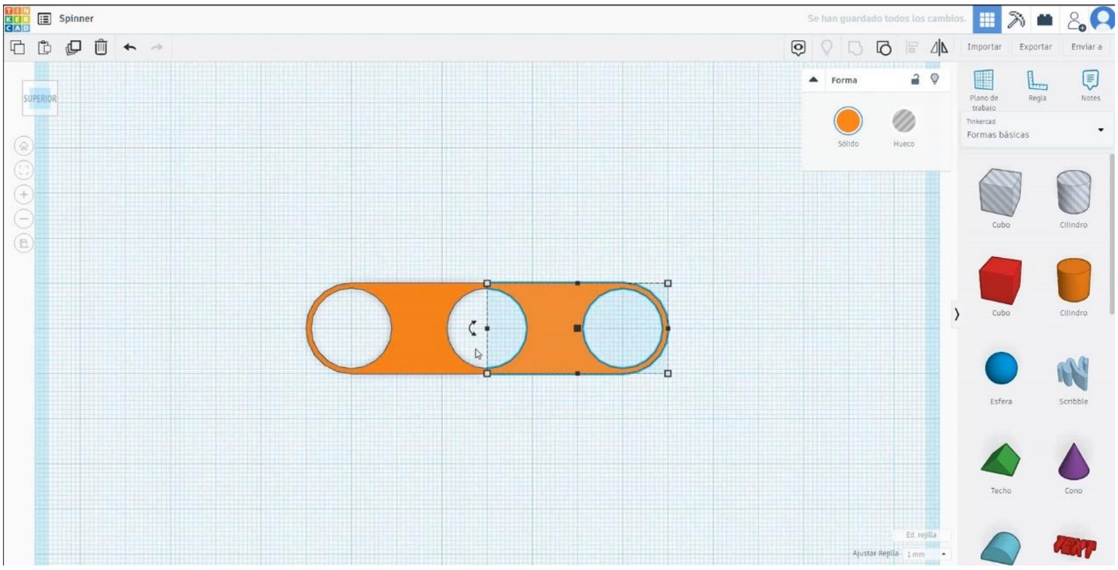




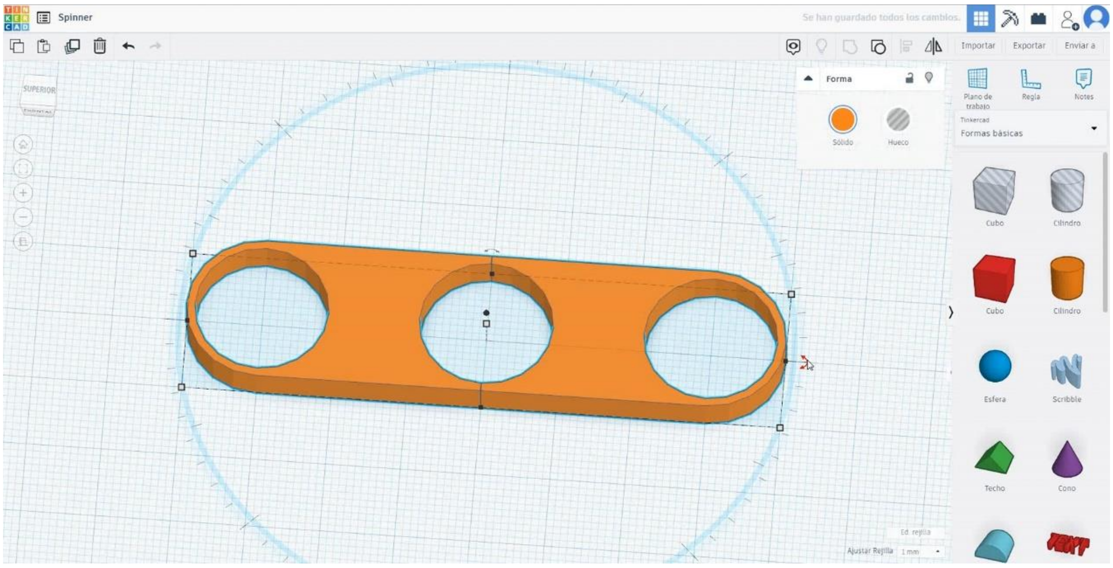
5. Select the object, then press duplicate , flip and the horizontal arrow to create a copy. Move it to make it match in the central part. Select both objects and press group.

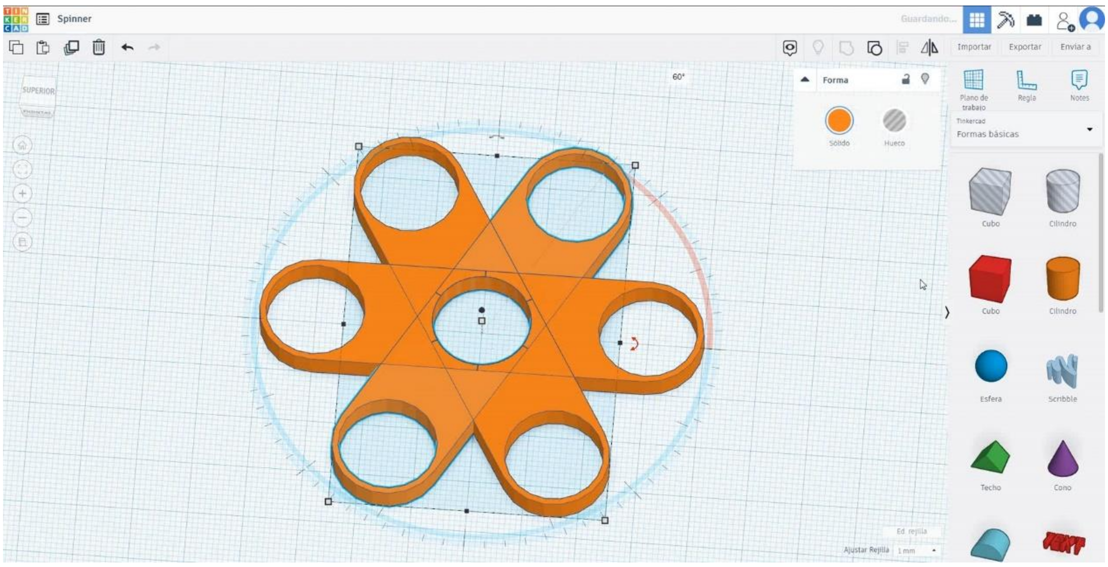




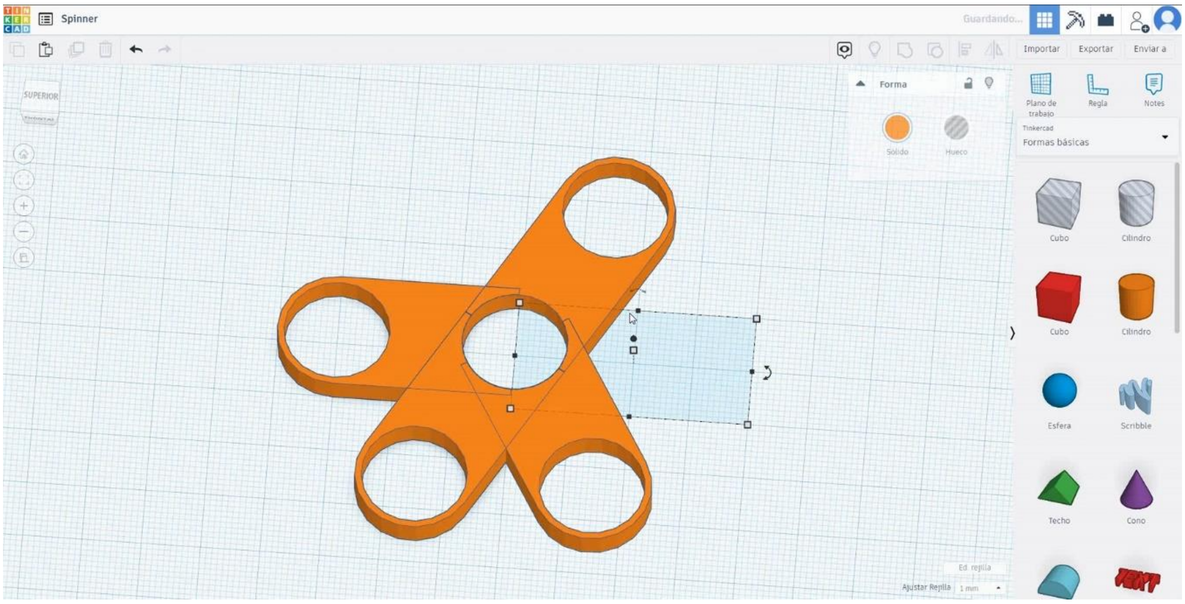


6. Now, we are going to rotate two times 60 degrees.

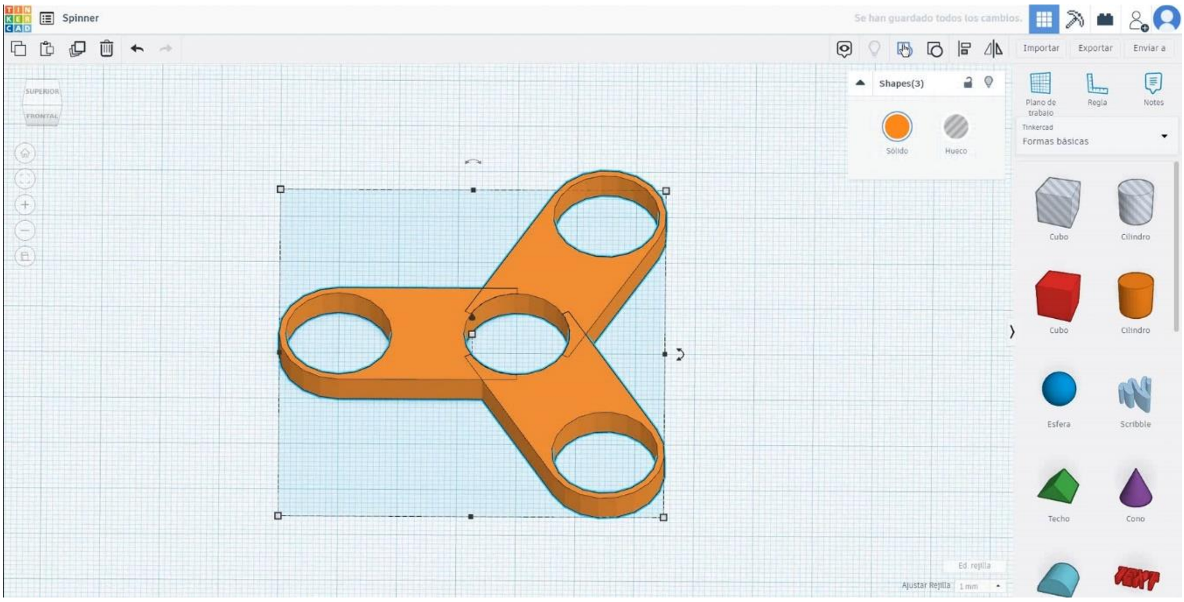




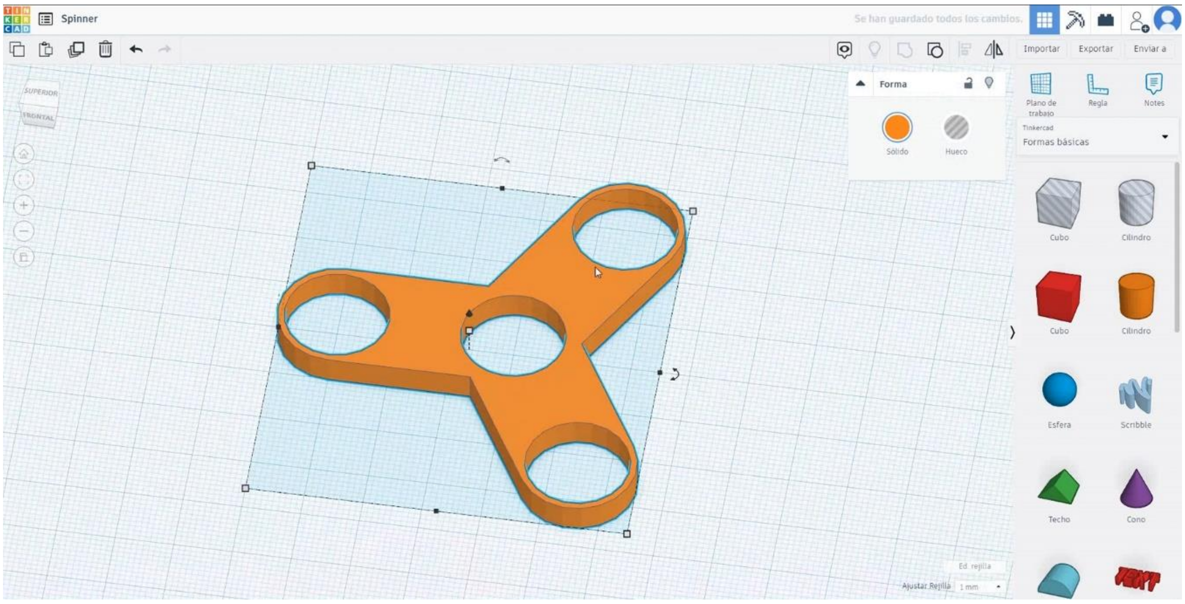
7. Select all the objects and press ungroup. Then select three and erase them.



8. Select the three objects and press group to merge them in one object.



9. Now, the spinner is finished.



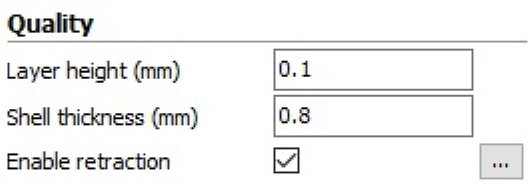
9.3.18.2 Spinner 3D printing seetings

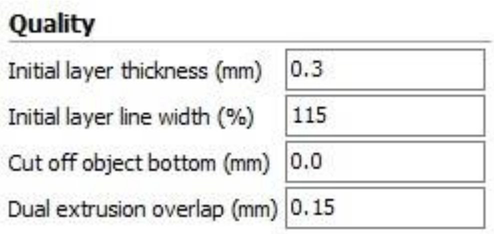
<u>Filament</u>

PLA
Diameter - 1.75 (mm)
Flow - 100%

<u>Quality</u>

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

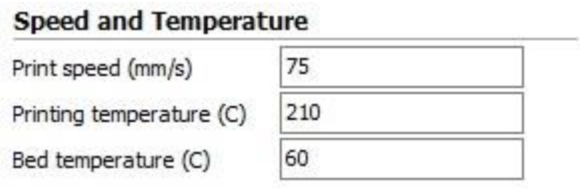


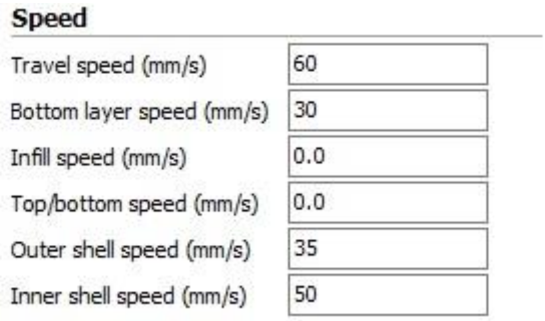


<u>Speed and Temperature</u>

Print Speed - 50 (mm/s)

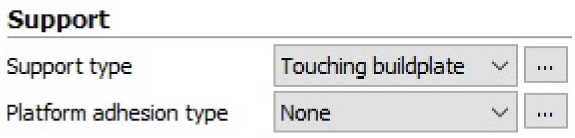
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





<u>Support Type</u>

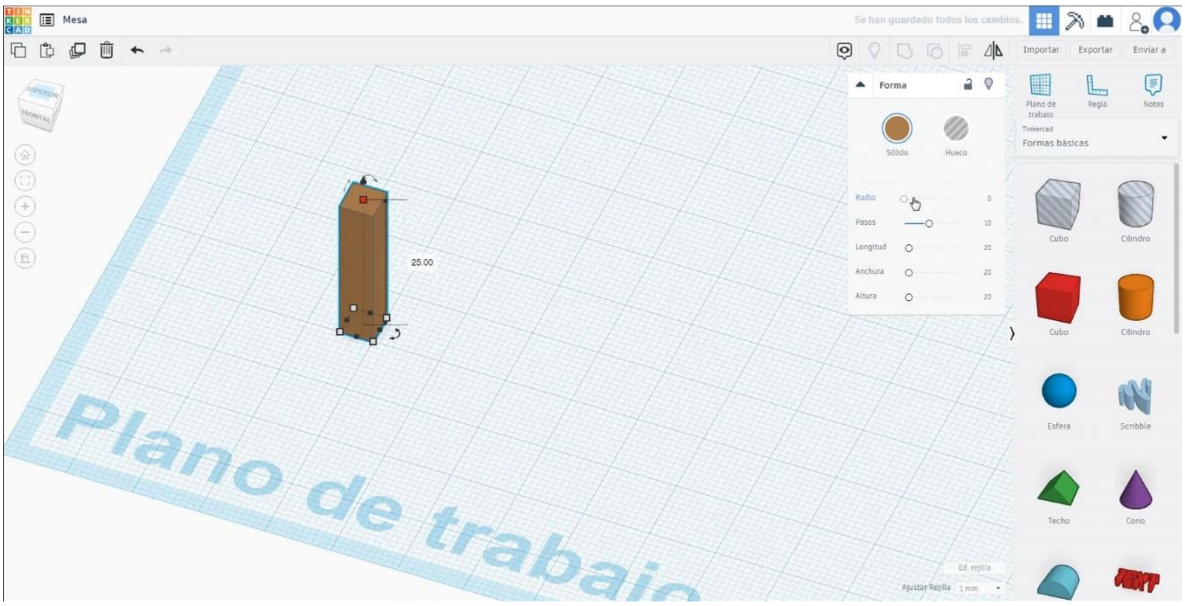
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

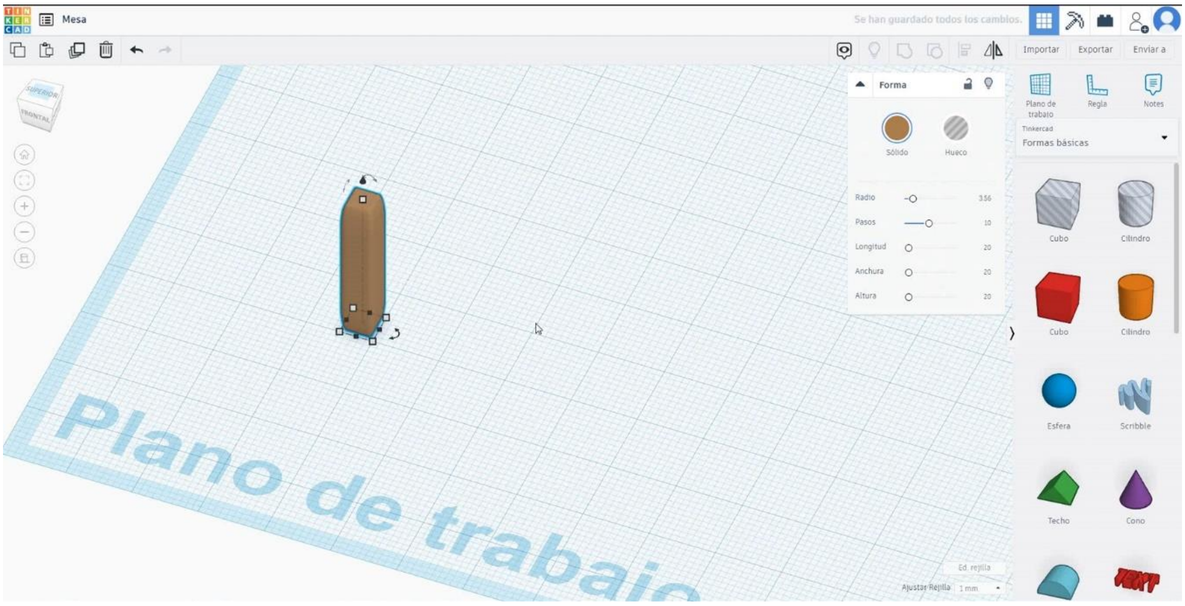


9.3.19 Part 19: Table

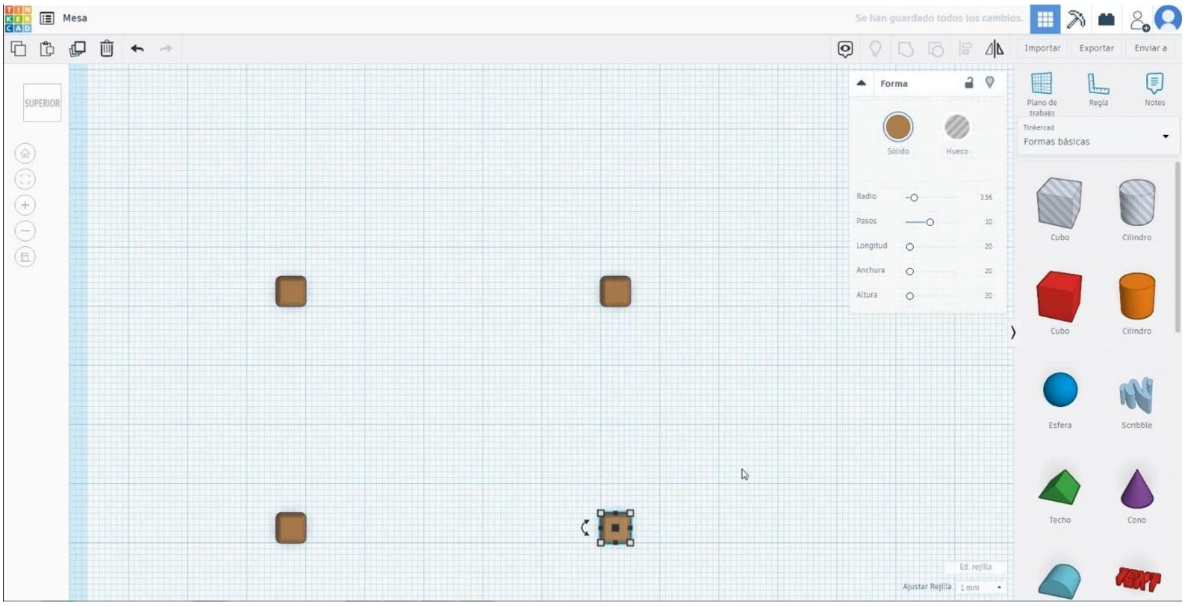
9.3.19.1 Table Design

1. We will star with the design with the table legs. First choose the cube and size it to 5x5x25 mm and change the option radius to make it rounded.

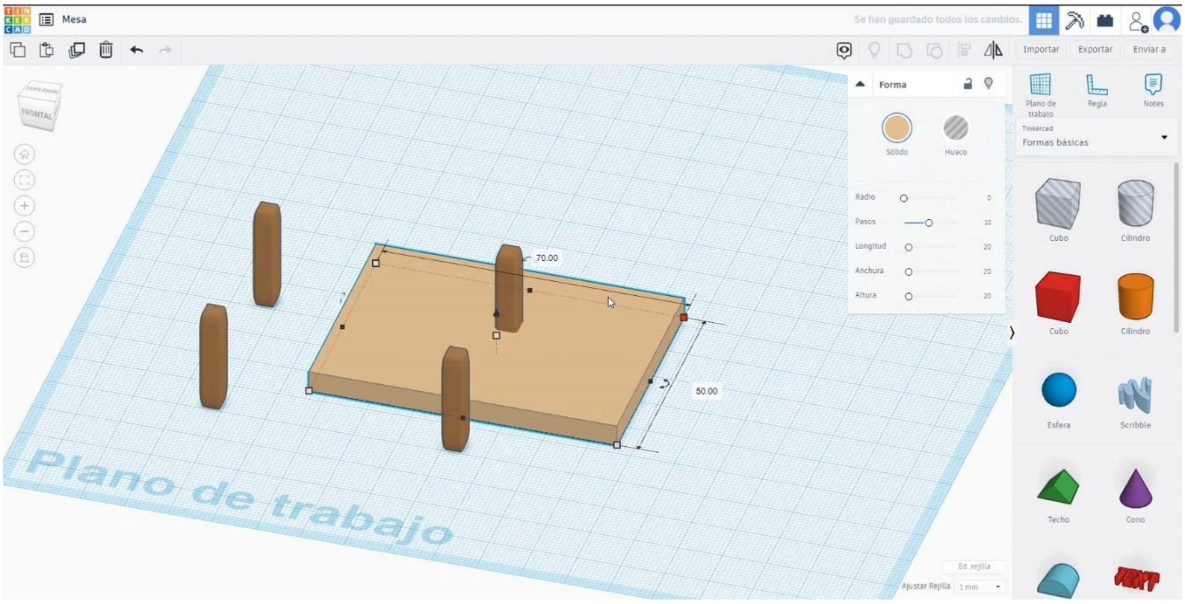


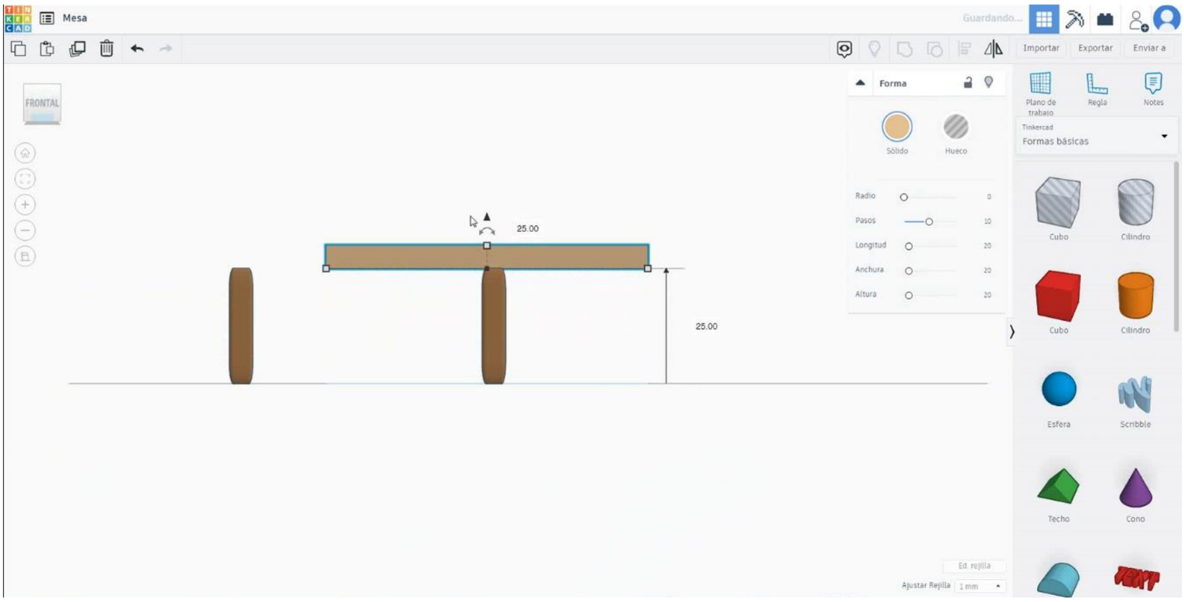


2. Copy the leg three times in a rectangular position.

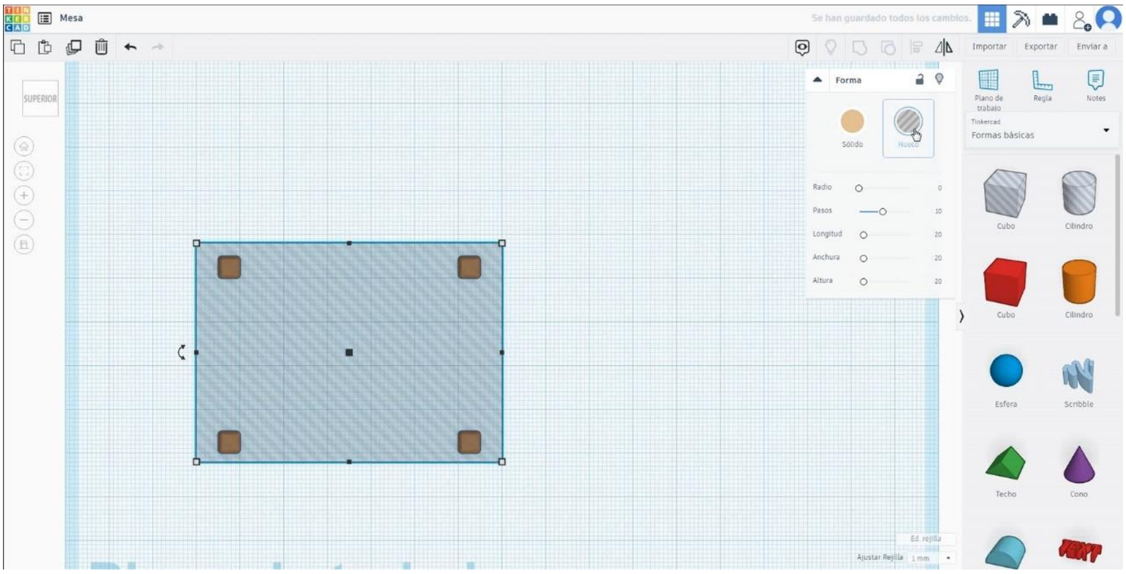


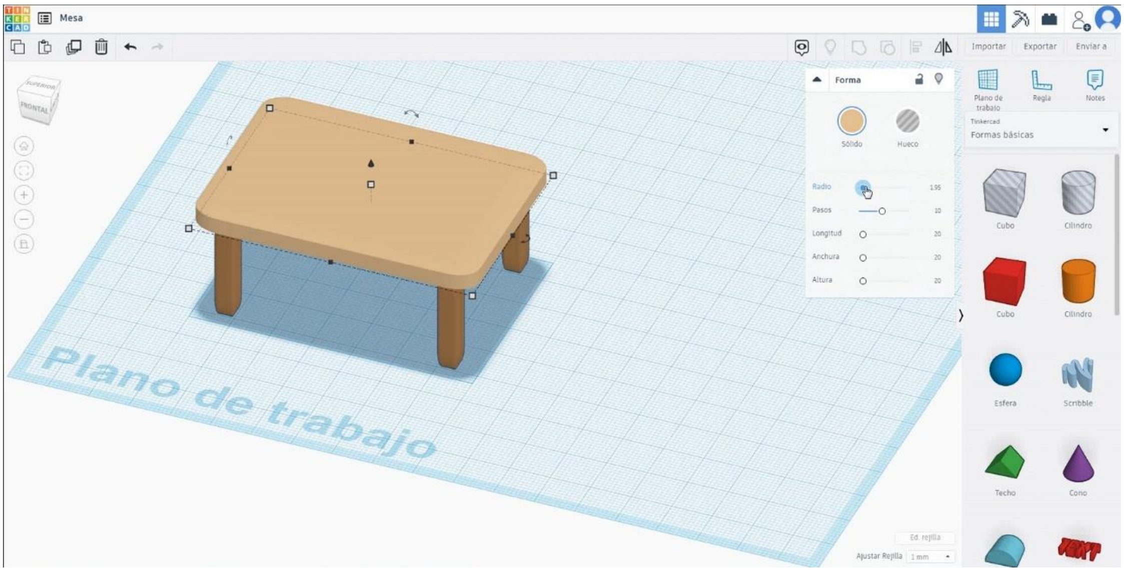
3. To create the table, choose the cube and size it to 70x50x5 mm. Then we will move to the height of 25 mm.



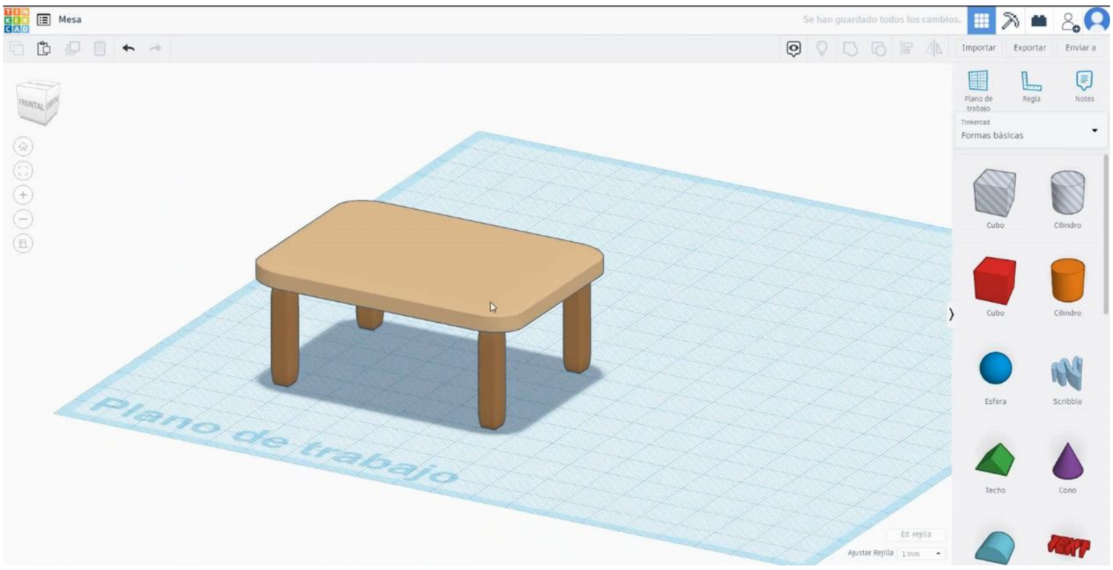


4. In order to center the table into the legs, we select hole mode and move it. After that we will select color mode again. Select the table and change the radius to make it rounded.





5. Now, the table is finished.



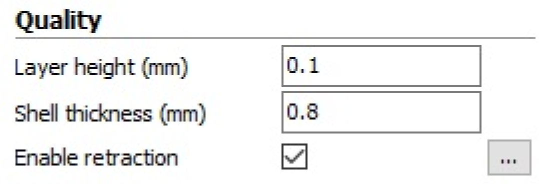
9.3.19.2 Table 3D printing seetings

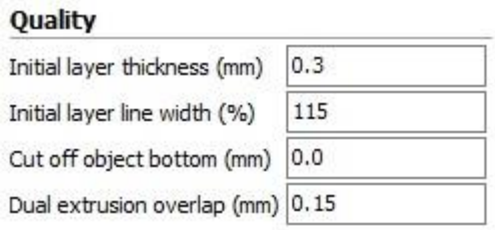
Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

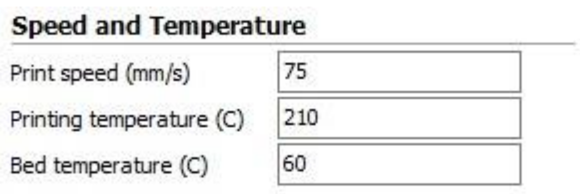
Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

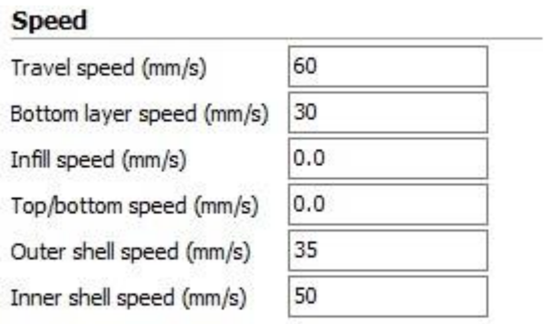




Speed and Temperature

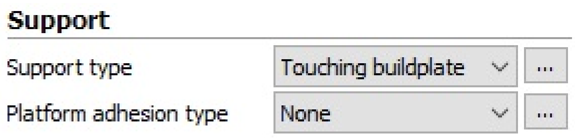
Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)





Support Type

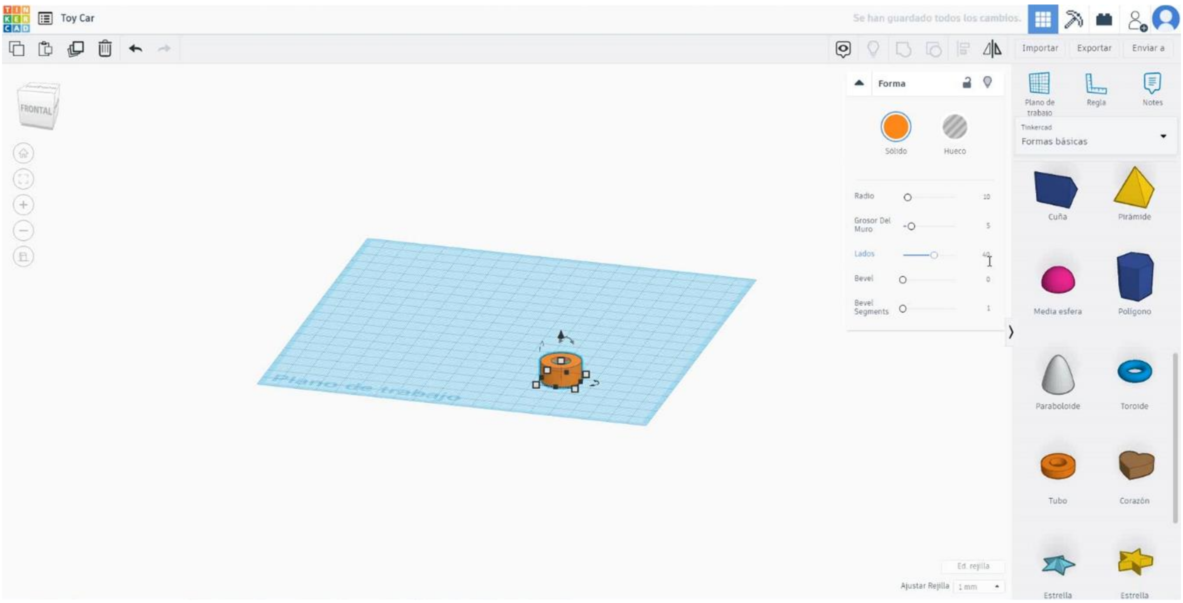
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

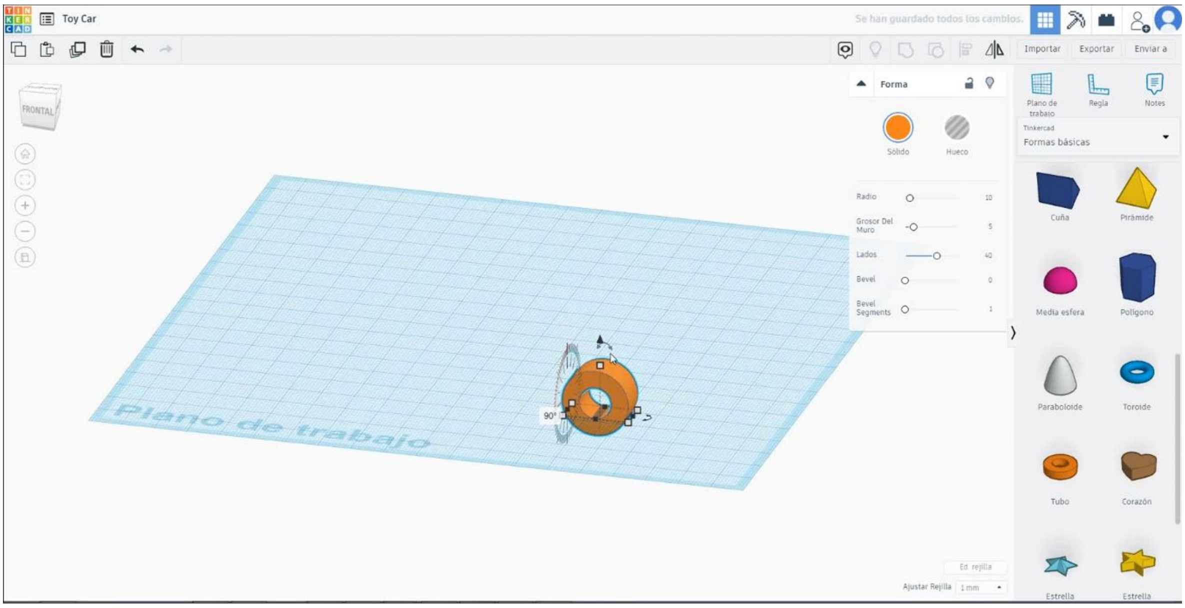


9.3.20 Part 20: Toy car

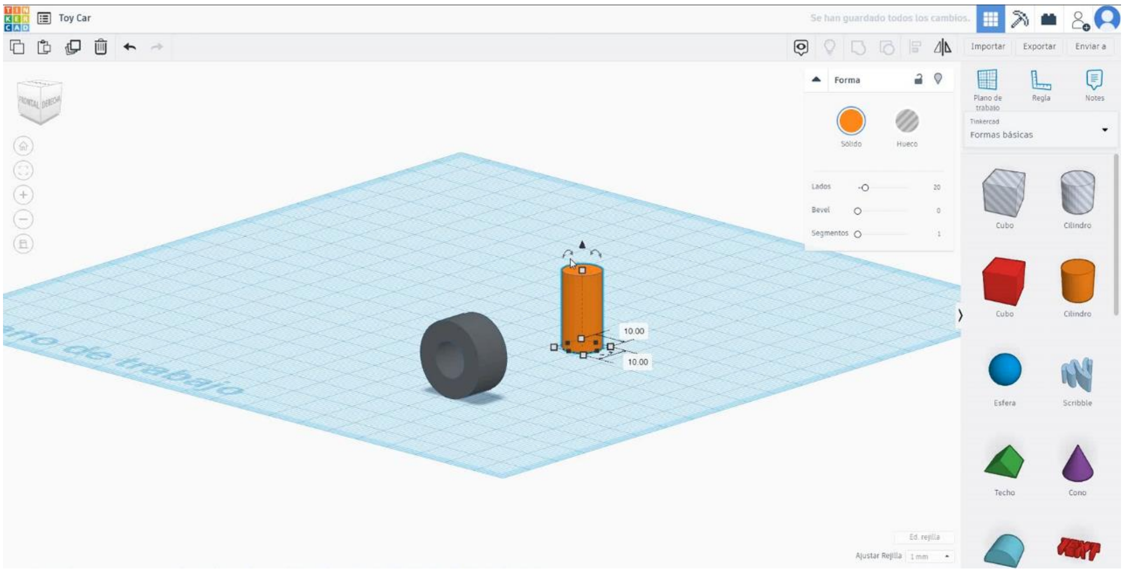
9.3.20.1 Toy car Design

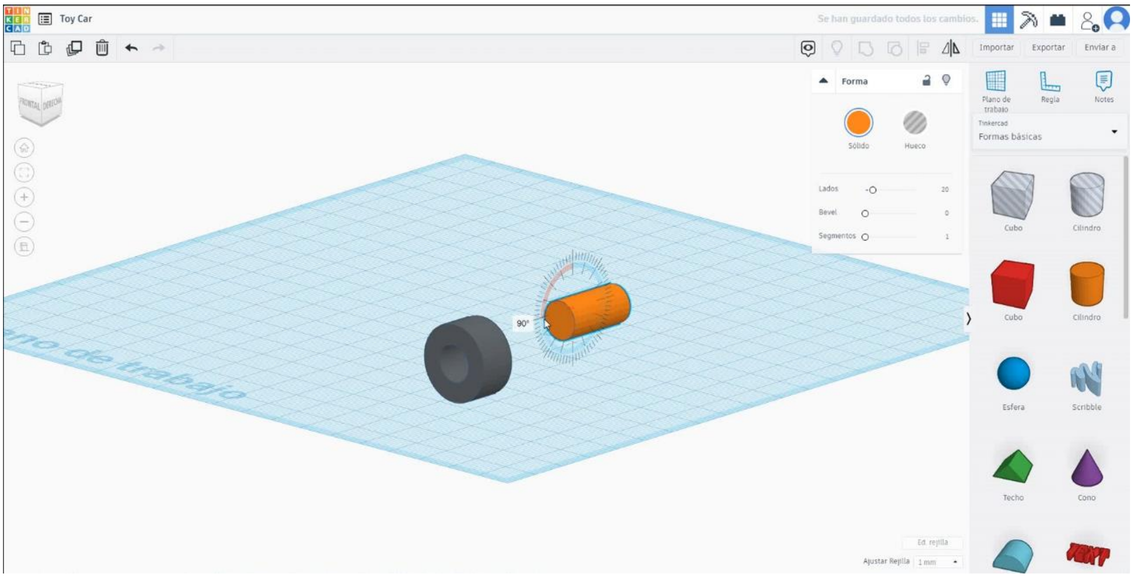
1. Choose the tube shape, size it to 20x20x10 mm with thick wall of 5mm and 40 sides and turn it 90 degrees. Then move it to Z 0 mm.

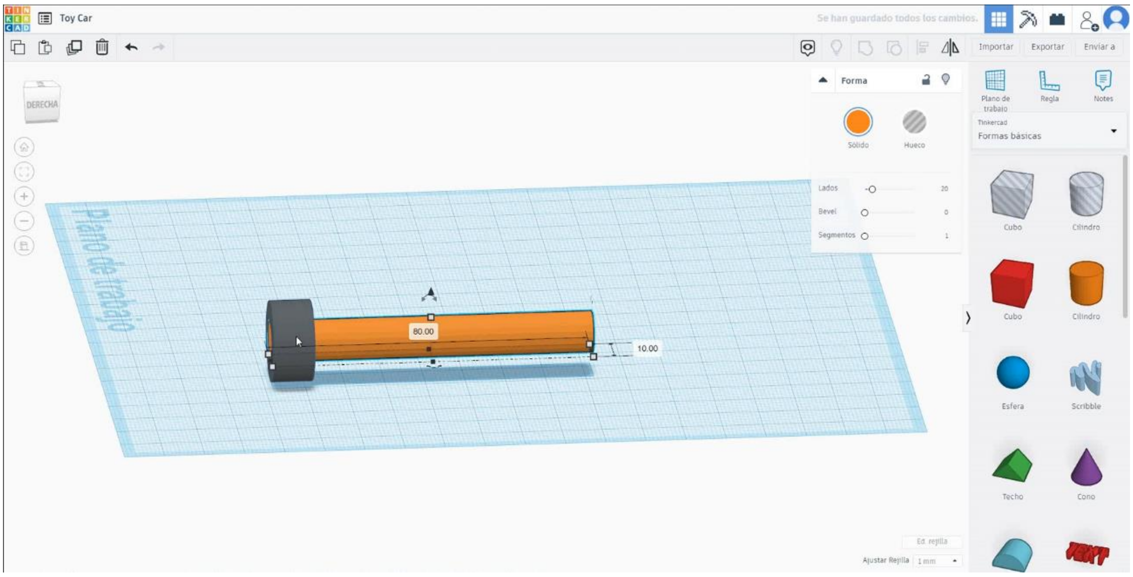


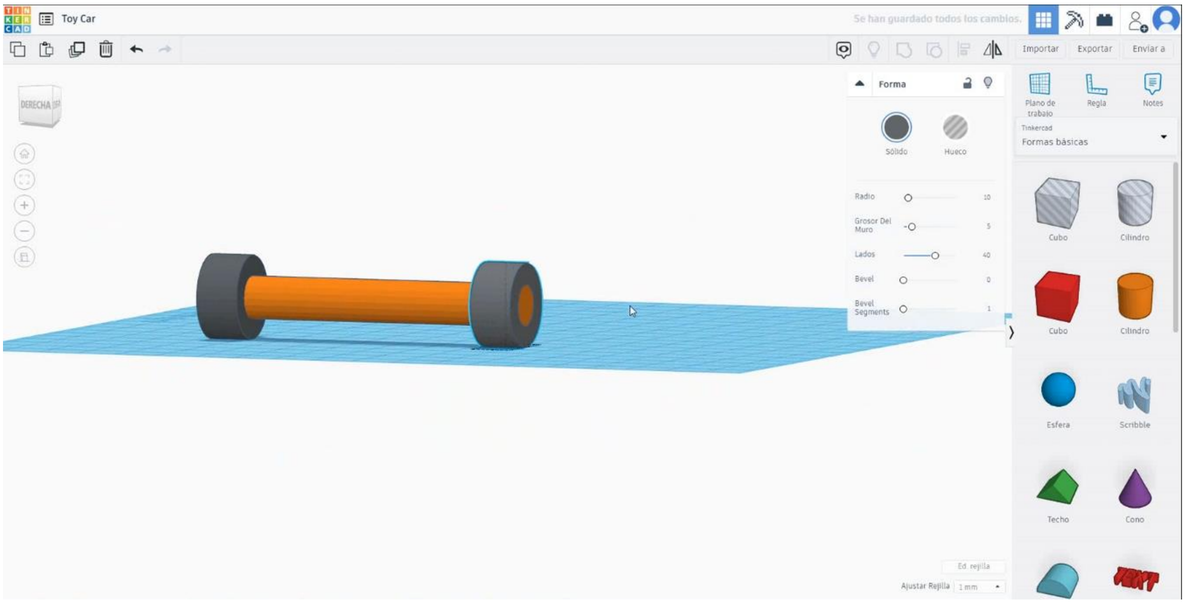


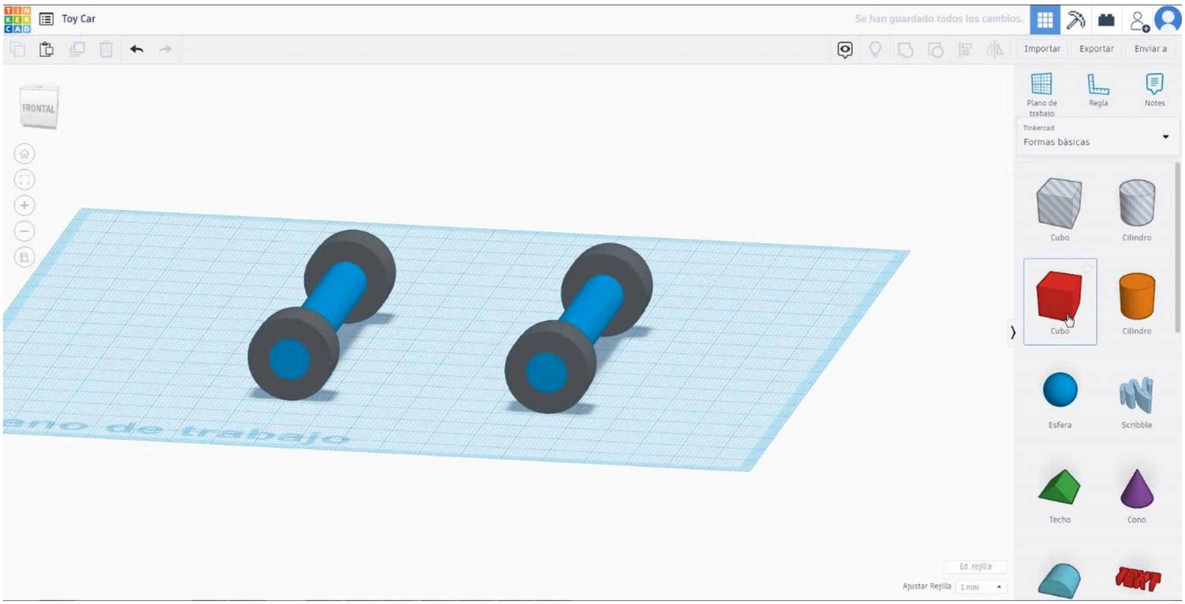
2. Choose the cylinder shape, size it to 10x10x80 mm, turn it 90 degrees and align it with in the center of the tube. Then copy the tube and move it to the opposite side to make the second wheel and copy all the set 60 mm to the left side.


Toy Car
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
10.00
10.00
Ajustar Rejilla

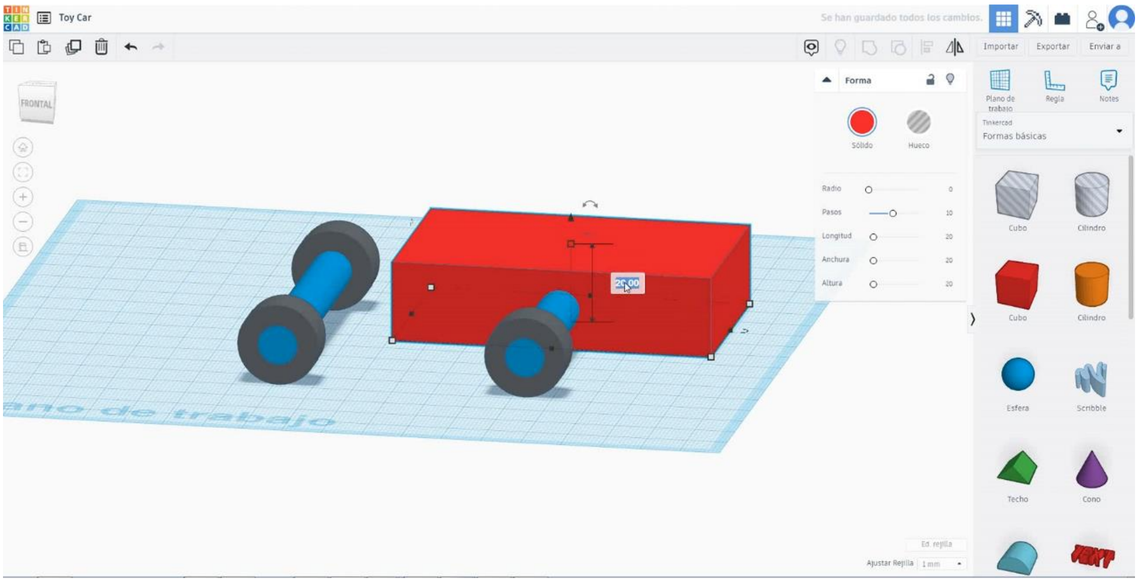

Toy Car
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
90°
Ajustar Rejilla

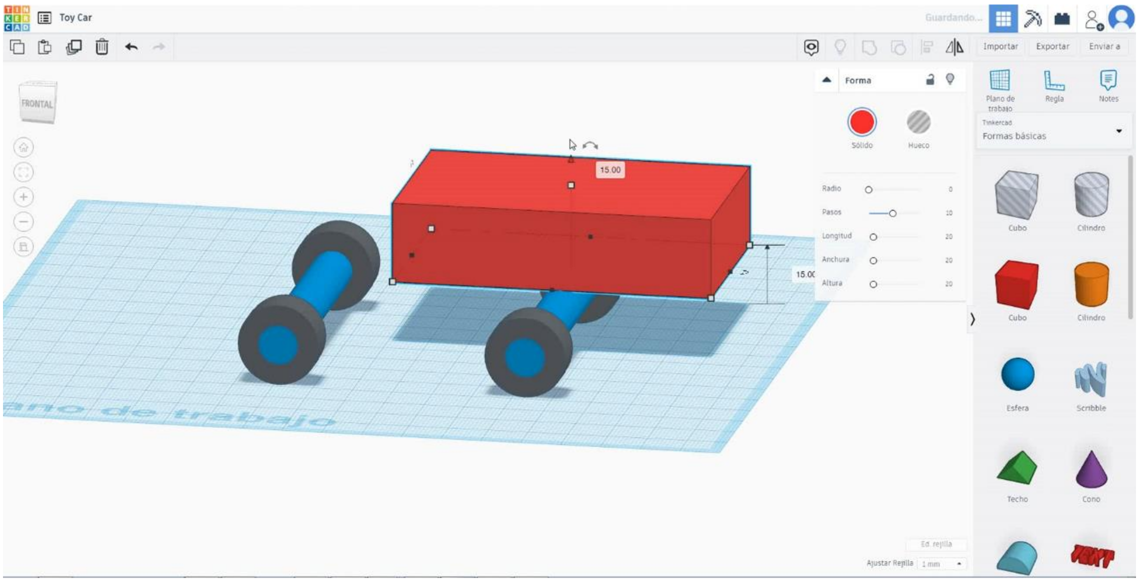

Toy Car
Se han guardado todos los cambios.
Importar
Exportar
Enviar a
DERECHA
Forma
Sólido
Hueco
Lados
Bevel
Segmentos
Plano de trabajo
Regla
Notes
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono
80.00
10.00
Ajustar Rejilla

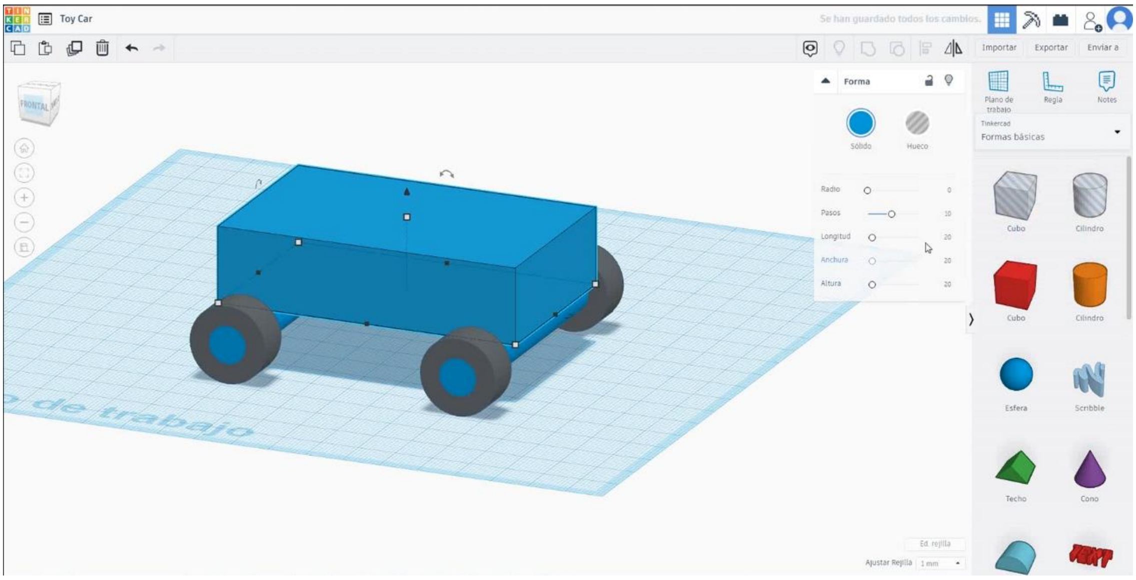




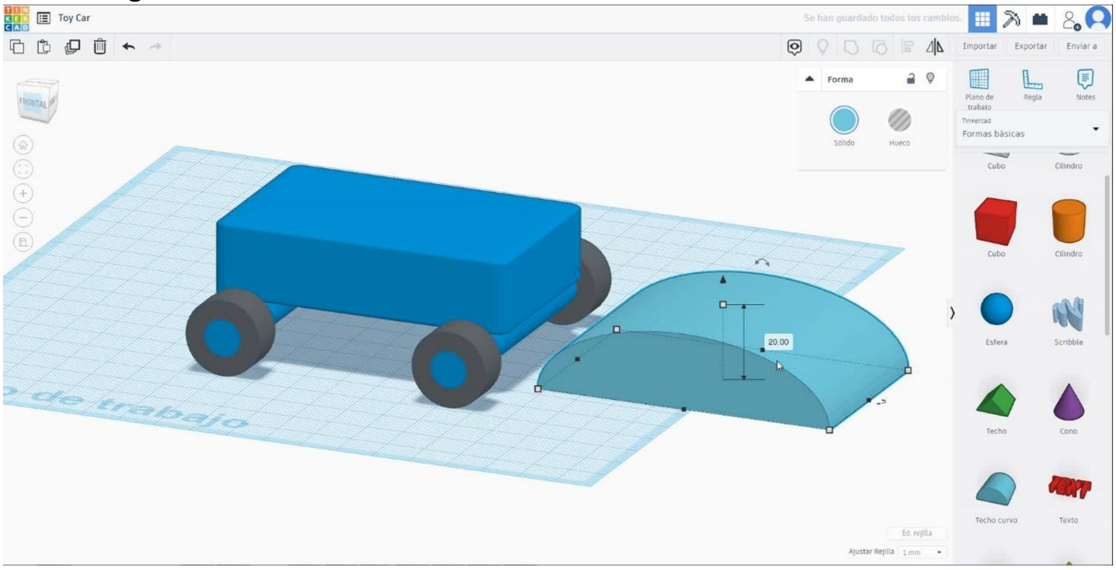
3. Choose the cube shape and size it to 80x50x20 mm and move it to the height of 15 mm. Then align in the center of the set.

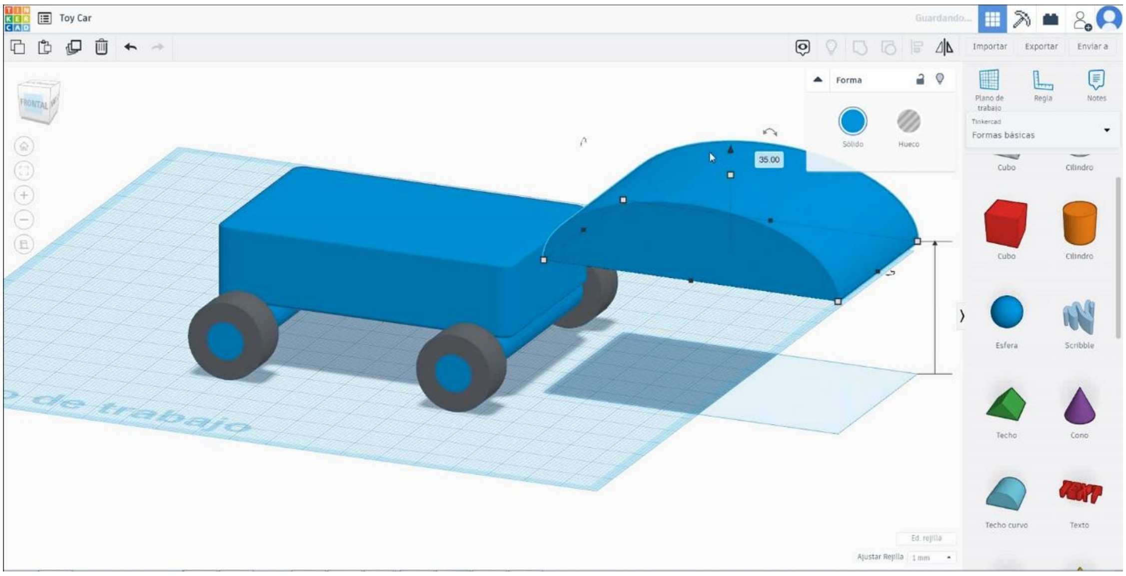

Toy Car
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono

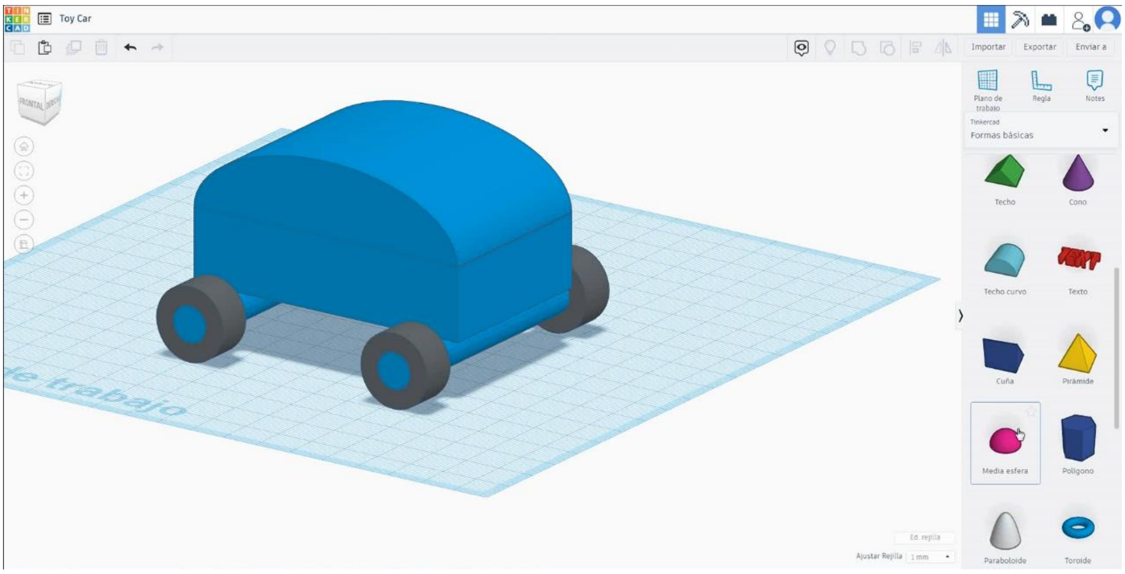

Toy Car
15.00
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono


Toy Car
Importar
Exportar
Enviar a
Forma
Sólido
Hueco
Formas básicas
Cubo
Cilindro
Esfera
Scribble
Techo
Cono

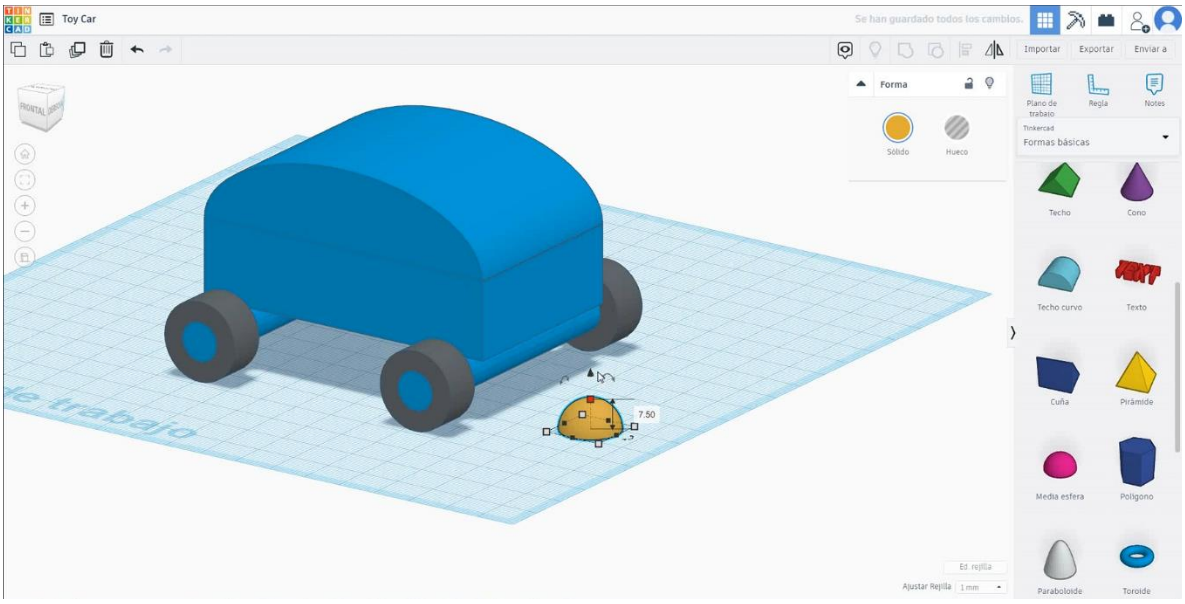
4. Choose the round roof shape and size it to 80x50x20 mm and move it to the height of 35 mm. Then align in the center of the cube.

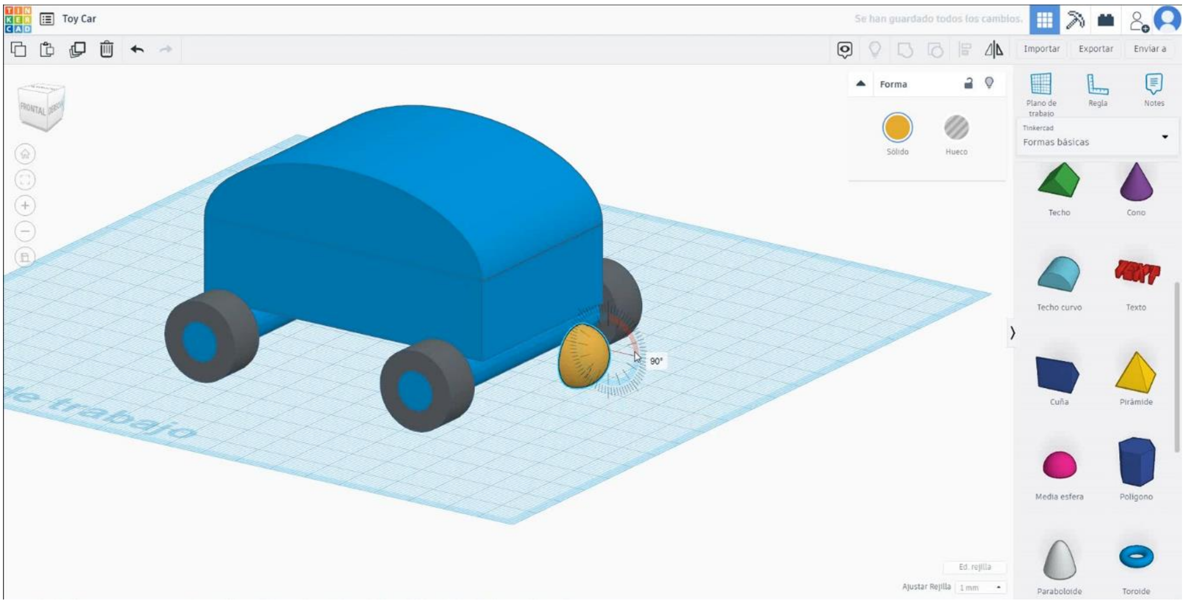




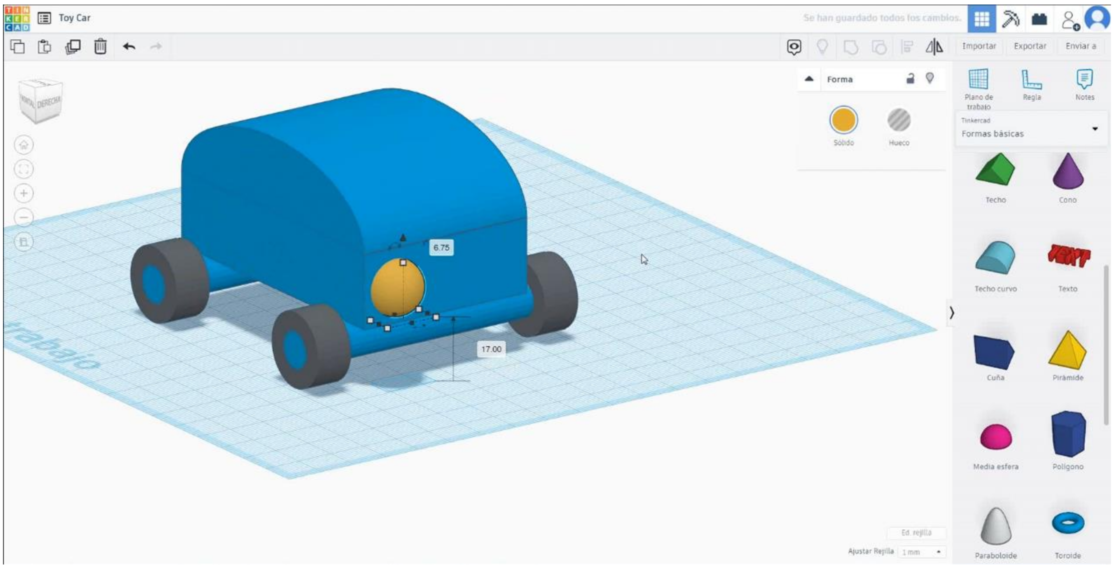


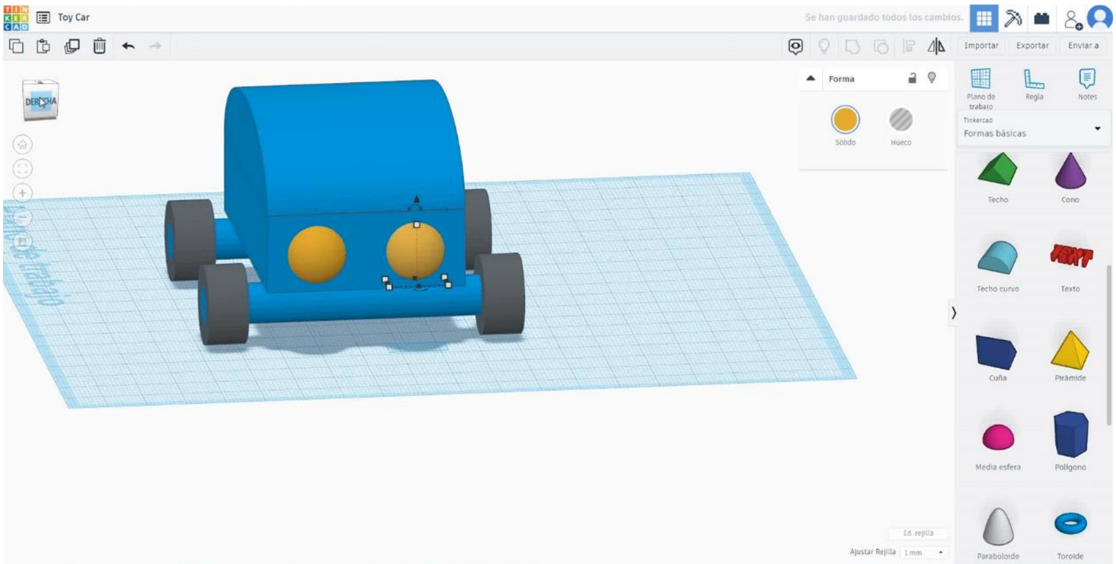
5. Select the shape half sphere, size it to 15x15x7.5 mm and turn it 90 degrees. Move it 17 mm in height and align with the right part of the cube. Then duplicate the half sphere to make the second light.



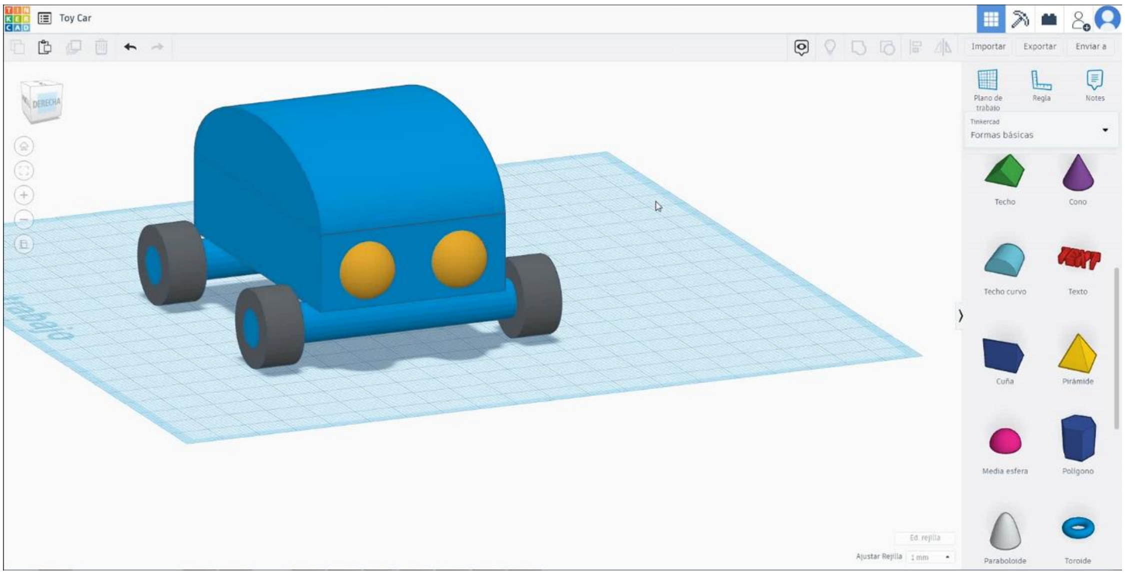


Co-funded by the Erasmus+ Programme of the European Union





6. Now, the car is finished.



9.3.20.2 Toy car 3D printing seetings

Filament

PLA
Diameter - 1.75 (mm)
Flow - 100%

Quality

Layer Height - 0.2 (mm)
Initial Layer Thickness - 0.3 (mm)
Shell Thickness - 0.8 (mm)
Bottom/Top Thickness - 1.2 (mm)
Fill Density - 20 (%)

Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑ ...

Quality	
Initial layer thickness (mm)	0.3
Initial layer line width (%)	115
Cut off object bottom (mm)	0.0
Dual extrusion overlap (mm)	0.15

Speed and Temperature

Print Speed - 50 (mm/s)
Travel Speed - 90 (mm/s)
Bottom Layer Speed - 30 (mm/s)
Printing Temperature - 215 (C)
Bed Temperature - 60 (C)

Speed and Temperature	
Print speed (mm/s)	75
Printing temperature (C)	210
Bed temperature (C)	60

Speed	
Travel speed (mm/s)	60
Bottom layer speed (mm/s)	30
Infill speed (mm/s)	0.0
Top/bottom speed (mm/s)	0.0
Outer shell speed (mm/s)	35
Inner shell speed (mm/s)	50

Support Type

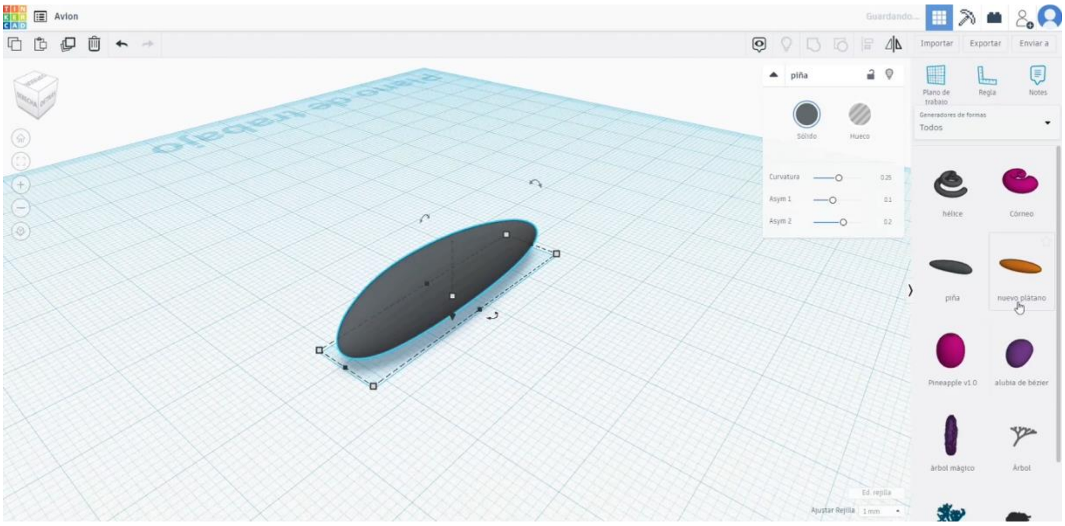
None / Touching Buildplate / Everywhere
Platform Adhesion Type - None / Brim / Raft

Support	
Support type	Touching buildplate
Platform adhesion type	None

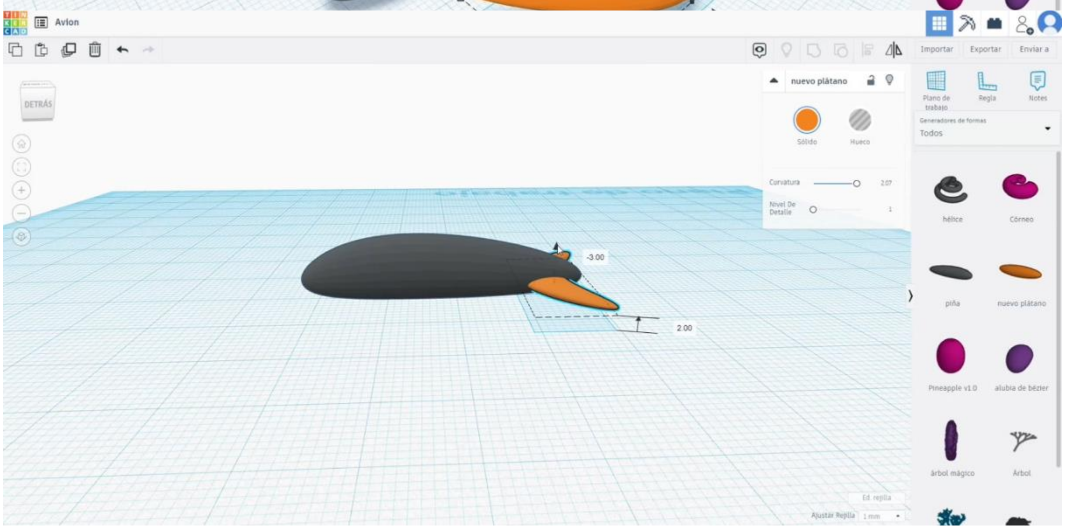
9.3.21 Part 21: Plane

9.3.21.1 Plane Design

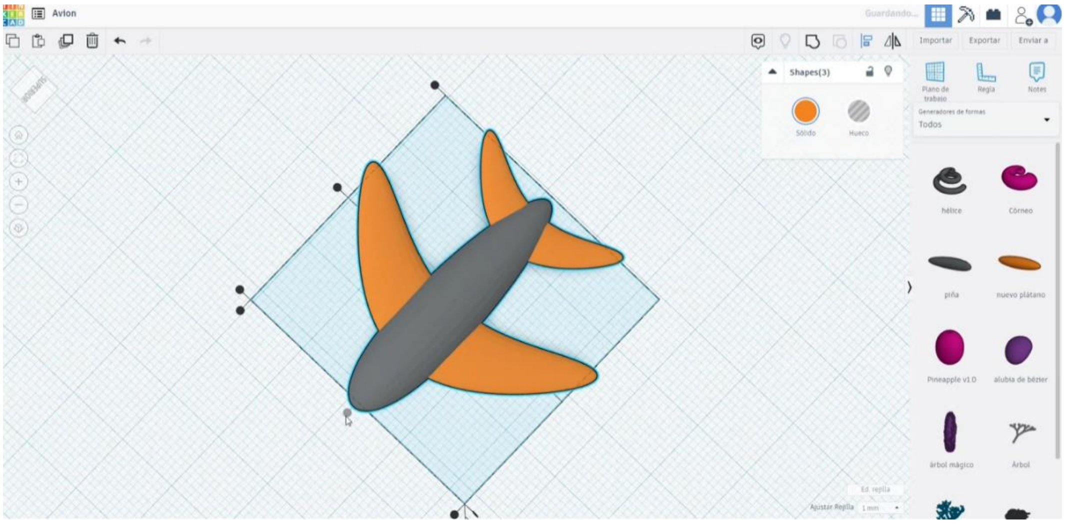
1. Build a pineapple, from the shapes generator list, and turn it 90 degrees.



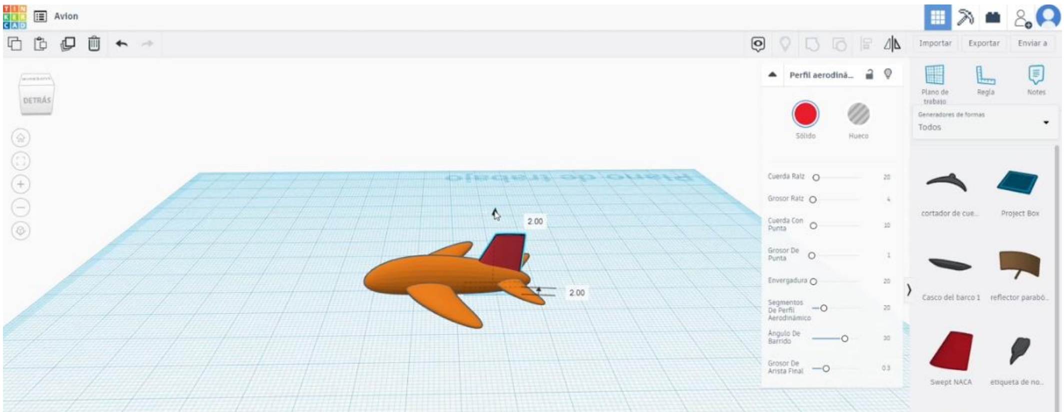
2. Draw a new banana, make it flatter and bend it editing the curvature. Turn it 90 degrees, scale it until fits with the main body. Place it like the tail of the plane.

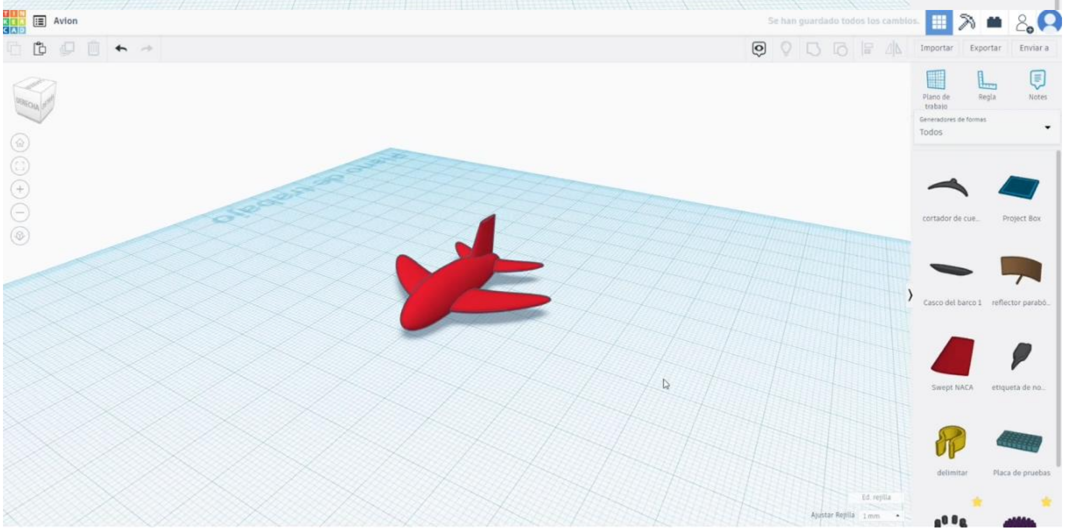


3. Copy the tail to créate the wings. Scale and place them like in the picture below. Align everything. Then press the group button.



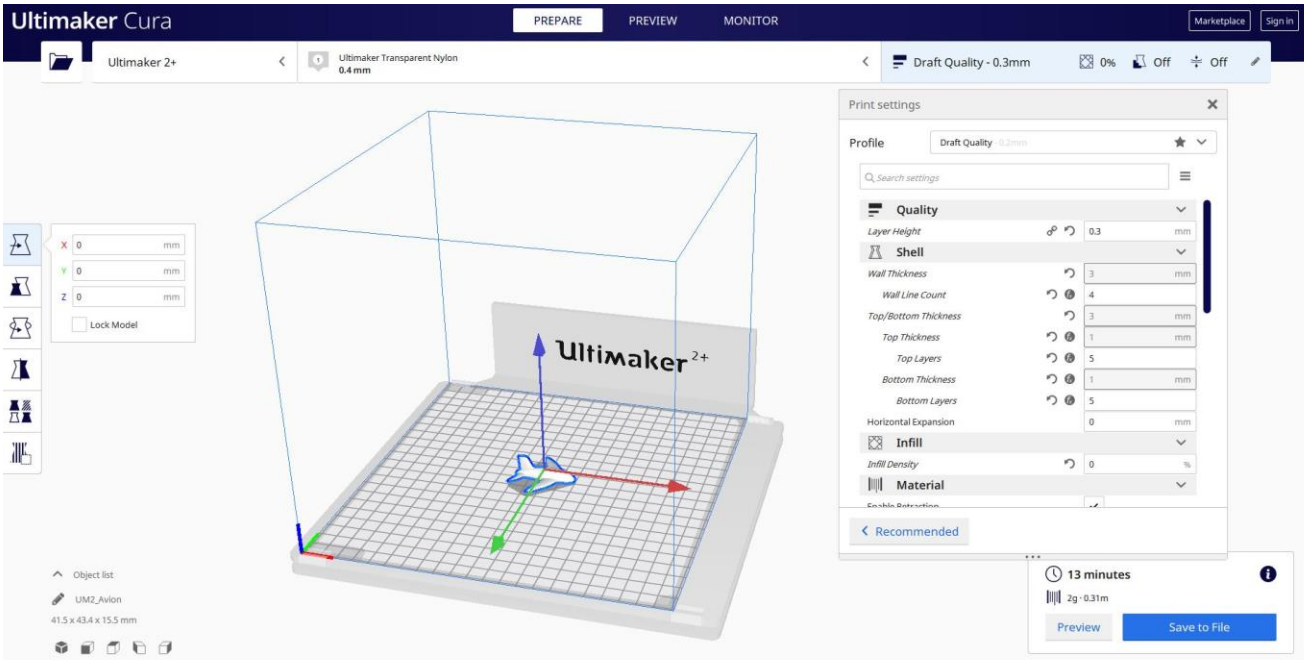
4. Now, look for the “Swept NACA“ in the shapes generators. Place it above the tail and scale it until it fits with the rest f the plane. Align it with the body and group them.



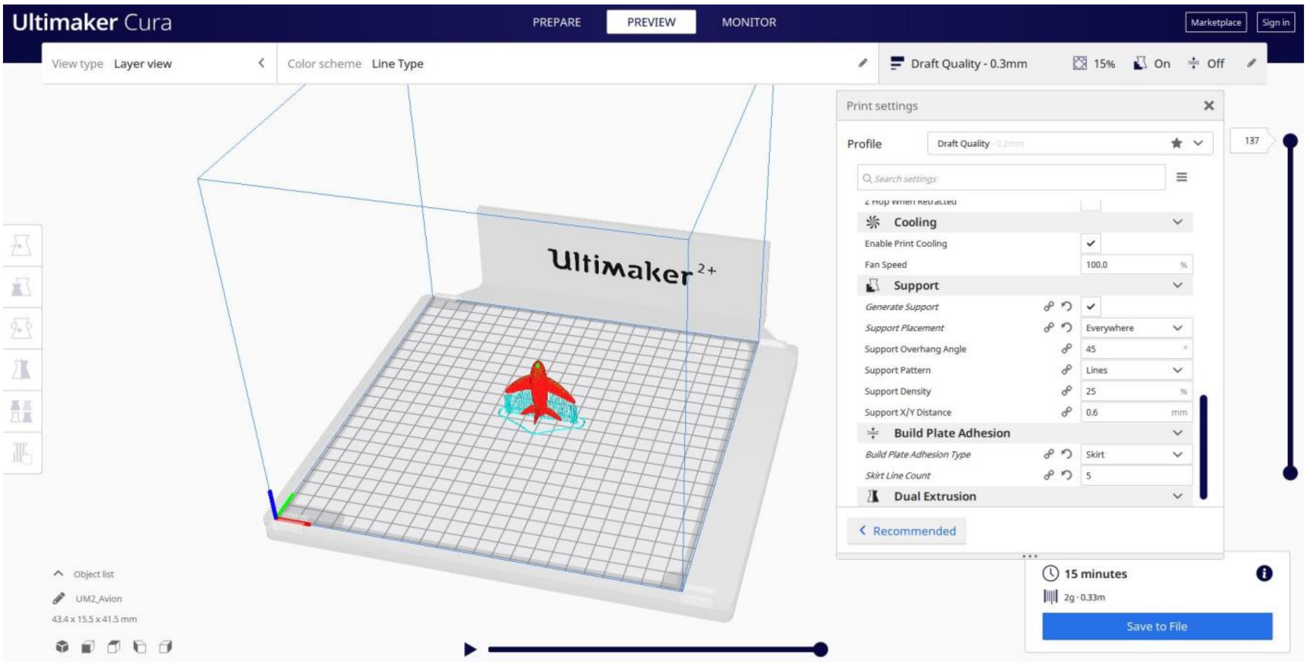


9.3.21.2 Plane 3D printing seetings

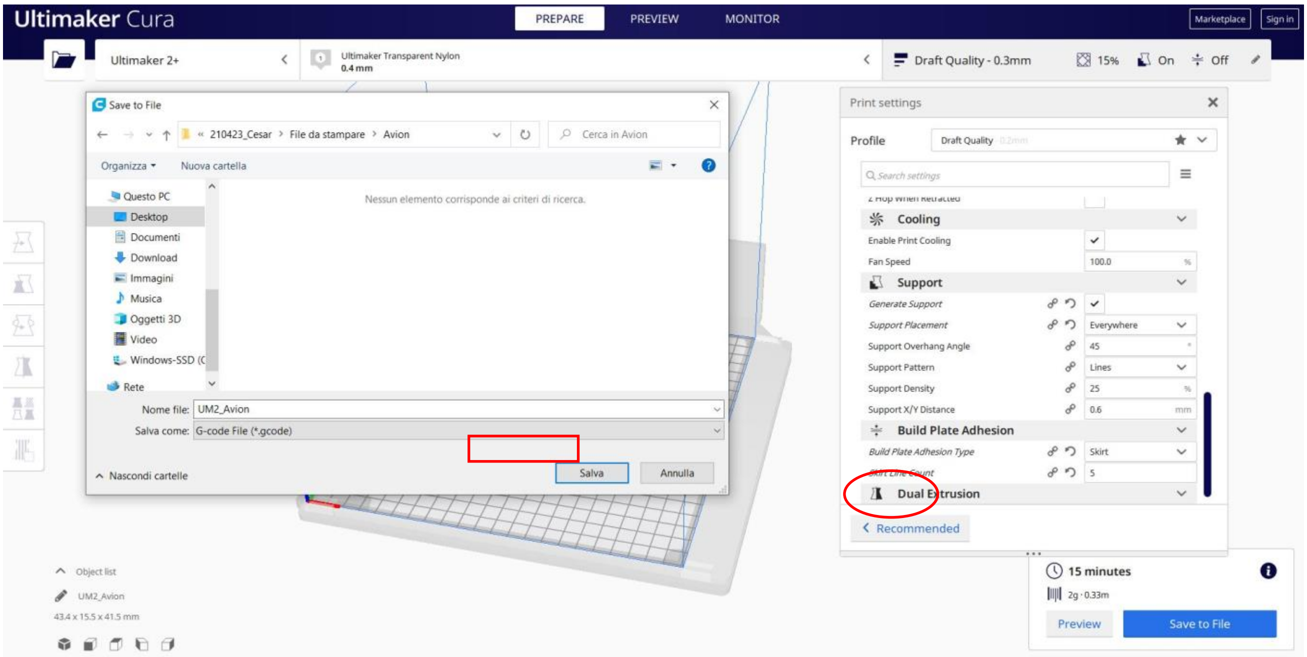
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



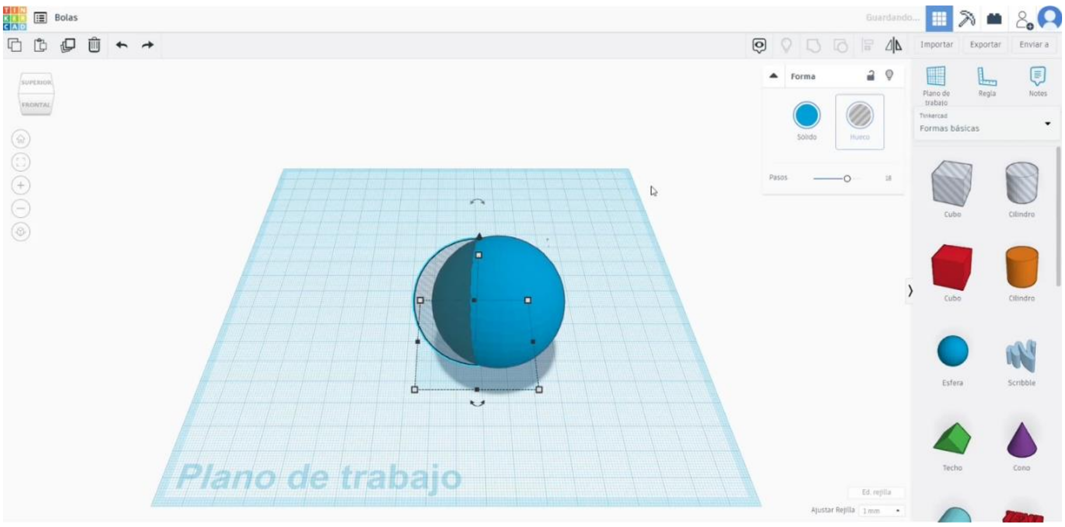
3. At this point I can save the ".Gcode" file to send to the machine.



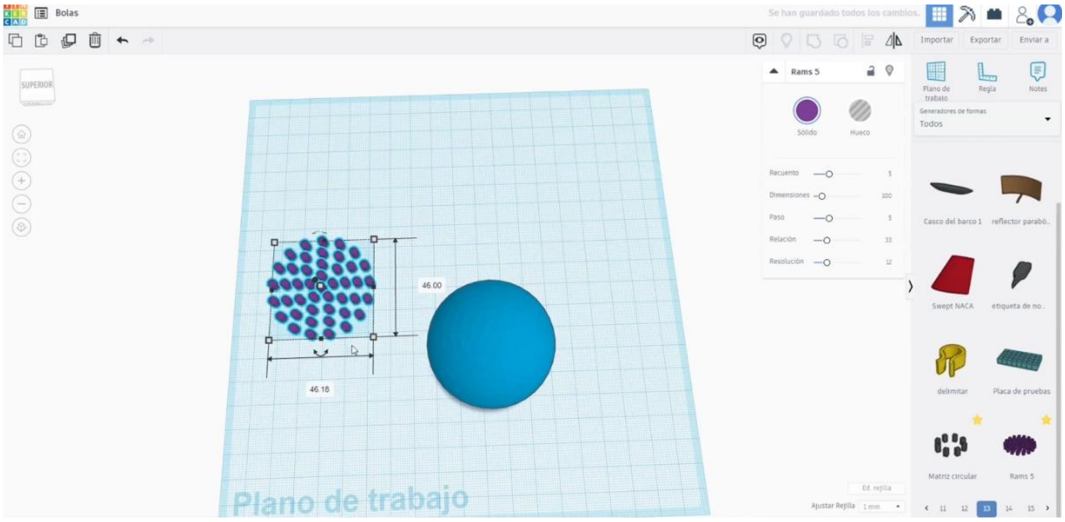
9.3.22 Part 22: Ball

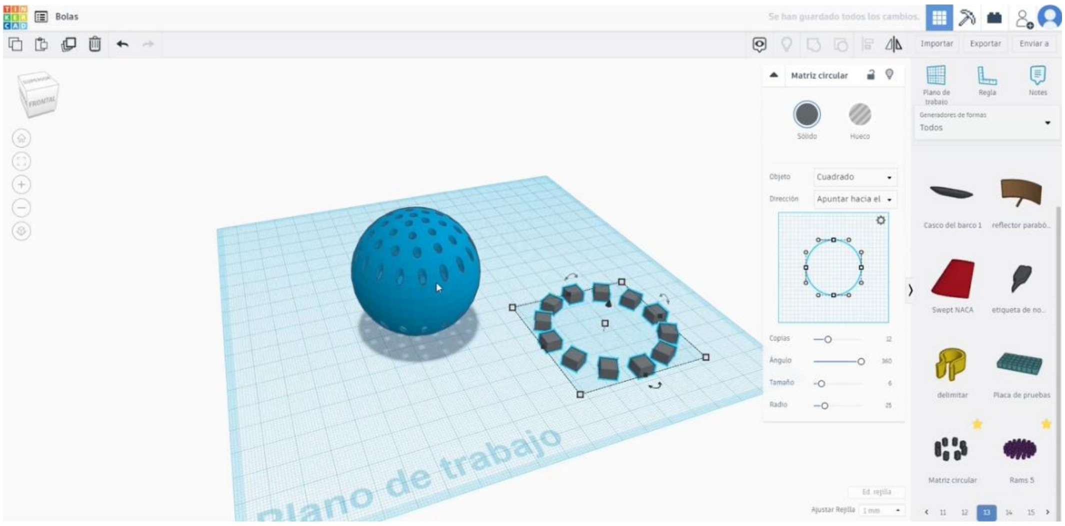
9.3.22.1 Ball Design

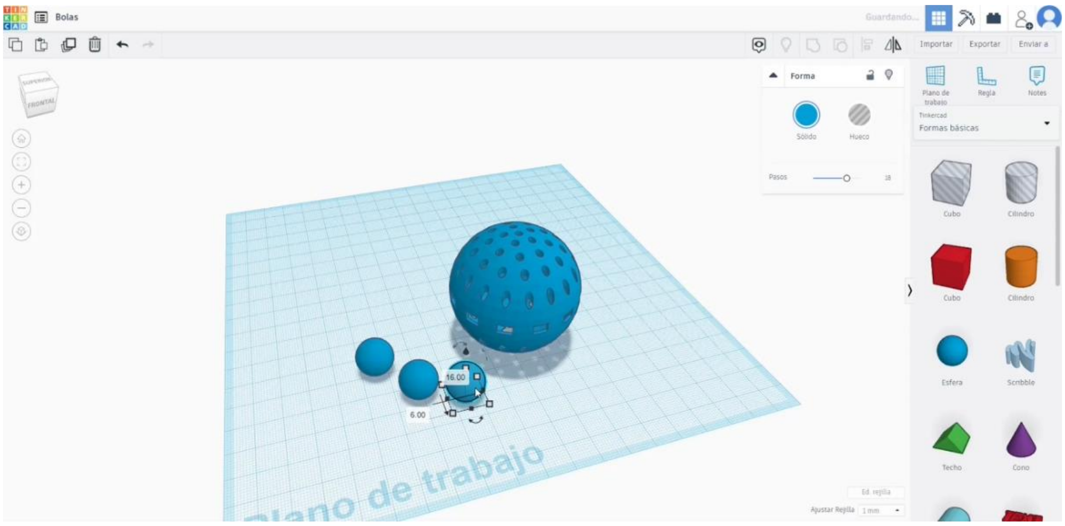
1. Draw a sphere, change the measurements to 50 x 50. Copy it and change the measurements of the second one to 48x48. Turn this into hole mode. Align them until they are one in the center of the other and group them together.



2. Build a *Rams 5* from the shapes generator list. Change the measurements to 46 of diameter. Increase the height until it exceeds the diameter of the ball. Align all the objects. Turn the Rams 5 into hole mode and group it with the ball.

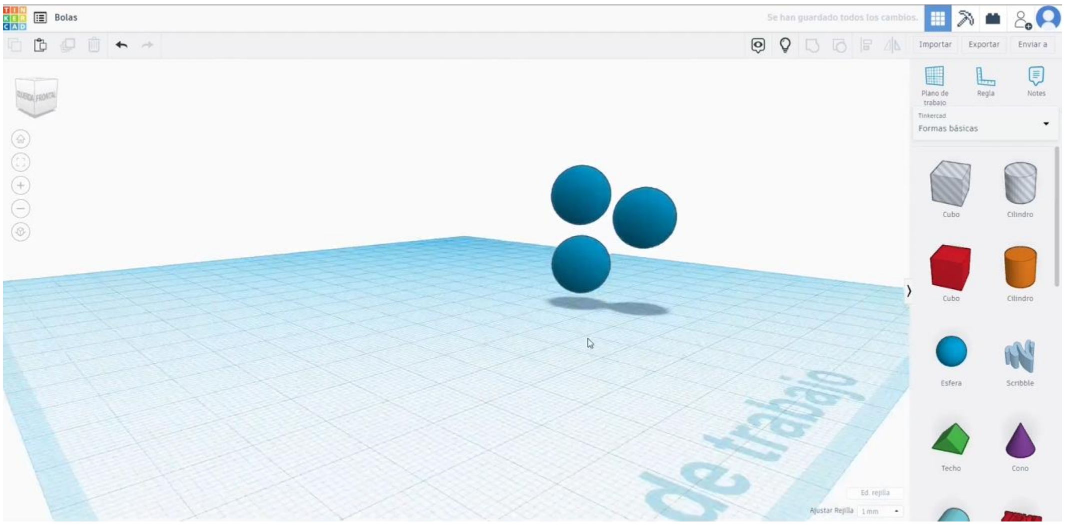


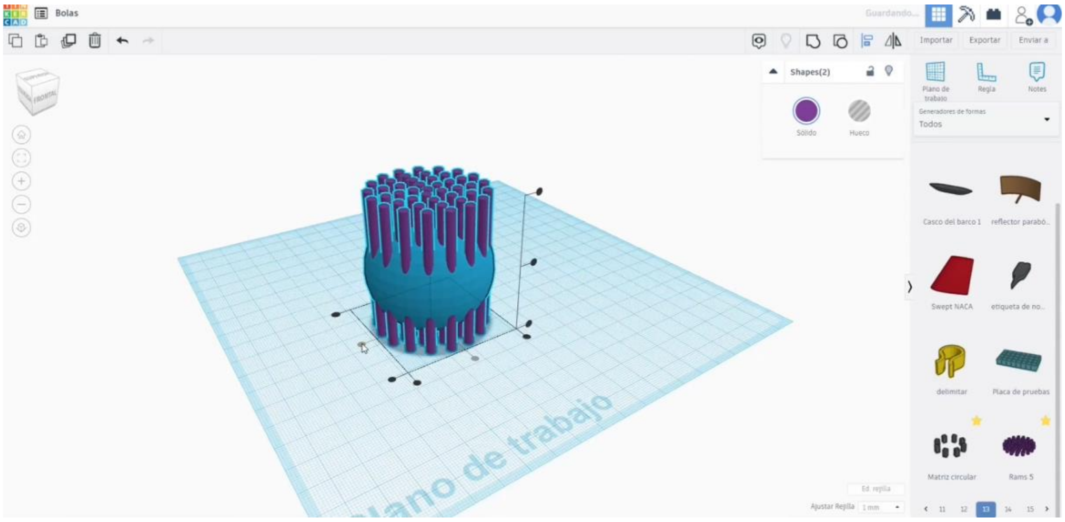


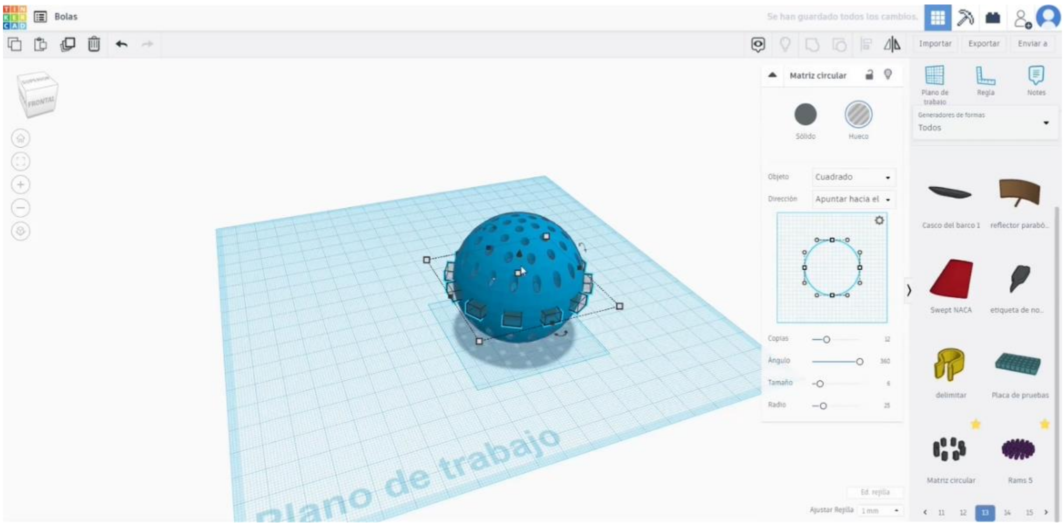


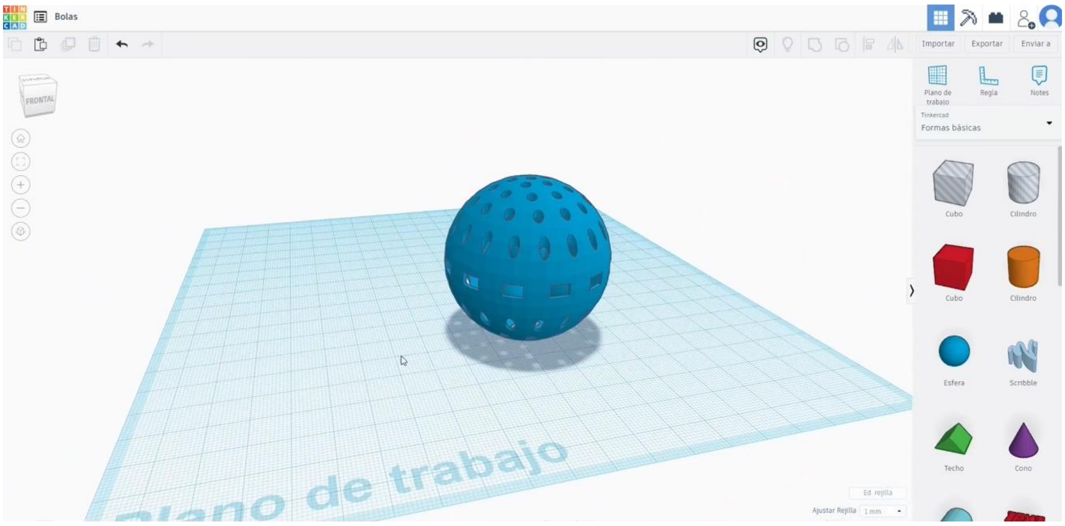
3. Create a circular matrix (12 copies, size 6, radius 25, height 4). Align the matrix with the ball. Turn it into hole mode and press group button.

4. Now, draw 3 spheres of diameter 14. Place them inside the big sphere, pay atention to don't overlay them. You can hide the big one to be sure than the small balls are floting inside withoug touching each other. You can hide the big one to be sure than the small balls are floting inside withoug touching each other.



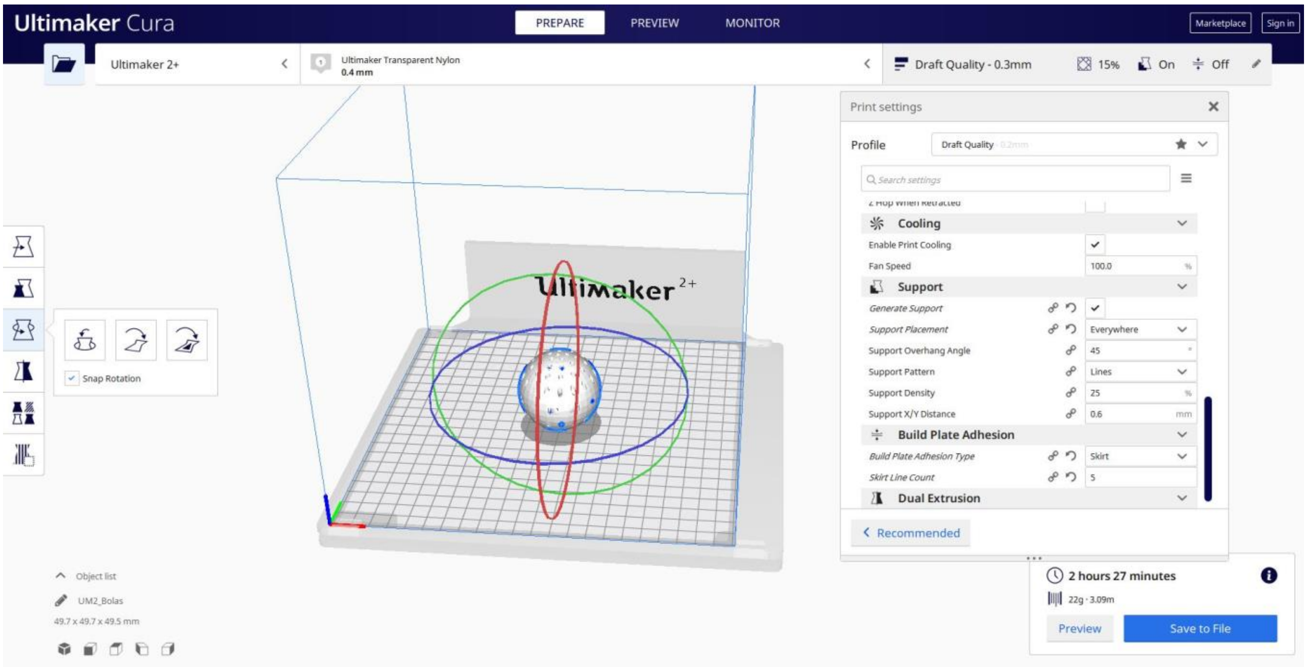




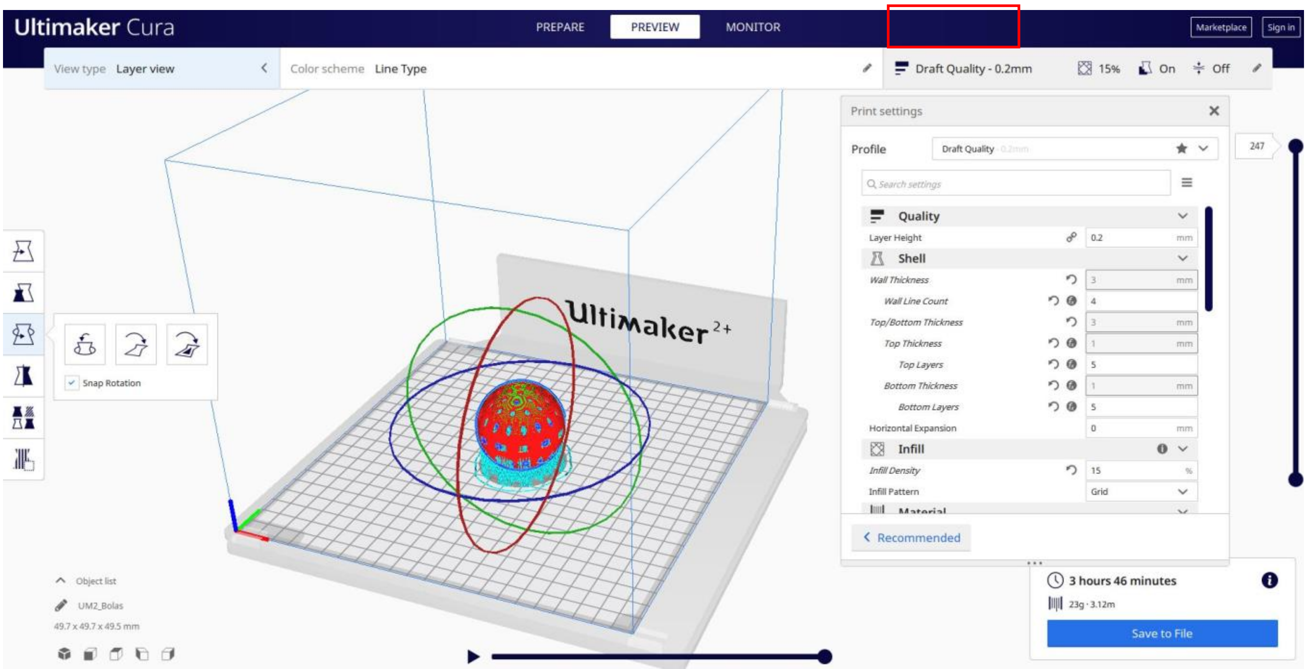


9.3.22.2 Ball 3D printing seetings

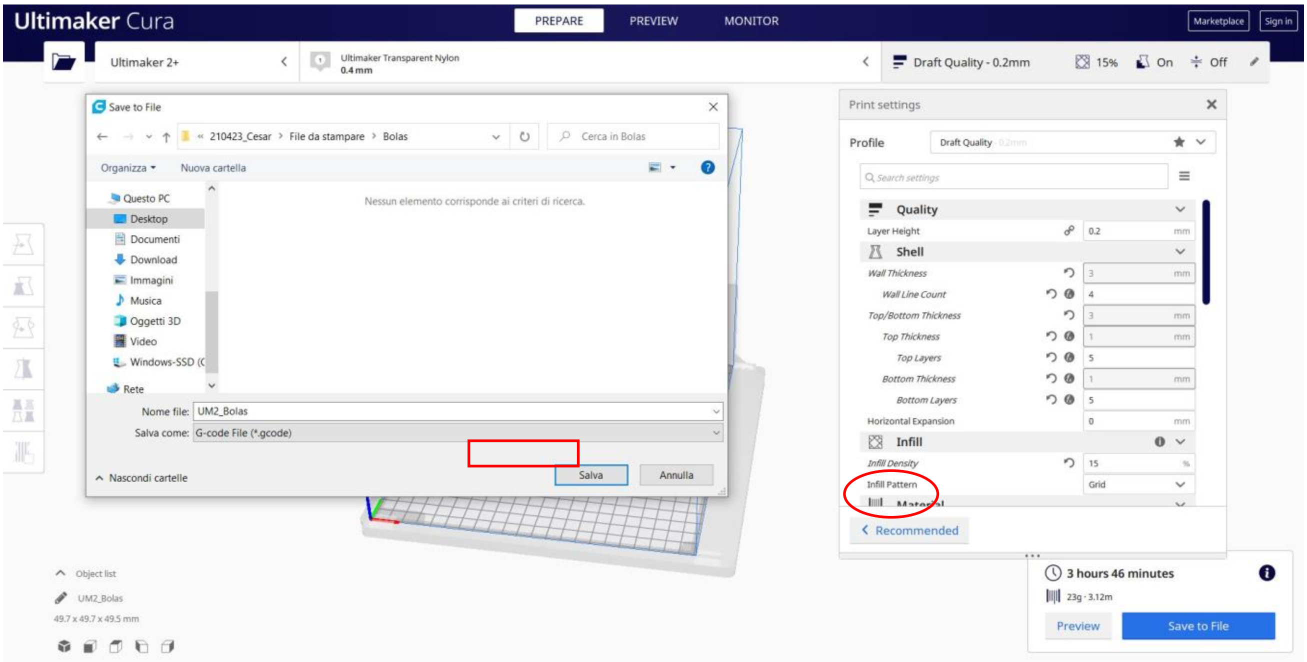
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



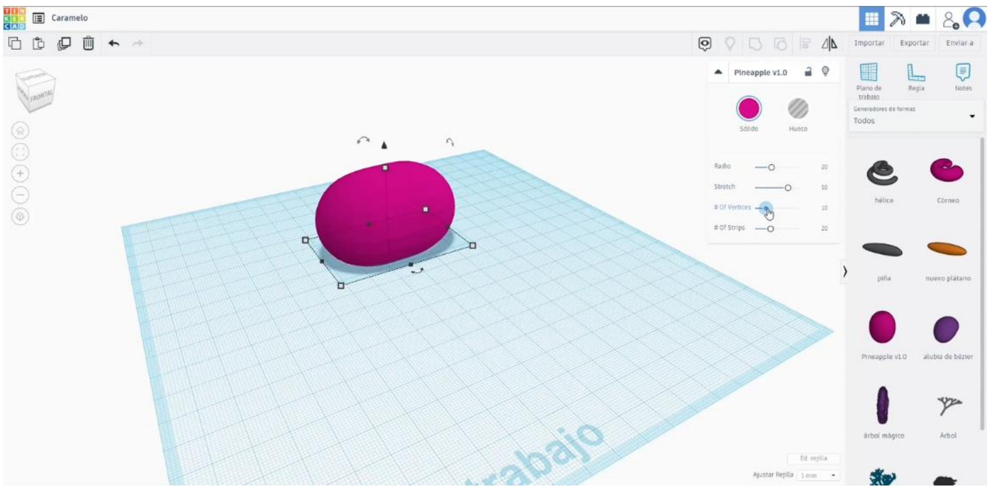
3. At this point I can save the ".Gcode" file to send to the machine.



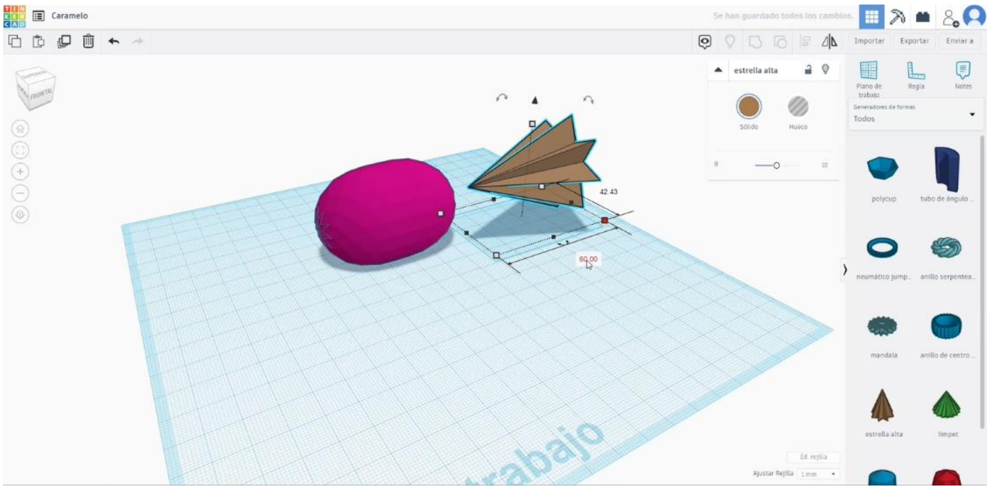
9.3.23 Part 23: Candy

9.3.23.1 Candy Design

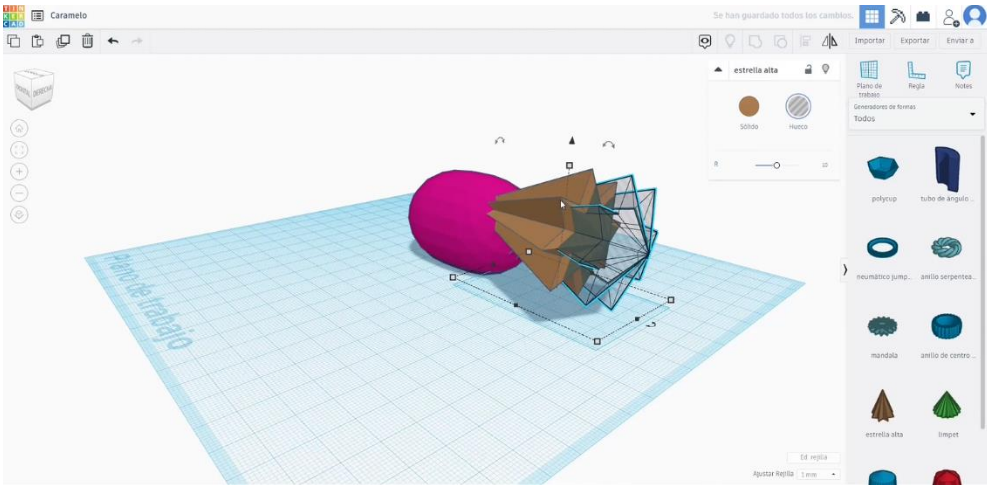
1. Draw a *Pineapple v1.0* (in shapes generators), turn it 90 degrees. Change the measurements to 20 x 50. Change the number of vertex and strips to make it more faceted.



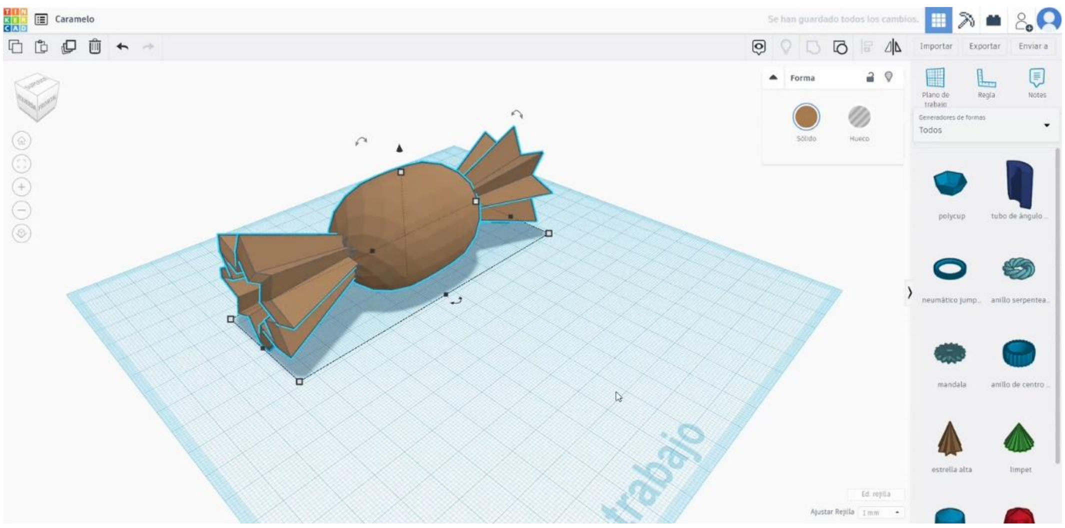
2. Buid a *high star.* Change its height to 60 and place it next to the other body.



3. Copy the star and place the second one like in the image below. Turn it into hole mode and group it with the other star.

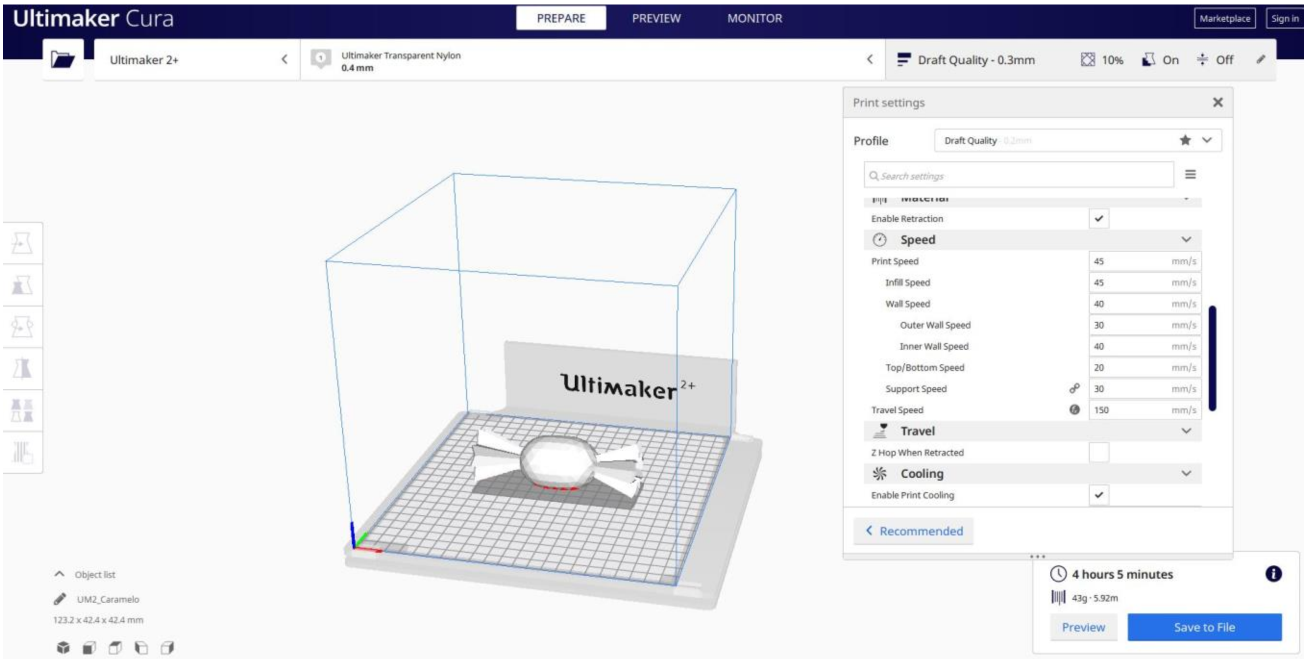


4. Do a symmetry of the hollow star that we obtained and place it in the other side of the central body. Align them all and merge them by pressing group button.

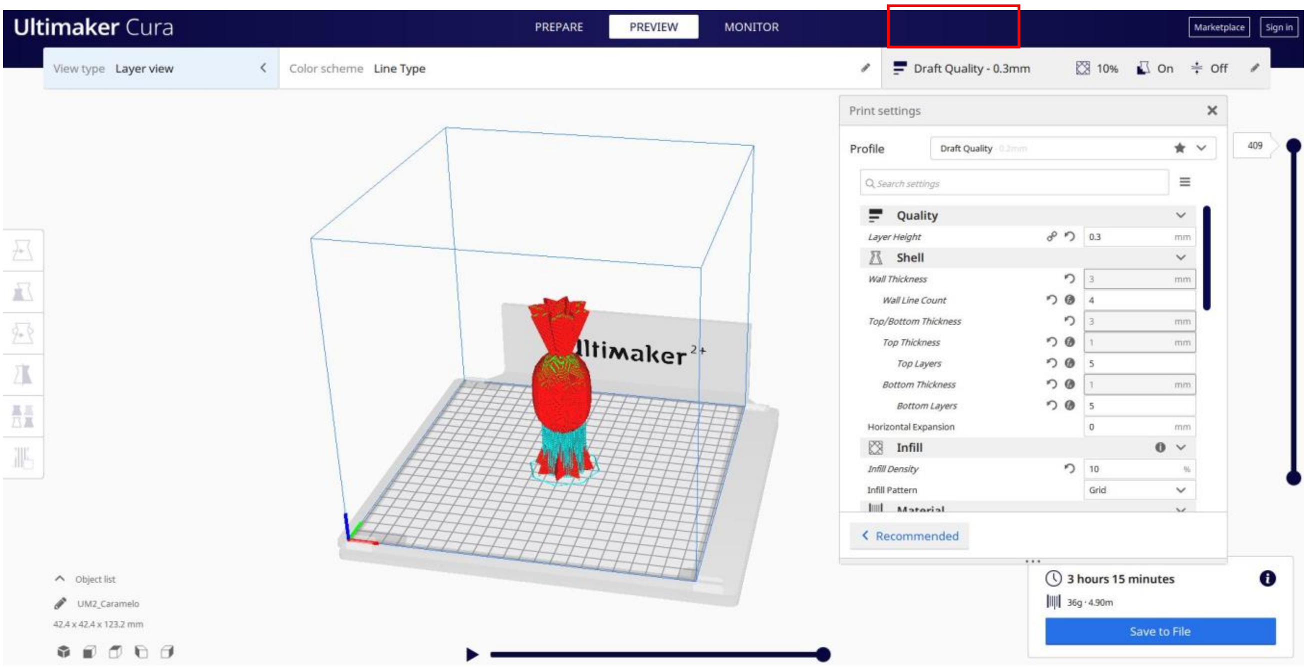


9.3.23.2 Candy 3D printing seetings

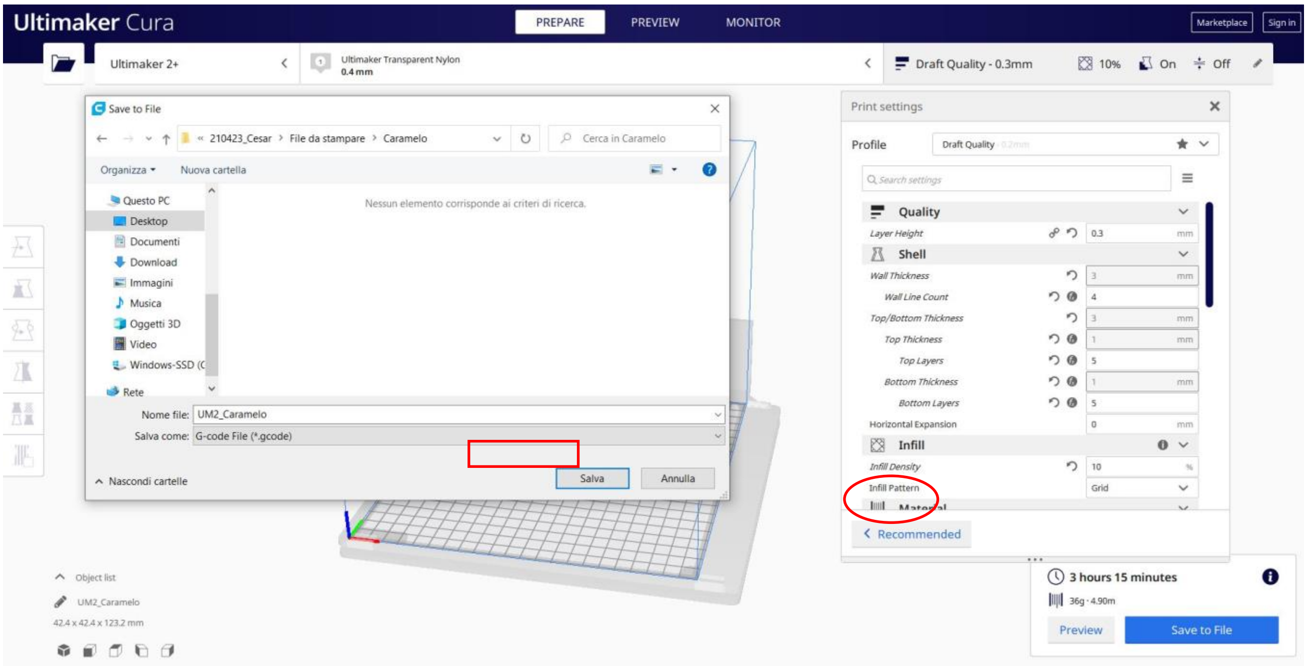
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



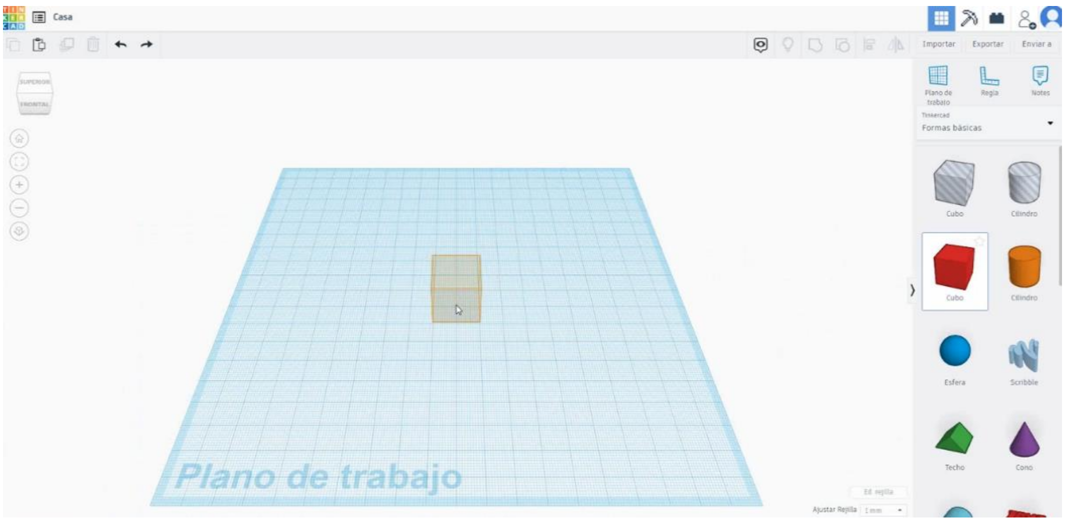
3. At this point I can save the ".Gcode" file to send to the machine.



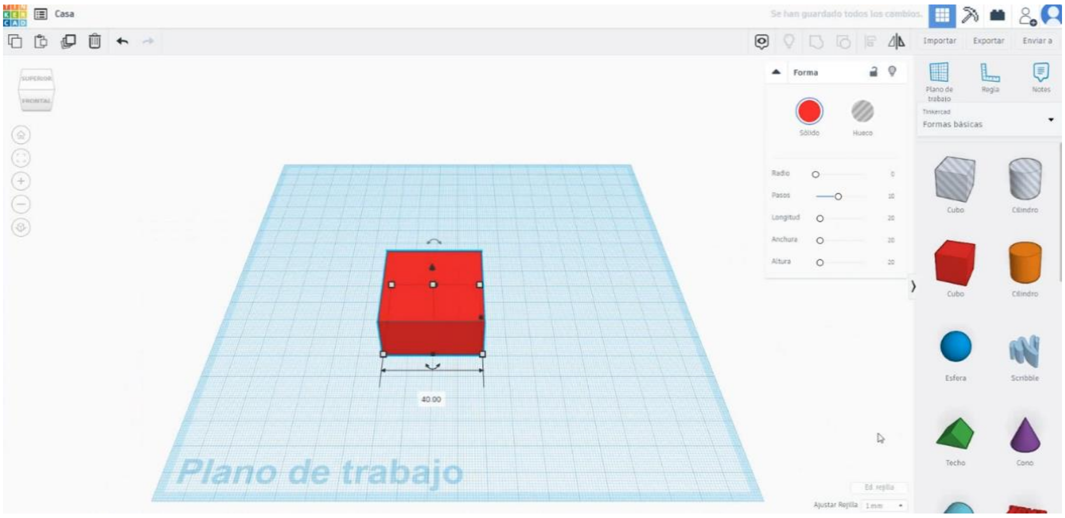
9.3.24 Part 24: House

9.3.24.1 House Design

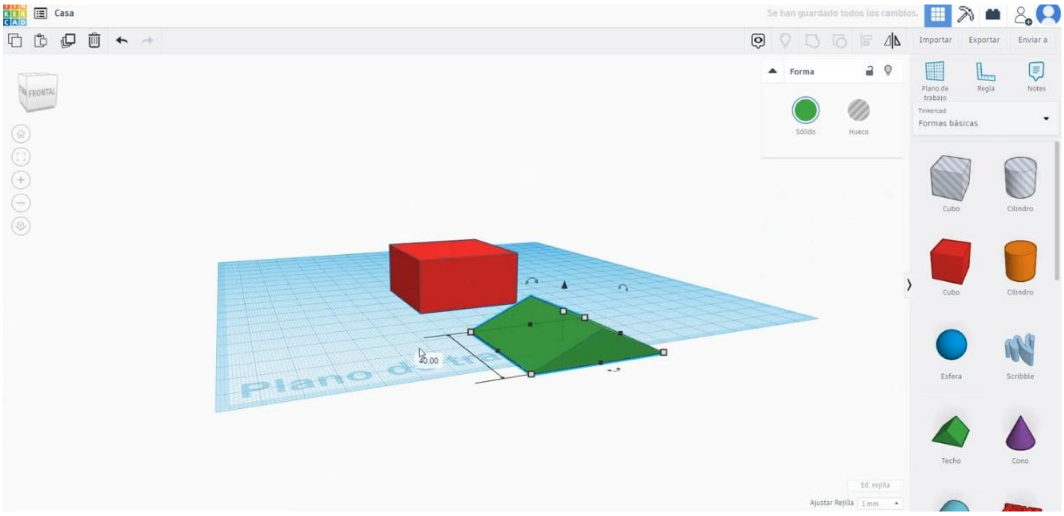
1. Build a Cube.



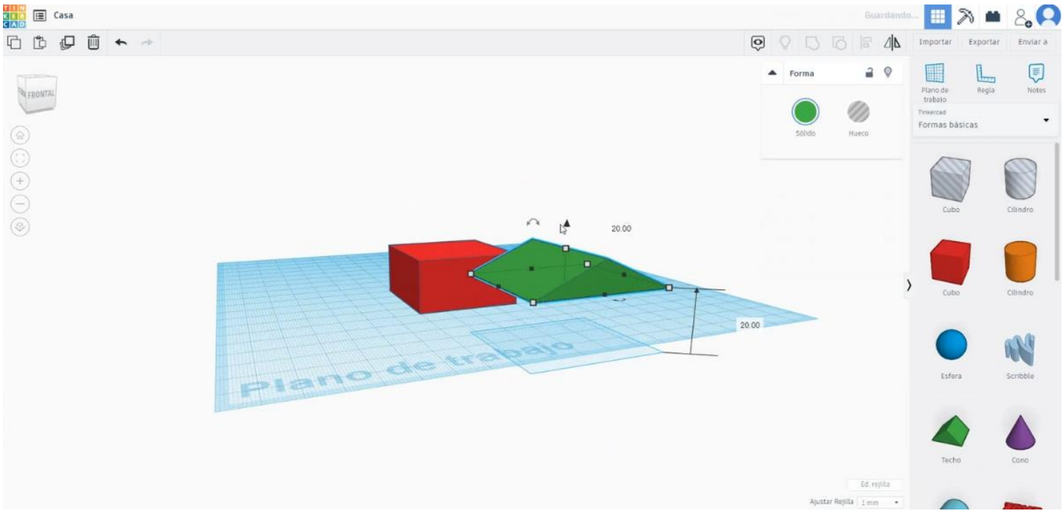
2. Change the measurements to 40 x 40 x 20.



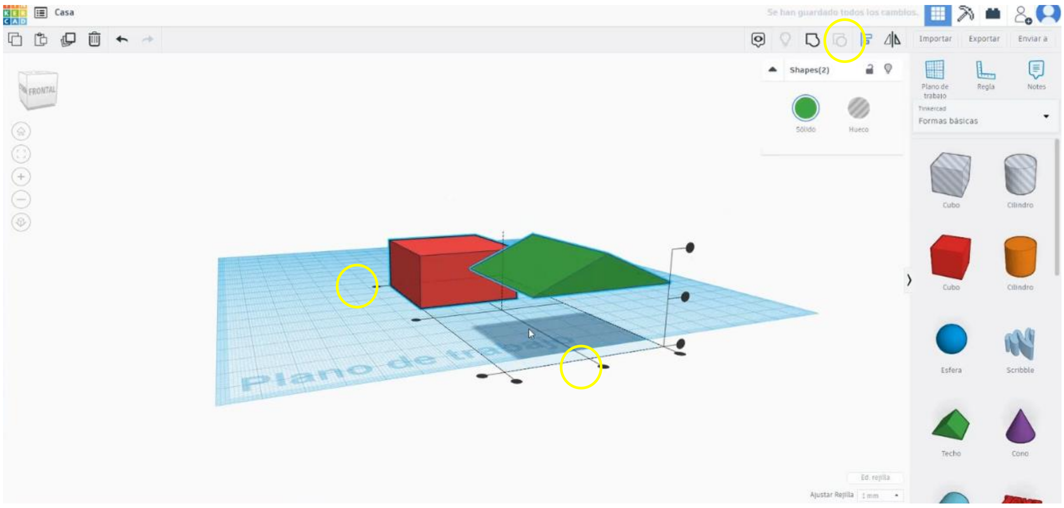
3. Now build the roof and change the width to 45 and the Depth to 40.



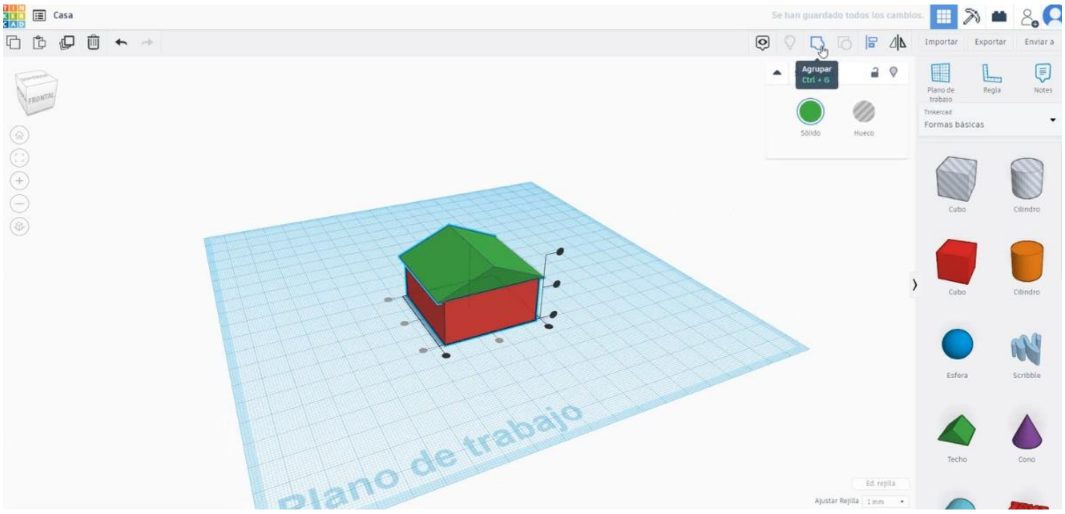
4. Raise the roof up to 20 high.



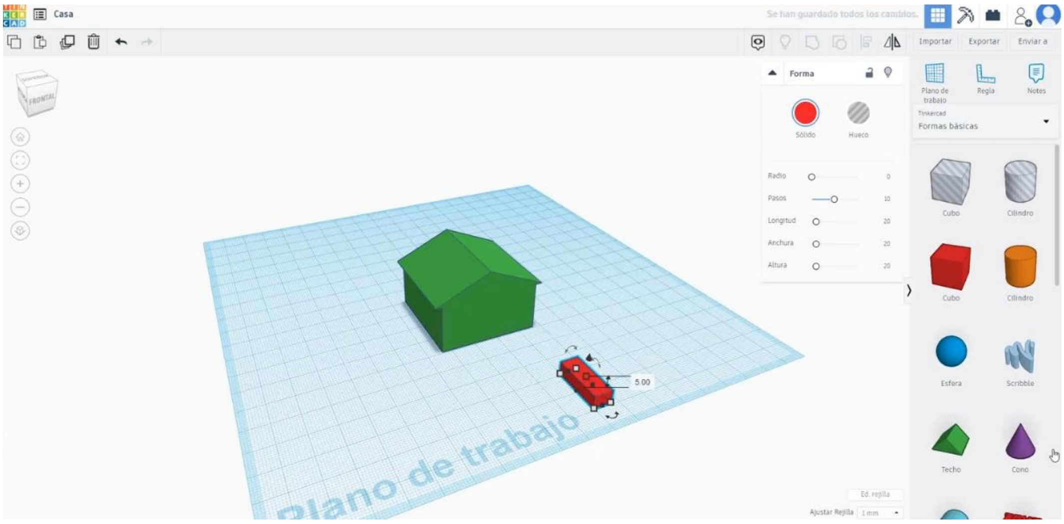
5. Align both figures selecting the two of them by pressing “Shift”.



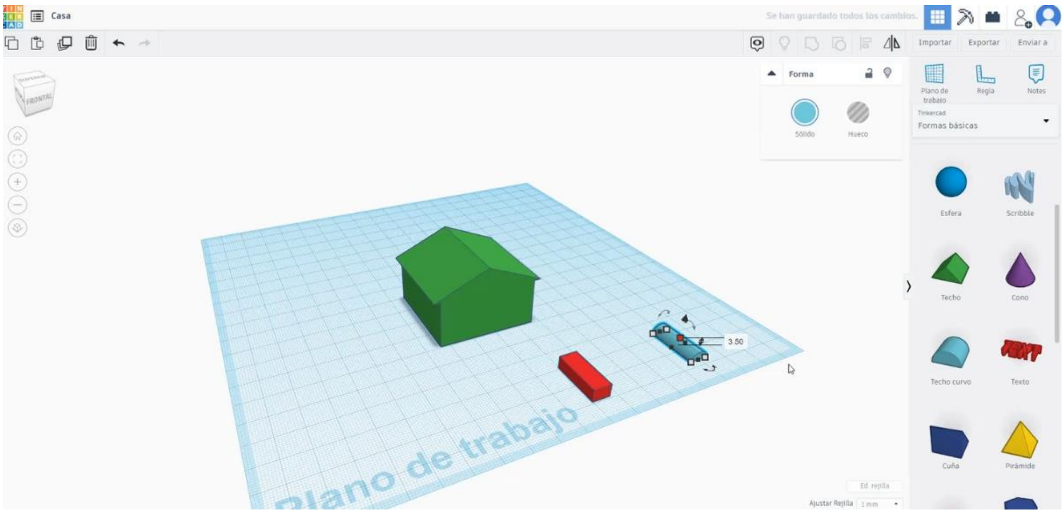
6. Then, press “Group” to merge them into one.



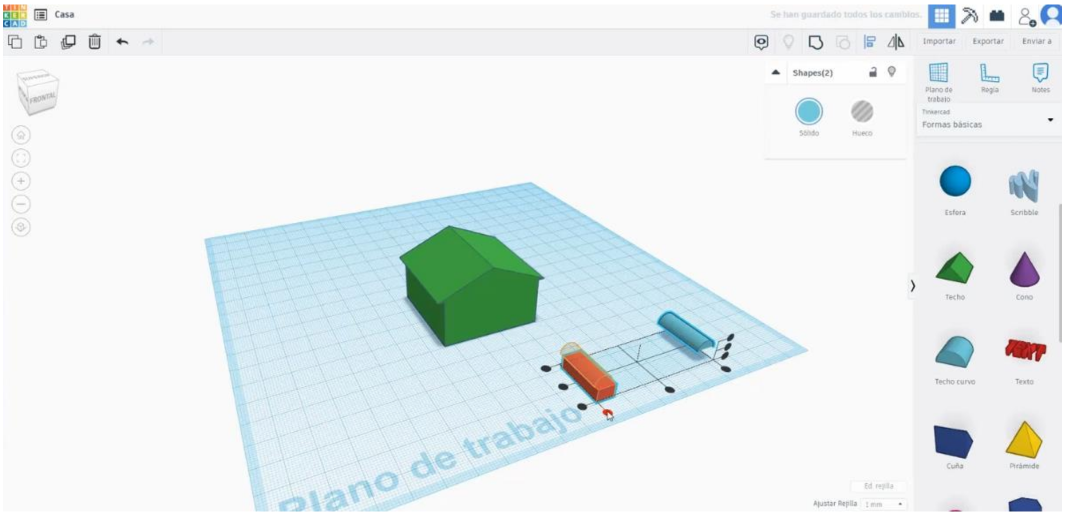
7. Select a new cube and change their measurements to 7 width and 5 height.



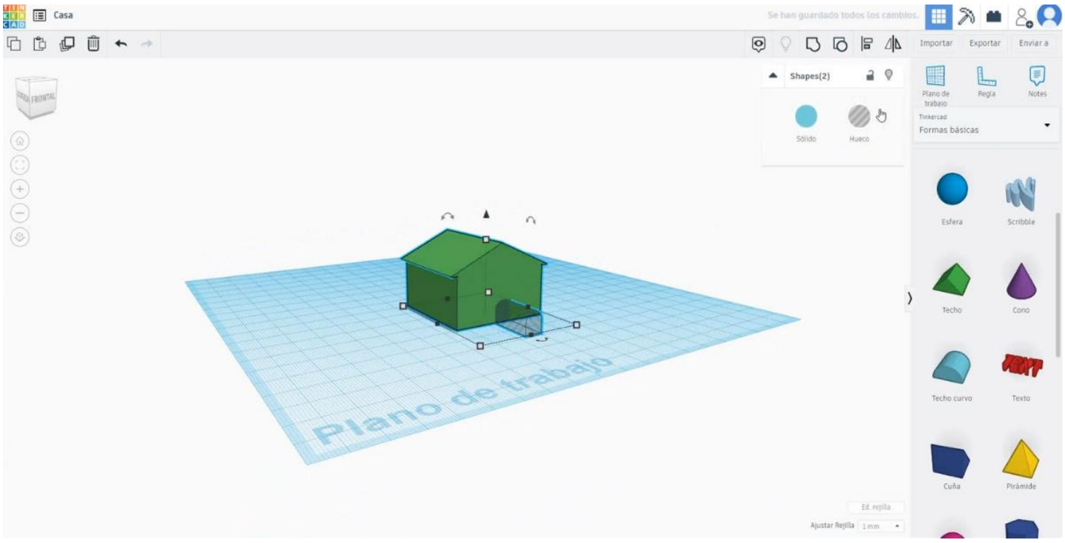
8. Now, draw a “Round Roof”, 7 width and 3.5 height. Raise it up to 5 high.



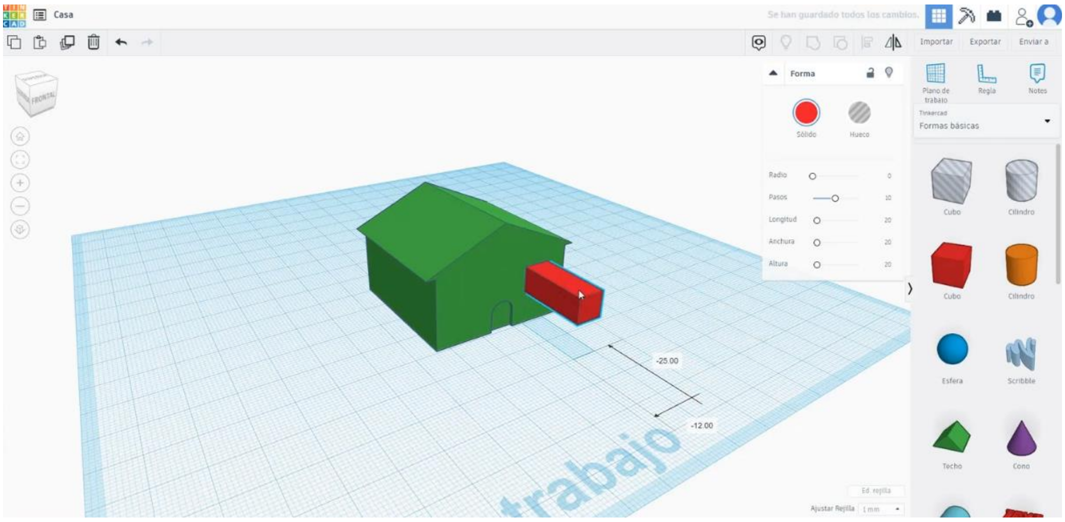
9. Select the cube and the round roof and align them. Then, group them.



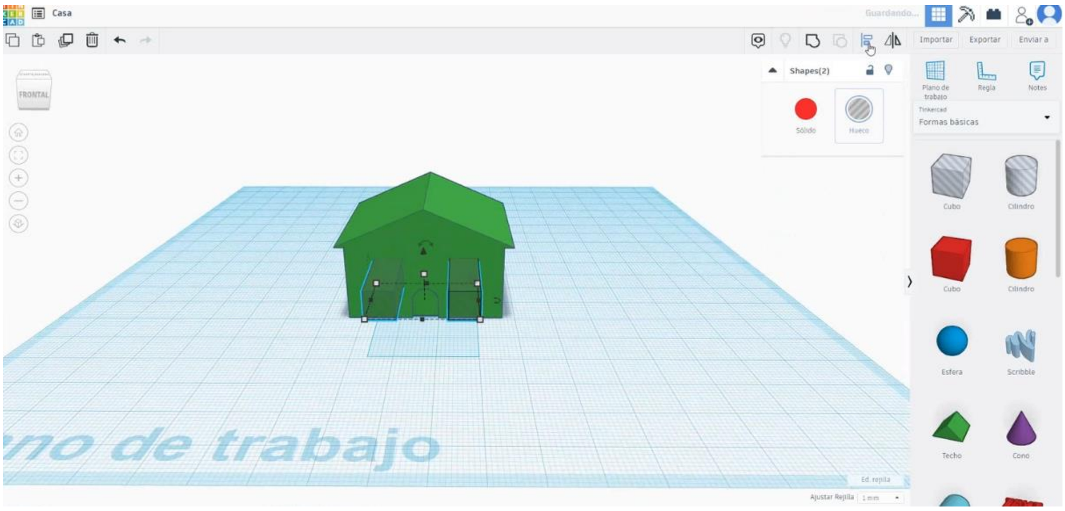
10. Place the door in the center of the house and change it to hole mode. Then, group them.



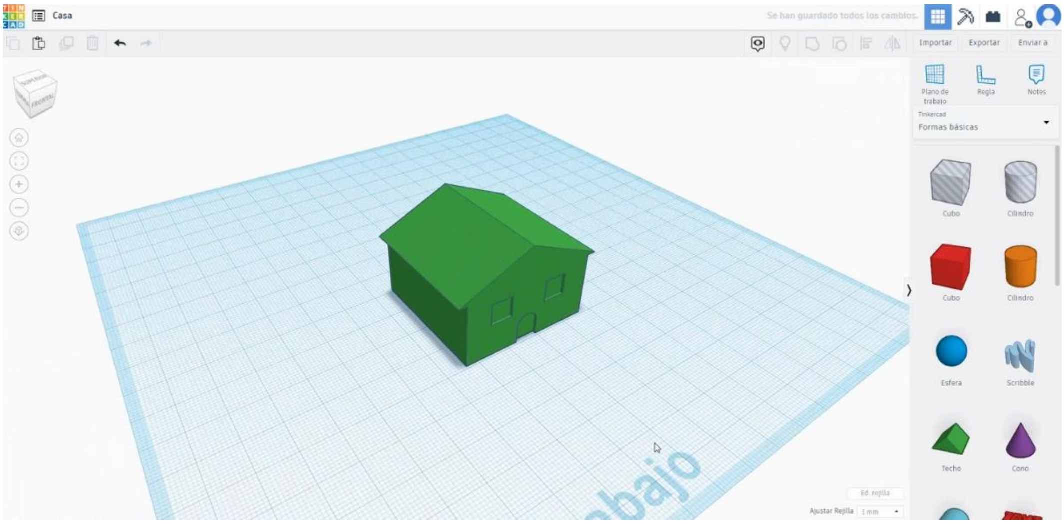
11. Use a cube 7 x 7 to make the window. Place it in the front side of the house.



12. Copy this window pressing the “Alt” button, move the new one to the other side. Then change them to hole mode and group the house and the two Windows.

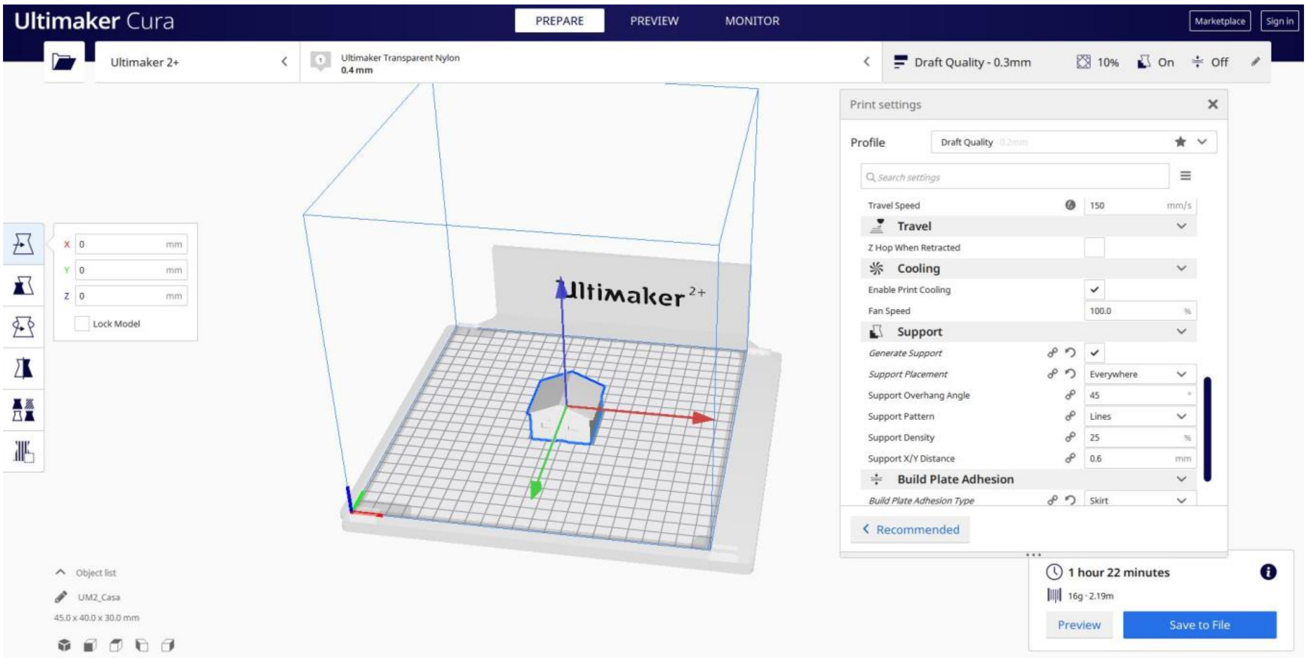


13. The house is finished.

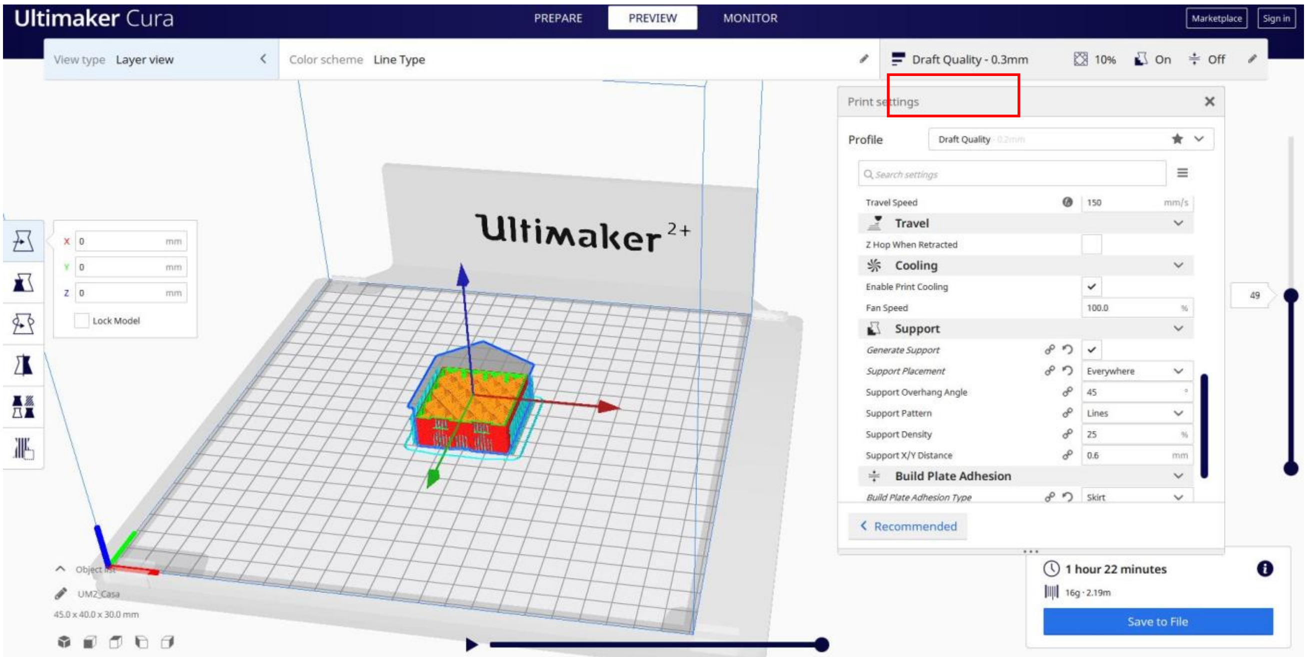


9.3.24.2 House 3D printing seetings

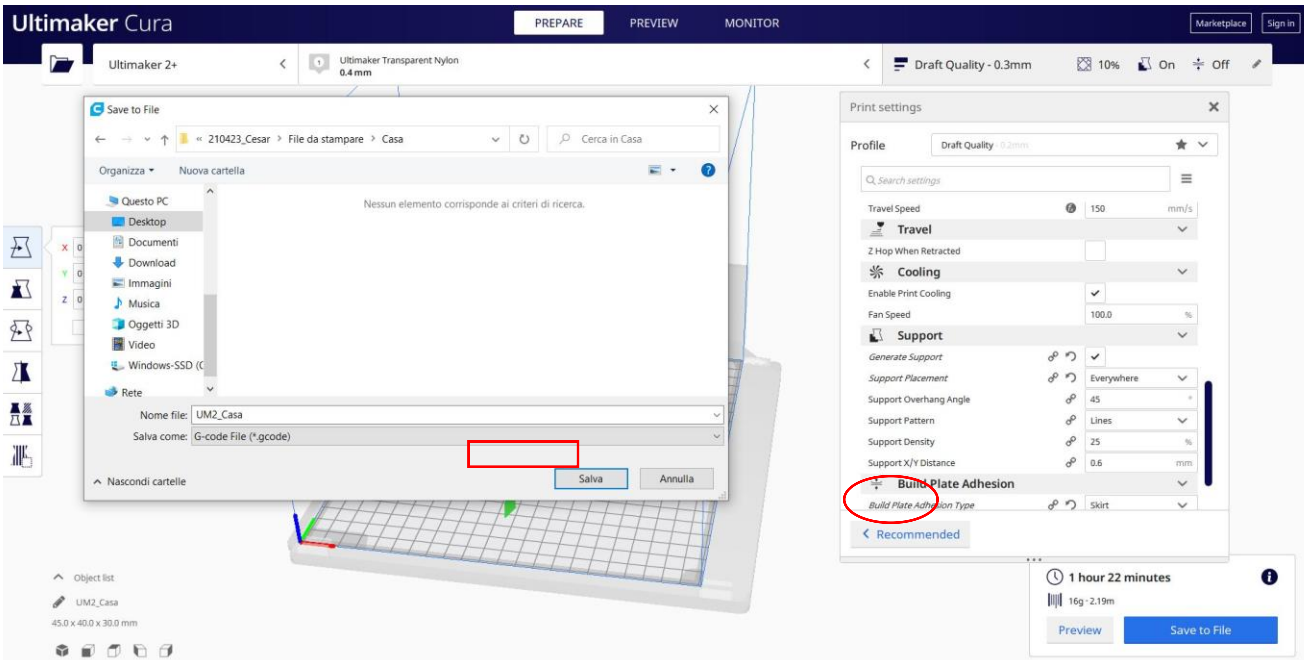
I. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



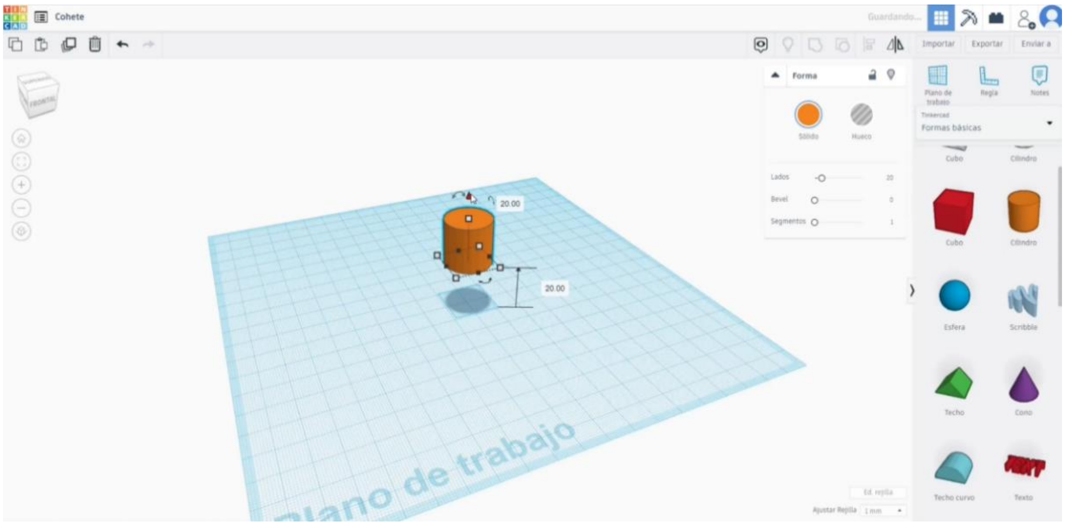
3. At this point I can save the ".Gcode" file to send to the machine.



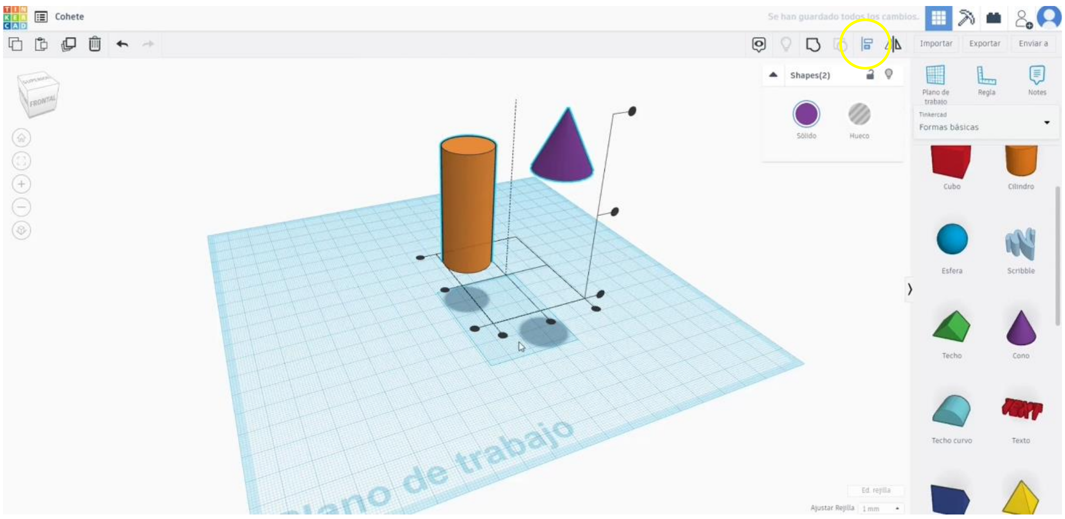
9.3.25 Part 25: Rocket

9.3.25.1 Rocket Design

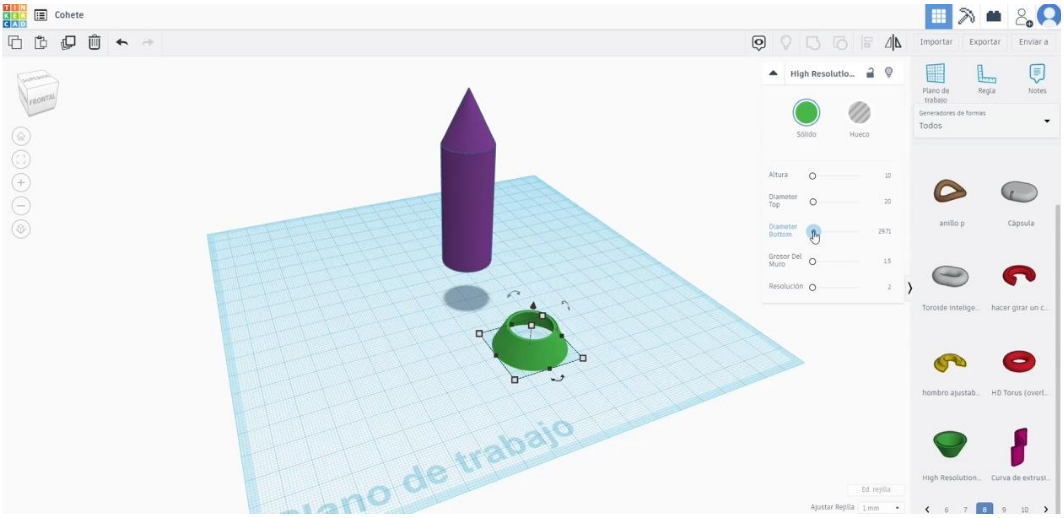
1. Build a cylinder. Place it to 20 above the work plane and change its height to 50.



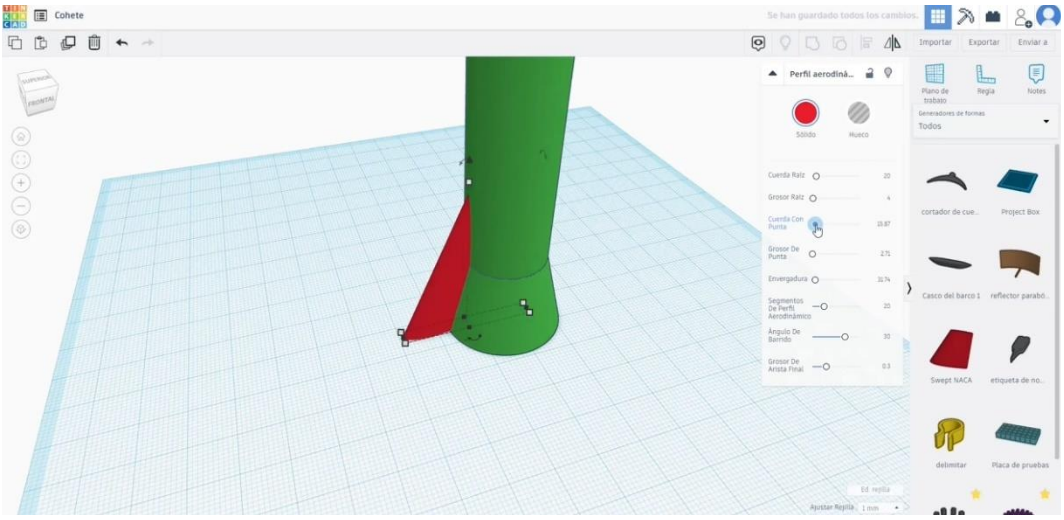
2. Draw a cone and align it with the cylinder. Then, group them.



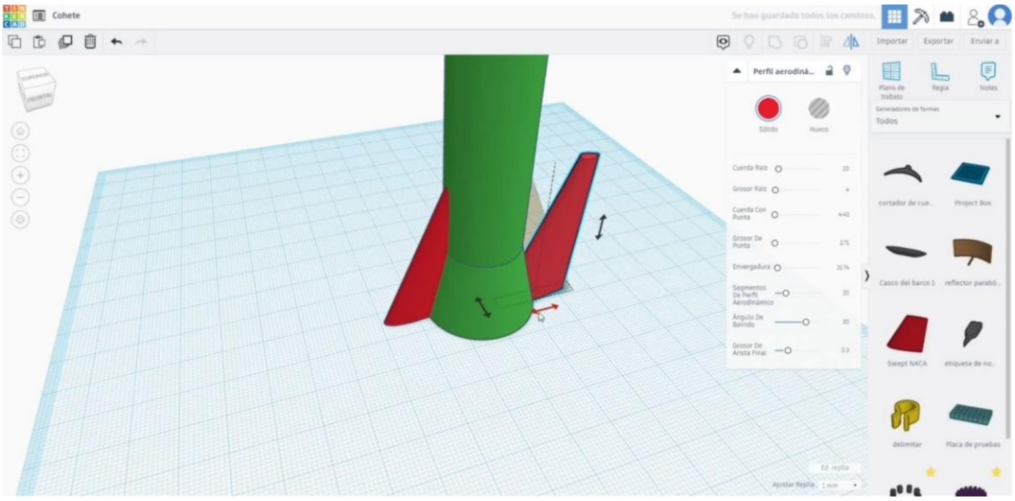
3. Look for the “high resolution tube…” in Shapes generators and place it under the cylinder. Change the measurements: Diameter top 20 and Diameter bottom 30. The height 20. Align and group them.



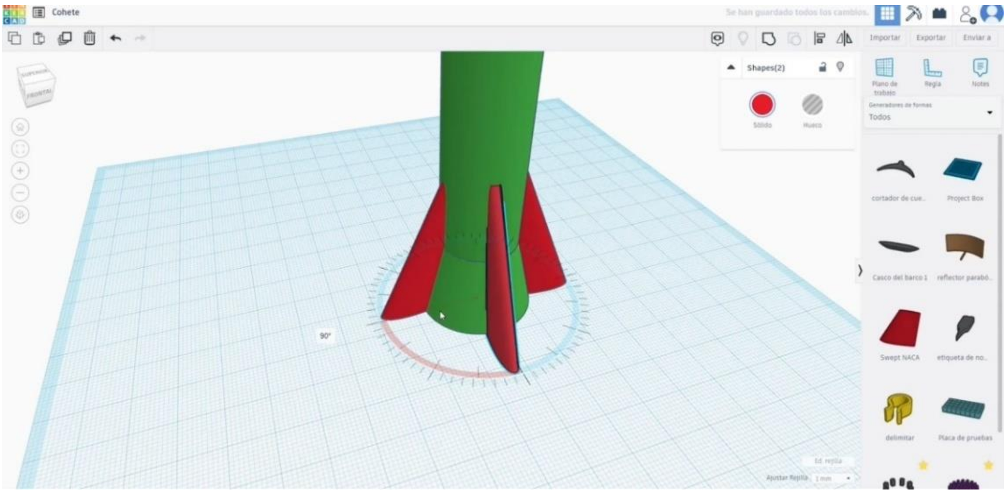
4. Build a “Swept NACA” and change the dimensions as you need.



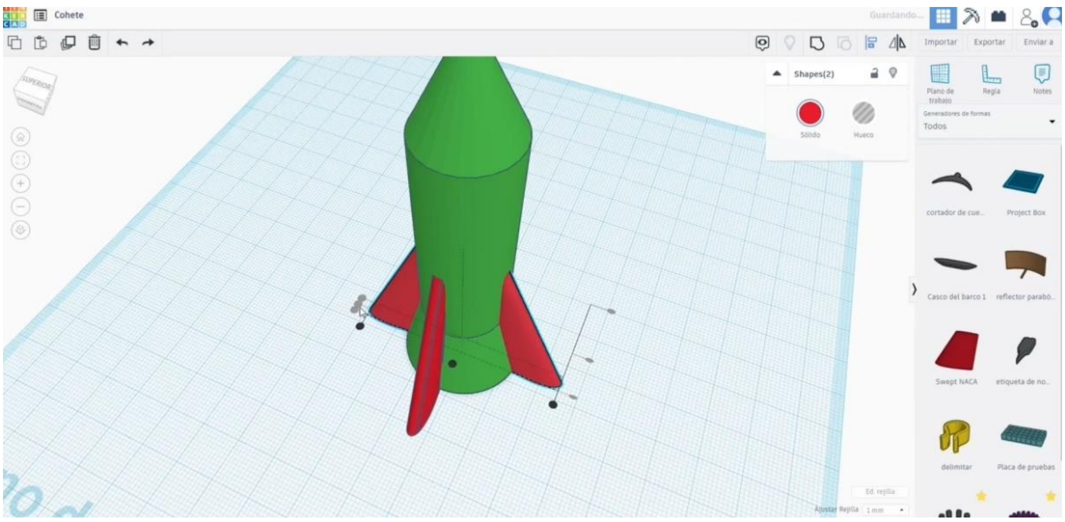
5. Copy the Swept Naca and place it in the other side. Press the symmetry button.



6. Copy this two pieces and rotate them to get four pieces in total.

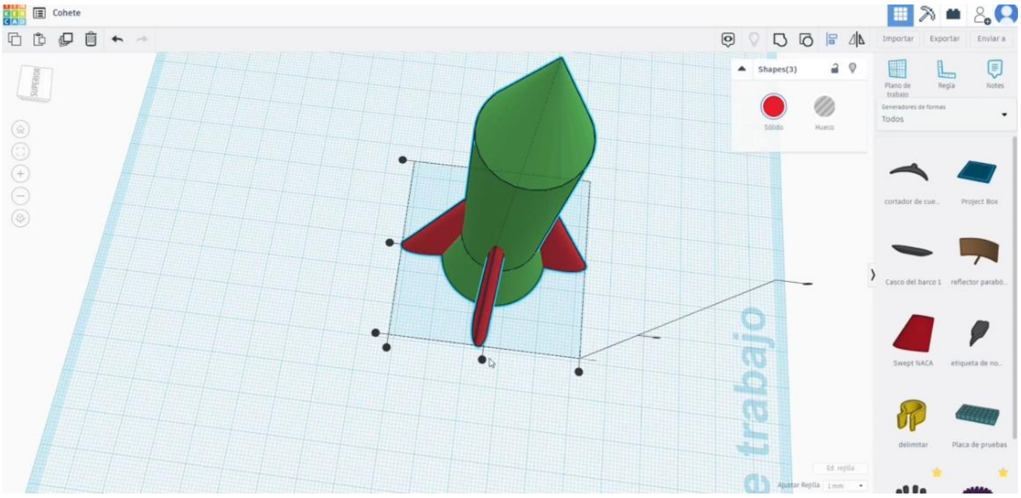


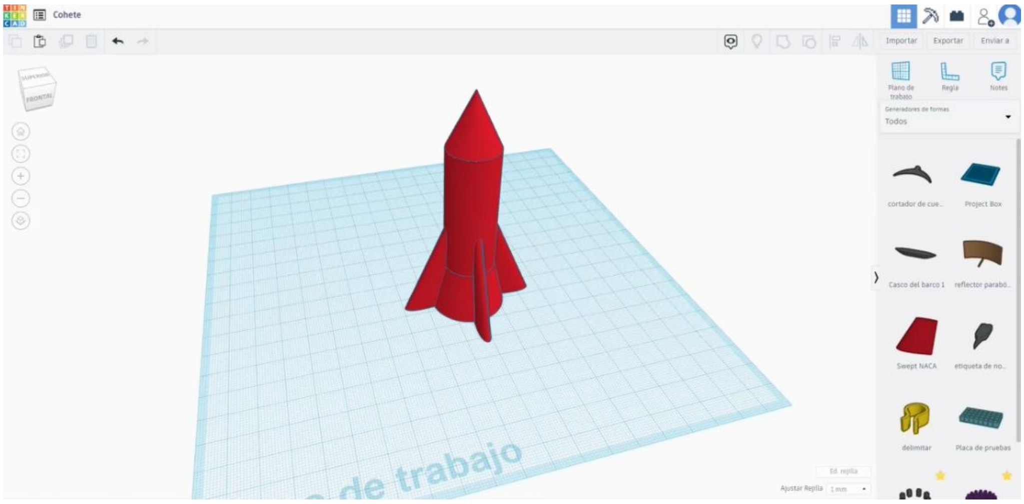
7. Select the two collinear Swept NACA, align and group them. Repeat this action with the other two.



8. Align now the main body of the rocket with the 4

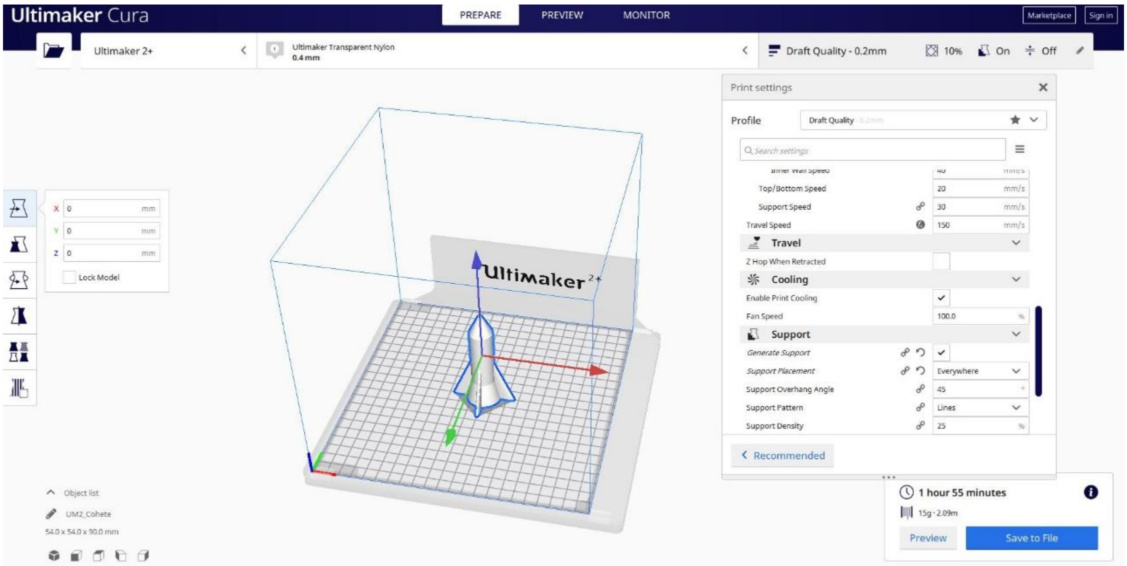
Swept NACA. Merge all the parts pressing “Group” button.



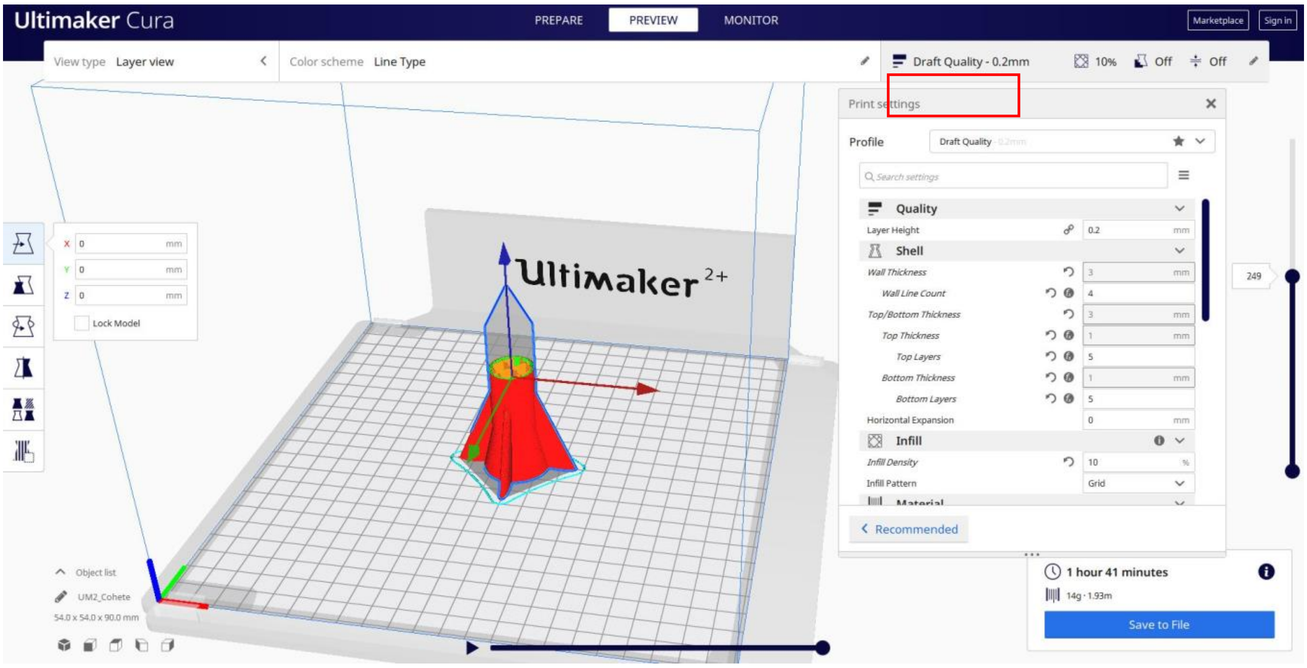


9.3.25.2 *Rocket 3D printing seetings*

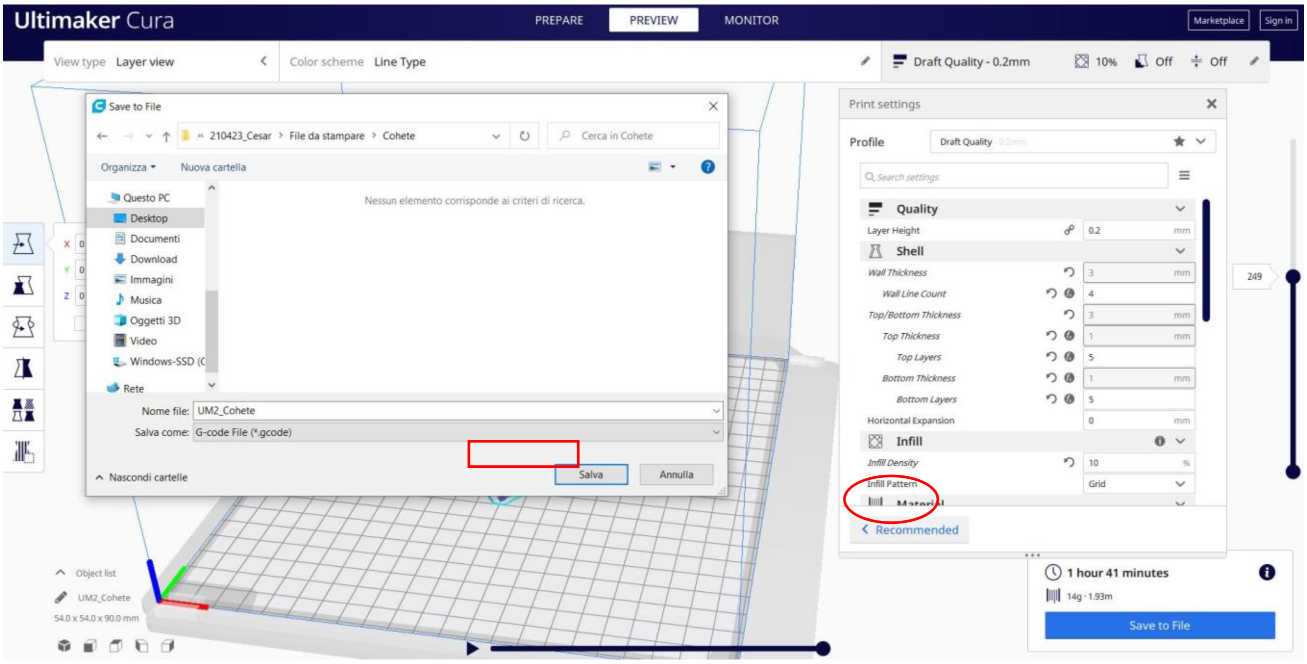
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



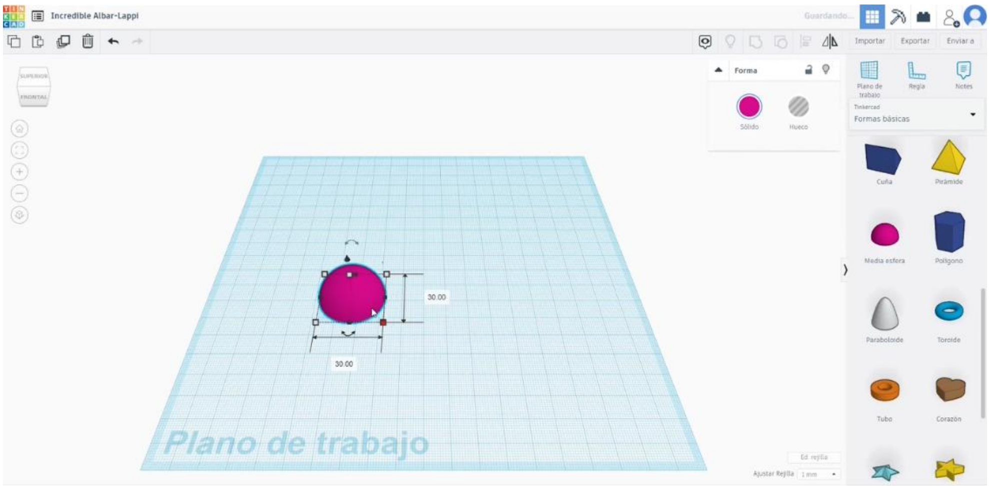
3. At this point I can save the ".Gcode" file to send to the machine.



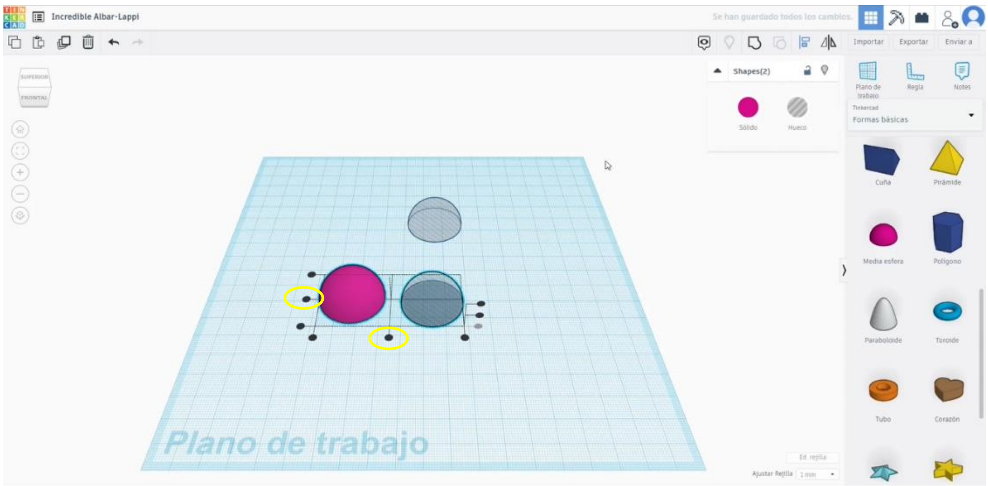
9.3.26 Part 26: Strainer

9.3.26.1 Strainer Design

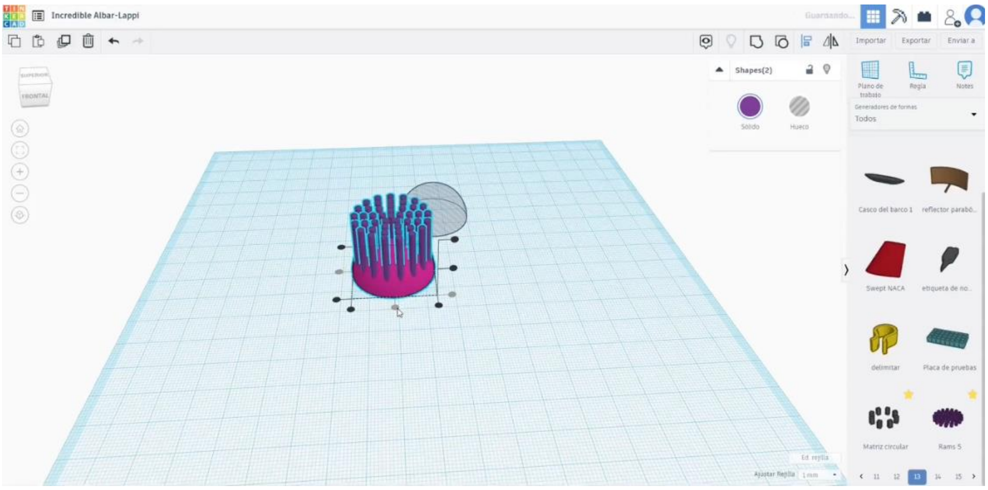
1. Start with half sphere and modify the measurements to 30x30 pressing shift, to keep the proportions.



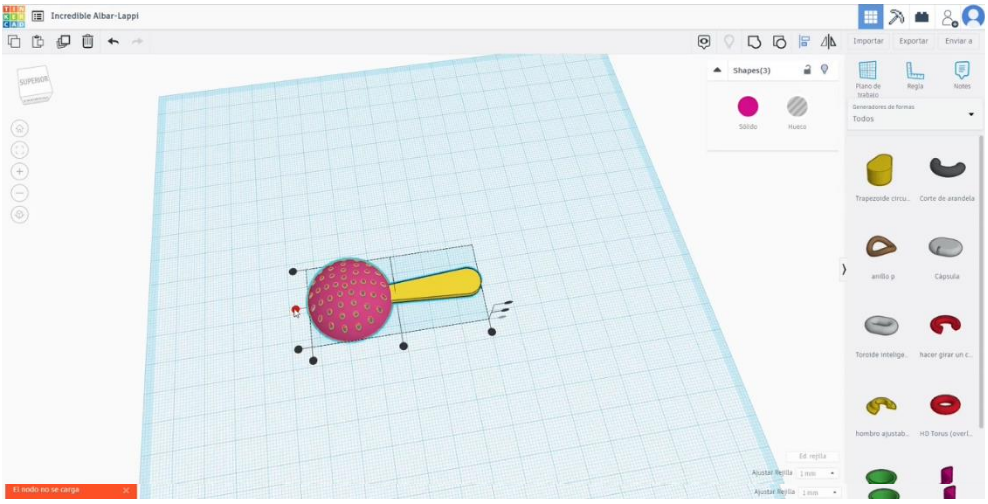
2. Copy this half sphere, pressing Alt button, and scale it to 28x28. Change it to hole mode and made another copy with the same size. Then, align the first half sphere and one of the others. With this two shapes selected, press group button.



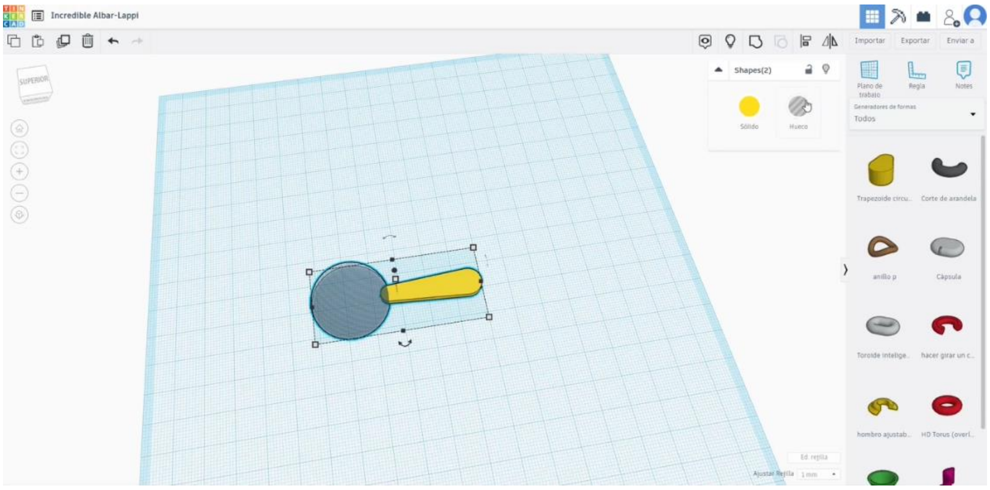
3. Now look for the Rams 5 in the Shapes generators. Give to it a measurement of 27 of diameter and increase its height, then center it with the first half sphere. Convert the Rams 5 into hole mode. Select the rams 5 and the half sphere and press group button.



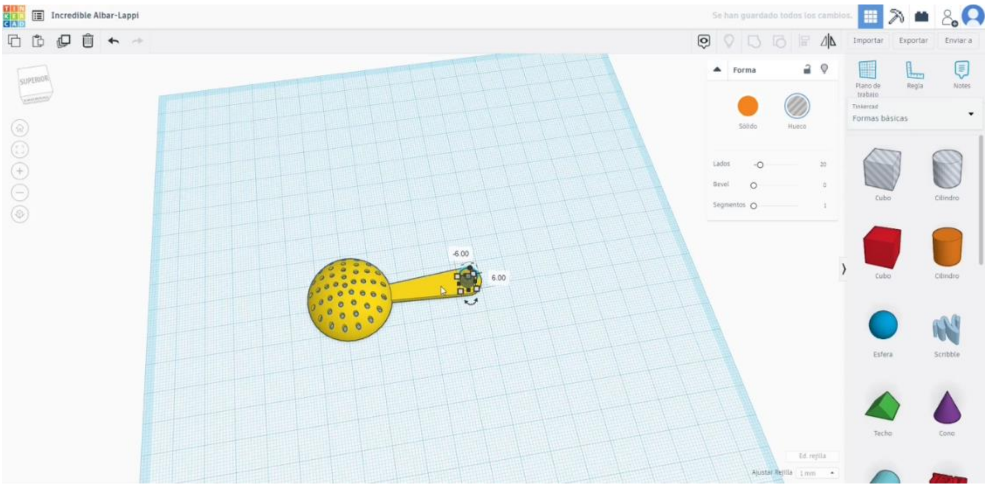
4. Select the new half sphere with holes and the old one that we created. Align them. Afterwards, créate a Circular Trapezoid. Change the measurements to 5 and 3 in diameter and 29 long. Place it next to the half sphere.

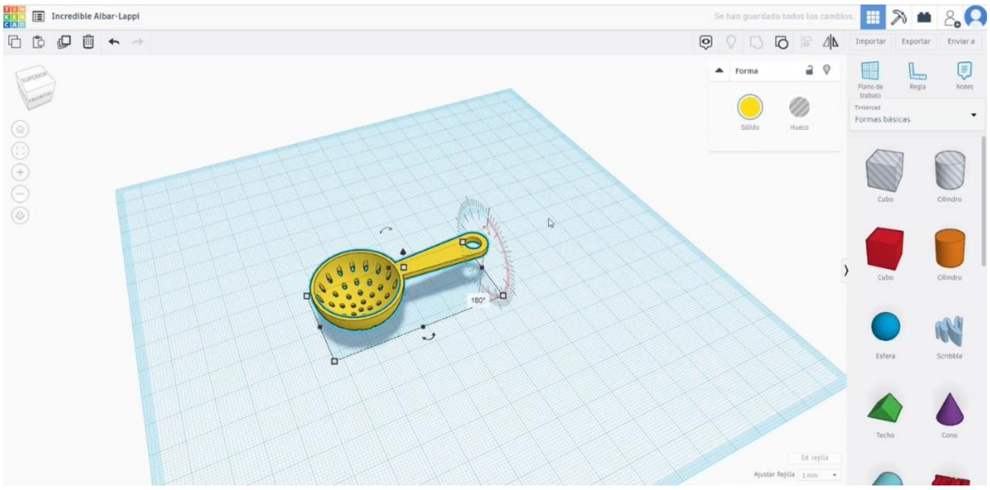


5. Once you have it aligned, press the light bulb button to hide the half sphere above. Then, select the trapezoid and the half sphere in the hole mode and press group button. Afterwards, press the light bulb button above, and the hiden half sphere will appear again.



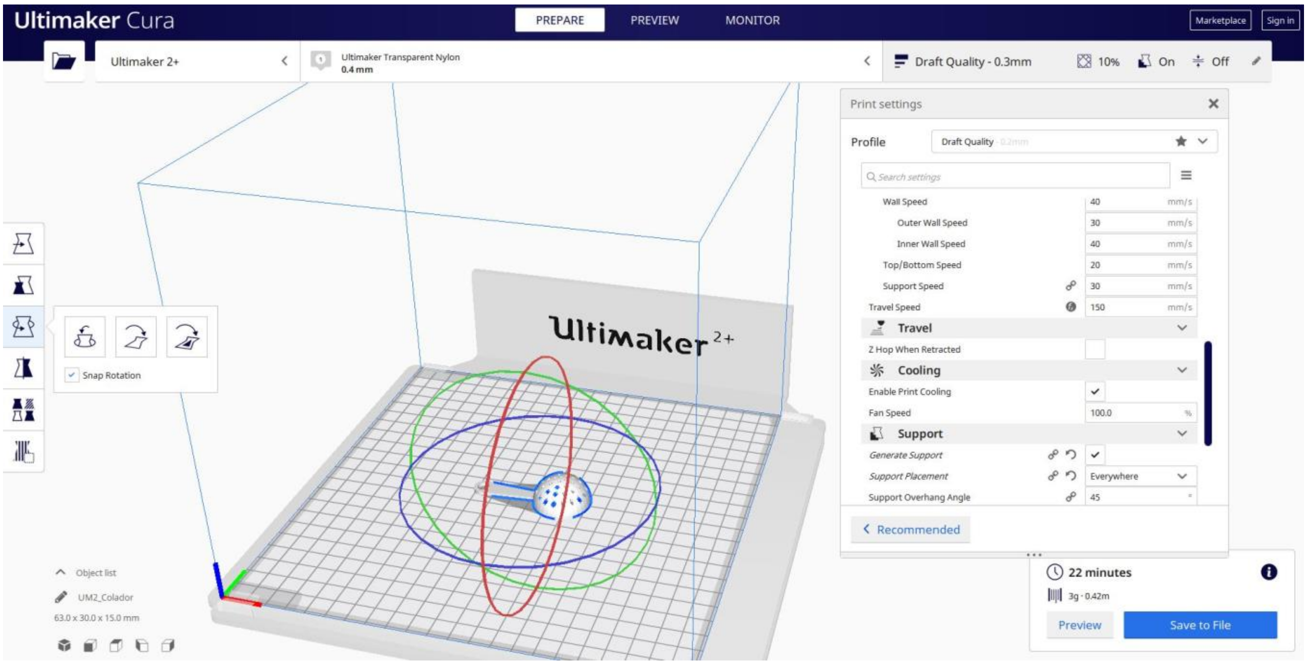
6. Create a hole cylinder. Scale it to 6 x 6 and place it in the end of the handle. Select everything and press group button.



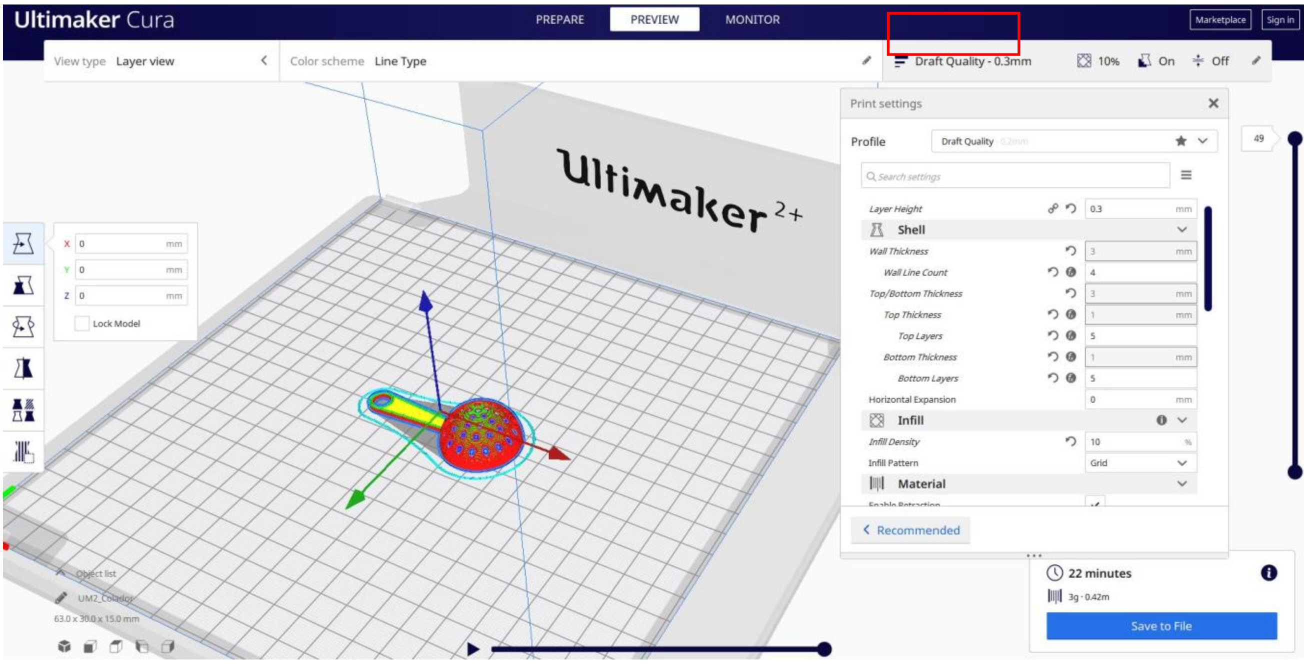


9.3.26.2 Strainer 3D printing seetings

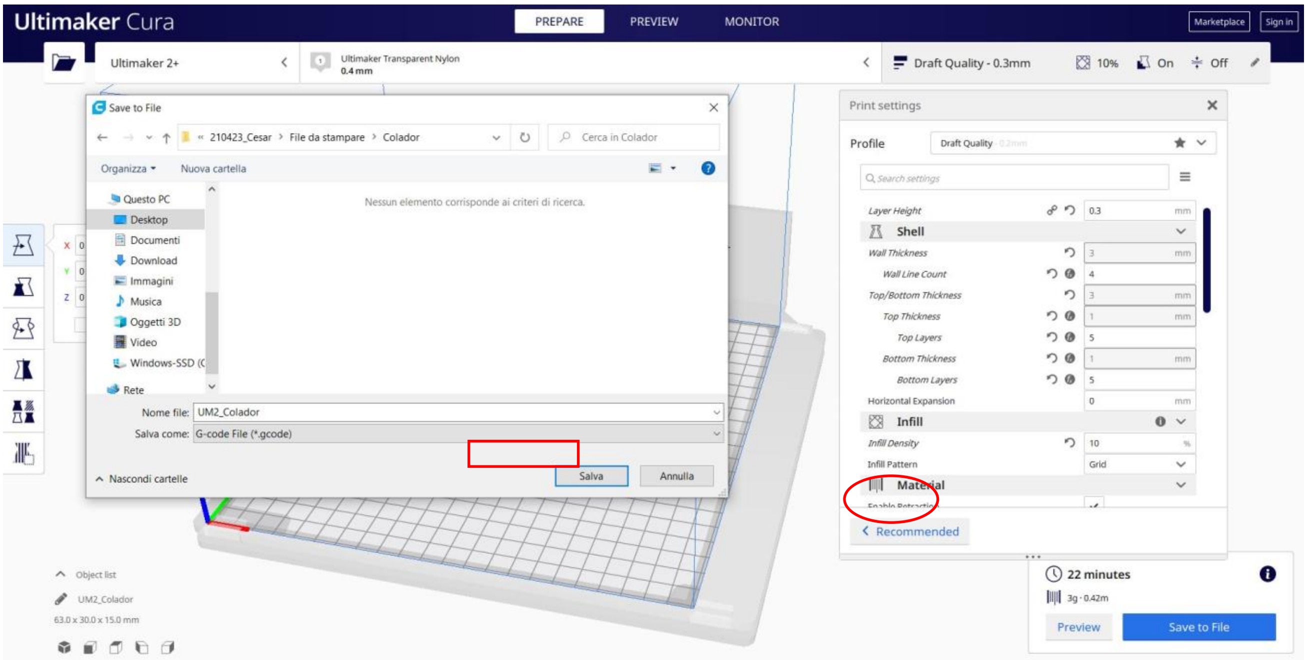
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



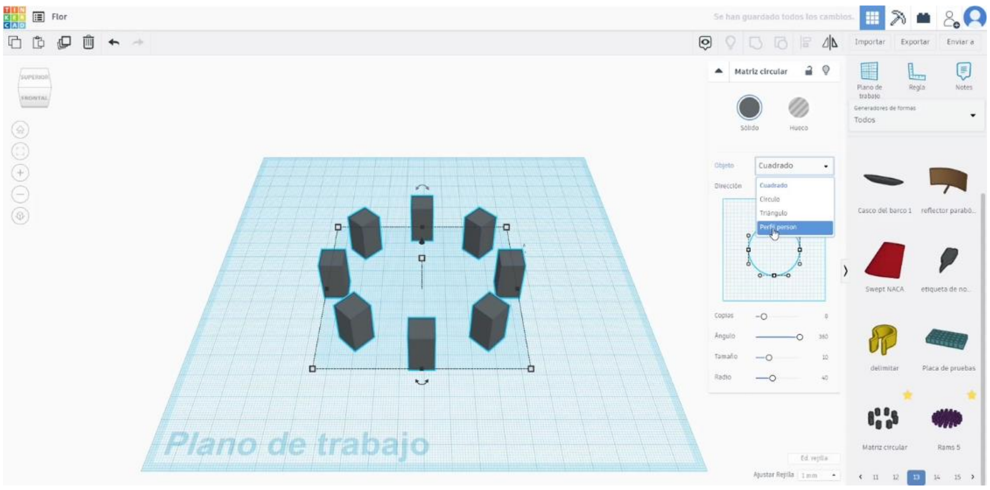
3. At this point I can save the ".Gcode" file to send to the machine.



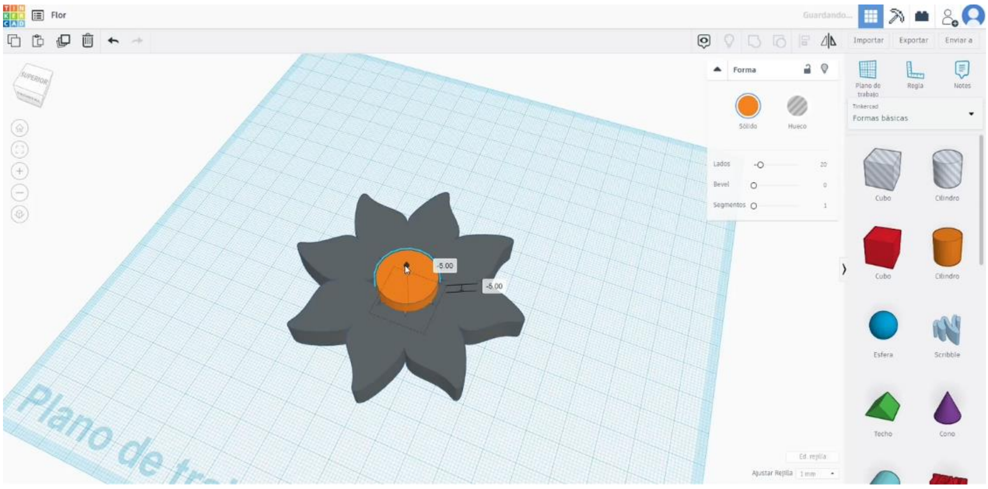
9.3.27 Part 27: Flower

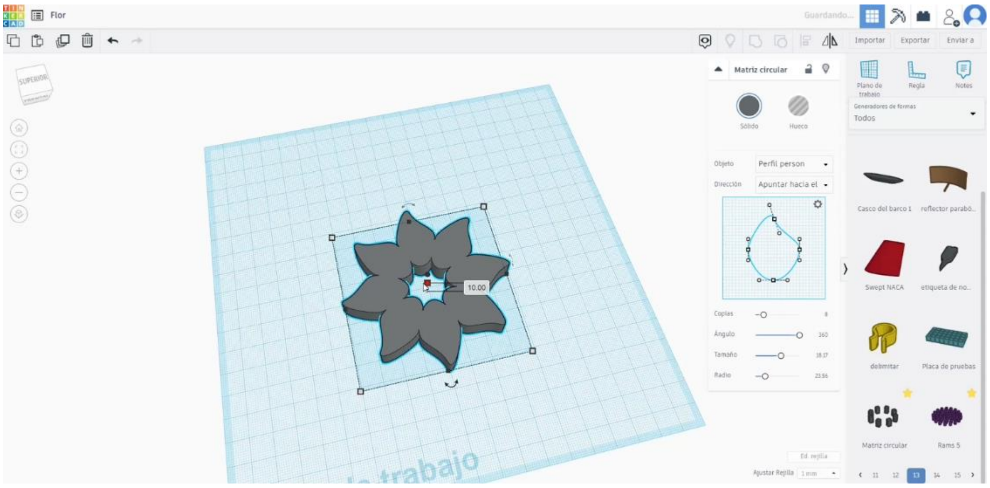
9.3.27.1 Flower Design

1. Build a circular matrix. Select the custom profile mode to draw the shape of the petals. Choose the size, radius, height… that you prefer.

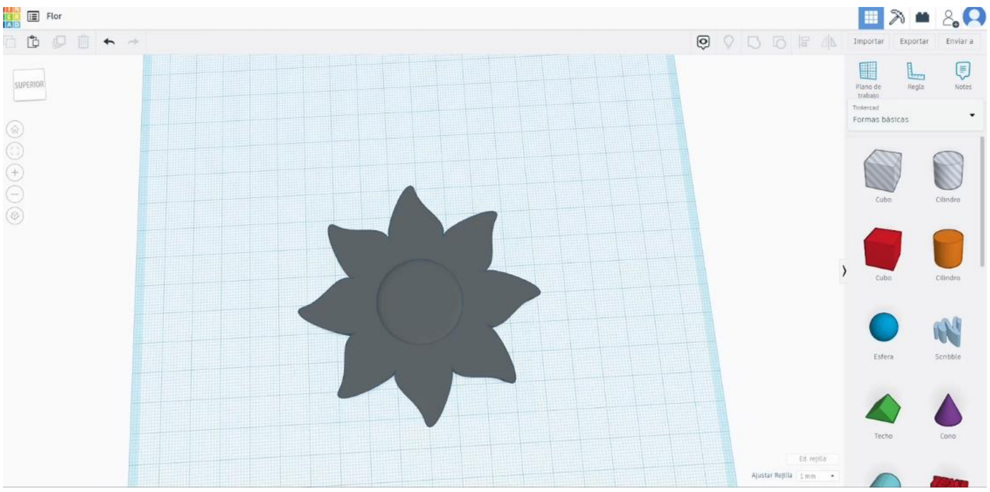


2. Draw a cylinder in the middle. Adapt the size, align it with the petals. Add bevel and segments to see the corner of the cylinder rounded and soft.



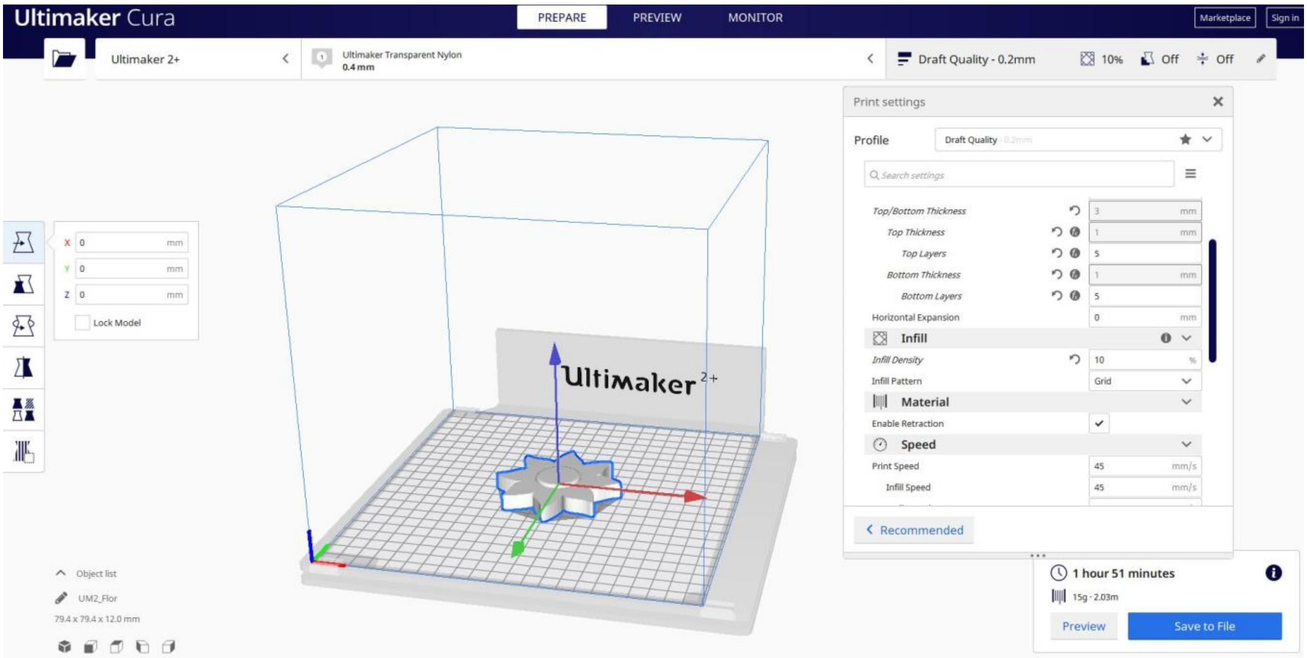


3. Select the cylinder and the petals and join them by pressing group.

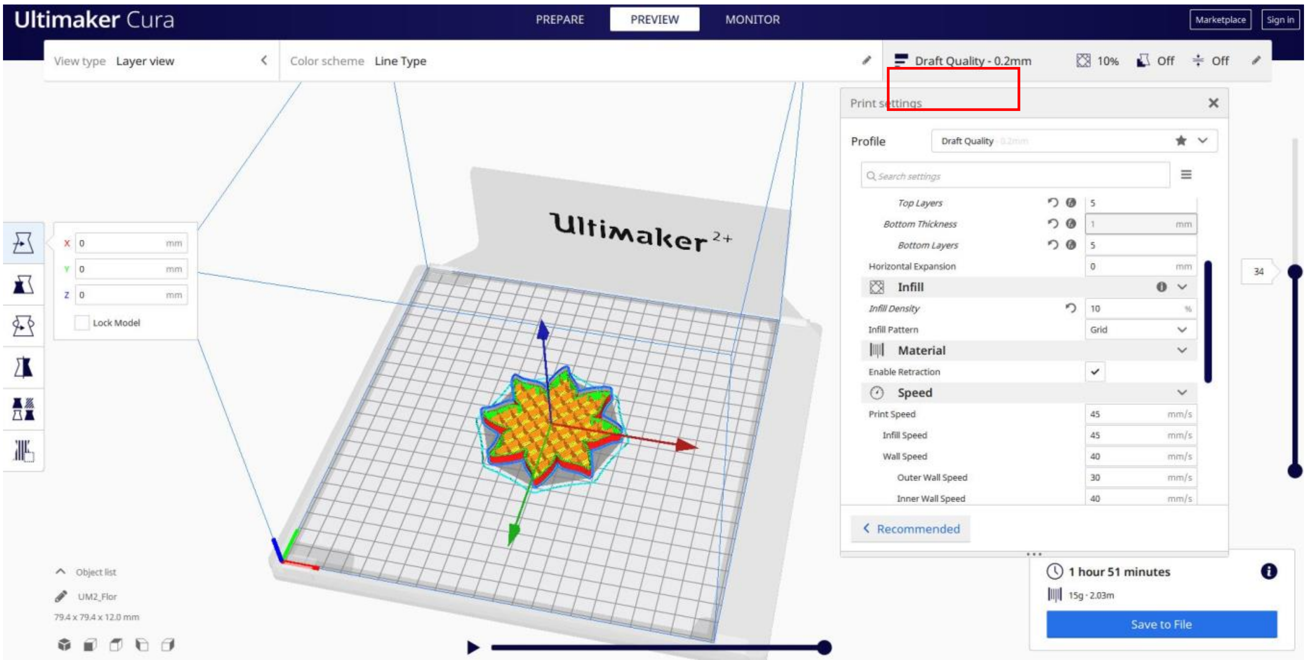


9.3.27.2 Flower 3D printing seetings

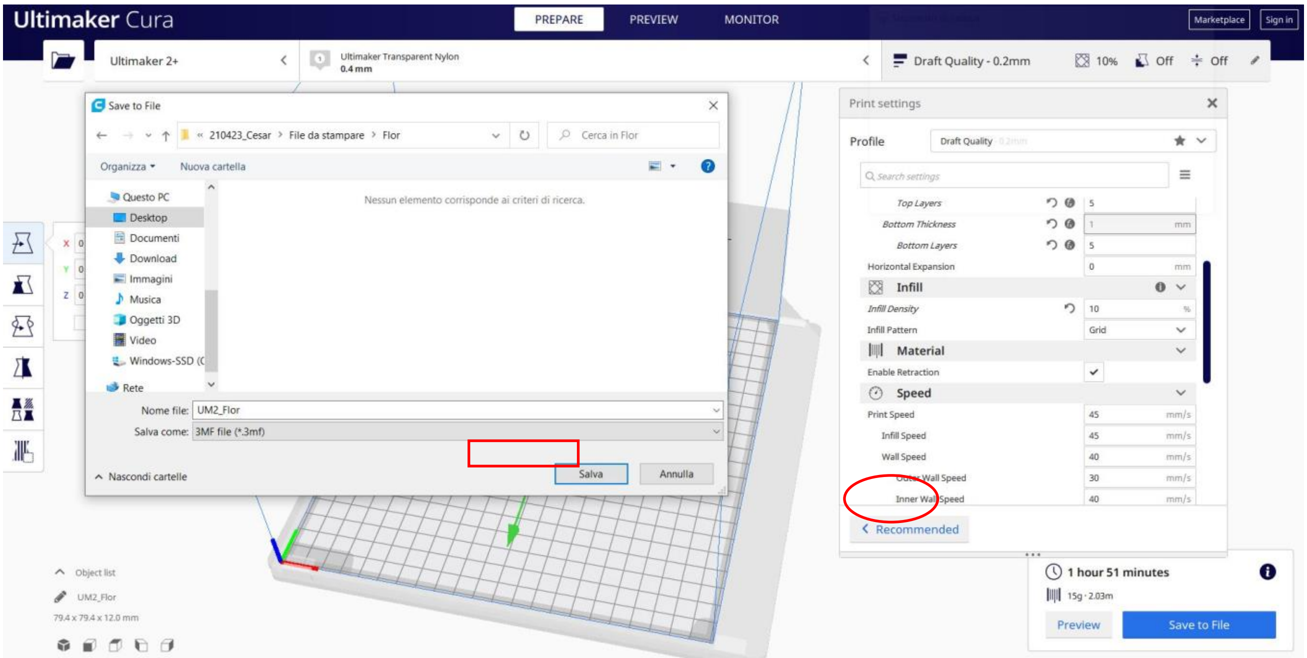
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



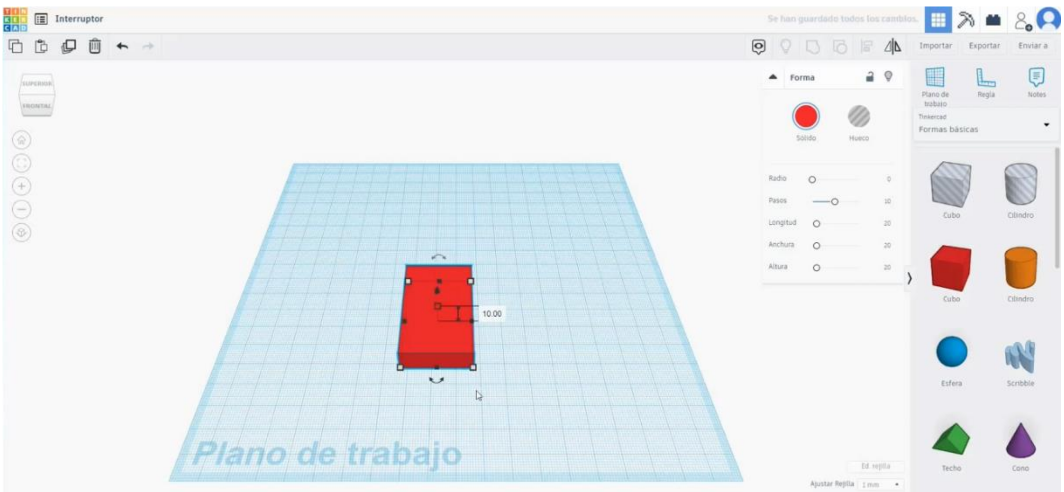
3. At this point I can save the ".Gcode" file to send to the machine.



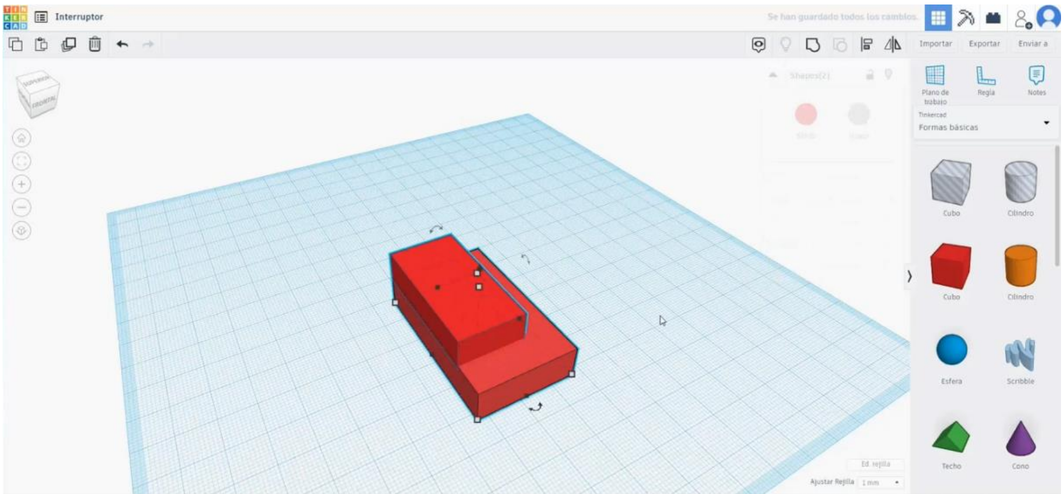
9.3.28 Part 28: Switch

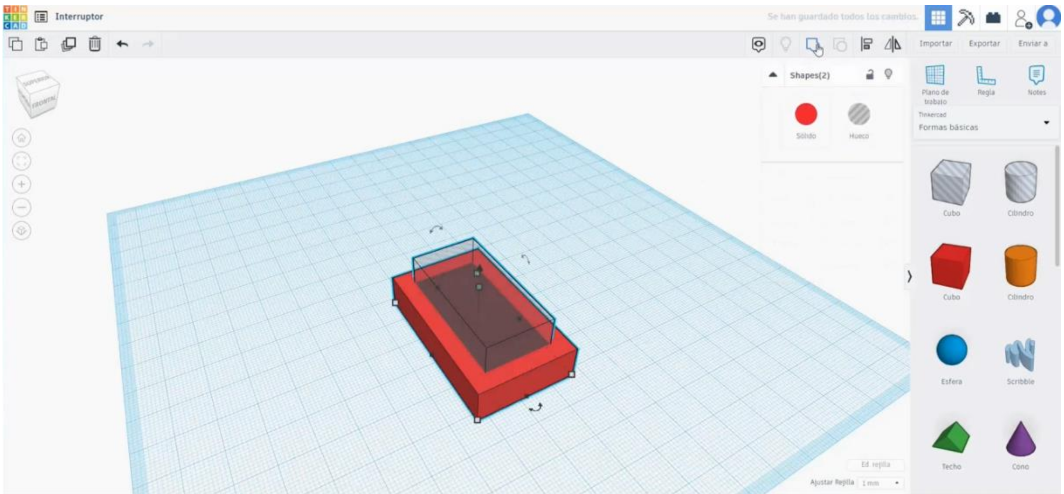
9.3.28.1 Switch Design

1. Start with a cube 50 x 30 x 10.

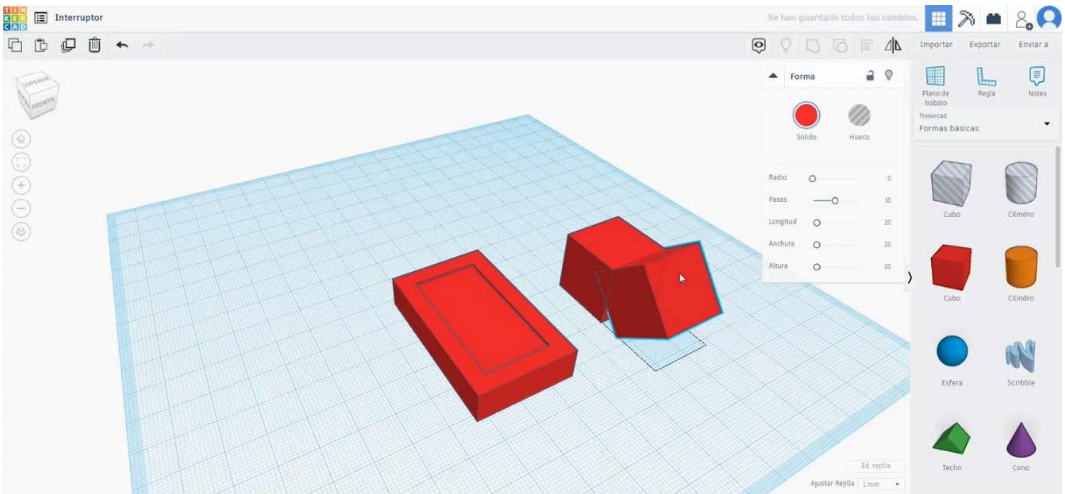


2. Copy de cube, place it to 9 height and change the measurements to 40 x 20 x 10. Align it with the other cube.

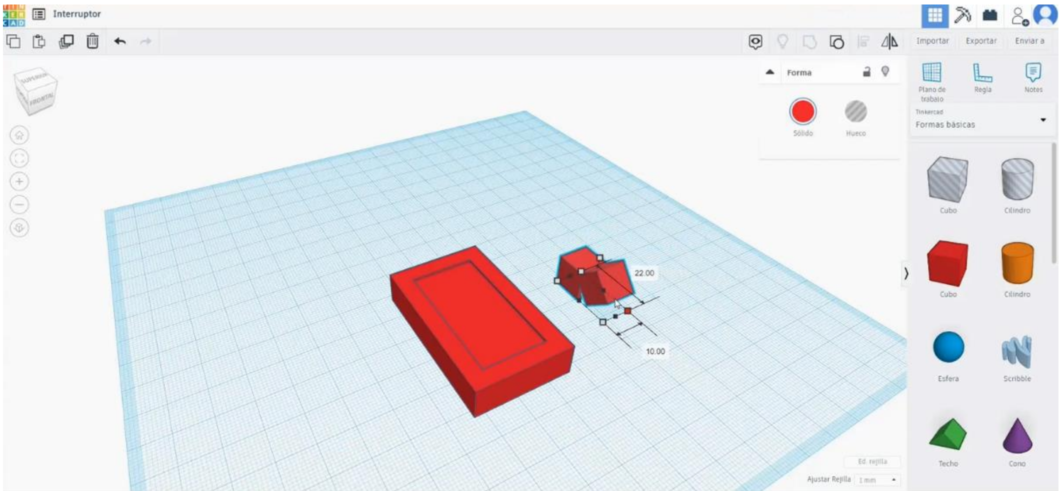


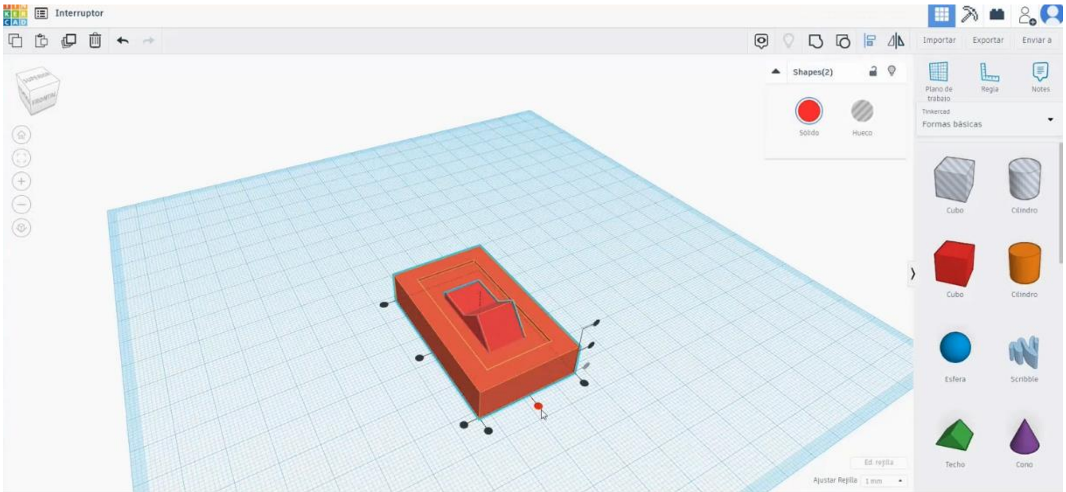


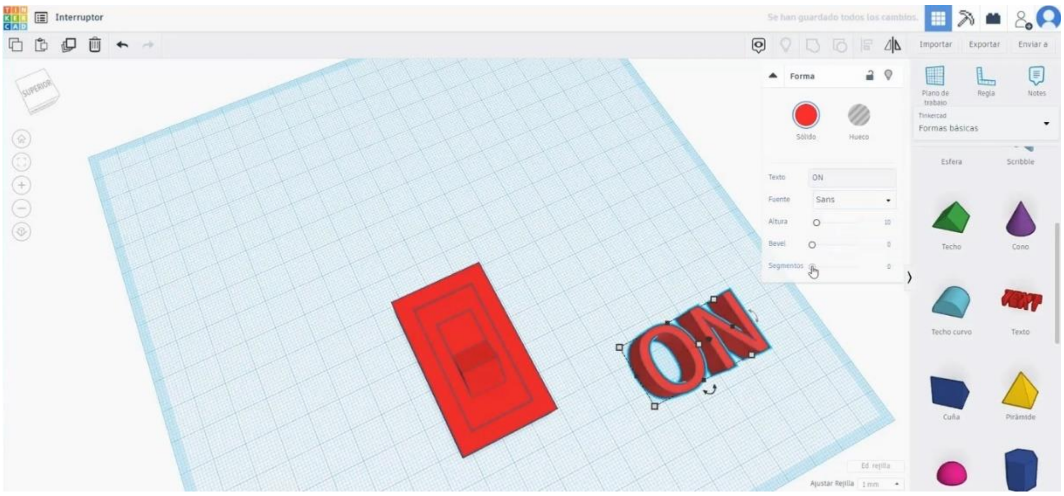
3. Change the top cube to hole mode and group the both figures.
4. Draw two cubes more. Rotate the second one 40 degrees and place it like in the image below. Align and group them.



5. Scale this two grouped cubes and place them in the center of the other figure. Align it all and press Group.

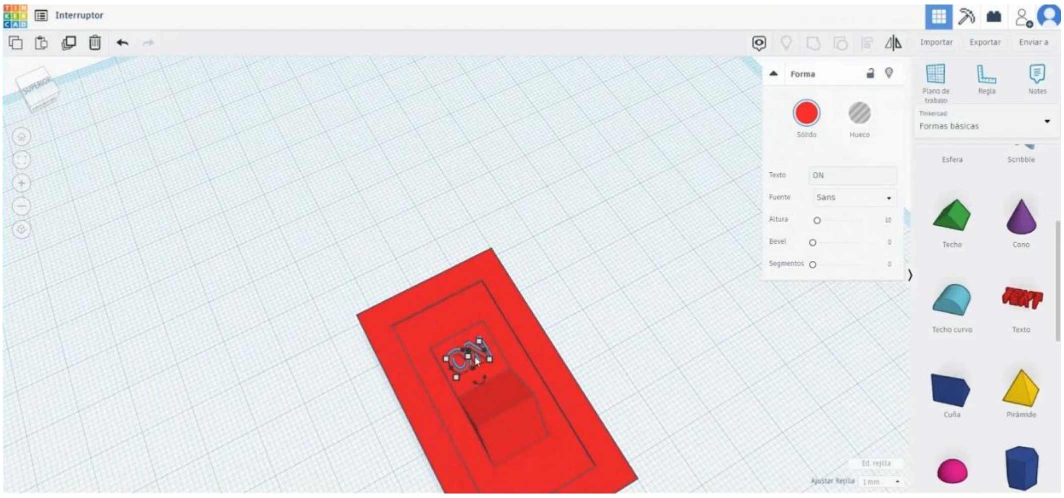




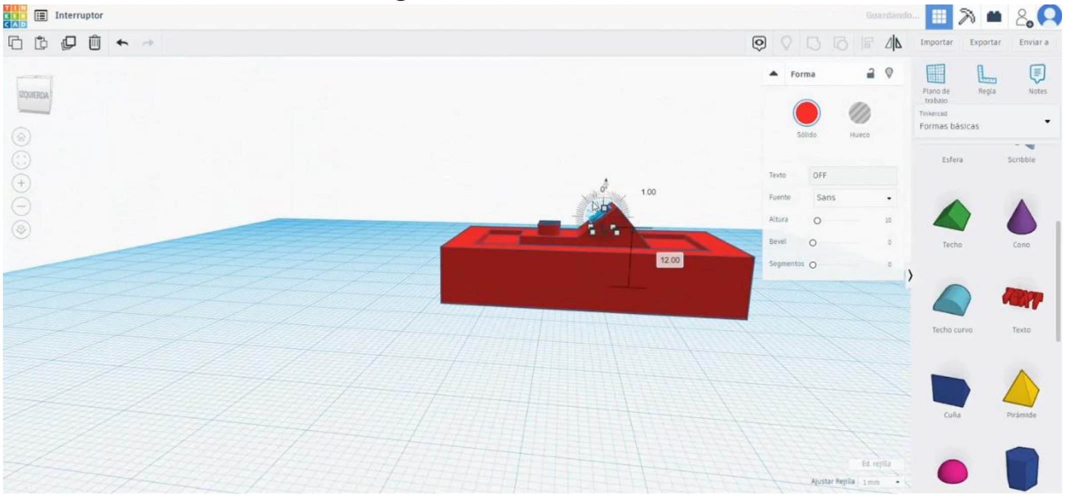


6. Create a Text and write “ON”. Select the Typography and Bevel that you prefer.

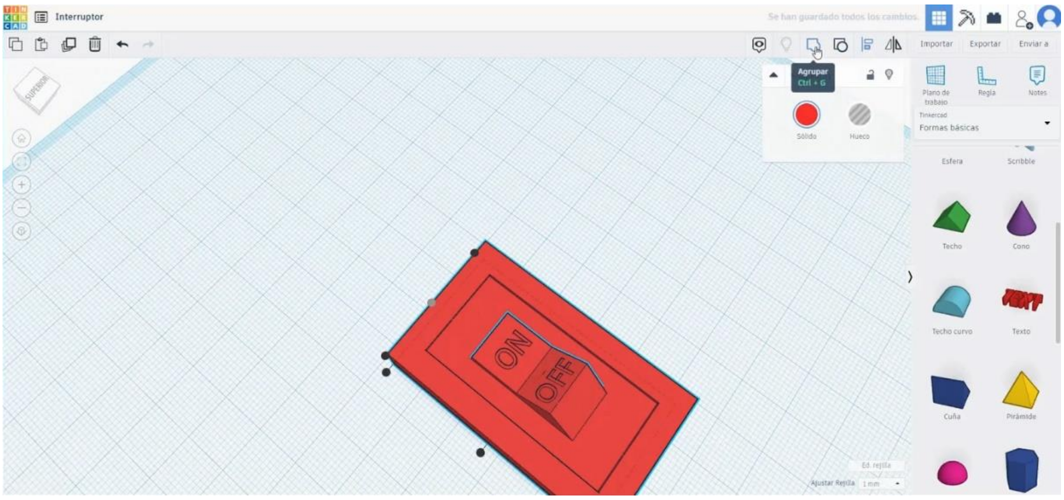
7. Scale it and place it like in the image below. Create a copy and write “OFF”.



8. Turn the “OFF” Text 40 degrees and place it in the inclined face that we made before. Align the text bodies with the other figures.

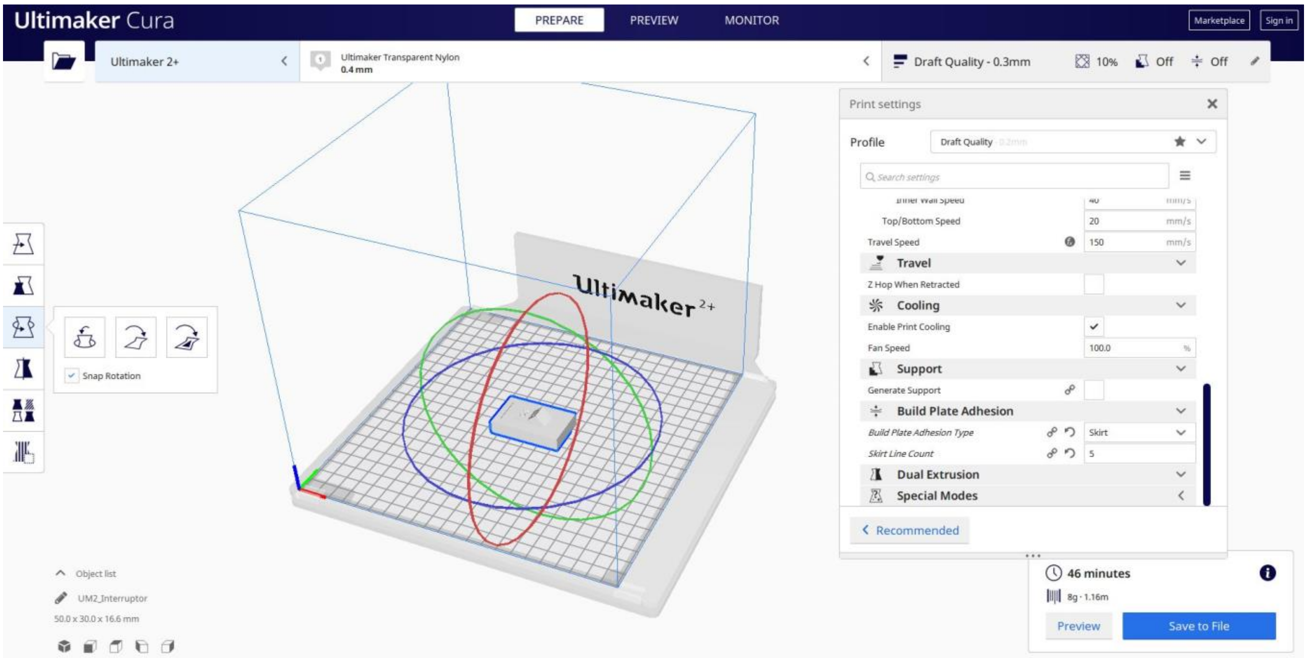


9. Select all the bodies (pressing “shift”) and press Group button.

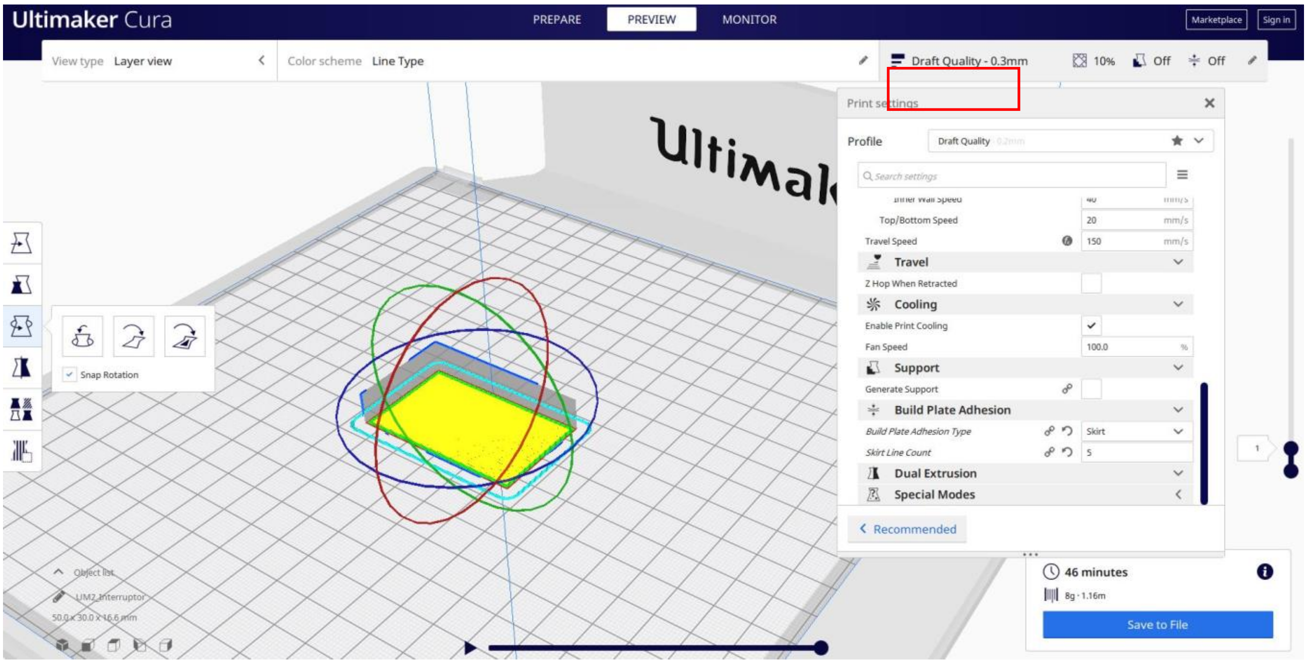


9.3.28.2 Switch 3D printing seetings

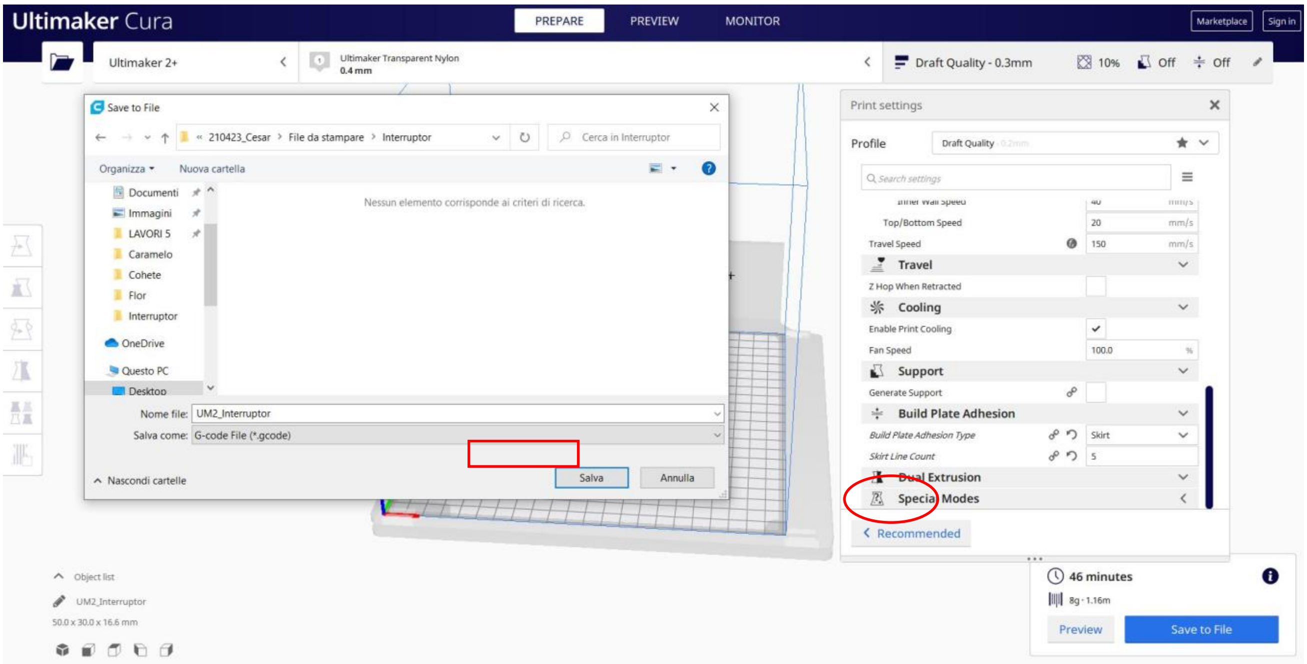
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



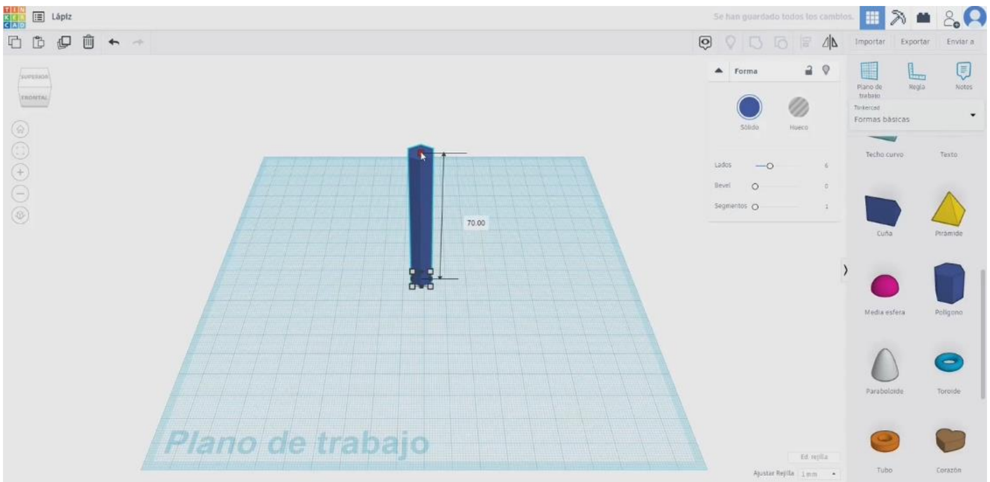
3. At this point I can save the ".Gcode" file to send to the machine.



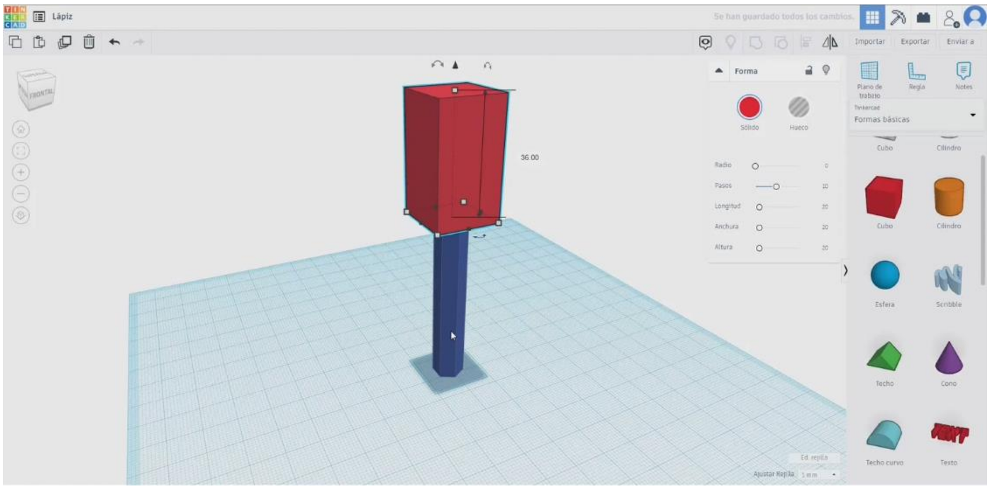
9.3.29 Part 29: Pencil

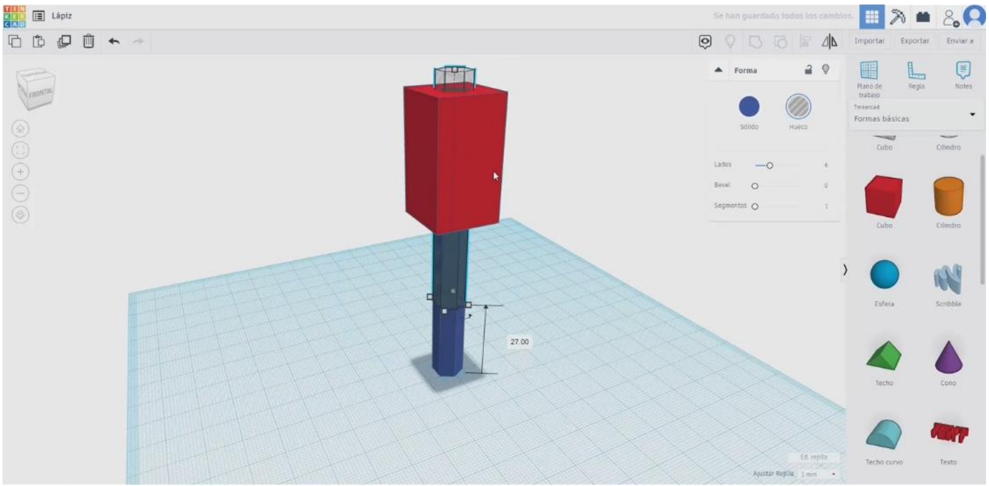
9.3.29.1 Pencil Design

1. Build a 6 sided polygon. Change the measurements to 10 x 8.66 x 70. Make a copy of this one and turn it into hole mode.

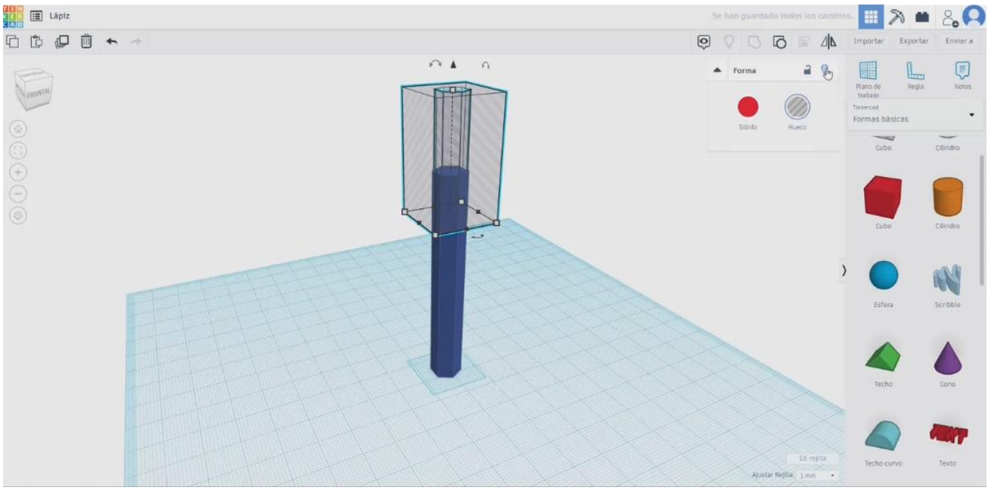


2. Create a cube wider than the polygon and place it above it. Elevate the hole mode polygon to place it like in the image below, and group them.



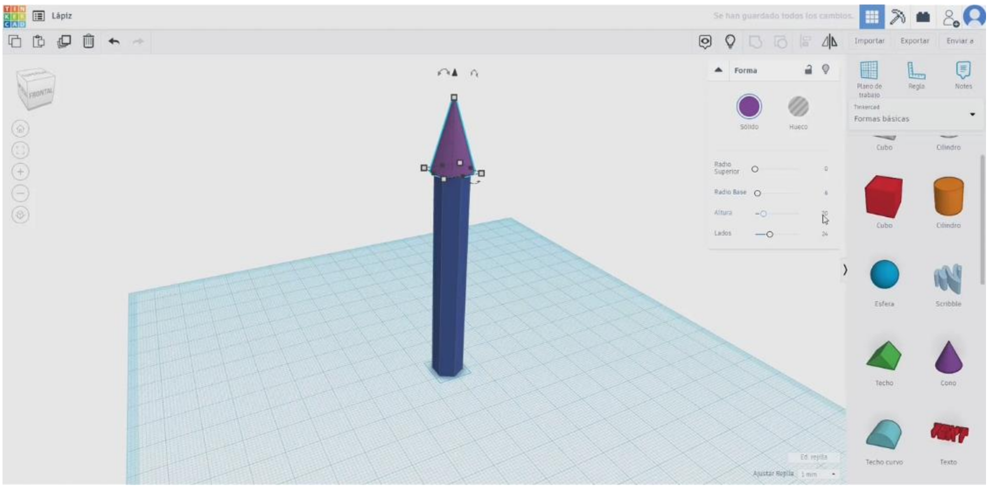


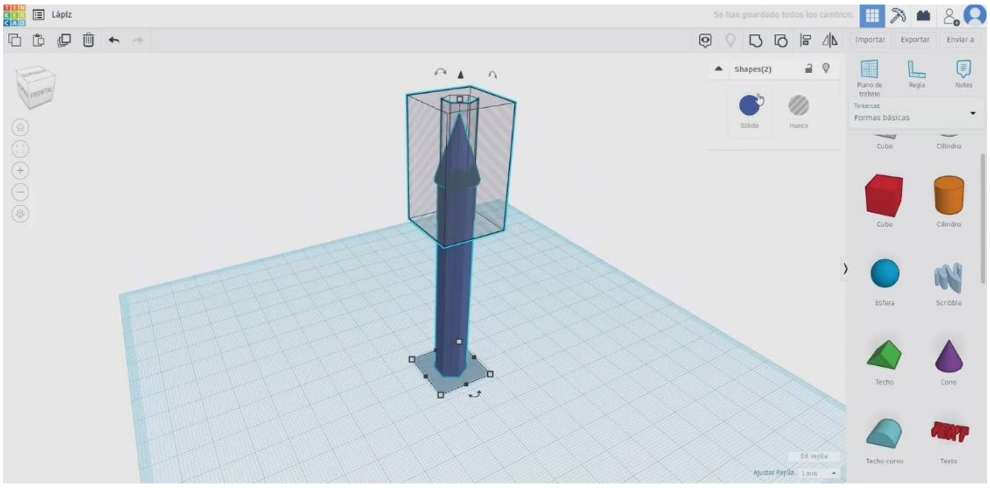
3. Turn the shape into hole mode. And press the light bulb button to hide it.



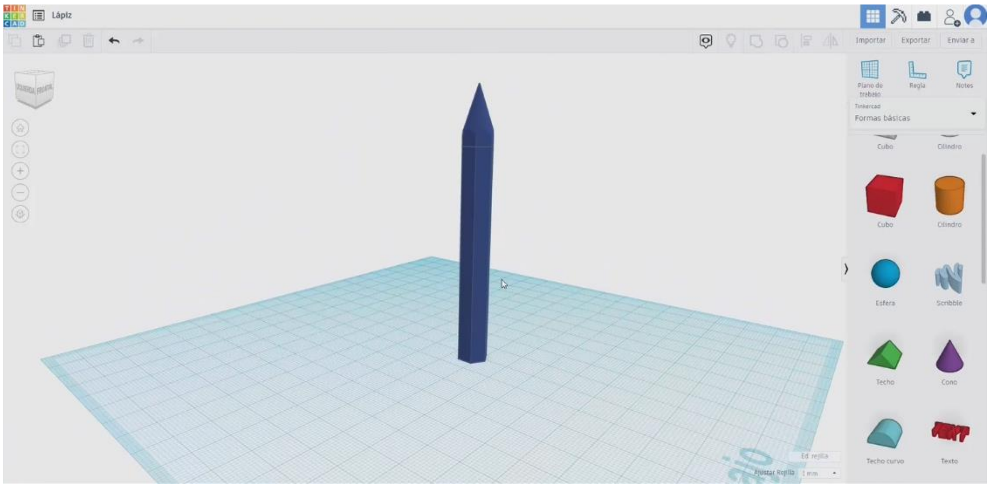
Co-funded by the Erasmus+ Programme of the European Union

4. Draw a cone, place it above the polygon. Change the base radius to 6 and the height to 18. Align it with the polygon. Merge the polygon and the cone. Now press the light bulb button above to show the cube we have hidden before.



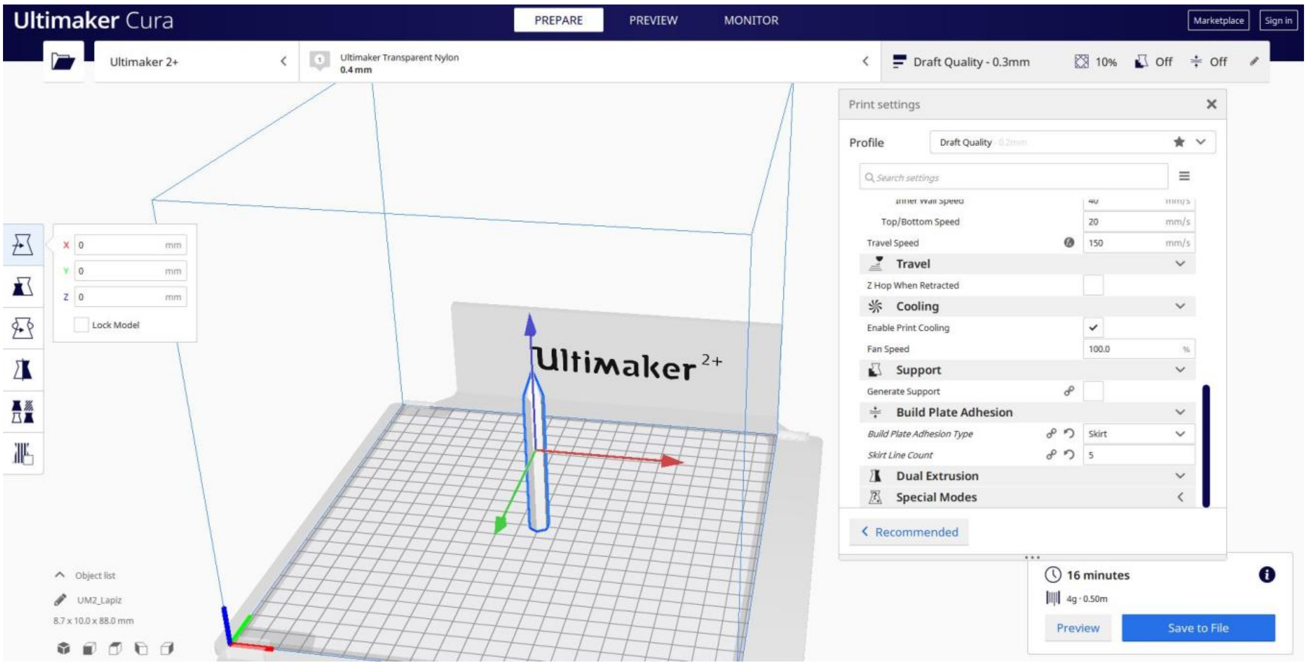


5. Select both bodies and press group button.

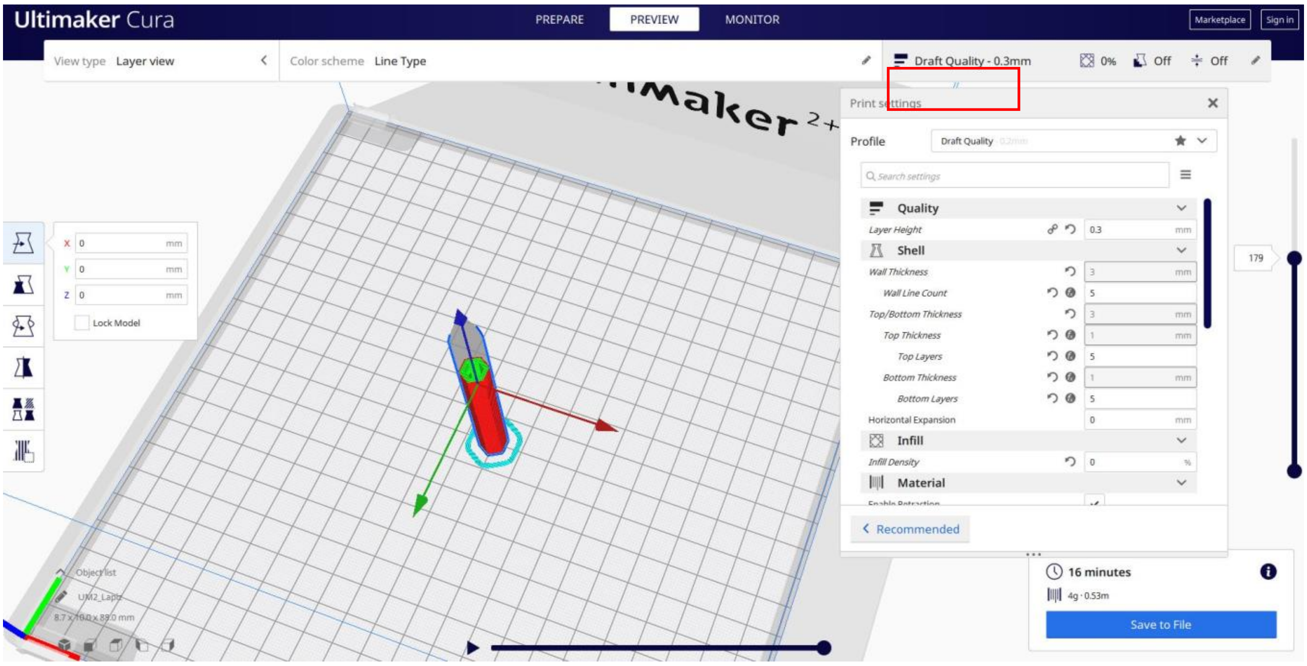


9.3.29.2 Pencil 3D printing seetings

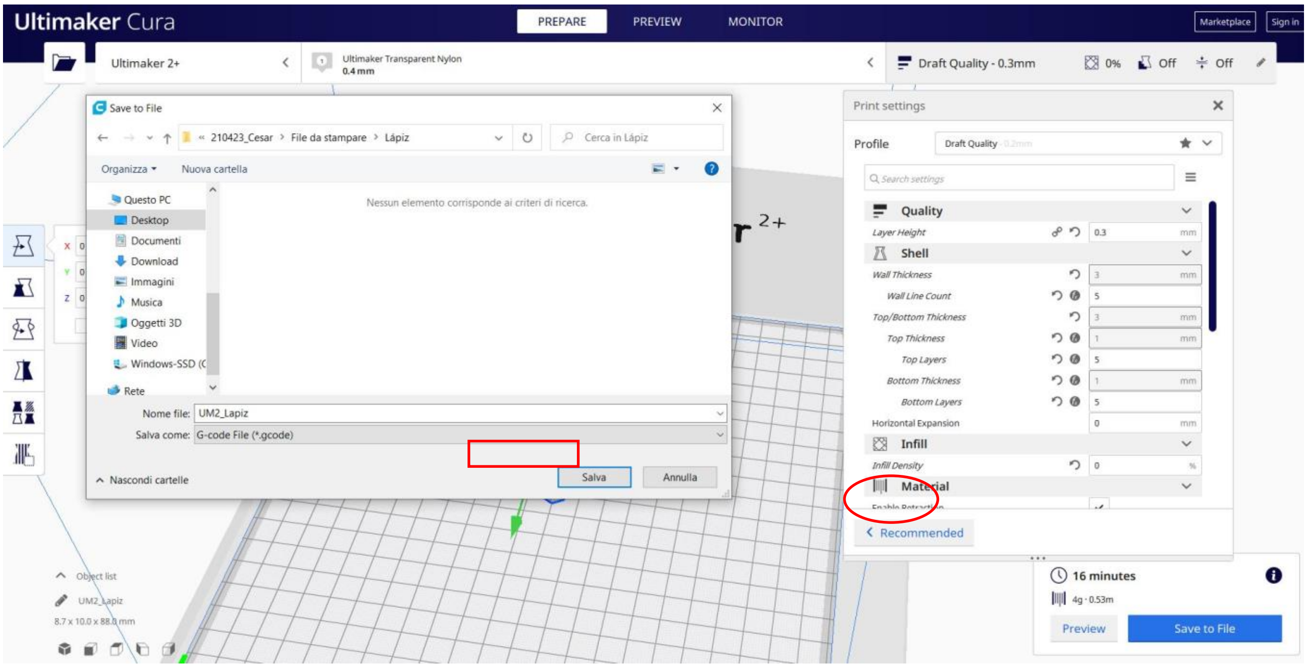
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



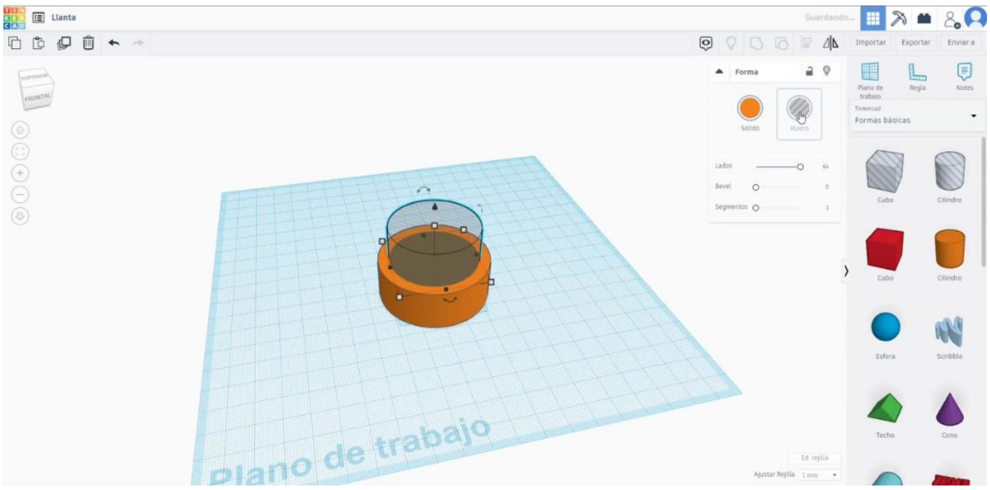
3. At this point I can save the ".Gcode" file to send to the machine.



9.3.30 Part 30: Rim

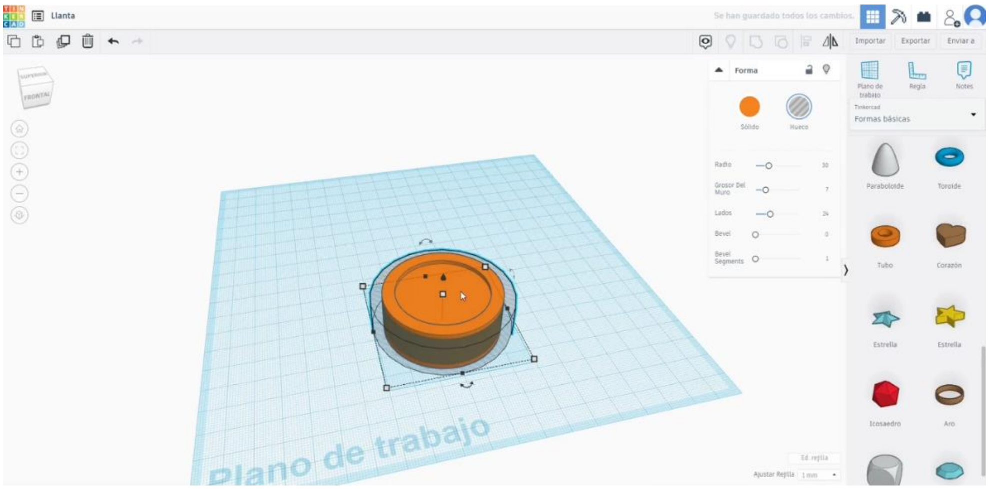
9.3.30.1 Rim Design

1. Build a 50x50x30 cylinder. Copy it and modify the second cylinder to 40 x 40 and overlap a Little bit the first one at the top. Select the second one and press the hole button.

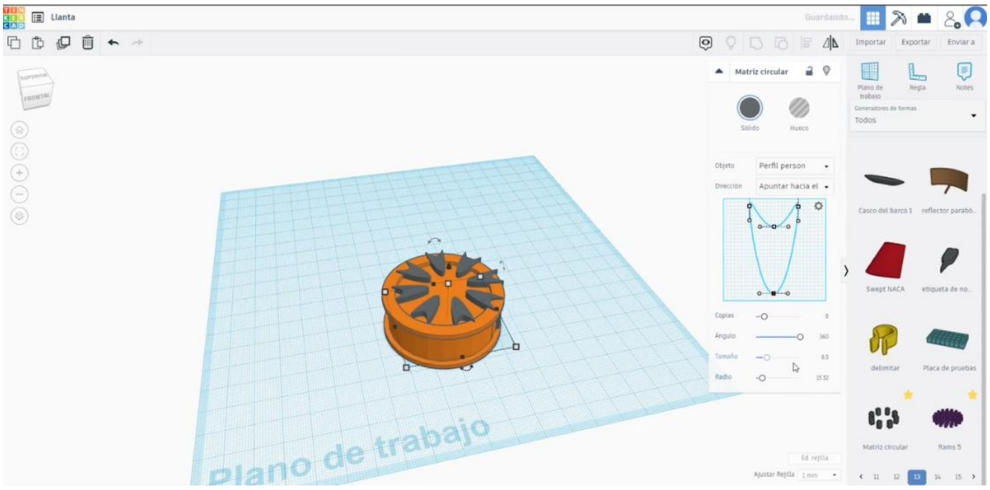


Co-funded by the Erasmus+ Programme of the European Union

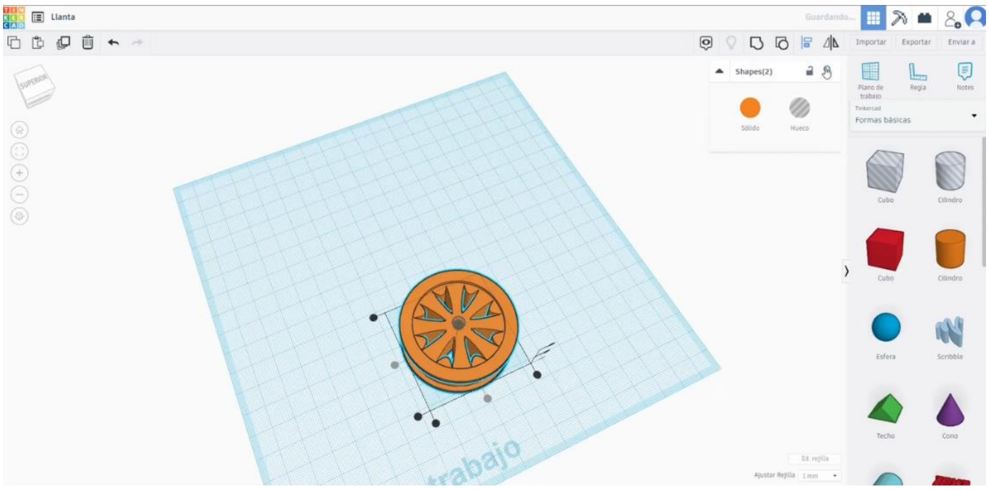
2. Create a tube, change the radius to 30, thickness 7, height 15. Align both bodies. Change the tube to hole mode. Then select the two bodies and group them.



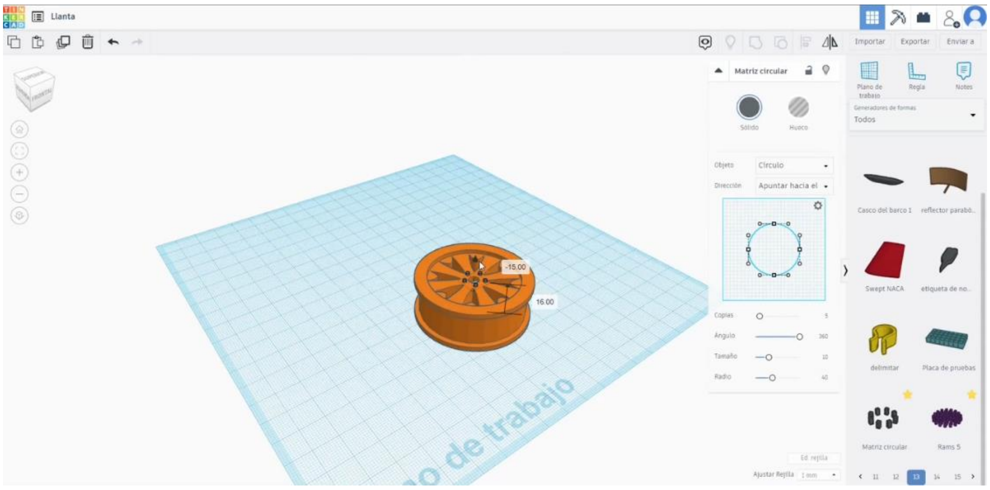
3. Draw a circular matrix. Select the personalized profile mode to draw the shape of the holes in the way you want. Adapt the size, copies, angle… to your design. Chang the matrix to hole mode. Align your design with the cylinder and press group button.



4. Now, draw a hole cylinder and place it in the center of the other body. Group them.

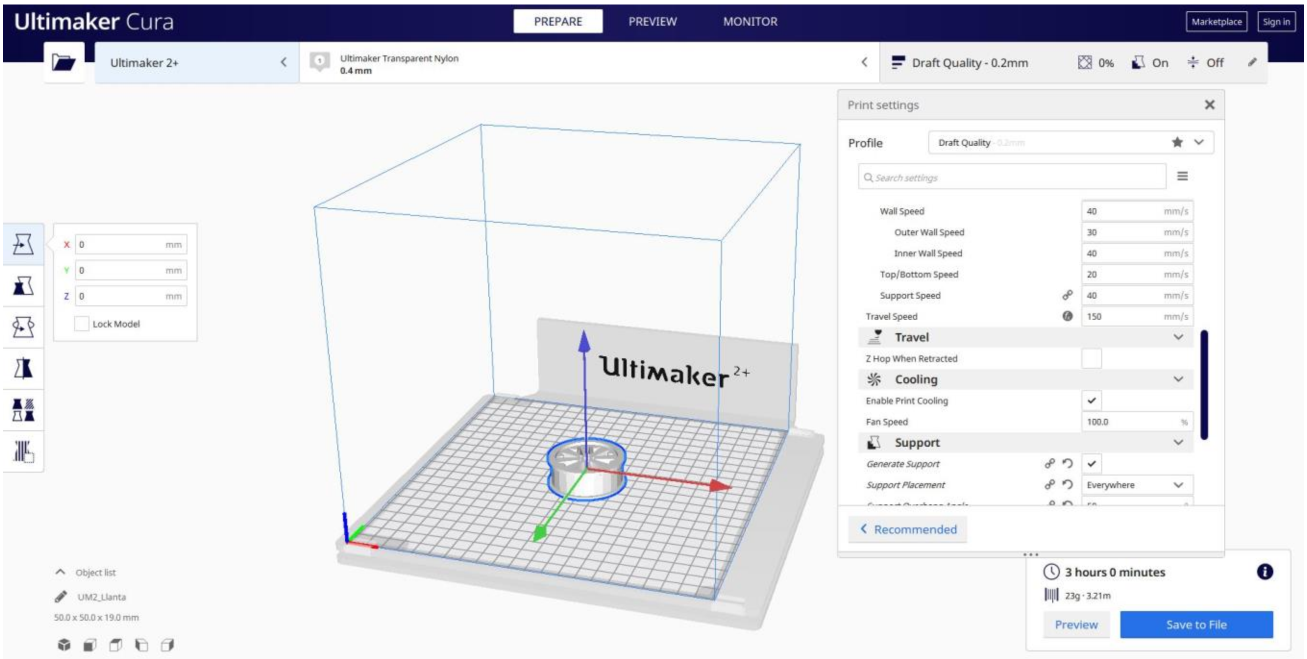


5. Create another circular matrix (5 copies, circles). Make it smaller until it fits in the center of the design, around the central hole. Join it with the other body.

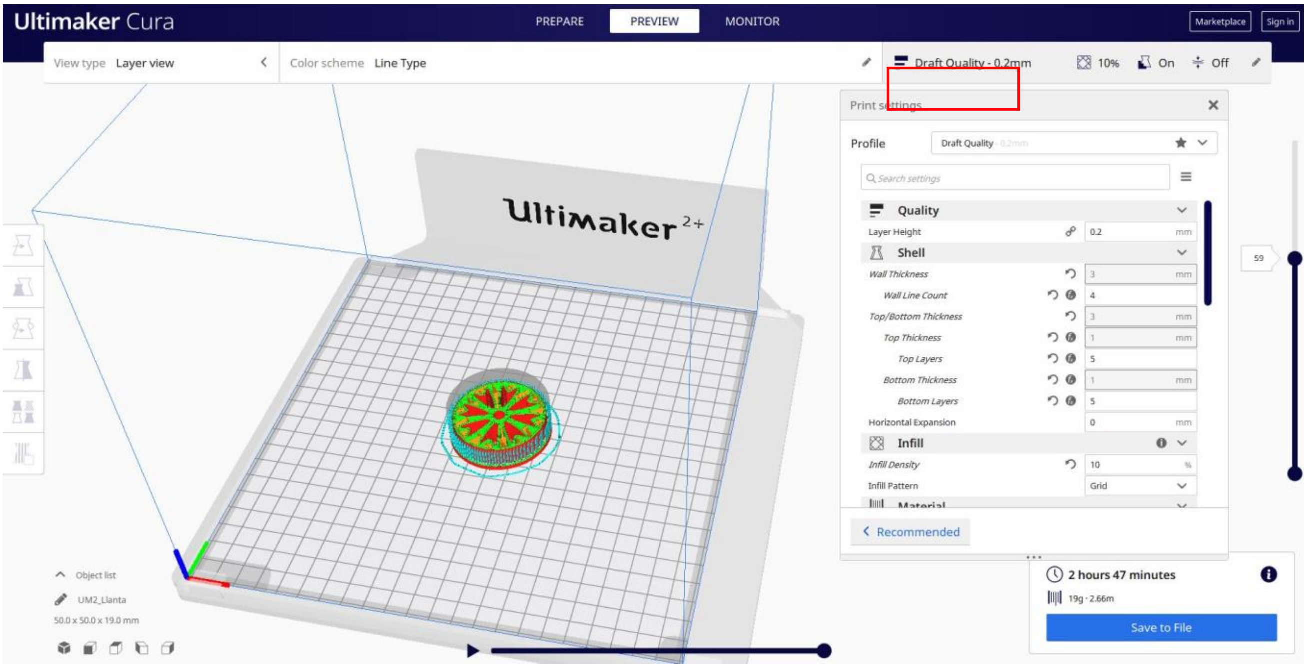


9.3.30.2 Rim 3D printing seetings

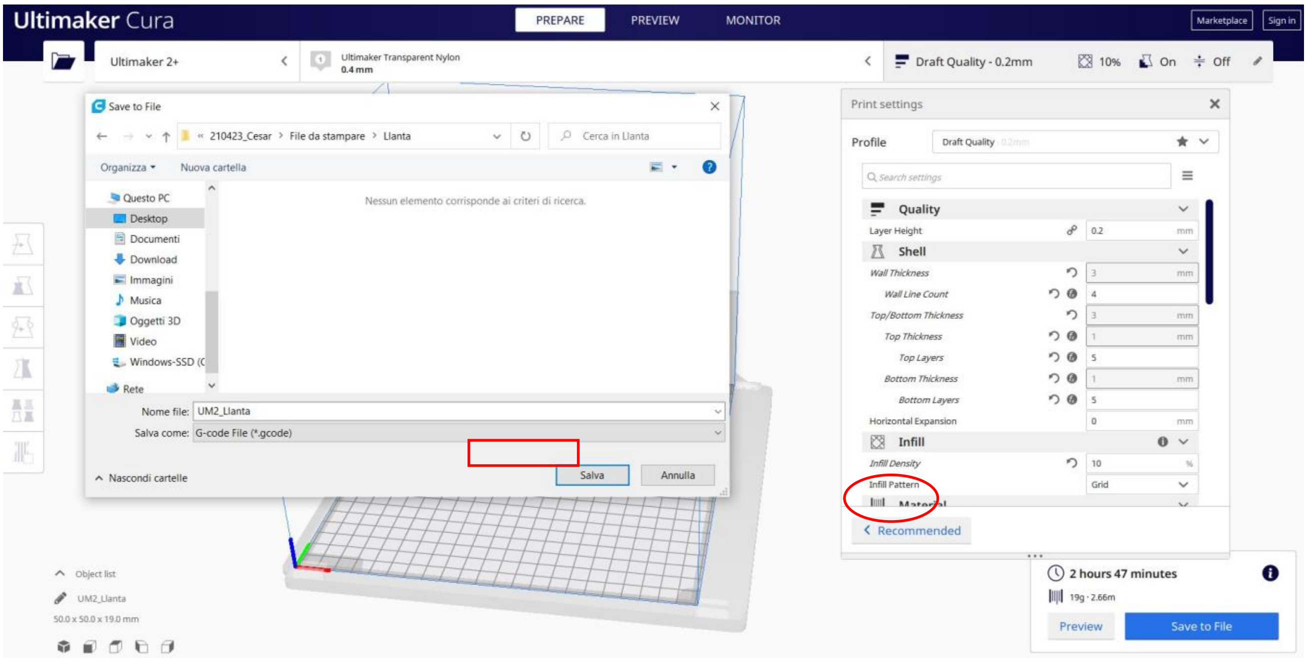
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



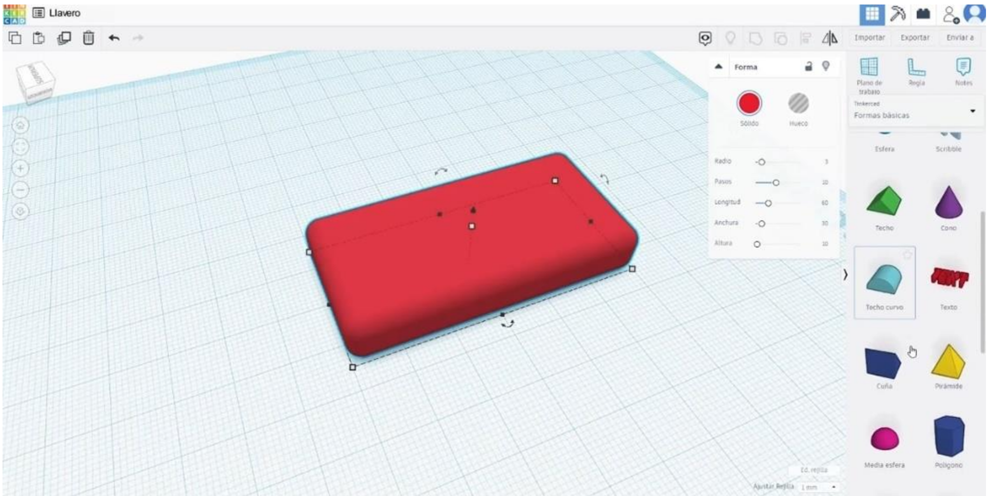
3. At this point I can save the ".Gcode" file to send to the machine.



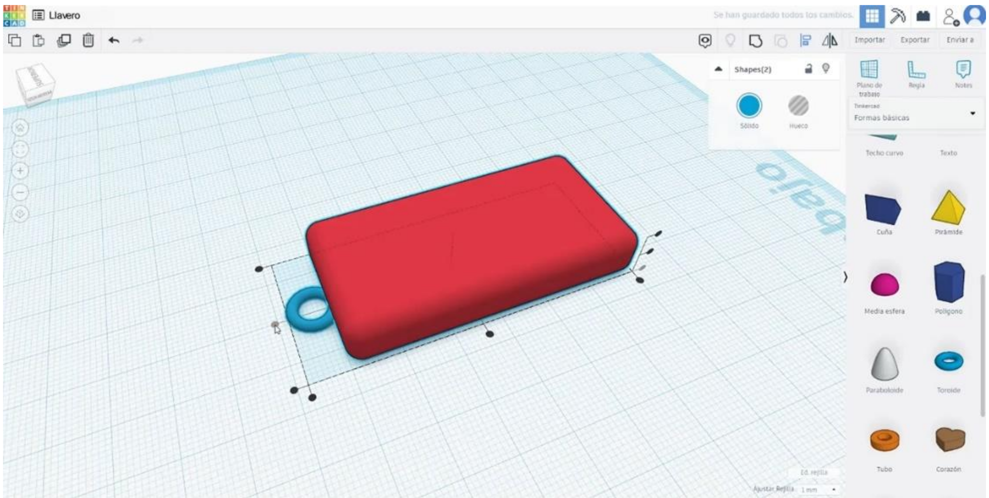
9.3.31 Part 31: Keyring

9.3.31.1 Keyring Design

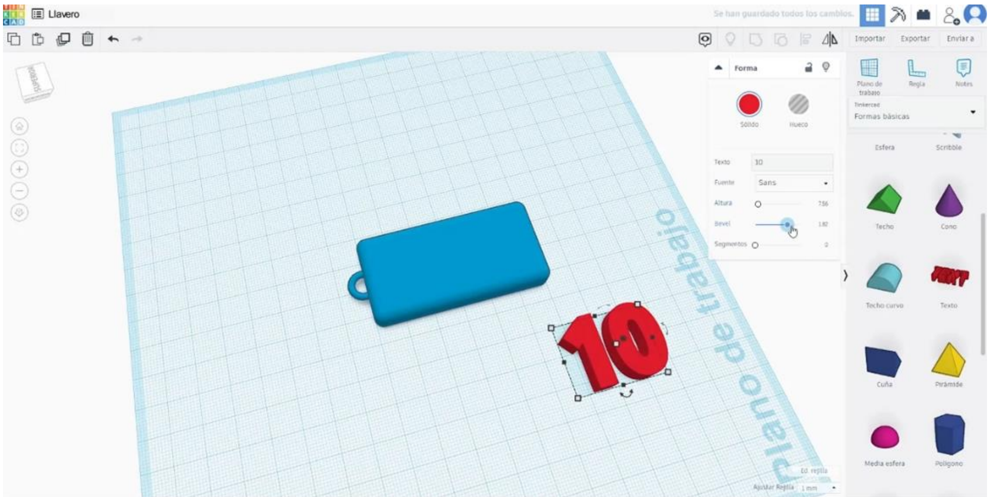
1. Start with a cube. Change the measurements to 60x30x10. Add a radius of 3.



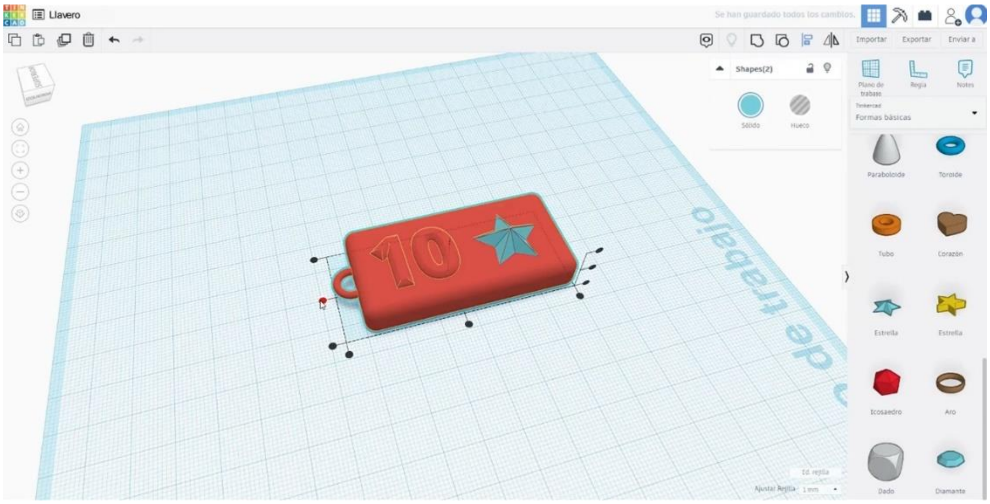
2. Add a torus, change the radius to 4 and the tube to 1. Select two pieces by pressing shift and align them. Group them.

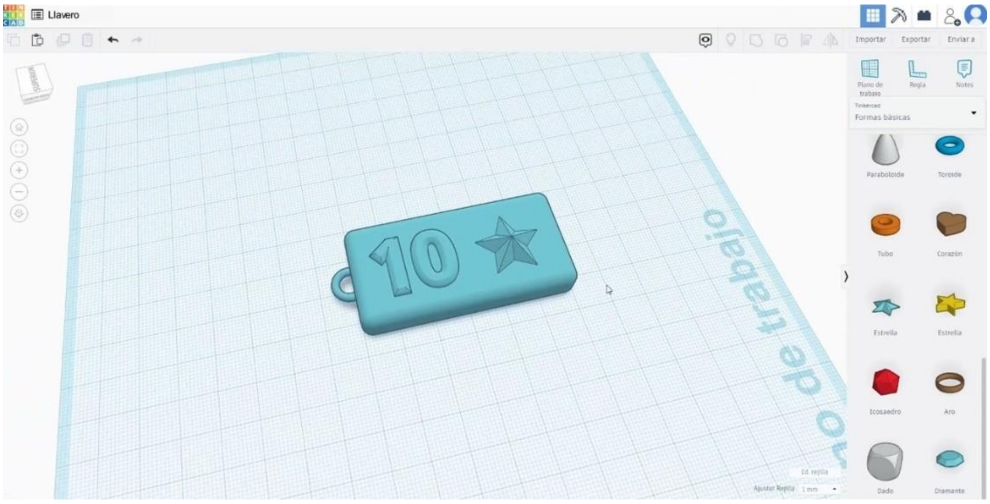


3. Create a text, write a short word or number, we wrote "10", change the typography, bevel… as you prefer. Scale it and place it aligned with the upper side of the cube that we created. Finally, select both bodies and merge them pressing group button.



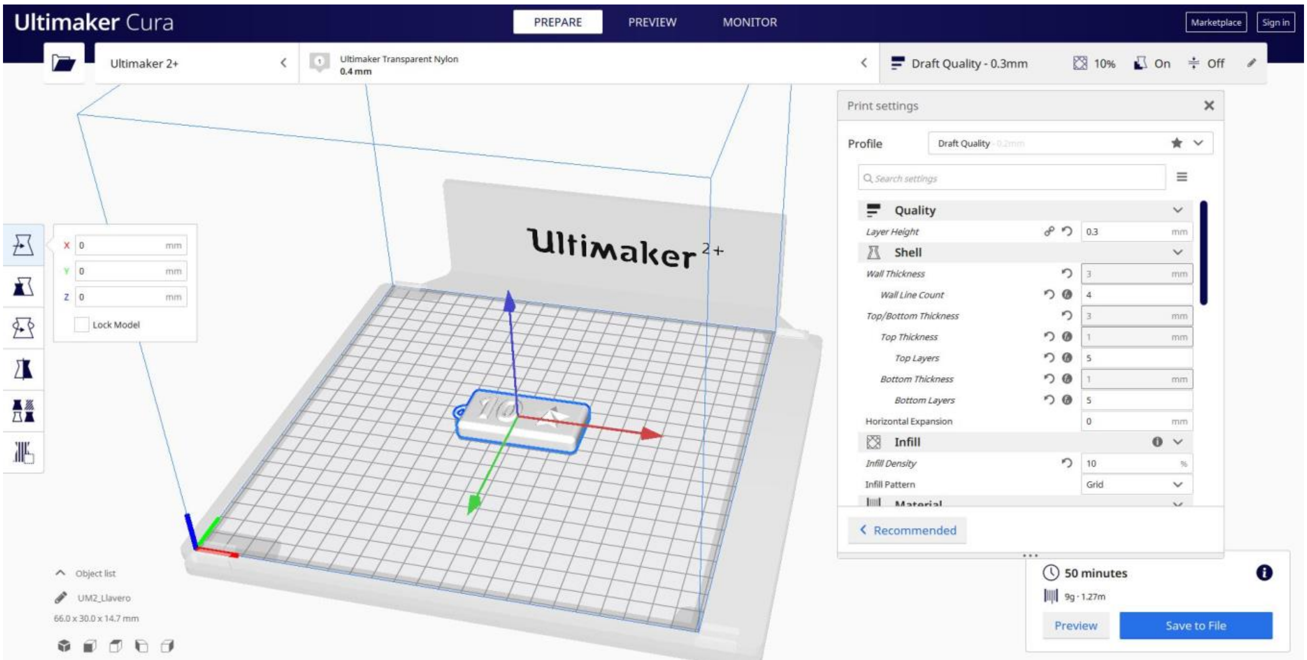
4. You can add for example a star next to the number. Scale and place it like in the picture below and group them.



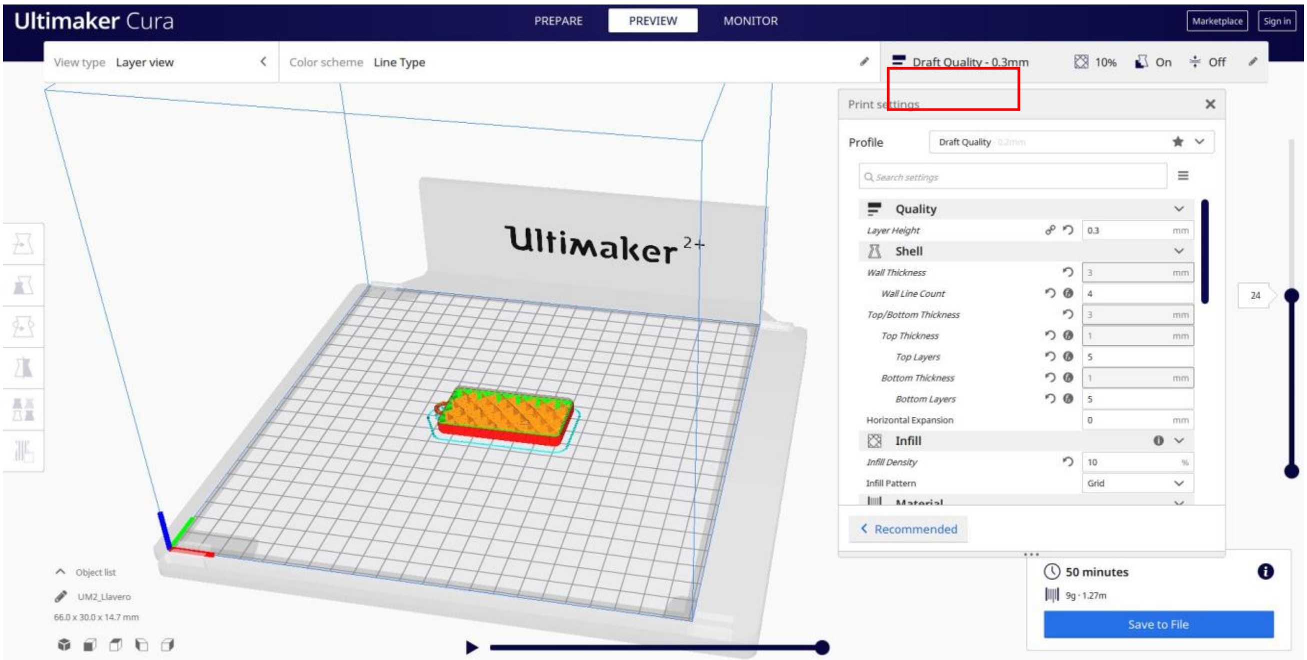


9.3.31.2 Keyring 3D printing seetings

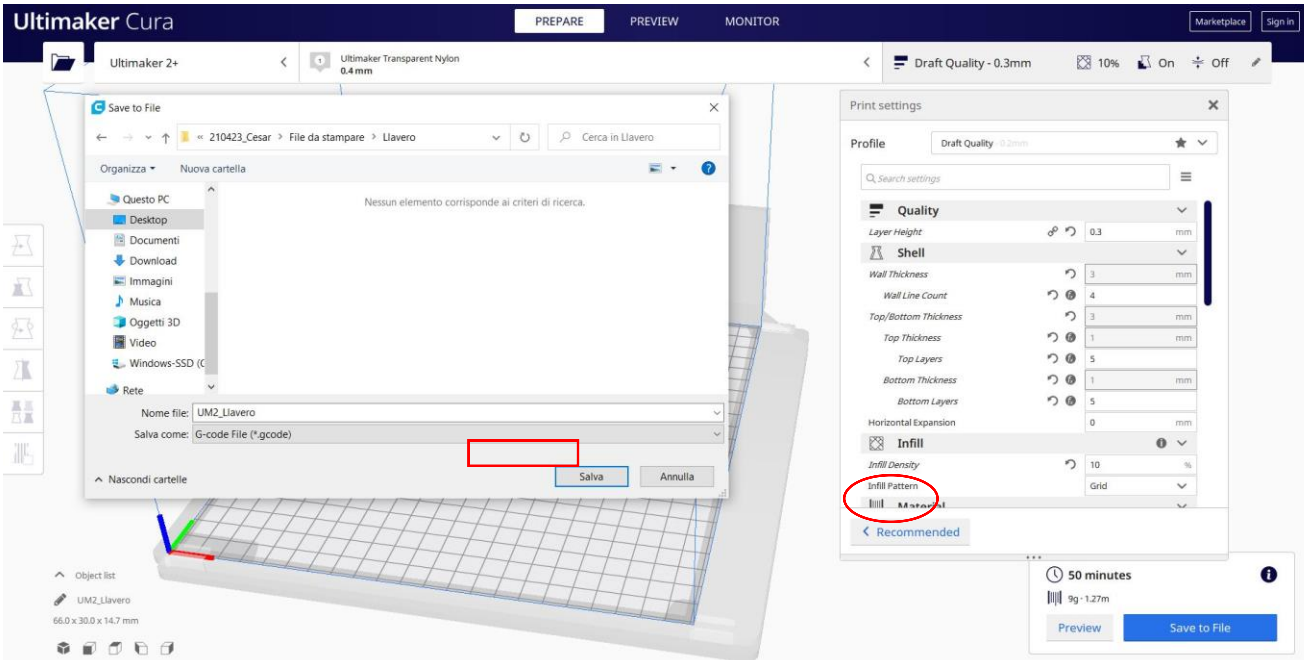
Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. Enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



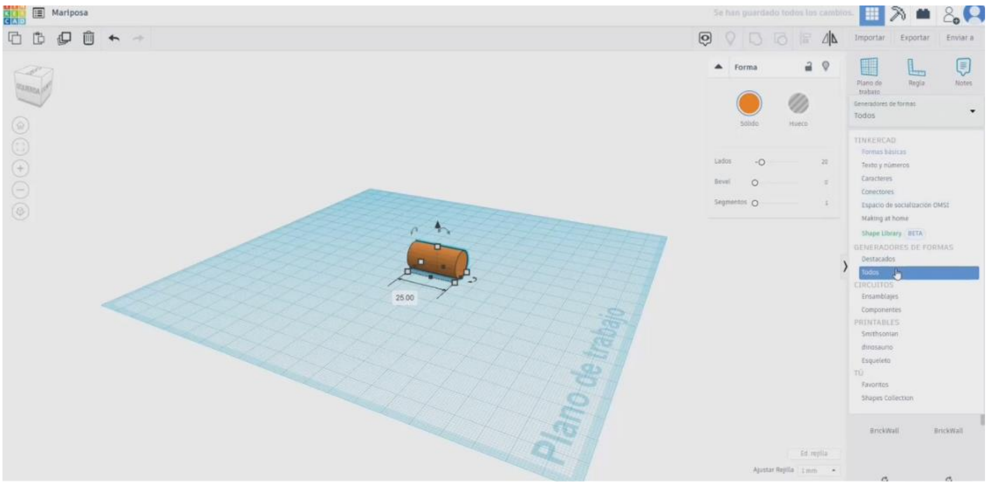
3. At this point I can save the ".Gcode" file to send to the machine.



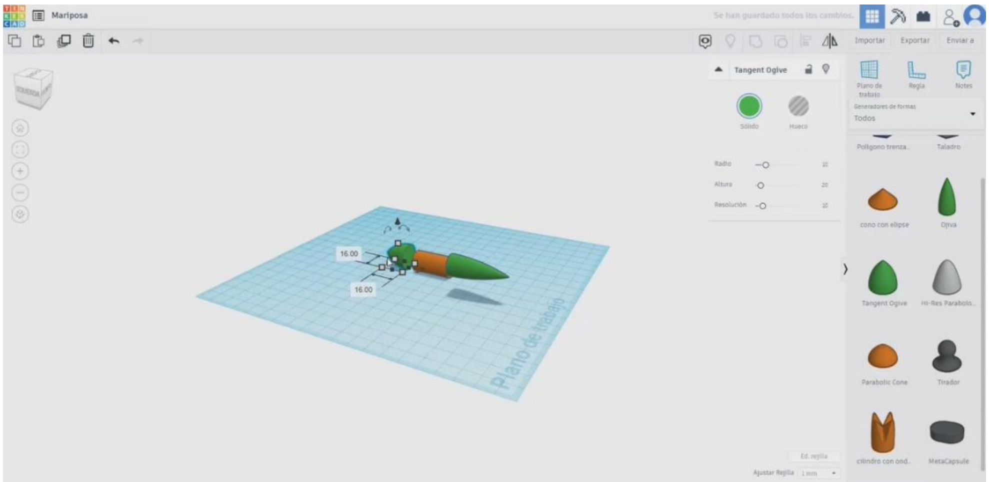
9.3.32 Part 32: Butterfly

9.3.32.1 Butterfly Design

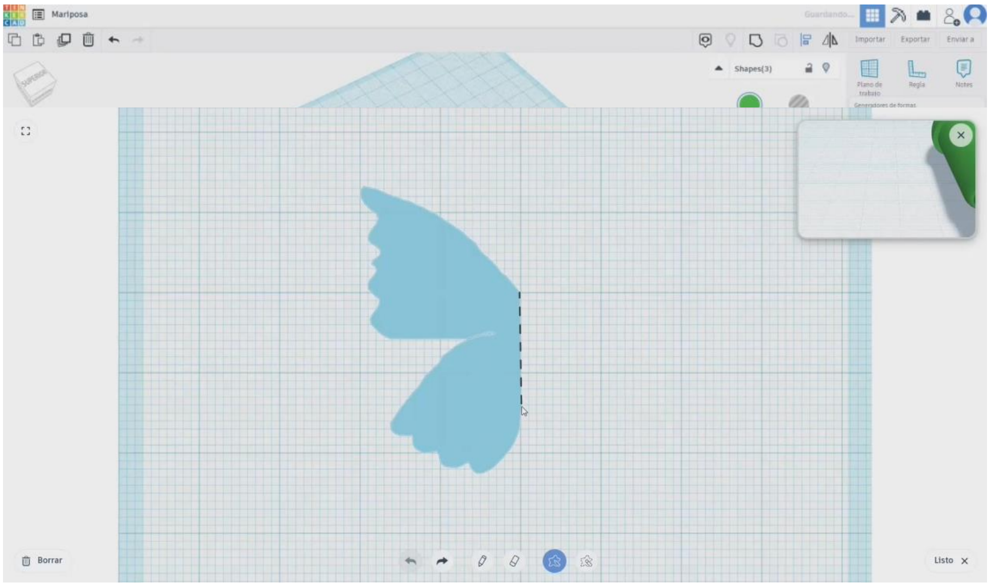
1. Build a cylinder. Turn it 90 degrees and change the measurements to 13x13x25.



2. In the list of shapes generators, look for an Ogive. Draw one and make it thinner and longer. After this, draw a Tangent Ogive, change the measurements to 16x16.

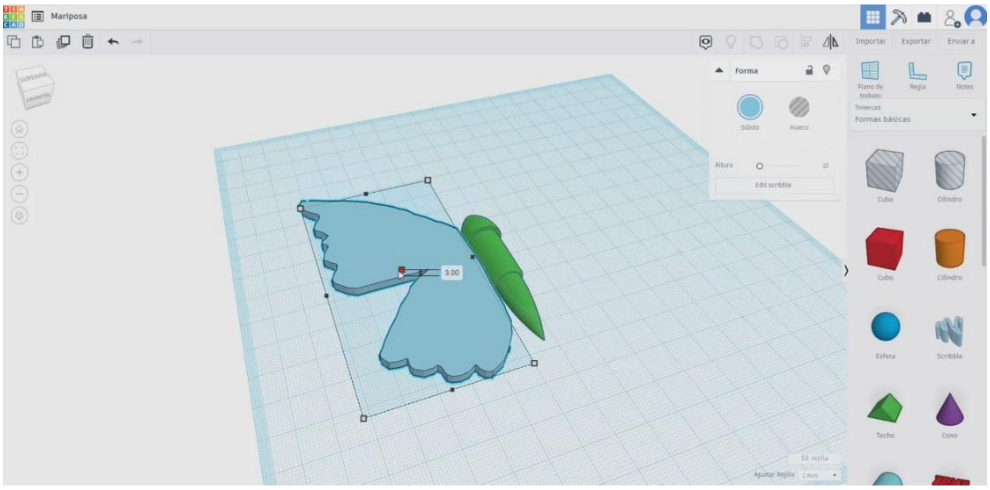


3. Select all the bodies and align them. Then group them.

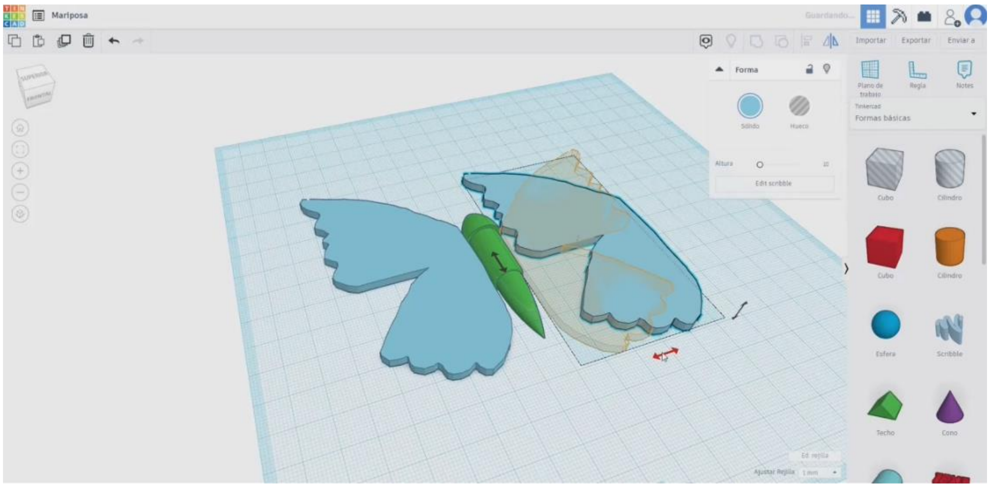


4. Now, in the basic shapes, choose the scribble tool and click next to the center body we built. Draw one of the wings with the shape you prefer.

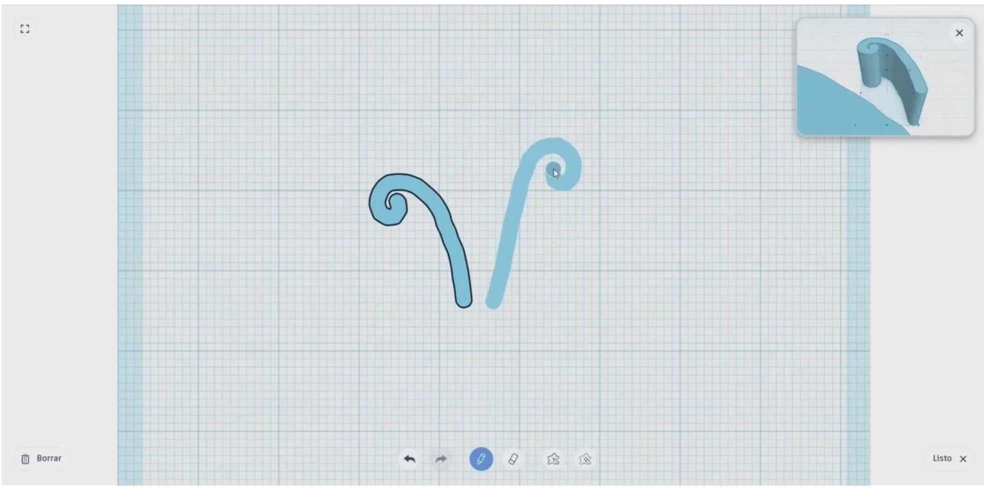
5. Scale the wing until have the proper size comparing it with the main body.



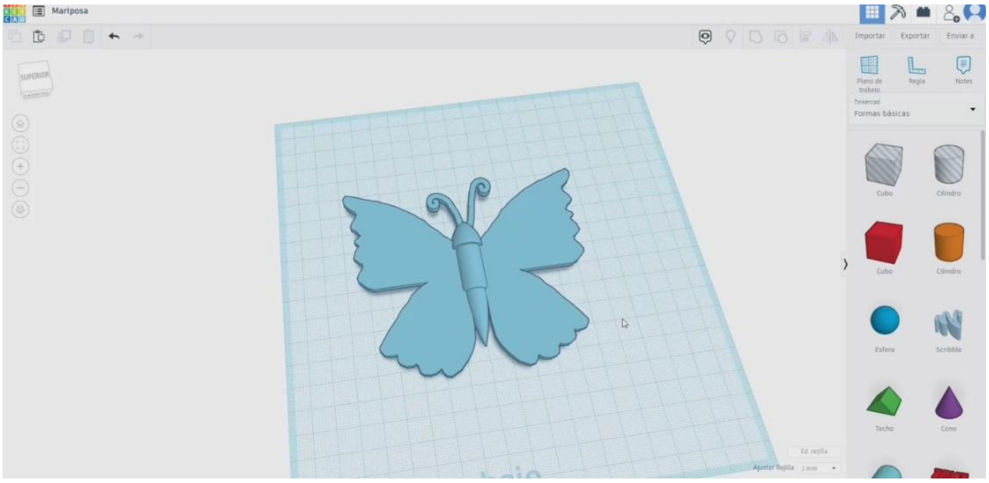
6. Copy the wing and use the symmetry button. Align the wings with the body and group them all.



7. Add another scribble and draw the antennae. When they are done, adapt the height and place them in the head of the body.

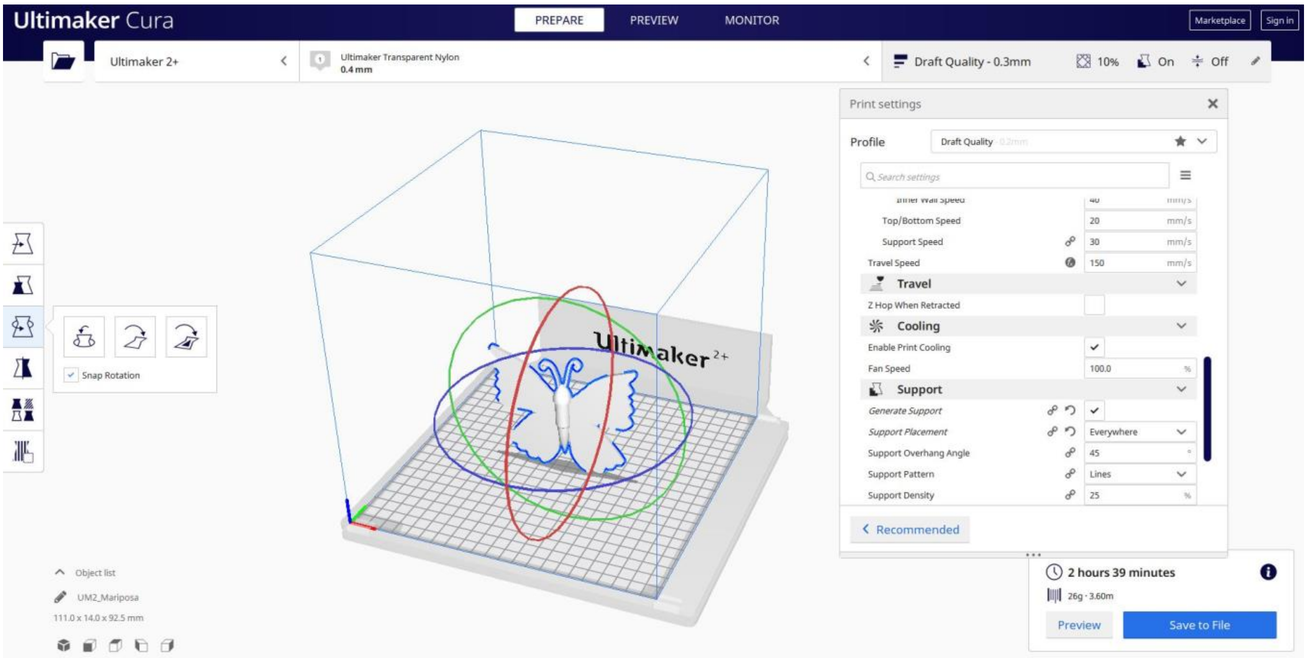


8. Select everything and press group button.

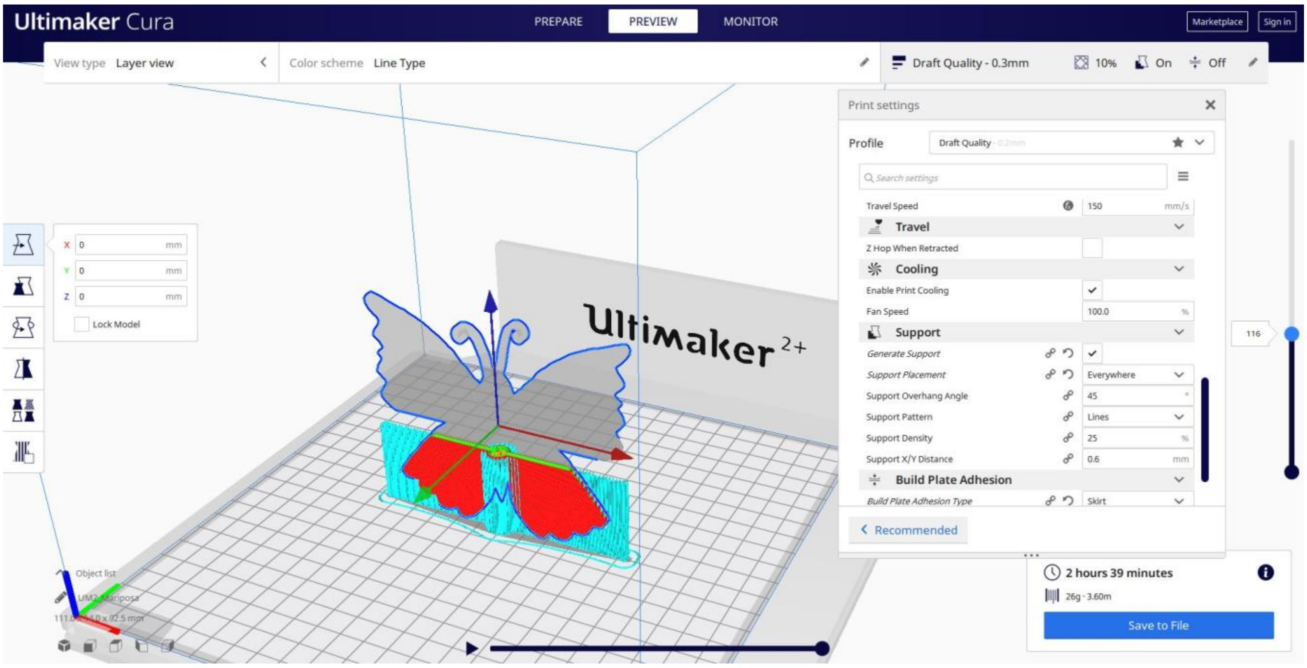


9.3.32.2 Butterfly 3D printing seetings

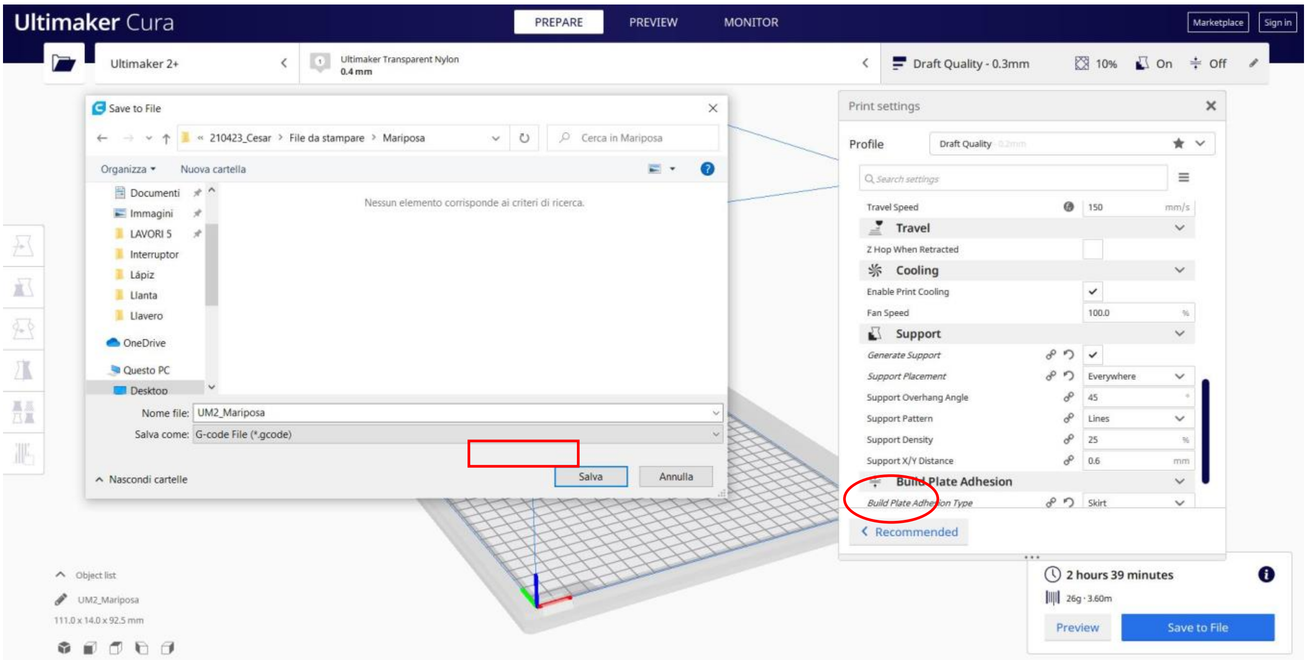
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



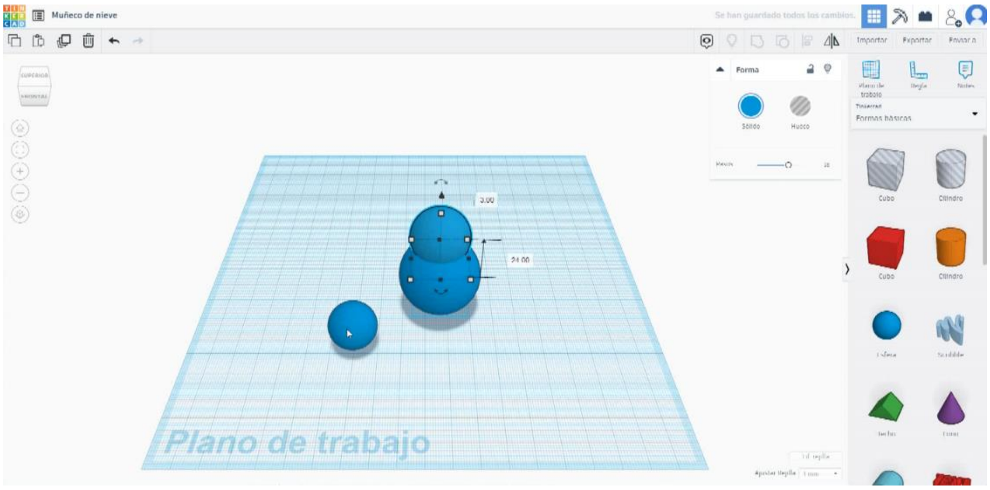
3. At this point I can save the ".Gcode" file to send to the machine.

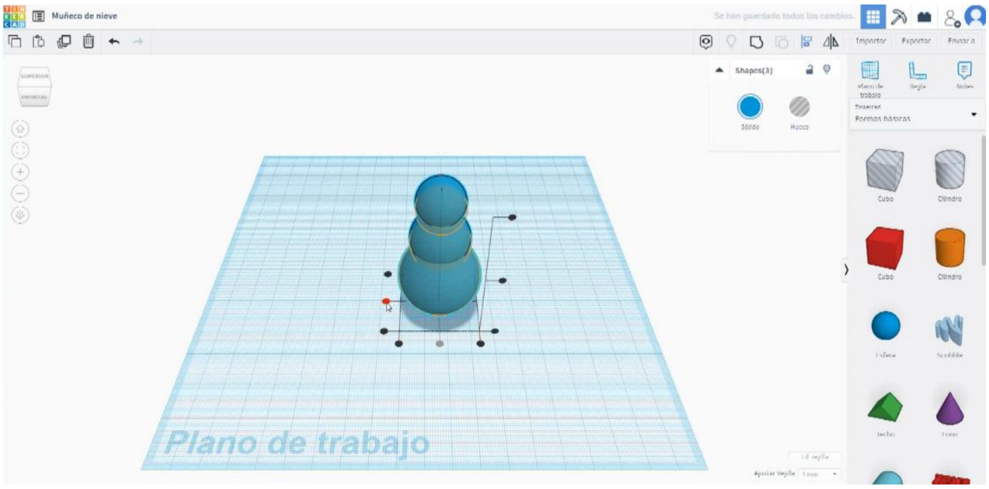


9.3.33 Part 33: Snowman

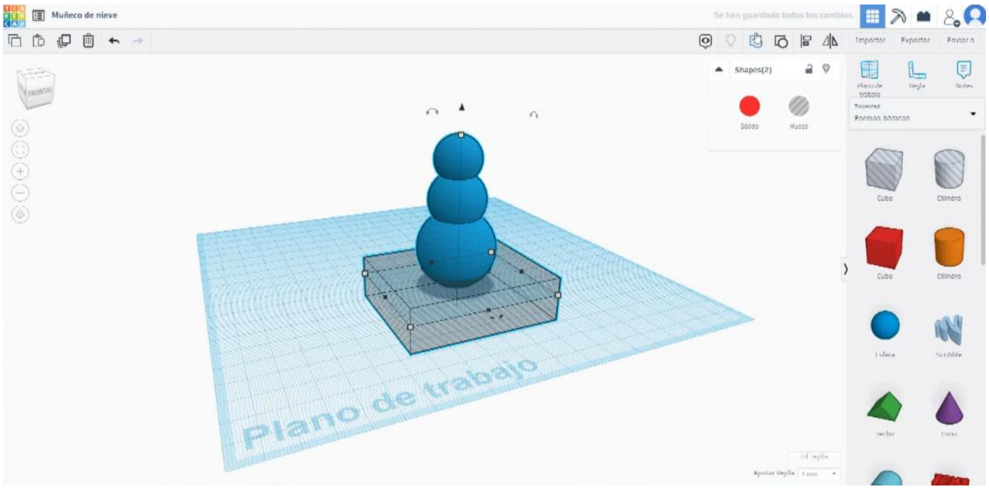
9.3.33.1 Snowman Design

1. Draw three spheres with different diameters, for example 35, 25 and 20. Place them one above the other one and select them all to align them vertically. Then, group the spheres.

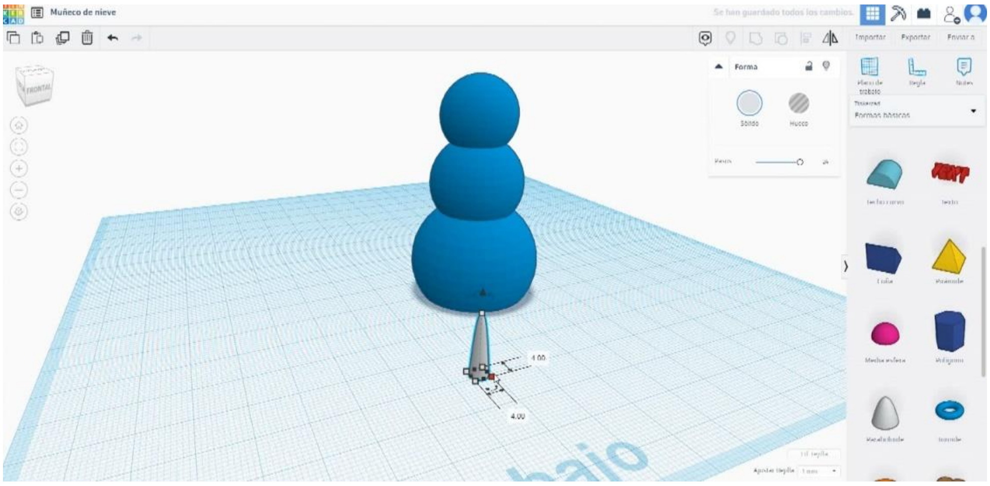


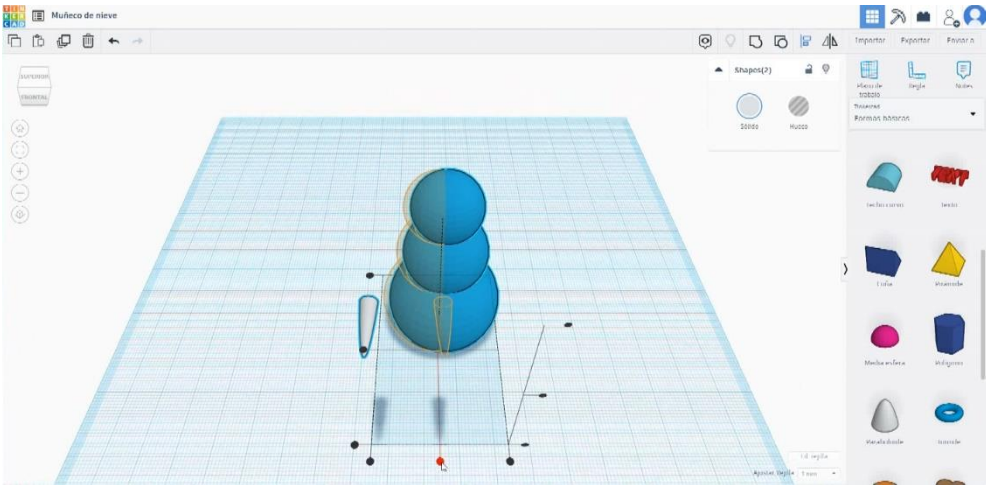


2. Build a cube, make it bigger tan the spheres and place it under them, a Little bit above the work plane. Change it to hole mode and group it with the spheres. This way we get a flat base for the snowman.

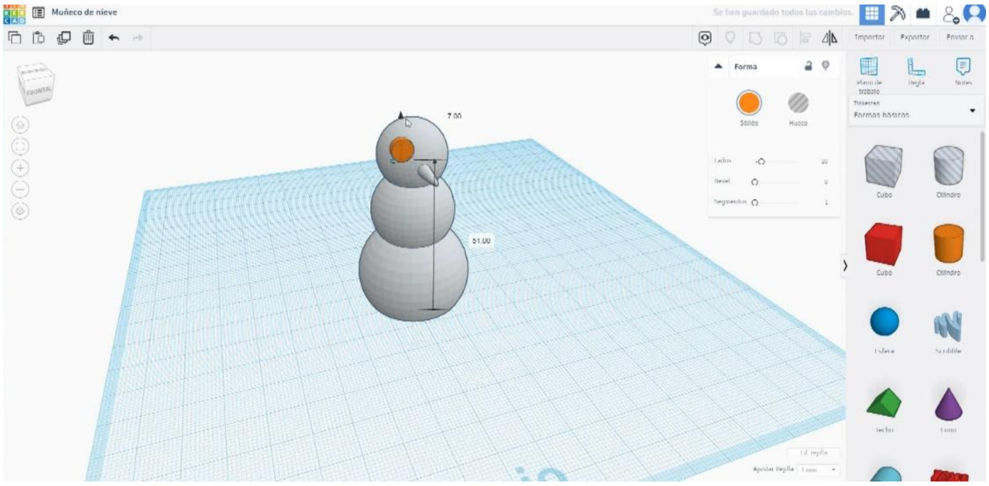


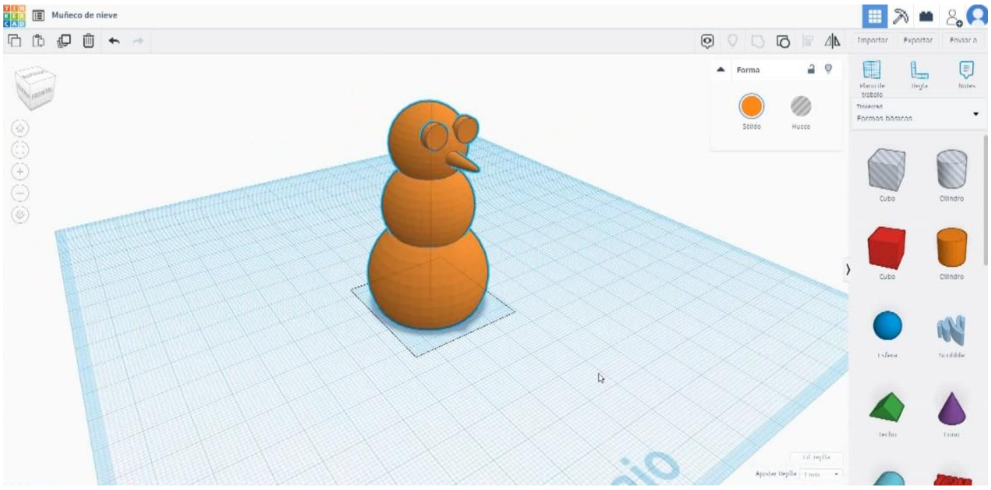
3. With a paraboloid we will do the nose. Change the measurements to 4 x 4 x 15. Rotate it 90 degrees and place it in the head of the snowman. Select the body and the nose and align them. Press Group button.





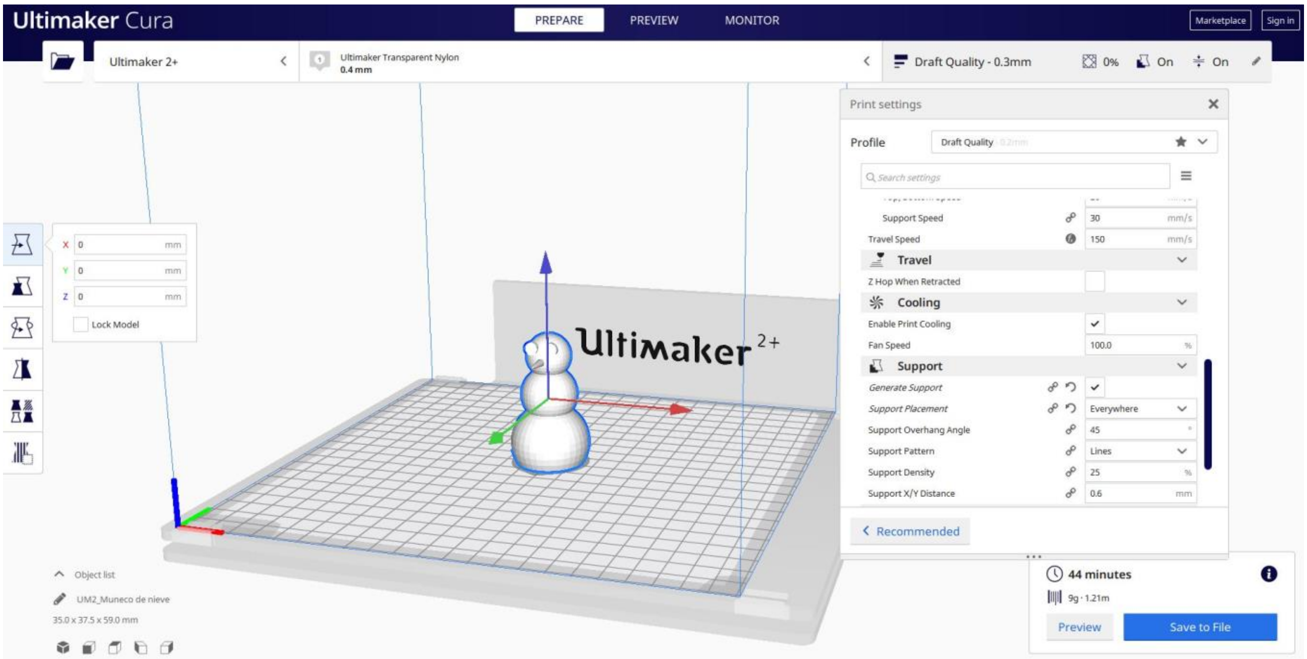
4. Use a cylinder to make an eye. Scale to 6 x 6 x 2 and place it above the nose, on one side. Copy it and place it in the other side. When you have them positioned, press group button.



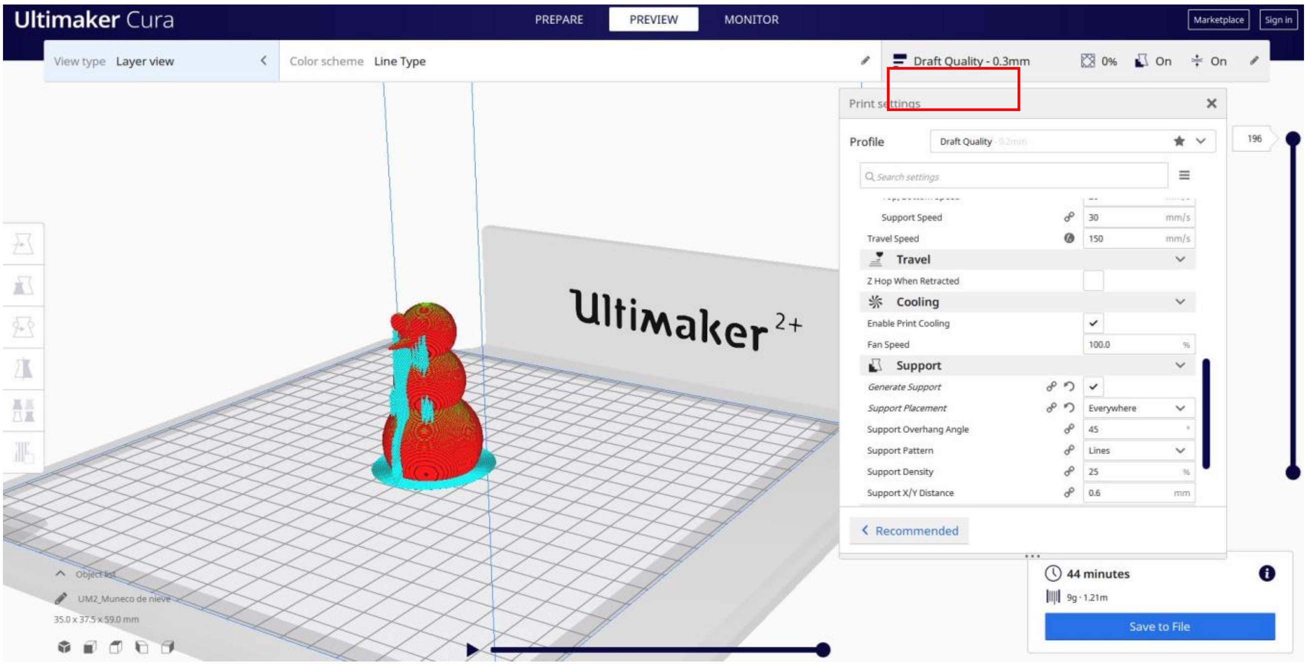


9.3.33.2 Snowman 3D printing seetings

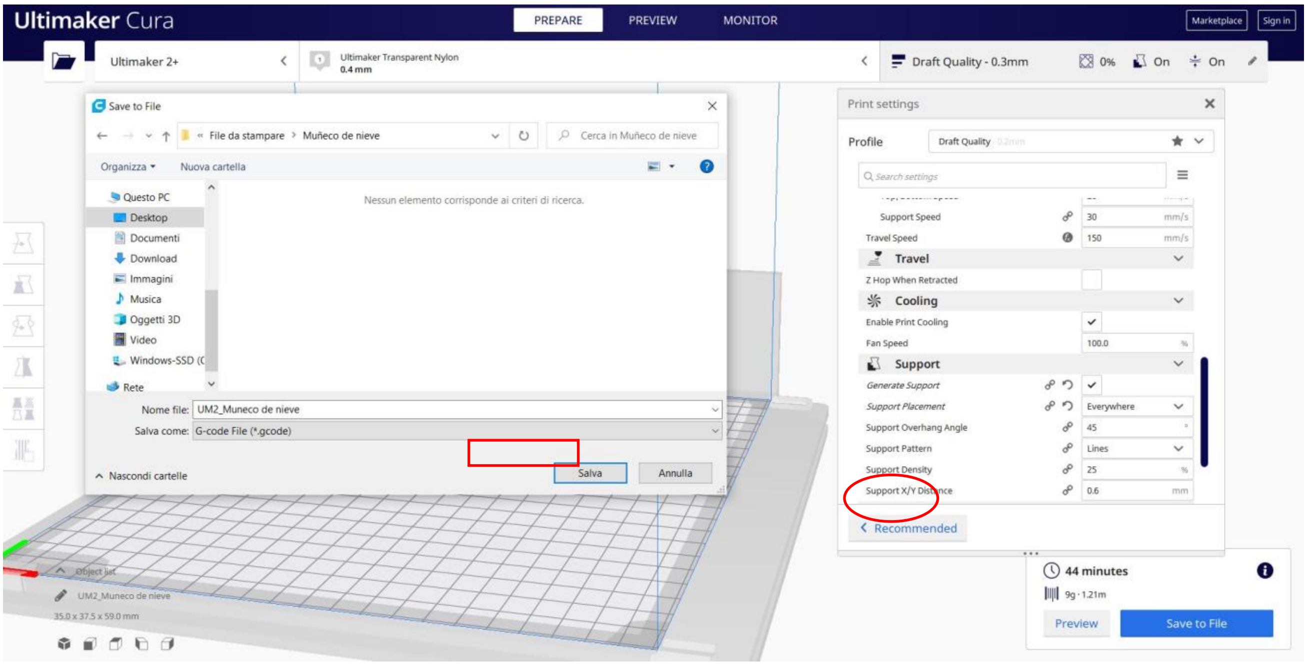
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



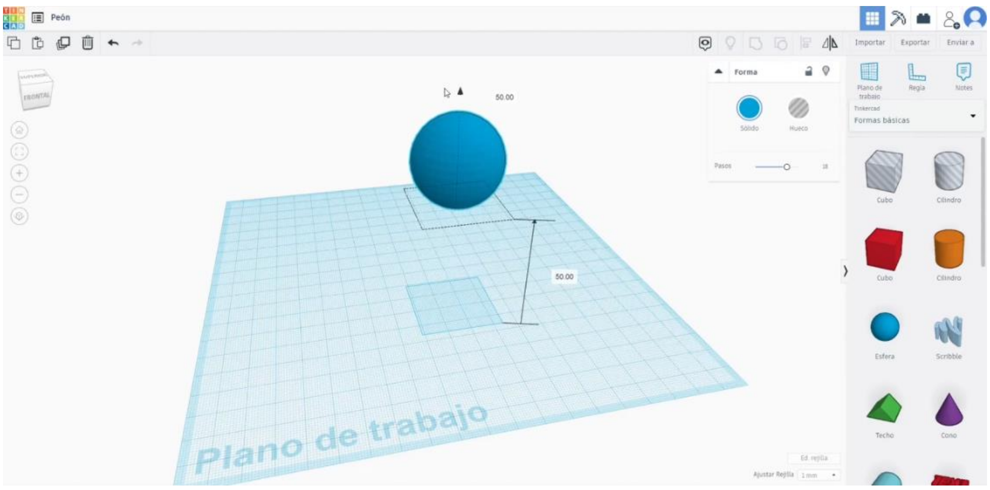
3. At this point I can save the ".Gcode" file to send to the machine.



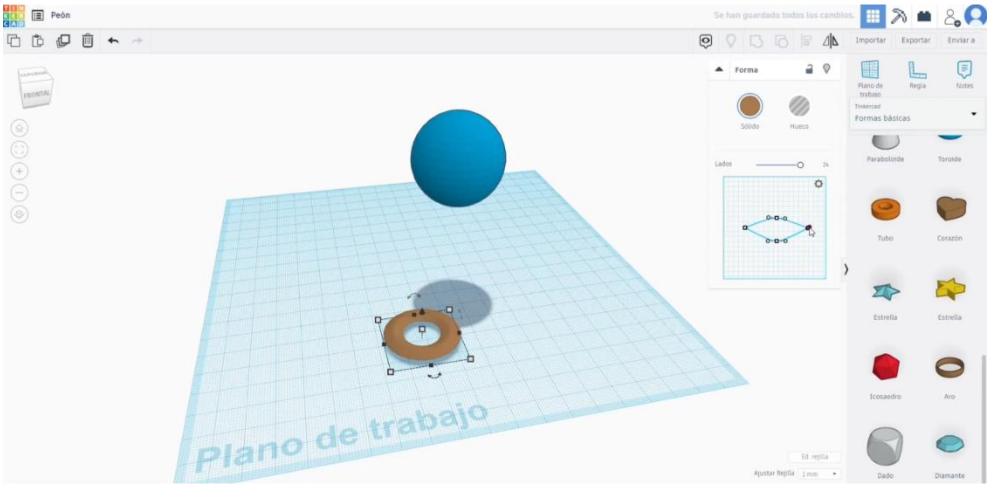
9.3.34 Part 34: Pawn

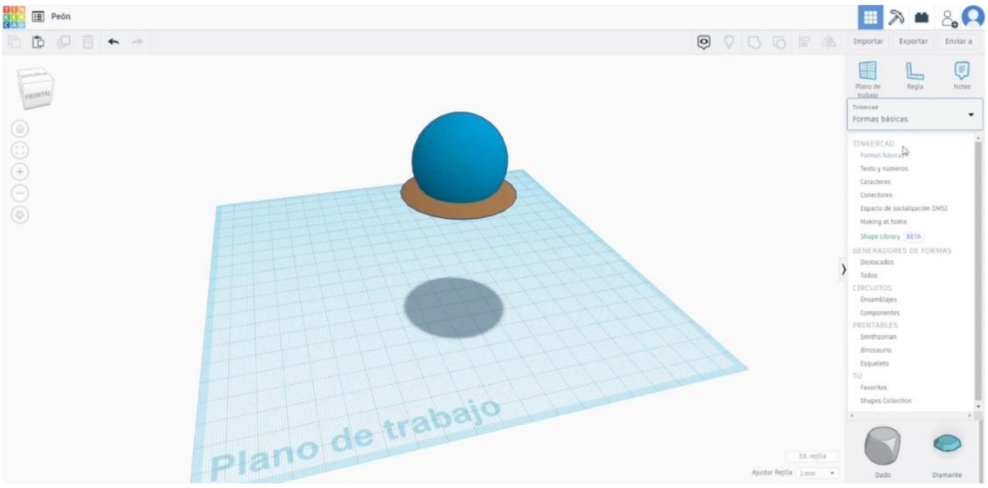
9.3.34.1 Pawn Design

1. Build a 35x35 sphere. Raise it up until 50 of height.

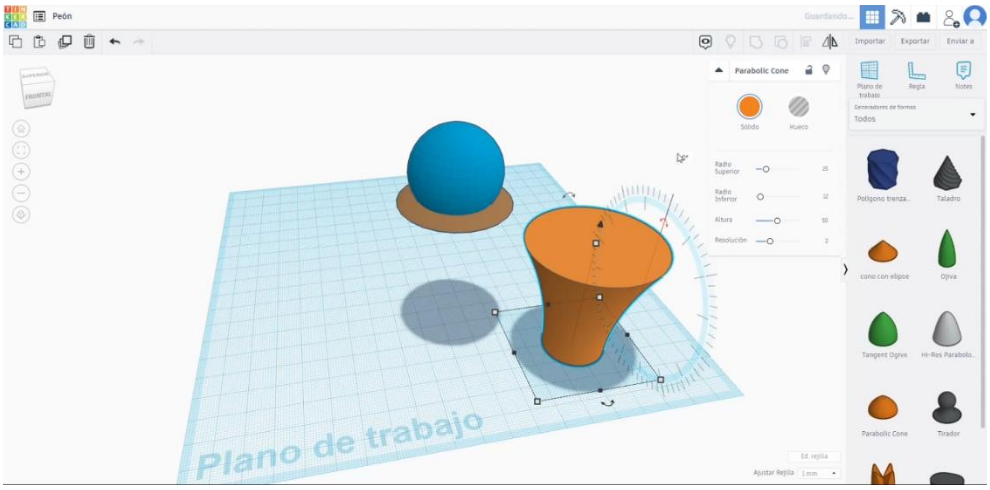


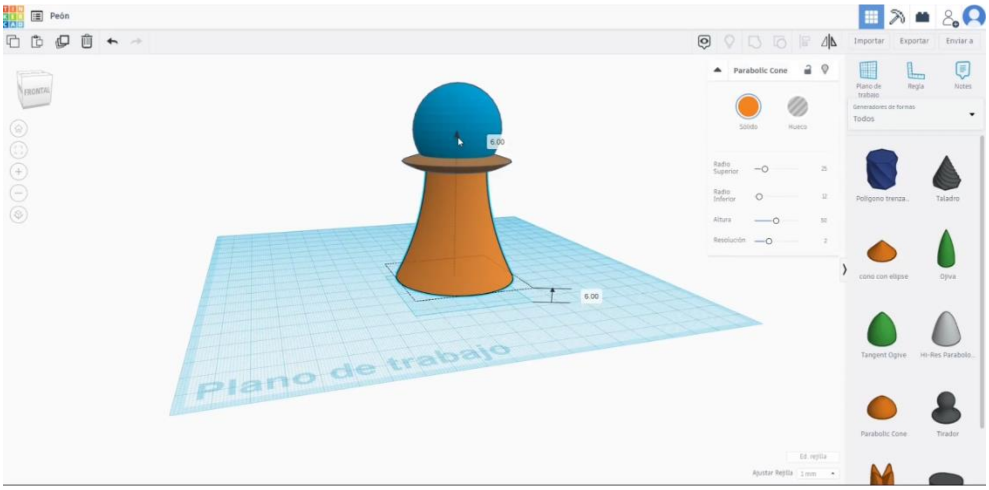
2. Create a ring, change the shape to get something similar to the images below.



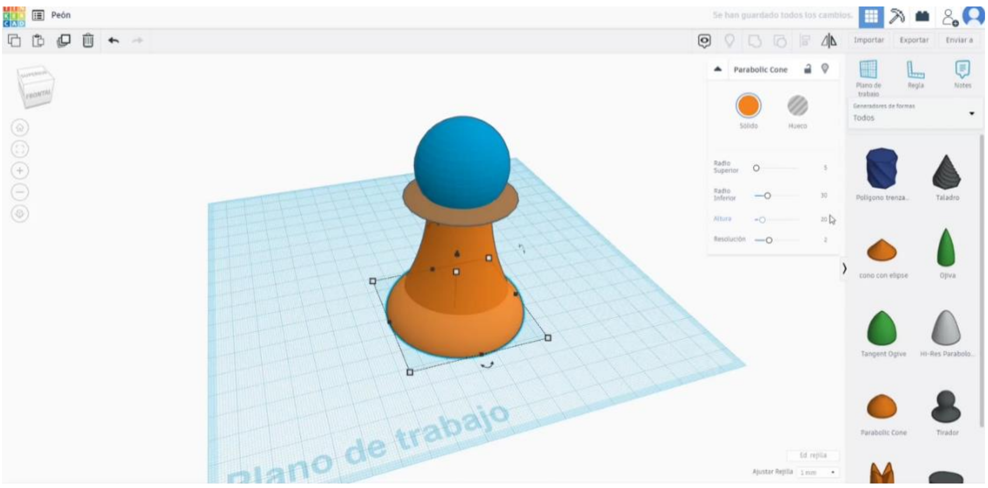


3. In the shapes generators list, look for Parabolic Cone. Edit the measurements (Top radius 25, low radius 12, height 50). Turn it 180 degrees. Place it under the sphere.

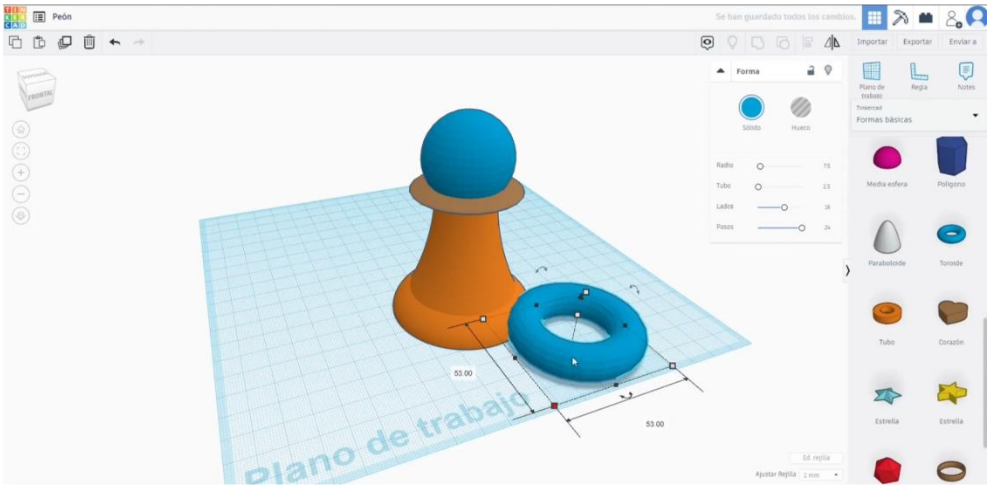


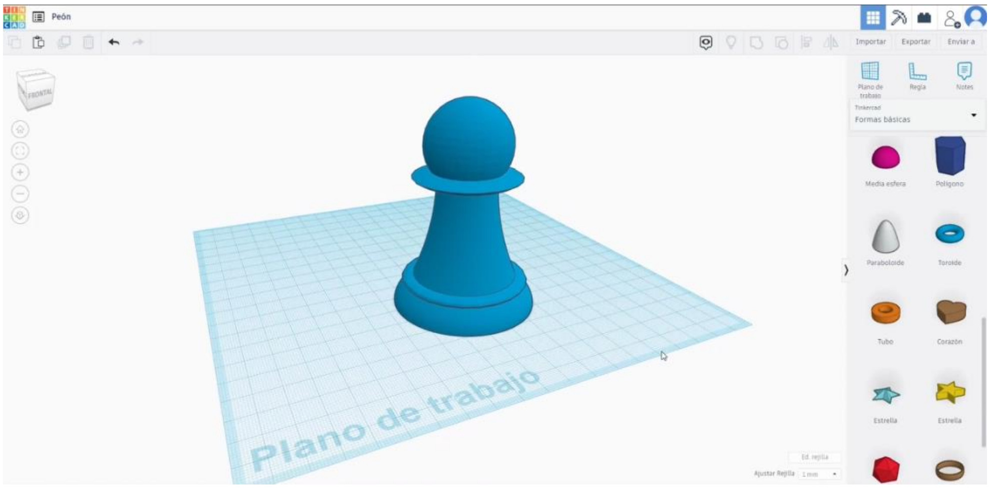


4. Draw anothe parabolic cone (top radius 5, low radius 30, height 20). Align it with the other bodies.



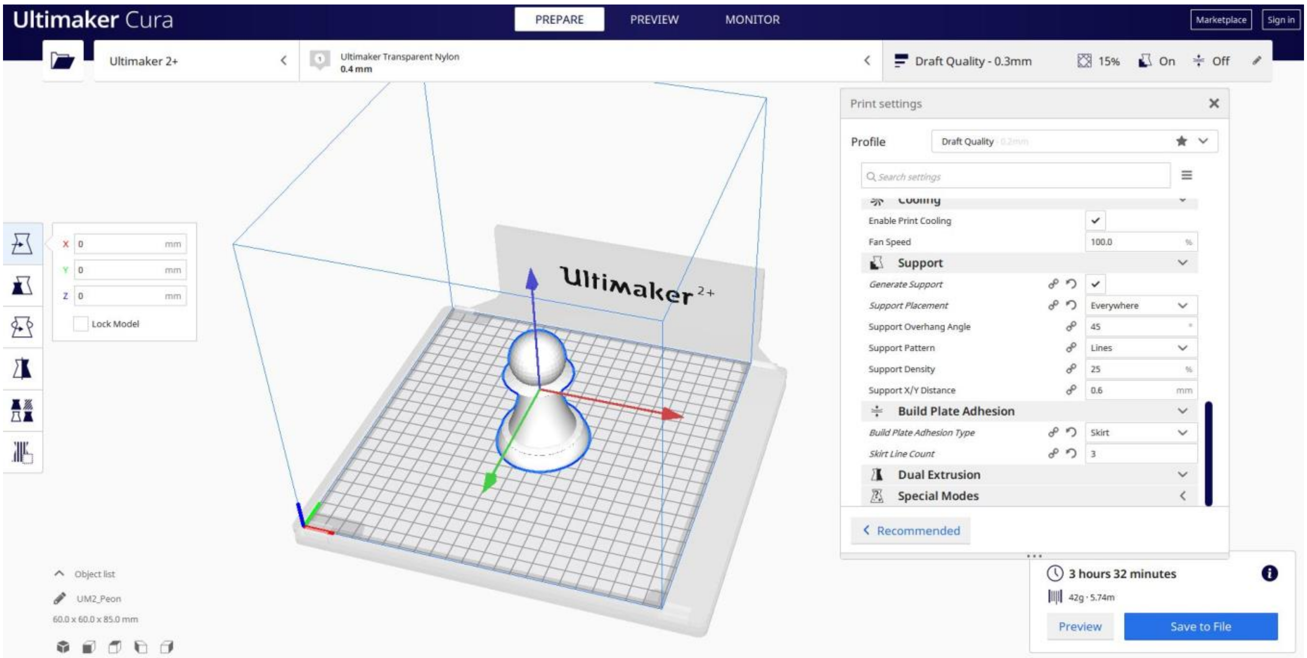
5. Build a 53x53x4 Toroid. Raise it up to 9. Align all the bodies and press the group button.



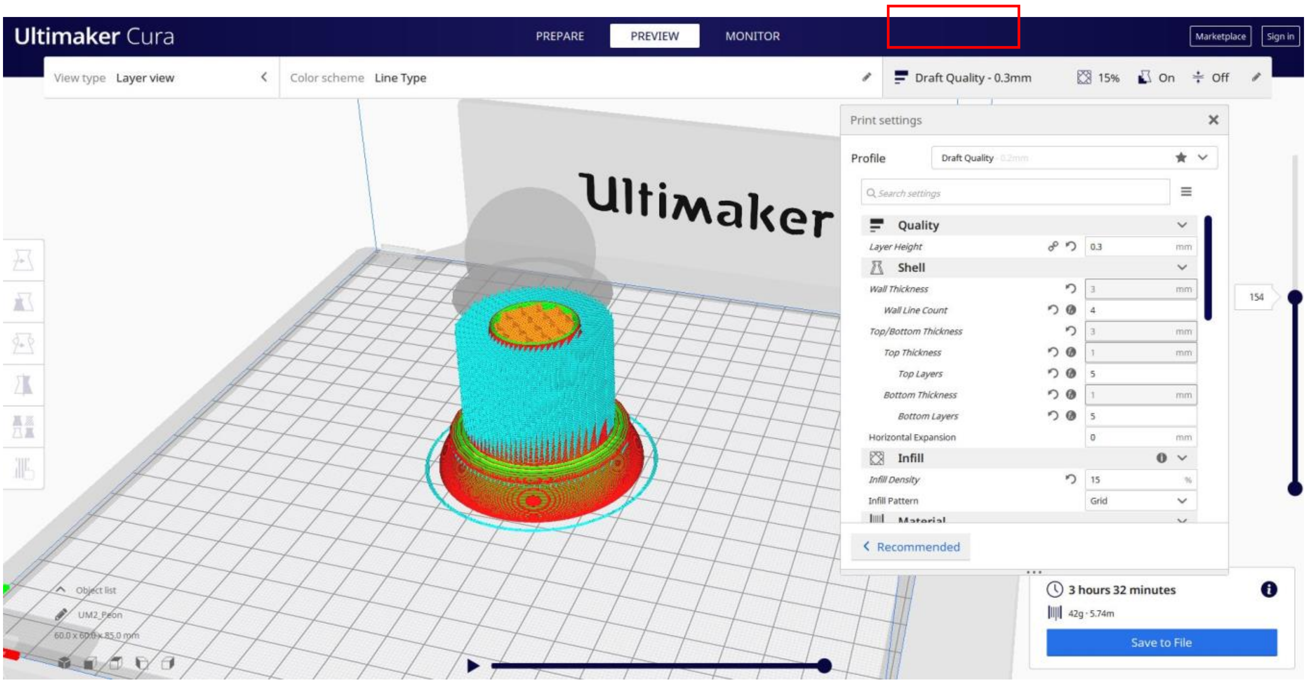


9.3.34.2 Pawn 3D printing seetings

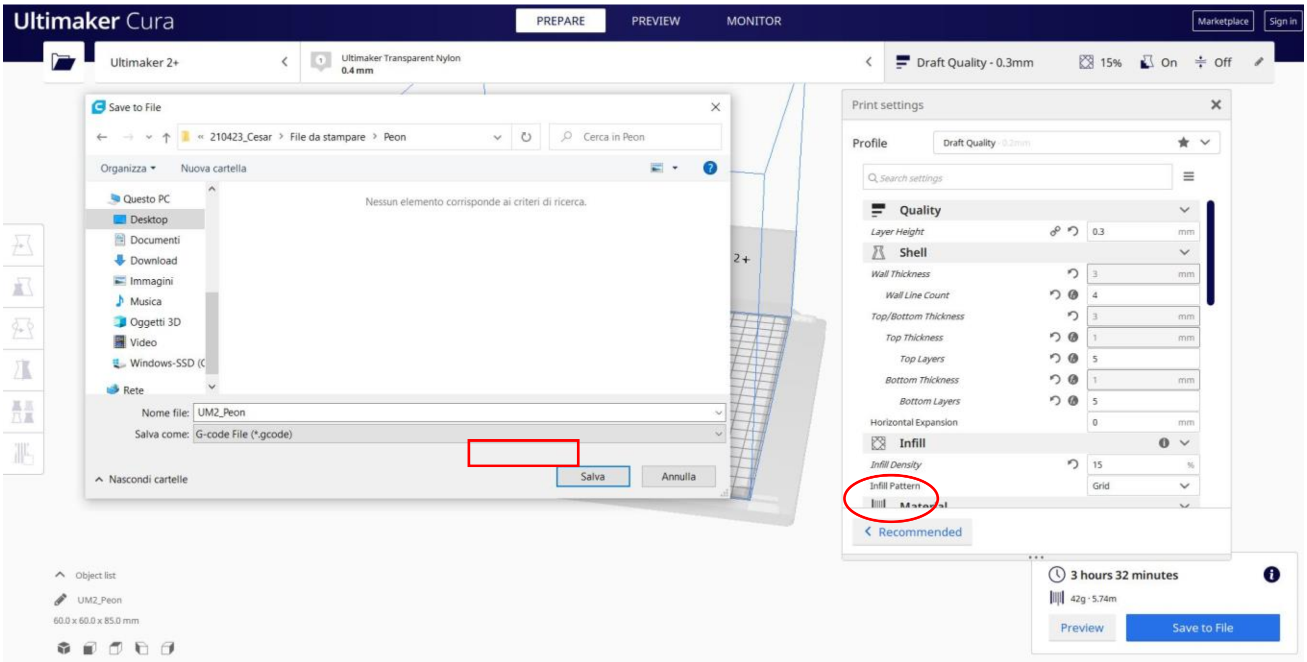
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



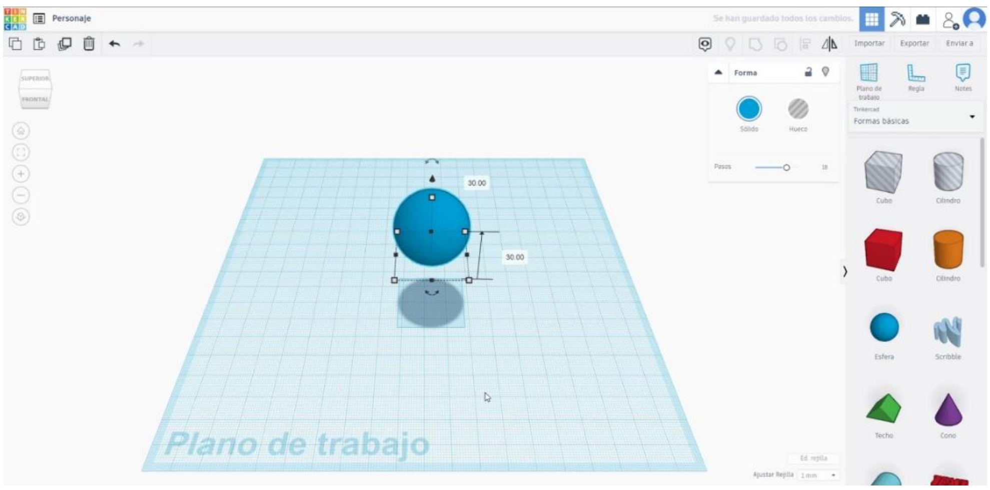
3. At this point I can save the ".Gcode" file to send to the machine.



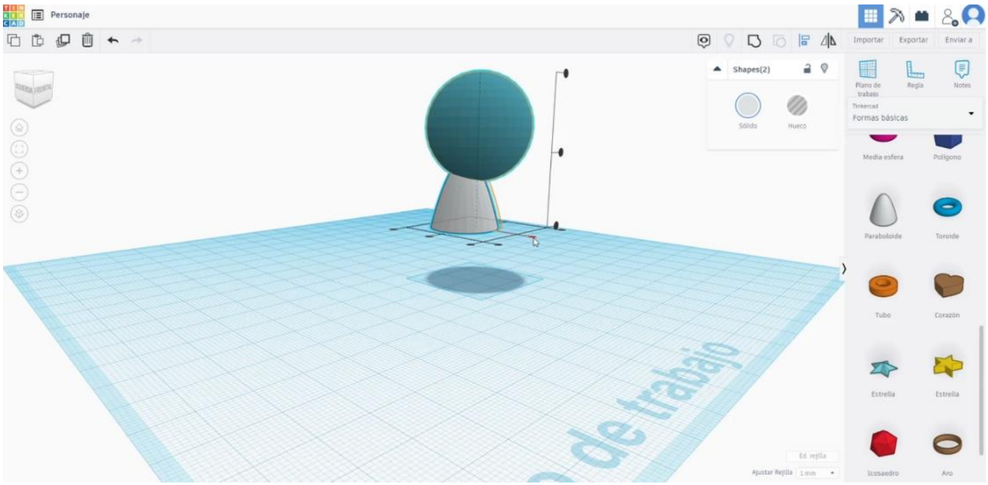
9.3.35 Part 35: Character

9.3.35.1 Character Design

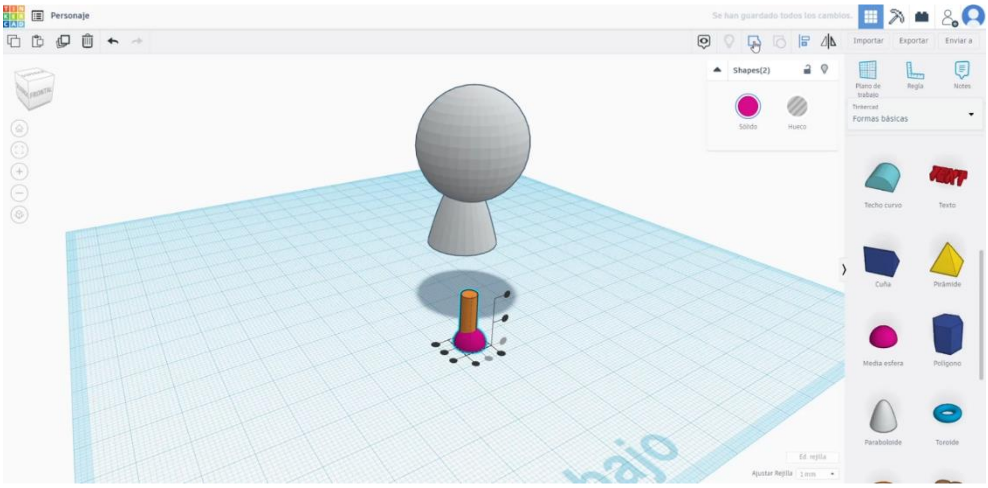
1. Build a 30x30 sphere. Raise it up until 30 height.



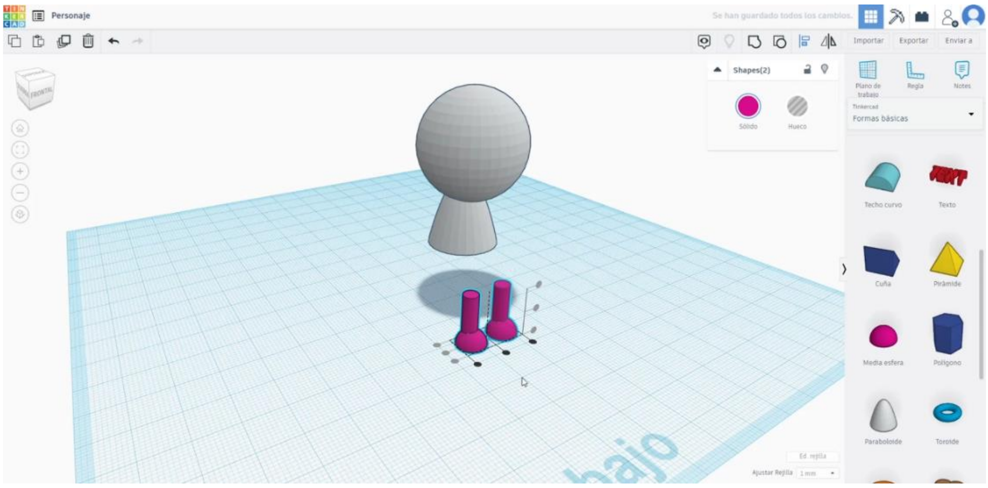
2. Create a paraboloid and place it under the sphere. Select both bodies and press group button.



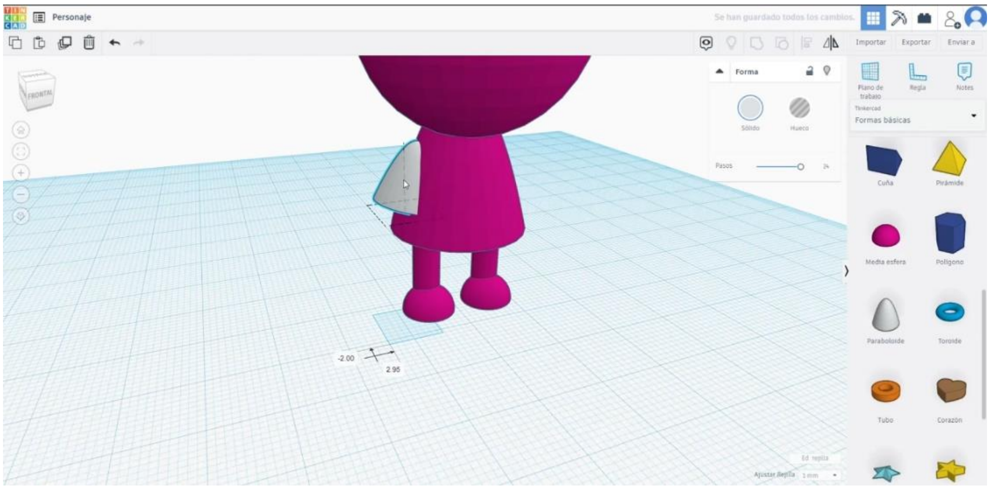
3. Draw a 4x4x14 cylinder and then a 9x9 half sphere. Align and group them like you can see below. Make a copy.



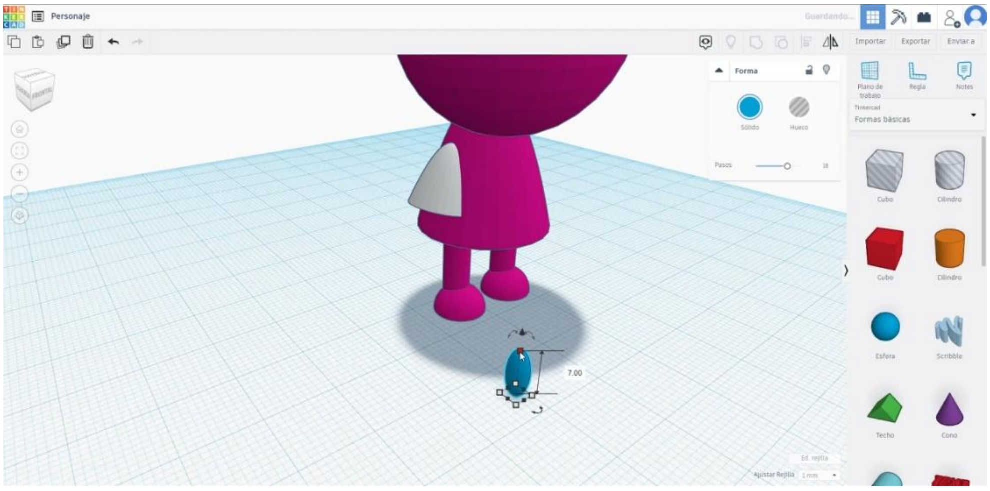
4. Align and group both legs and place them aligned under the body. Select all the bodies and group them.

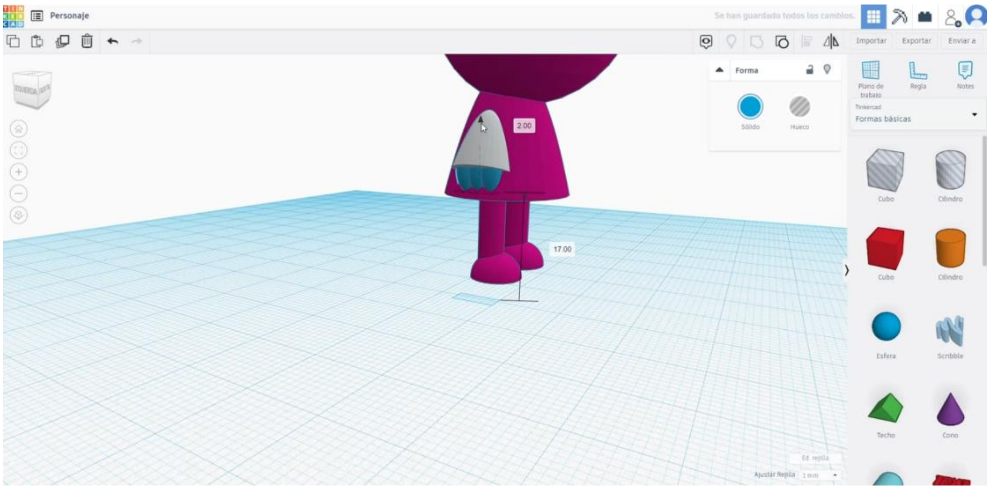


5. Draw a new paraboloid (9x9). Incline it a little bit and place it like in the image below.

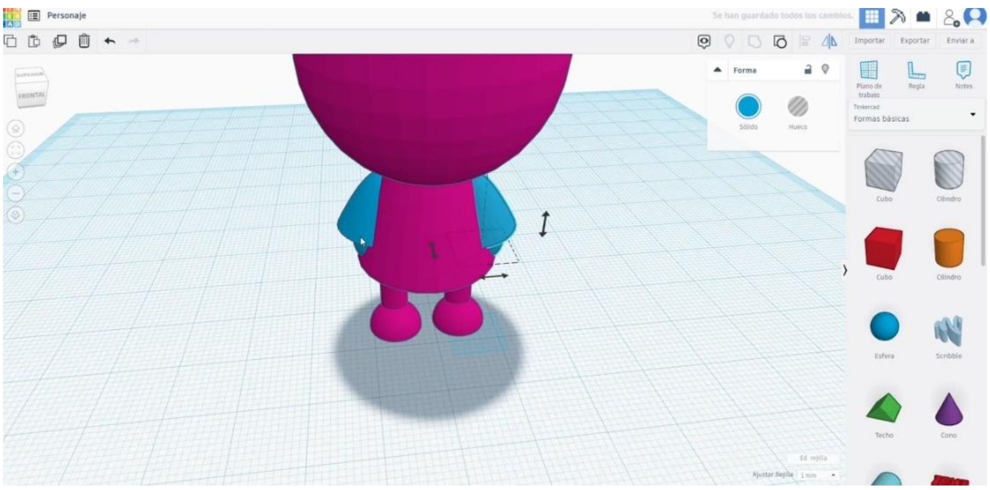


6. Build a sphere and change the measurements to 3x3x7. Make 2 copies, align them and group them. Place them under the last parabolid. Group the paraboloid and the 3 spheres.

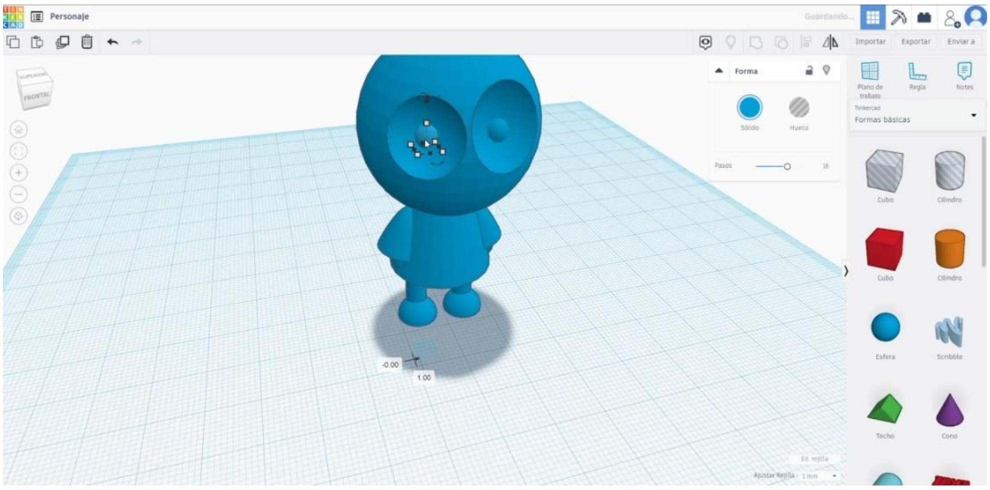




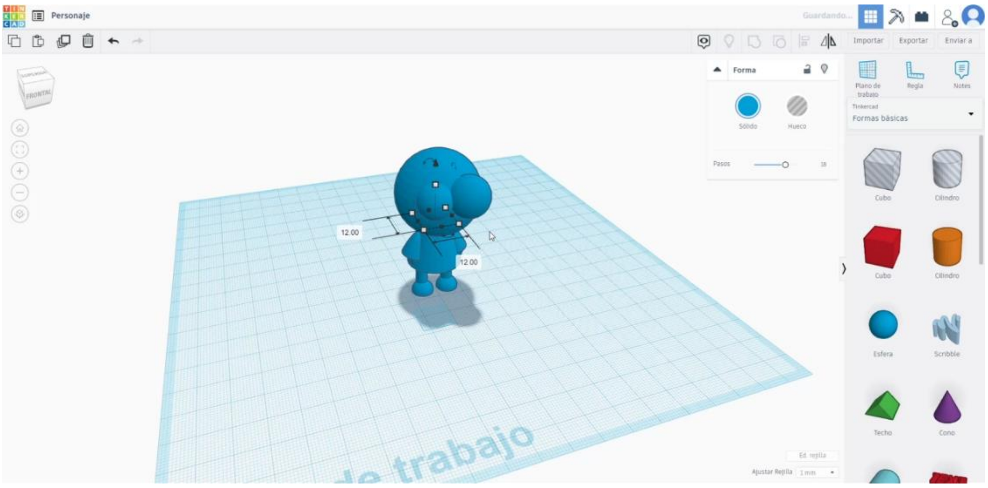
7. Do the symmetry of this last figure and place it on the other side of the main body. Align the both arms ant group them. Then align the arms with the rest of the body. Group them all.



8. Build a couple of spheres to create the eyes. For example, we make a 14x14 sphere and a 12x12 sphere. Place them in the face.

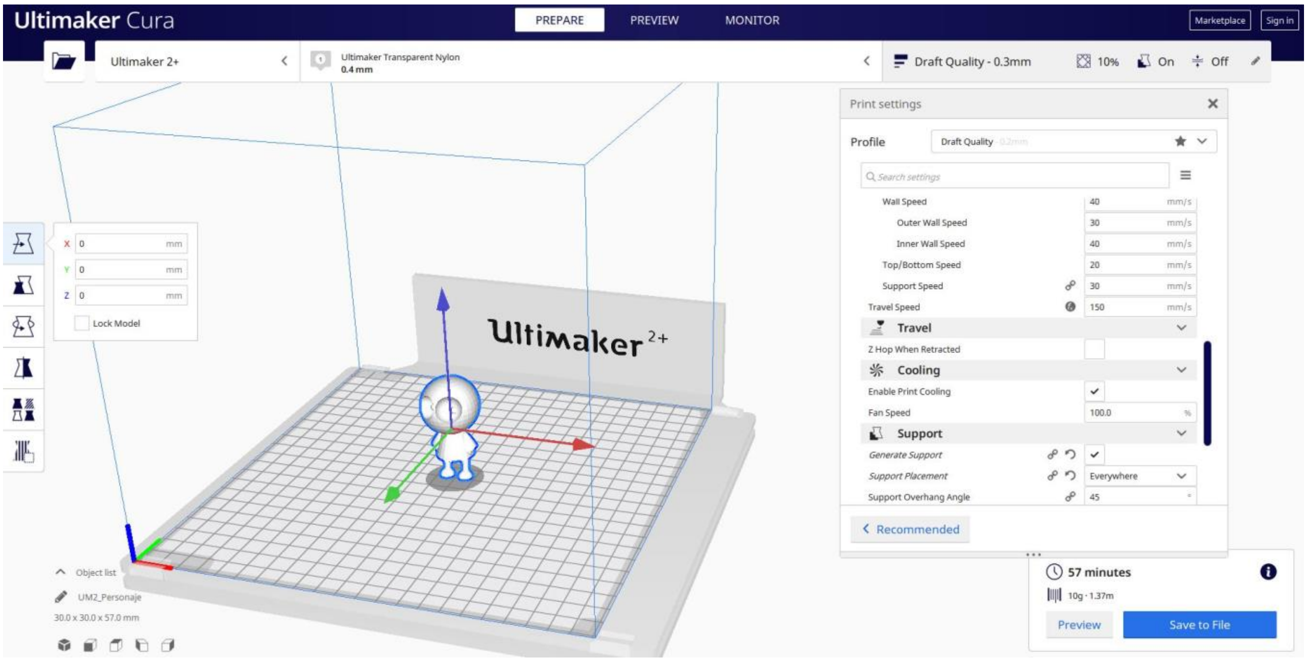


9. Change two spheres to the hole mode. Group them with the main body. Now draw a couple of small spheres (4x4) and place them in the center of the eyes. Select and group everything.

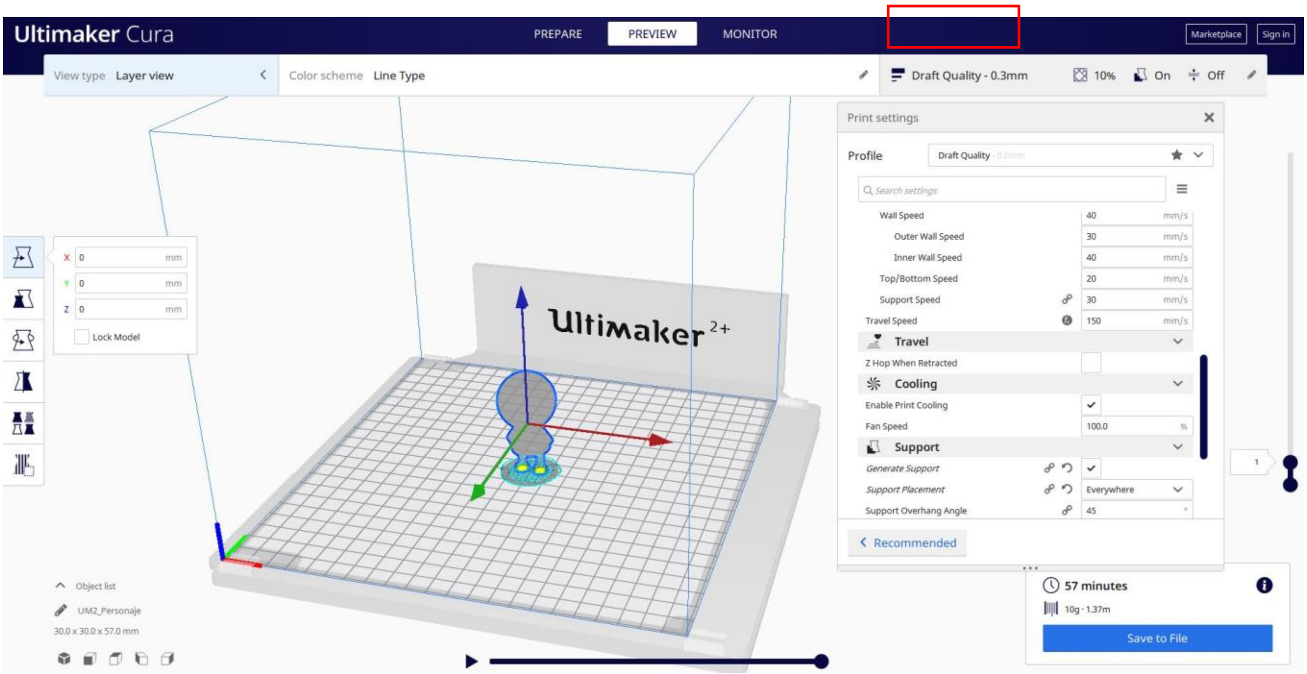


9.3.35.2 Character 3D printing seetings

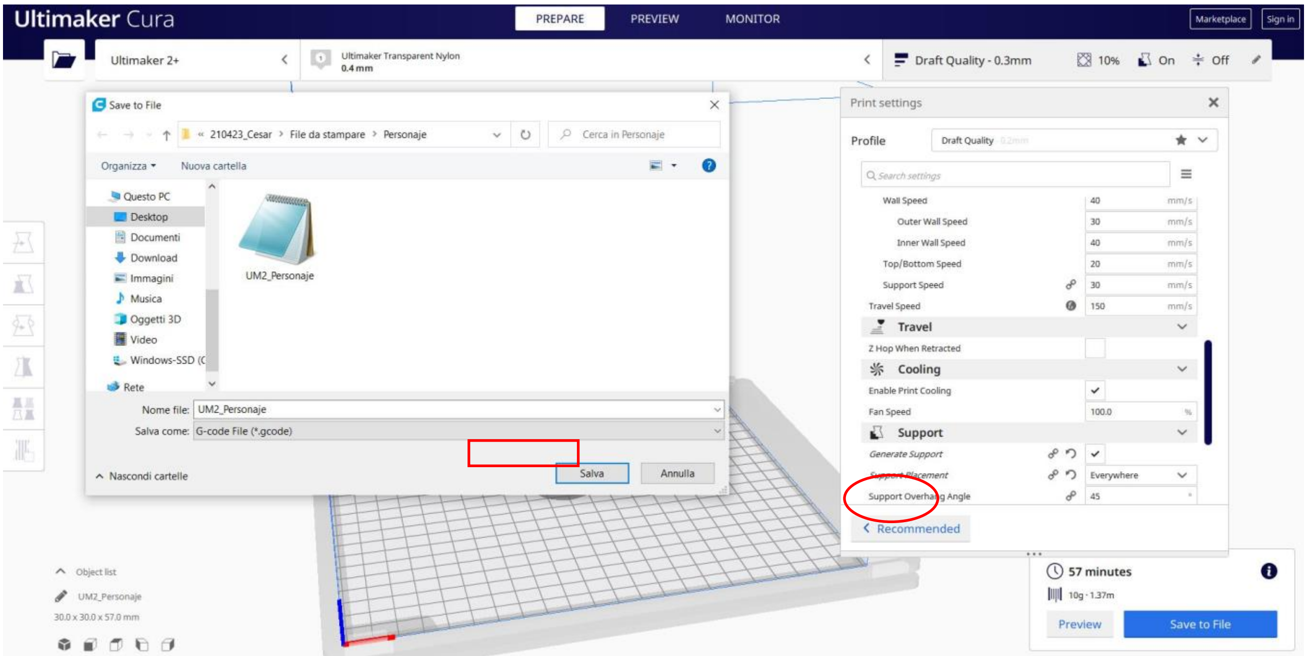
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



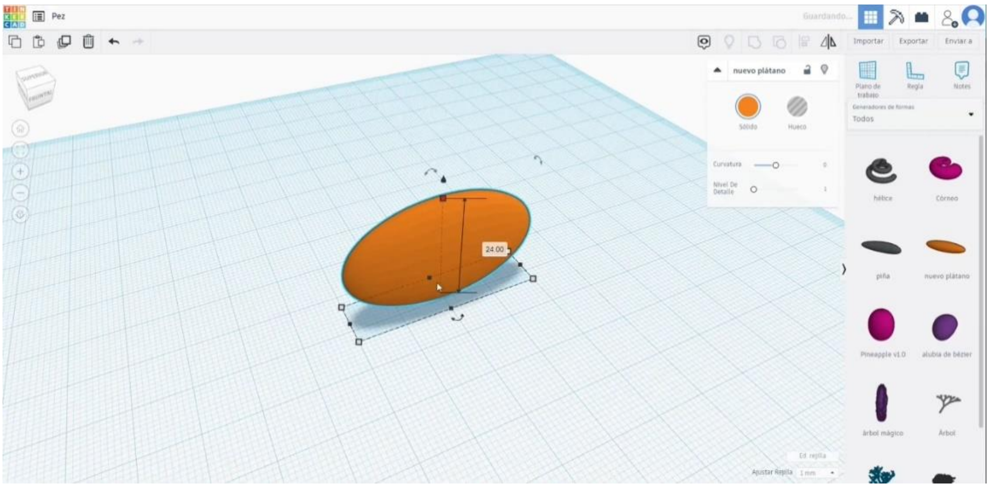
3. At this point I can save the ".Gcode" file to send to the machine.



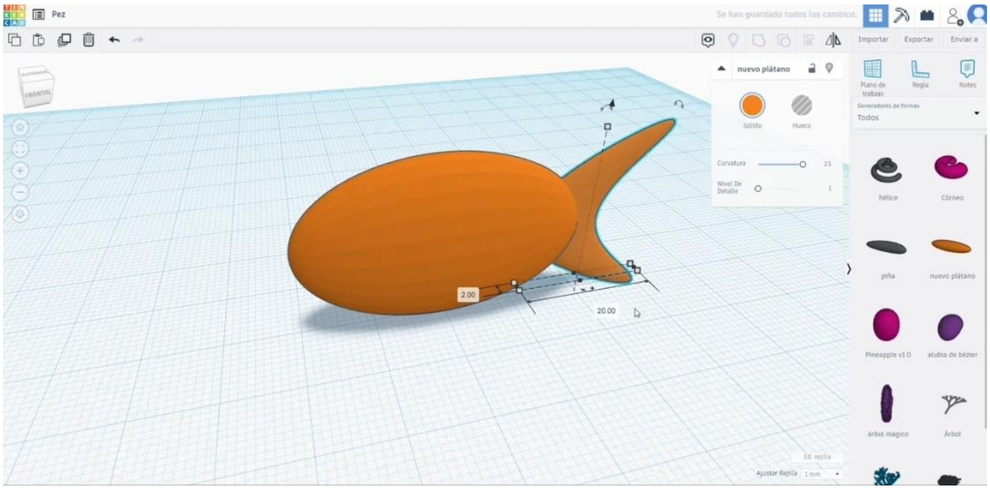
9.3.36 Part 36: Fish

9.3.36.1 Fish Design

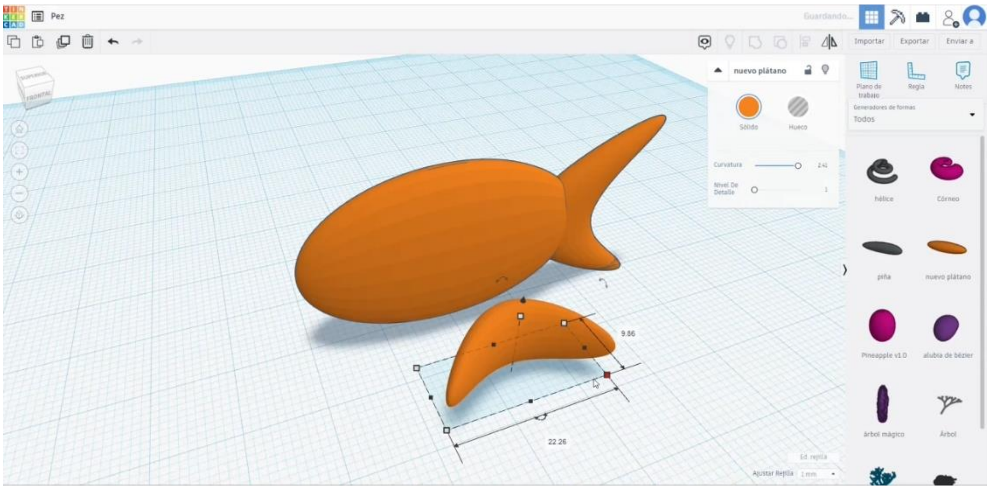
1. Build a *New banana*. Change the height to 24.



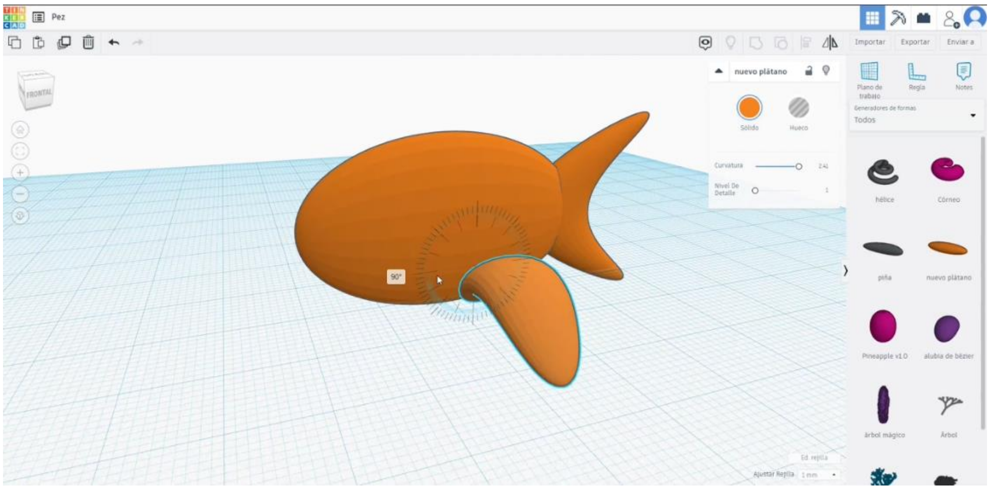
2. With another *New banana* créate the fish tail. Make it flatter (height 3) and bend it editing the curvature. Place it like you can see below. Select both bodies and press group button.

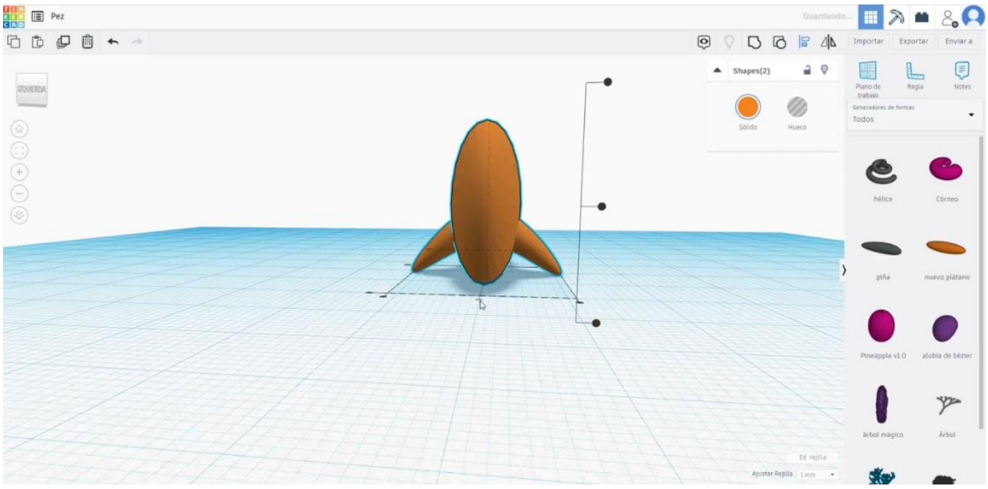


3. Draw again a new banana, bend it like in the picture and make it smaller.

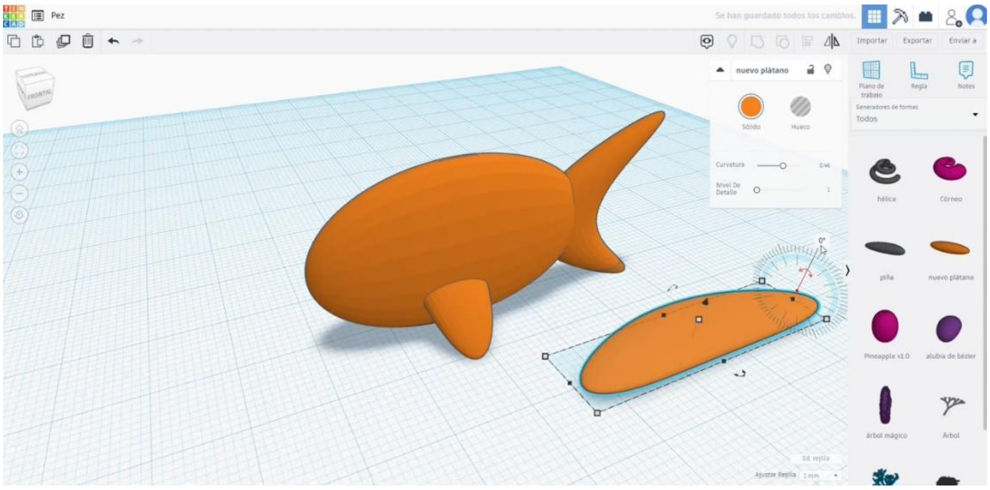


4. Turn it 90 degrees in both directions and place it like in the second picture below. Select everything and press the group button.

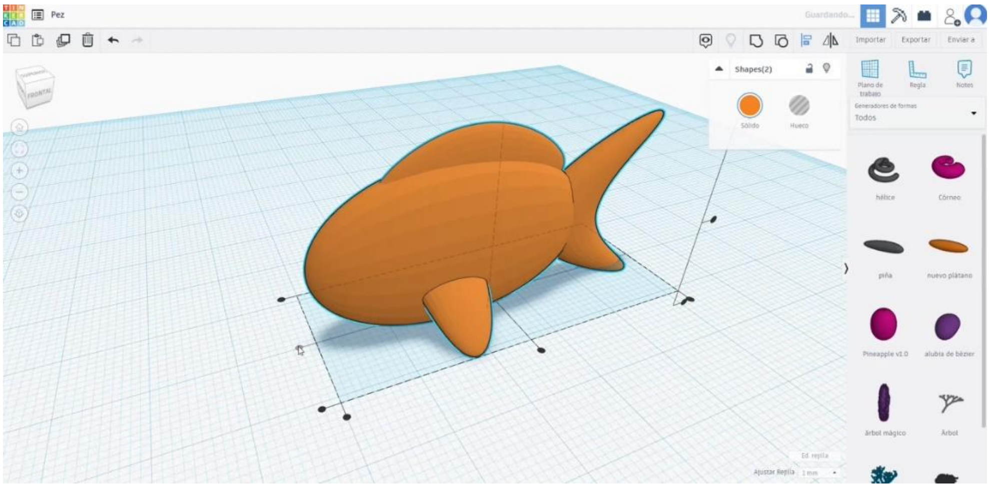




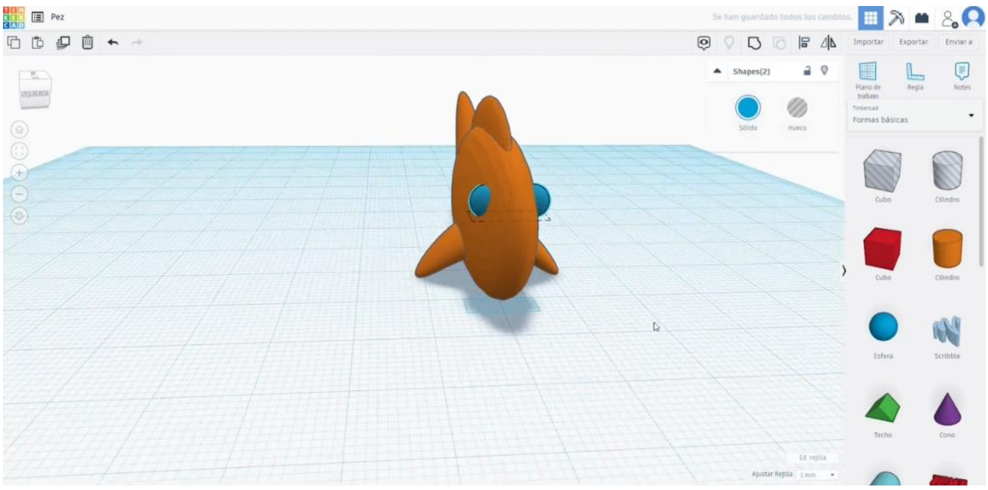
5. Use a *New Banana,* modify the curvature a Little bit and change the height to 3. Turn it 90 degrees.

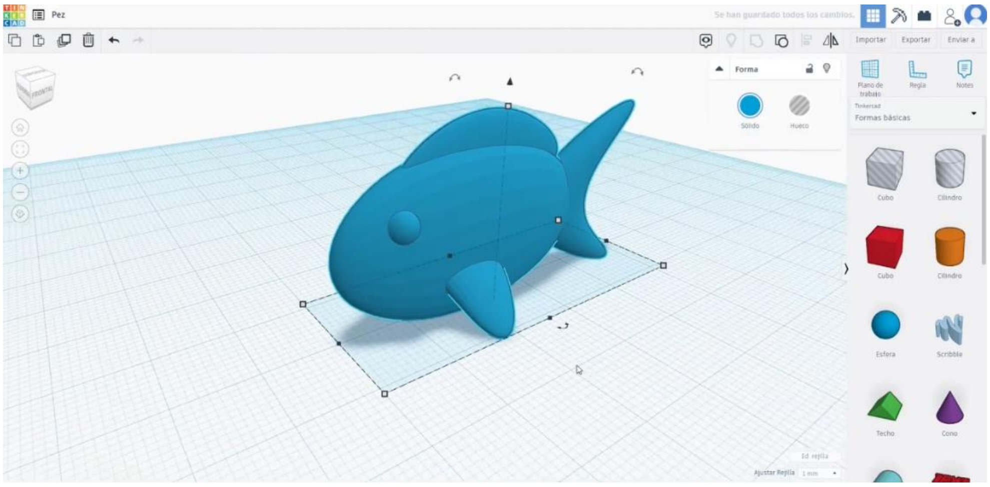


6. Place this last shape above the main body. Then align and group everything.



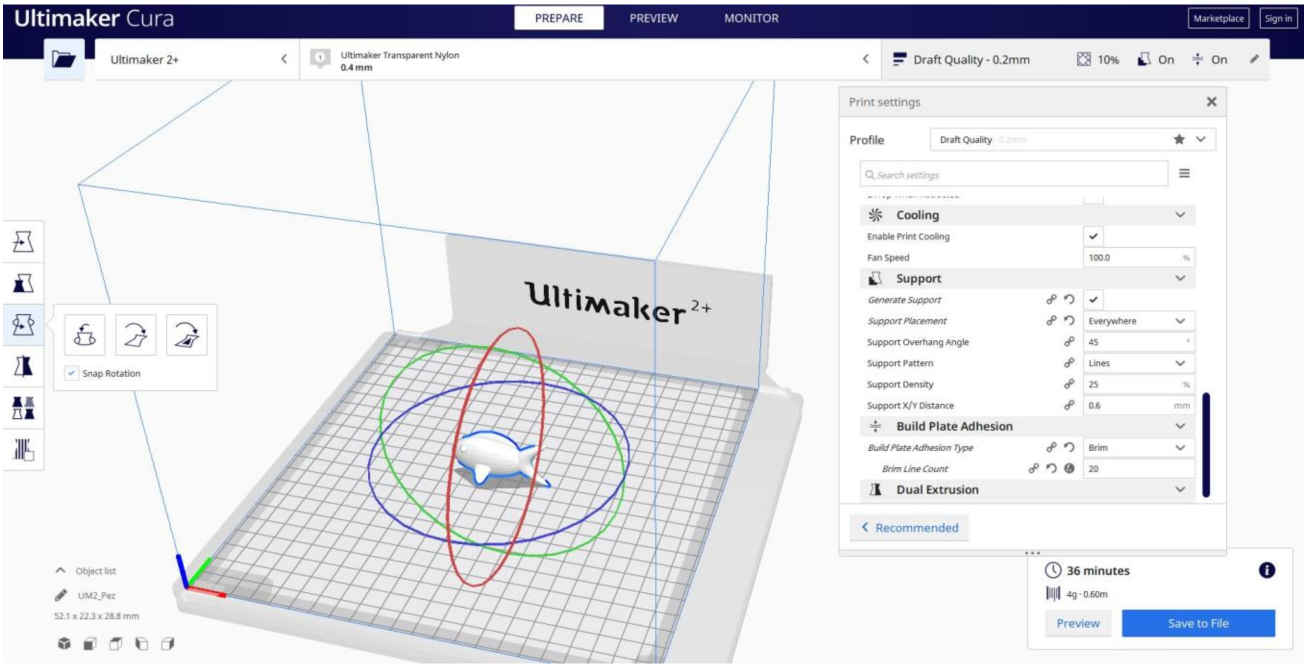
7. With a couple of spheres, make the eyes. Group them first and then align the two spheres with the main body. Select everything and group them.



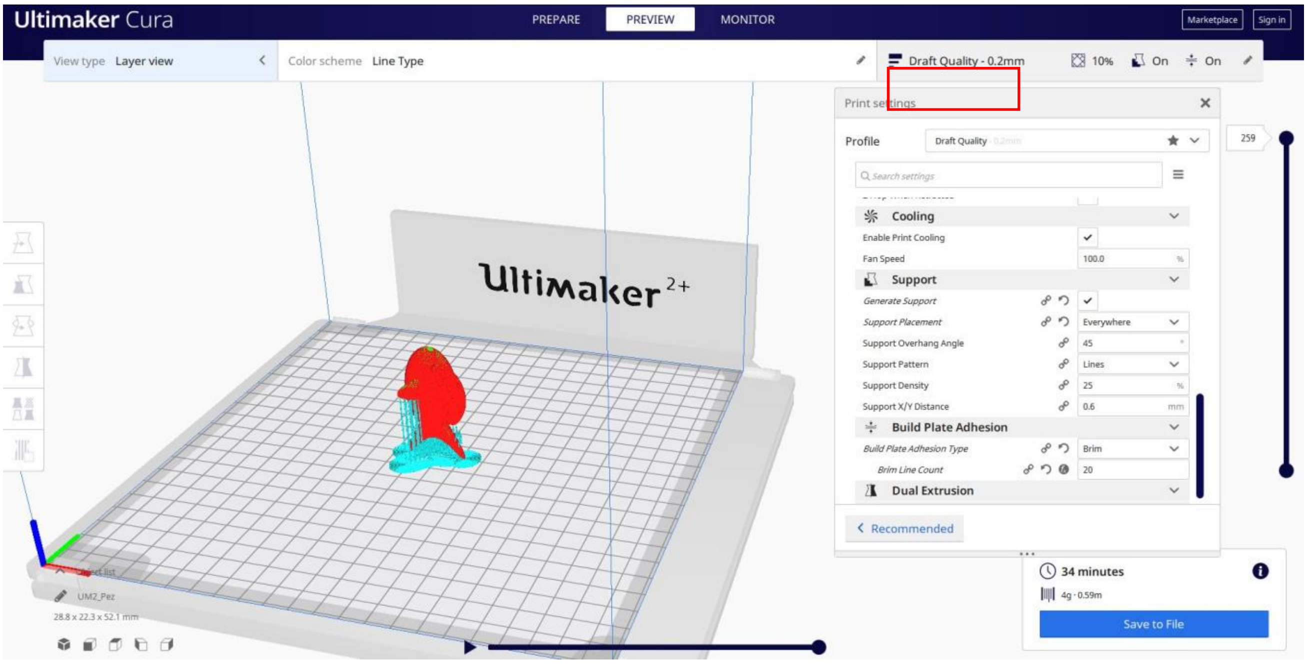


9.3.36.2 Fish 3D printing seetings

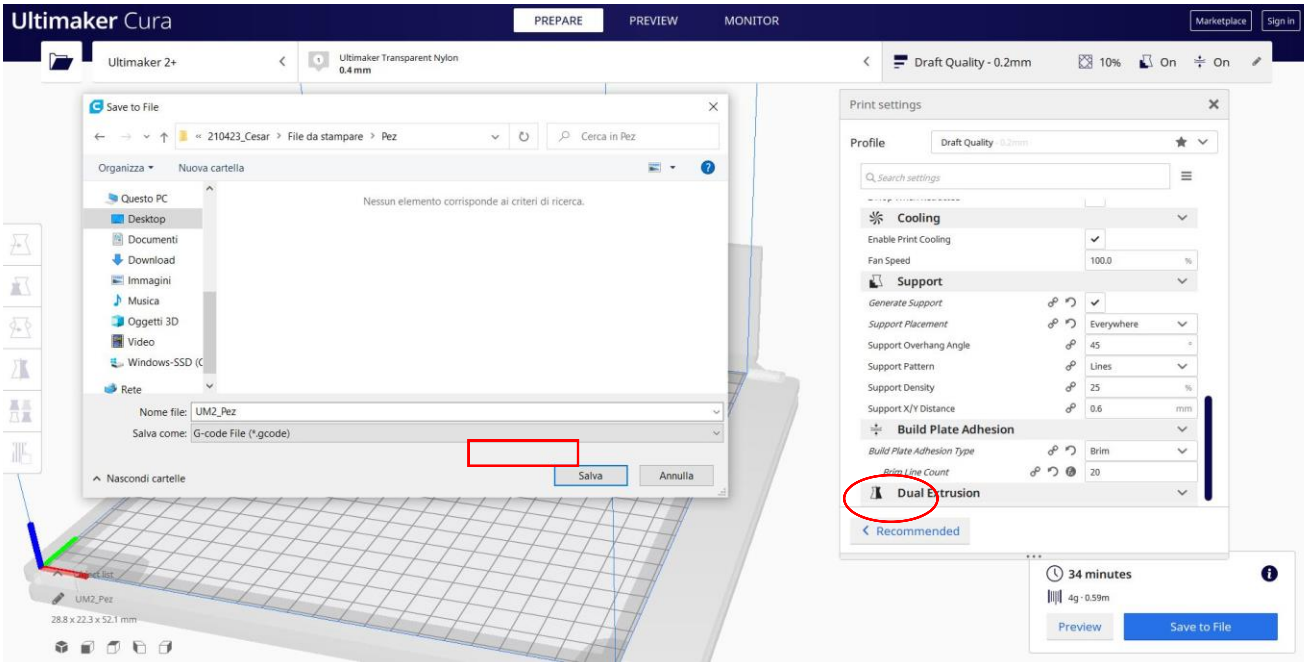
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



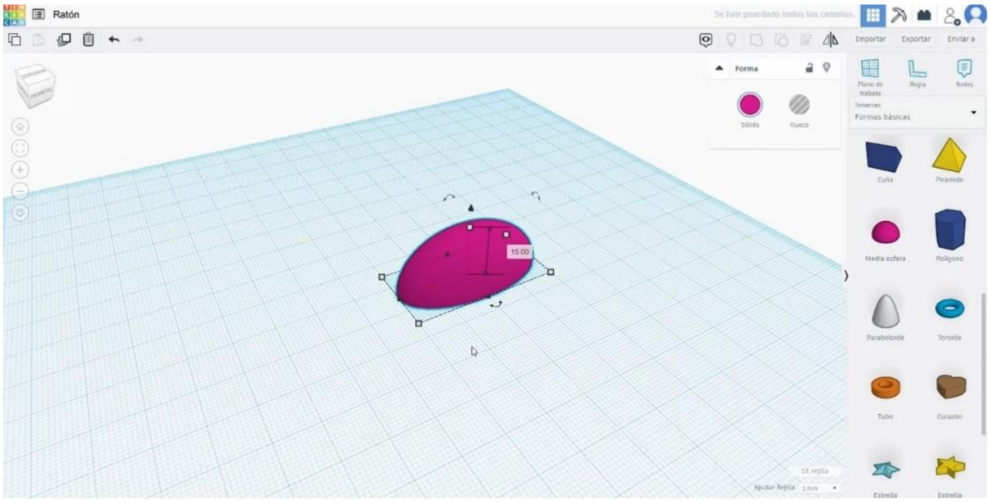
3. At this point I can save the ".Gcode" file to send to the machine.



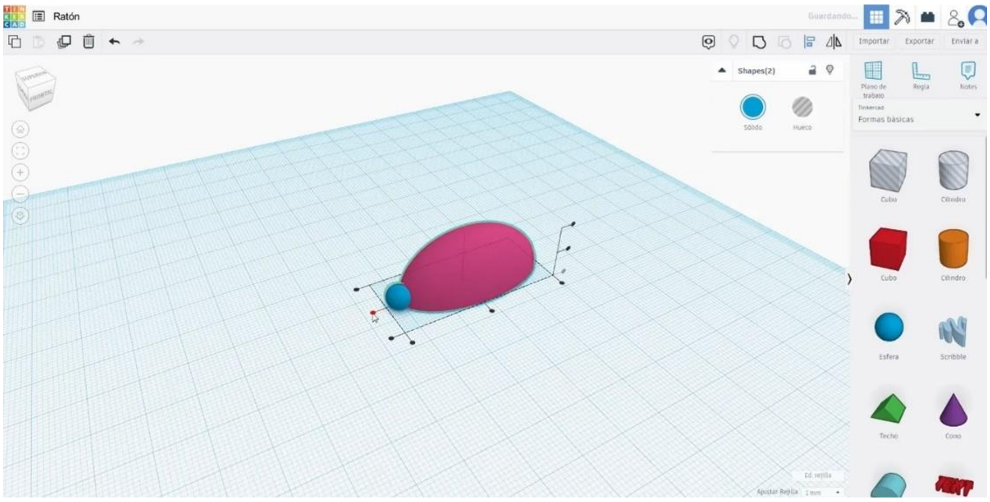
9.3.37 Part 37: Mouse

9.3.37.1 Mouse Design

1. Start with half sphere and modify the measurements to 40 x 20 x 15.

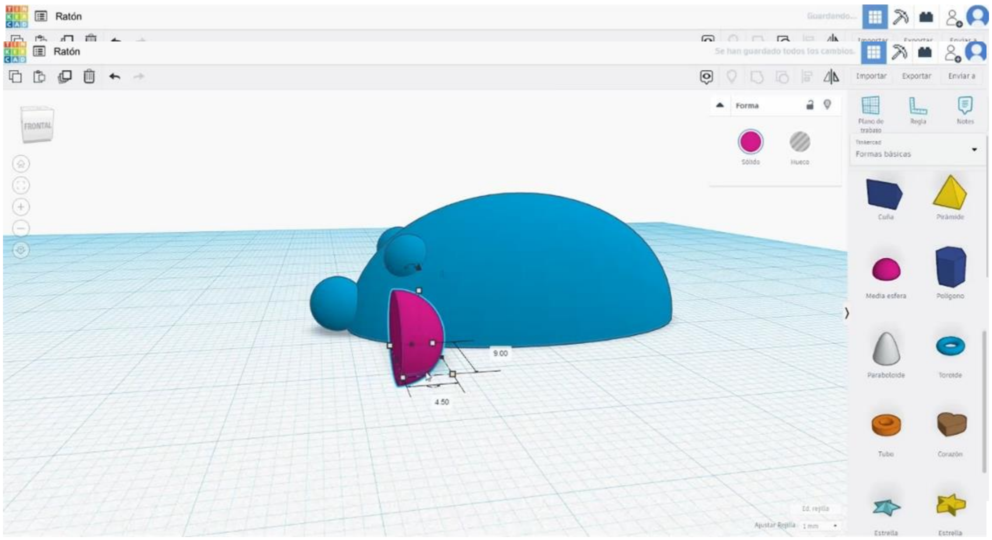


2. Add a sphere, scale it to 6 in diameter. Align it with the main body and press group button.



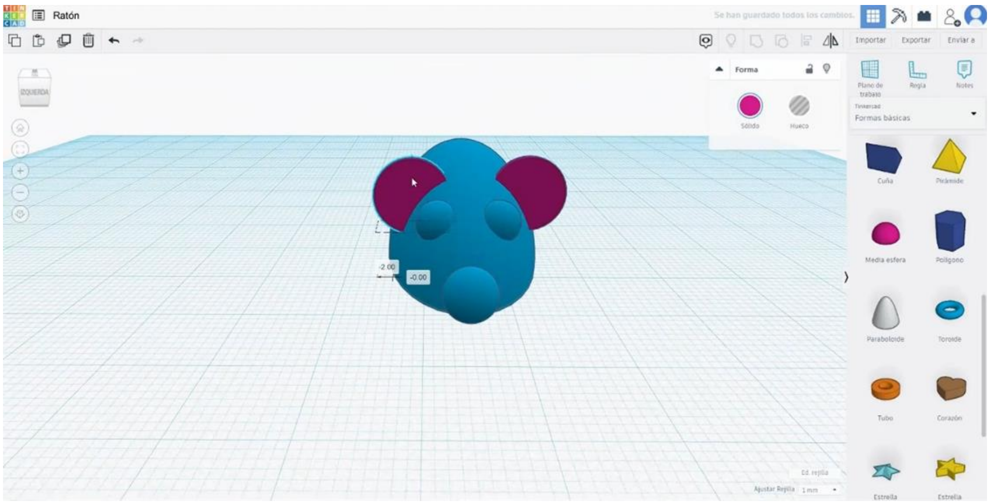
Co-funded by the Erasmus+ Programme of the European Union

3. With other sphere, changing its measures to 5 in diameter, we will do an eye. Place it in the proper place, and copy it yo do the same in the other side. Select and merge them with the other figures pressing Group.

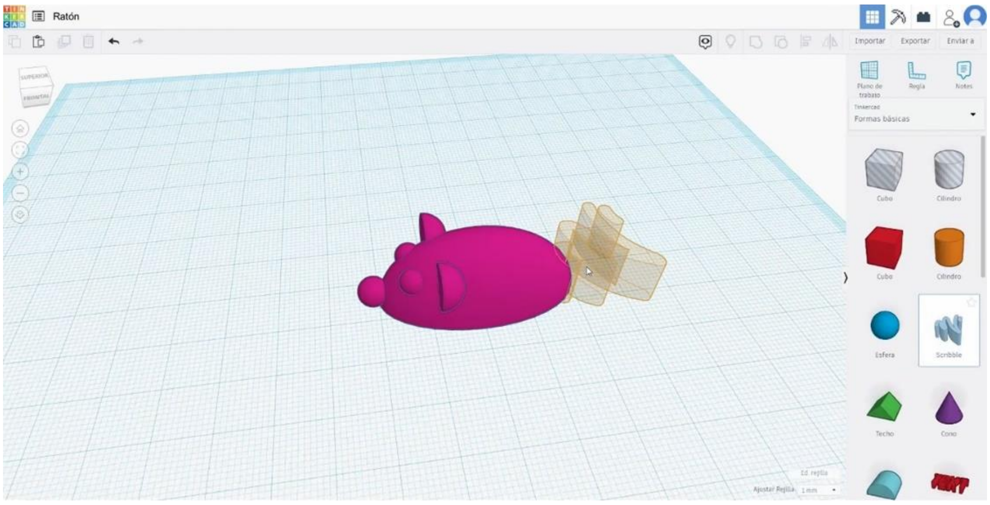


4. Build a half sphere, rotate it and scale it to 9 x 4.5. Place it behind one eye.

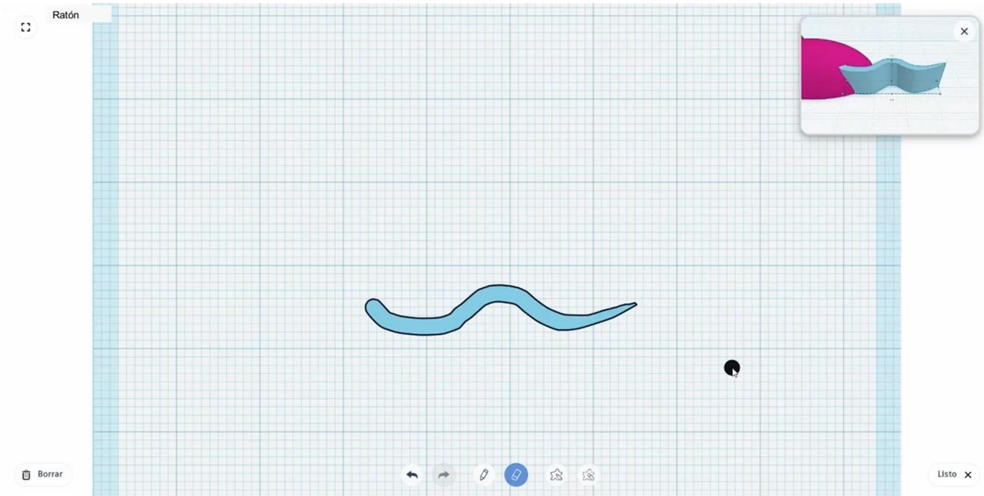
5. Copy the ear and place it in the other side. Then select them all and press group button.



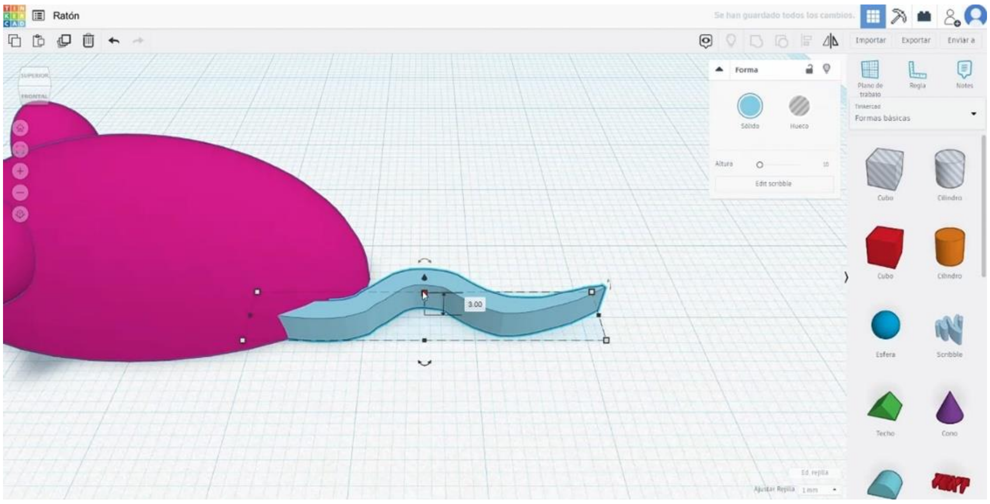
6. Create a “scribble” and place it where the tail will be.

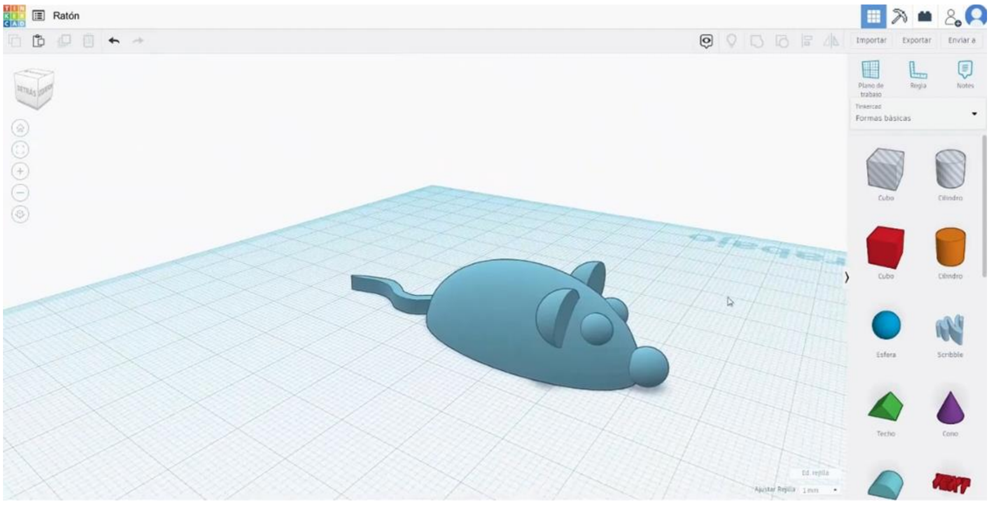


7. Then a different window will be opened. Draw the shape that you want for the tail.



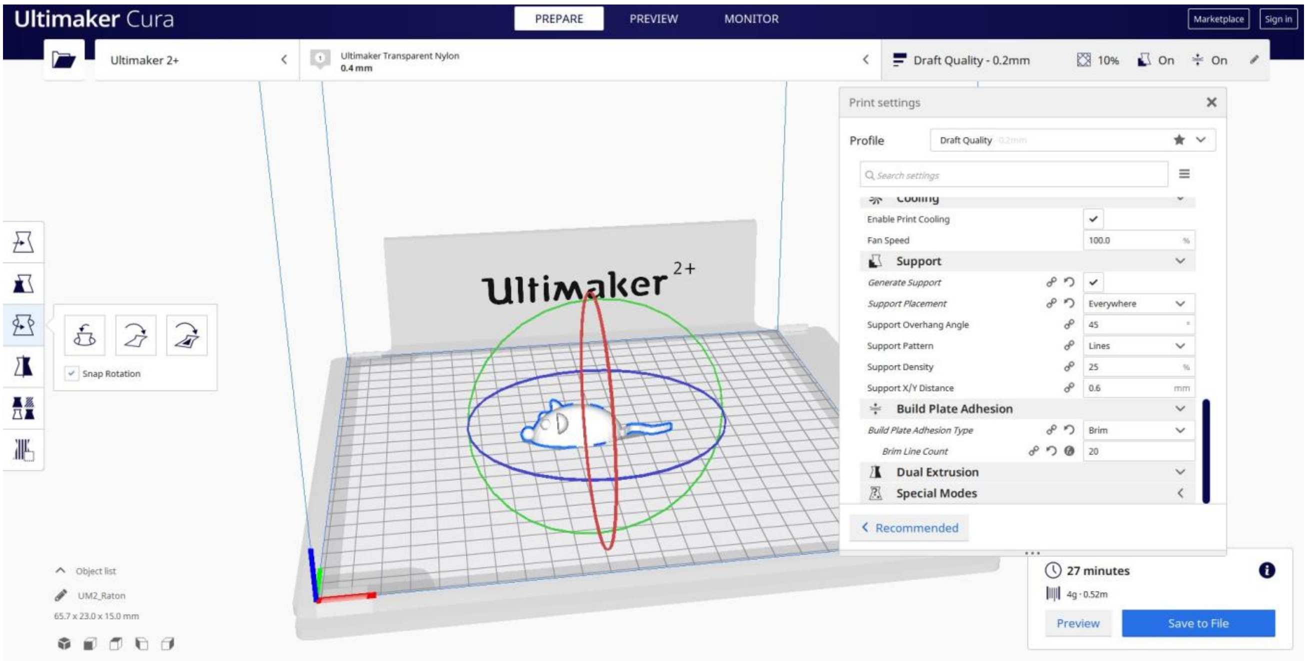
8. Change the height to 3 and place it in the center of the body. Join the tail to the body pressing group button.



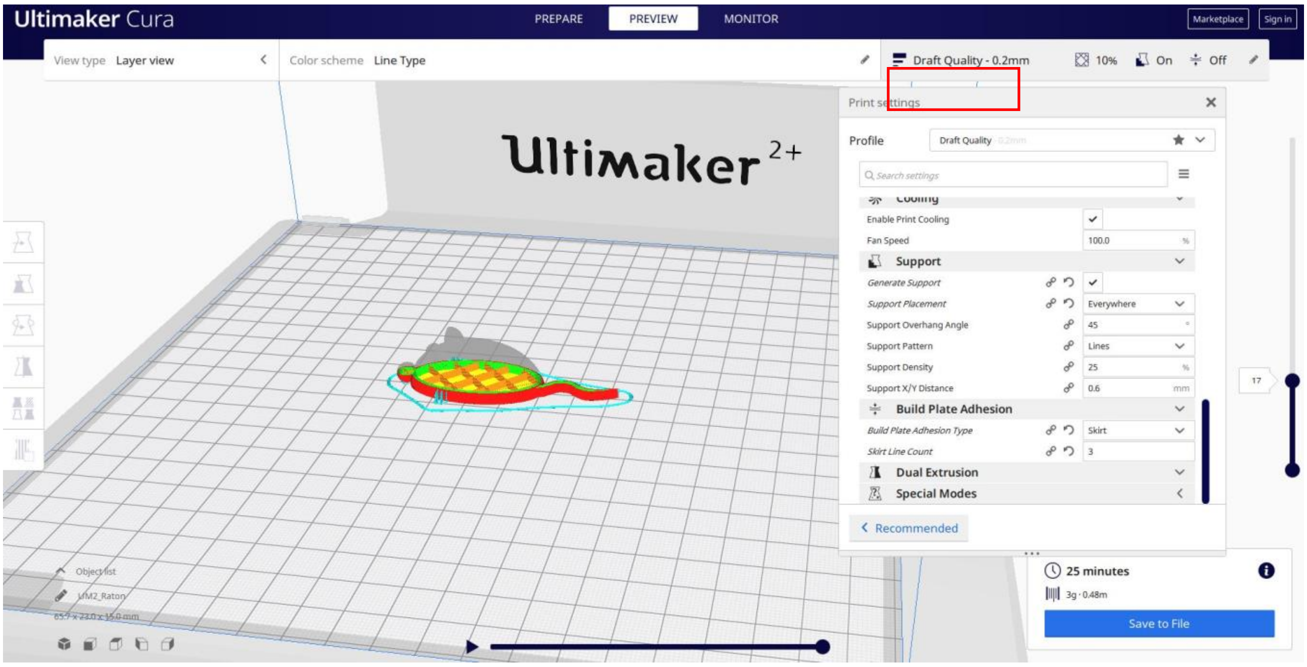


9.3.37.2 Mouse 3D printing seetings

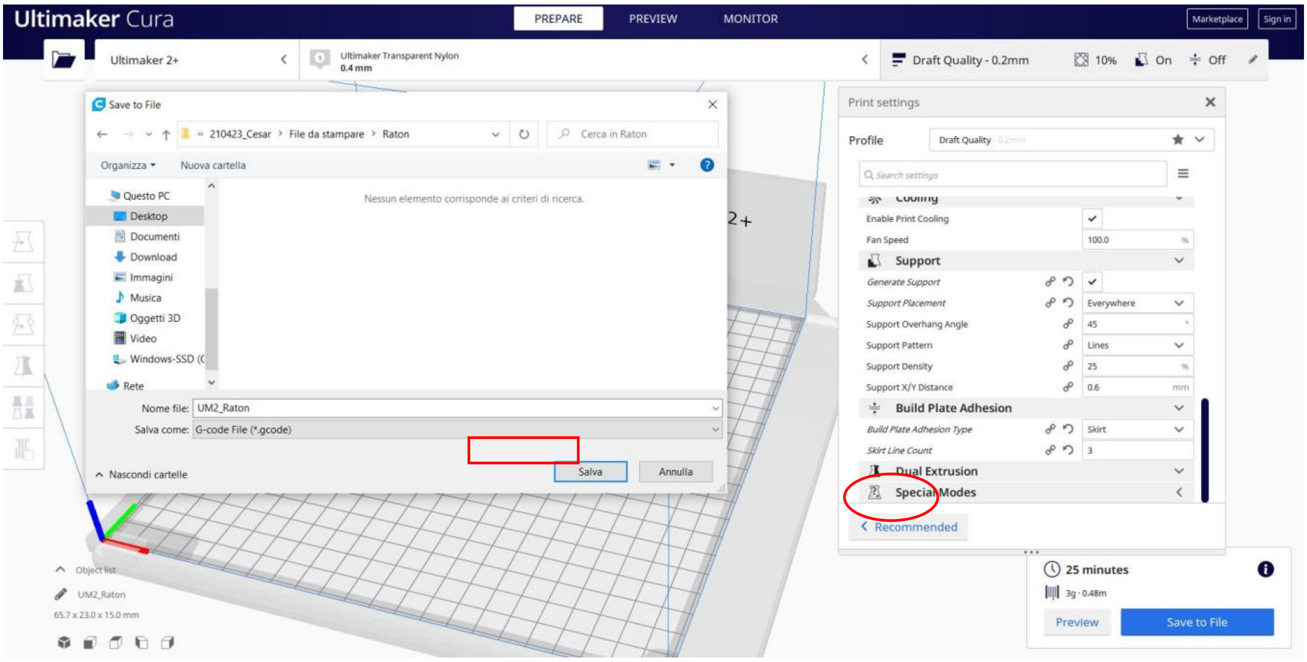
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



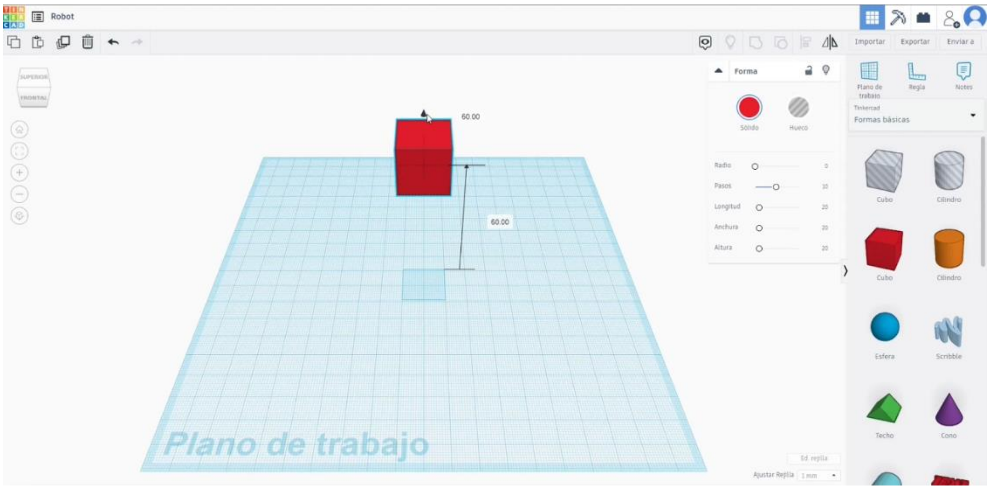
3. At this point I can save the ".Gcode" file to send to the machine.



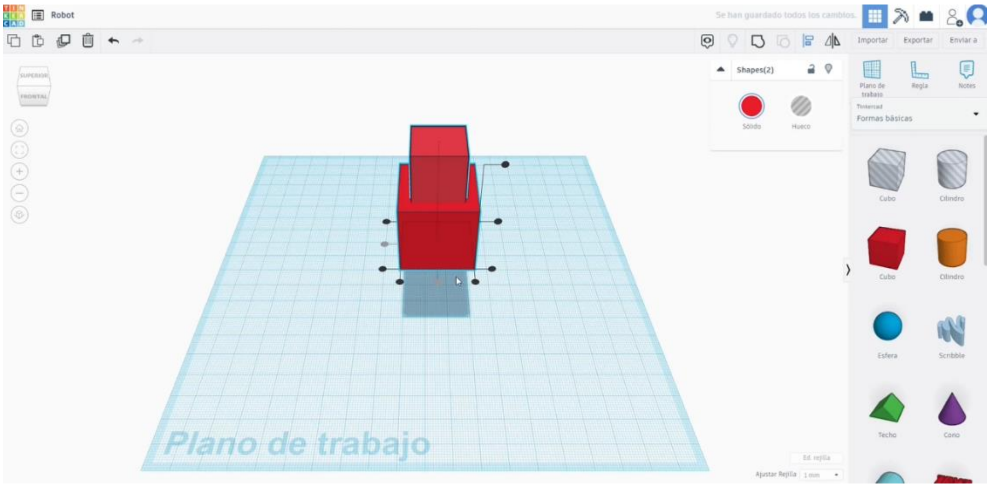
9.3.38 Part 38: Robot

9.3.38.1 Robot Design

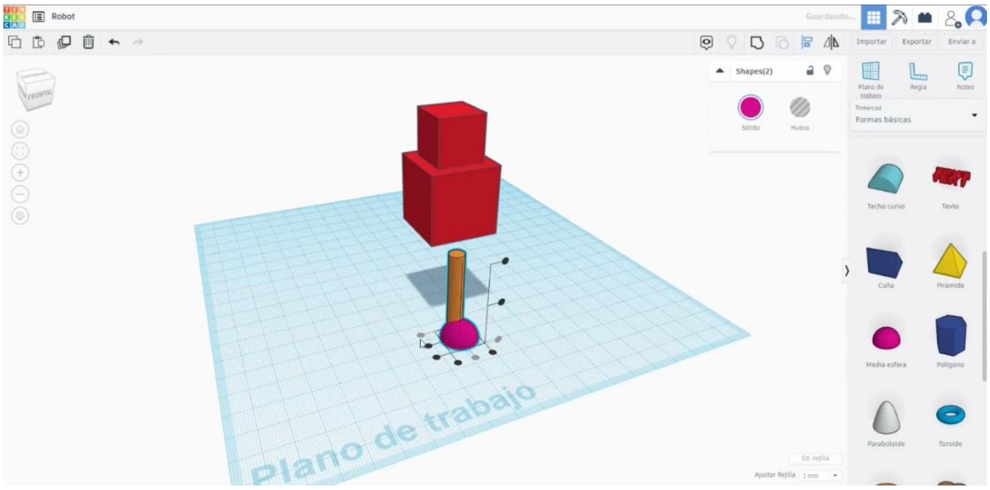
1. Build a cube, raise it to 60 high.



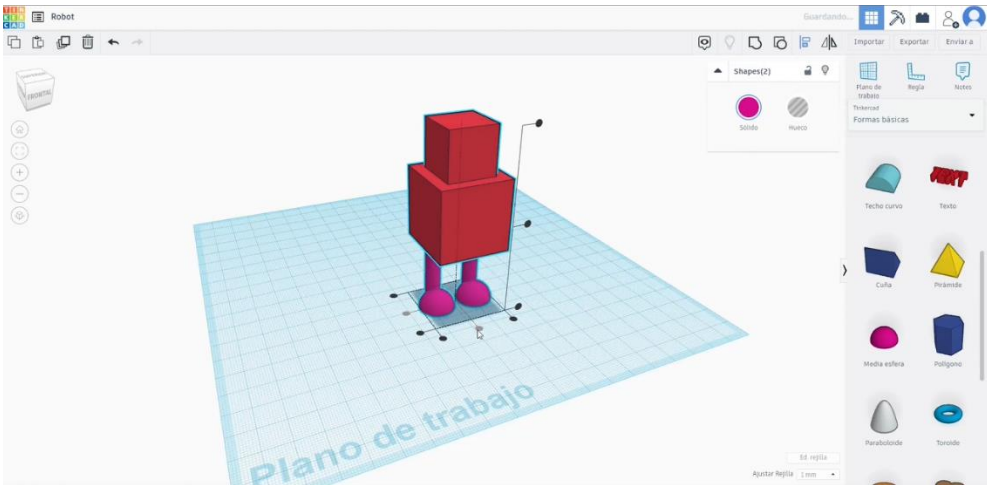
2. Add other cube (30x30x30). Put it under the other one. Raise it 30. Align both cubes and join them.



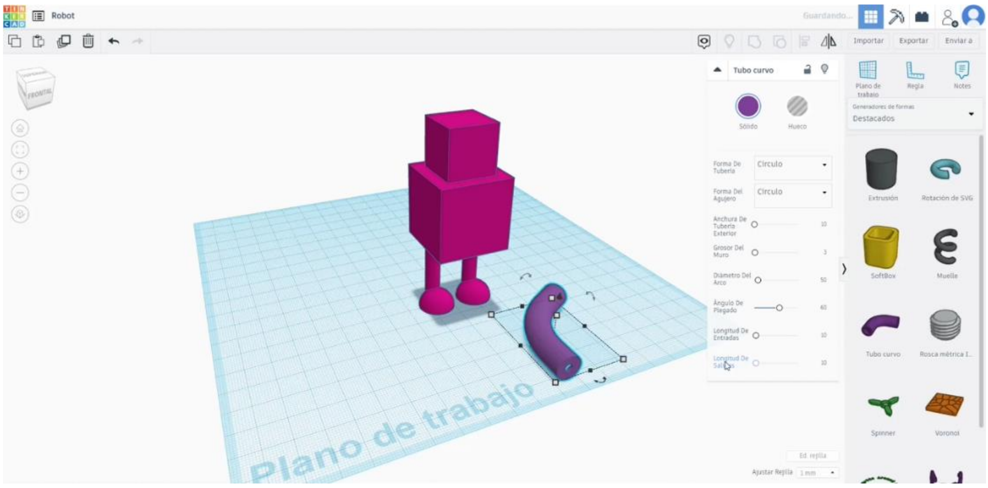
3. Make a 6x6x36 cylinder. Add a half sphere and scale it to 15x15. Then align the cylinder and the half sphere like you can see below, and press group button.



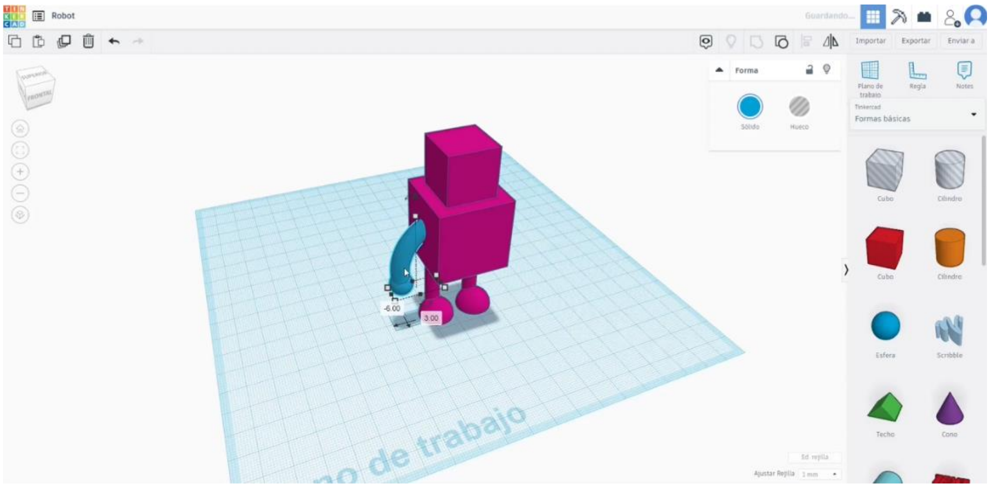
4. Create a copy of the leg, align both legs and group them. Then place them aligned under the cubes. Select all of them and press the group button.



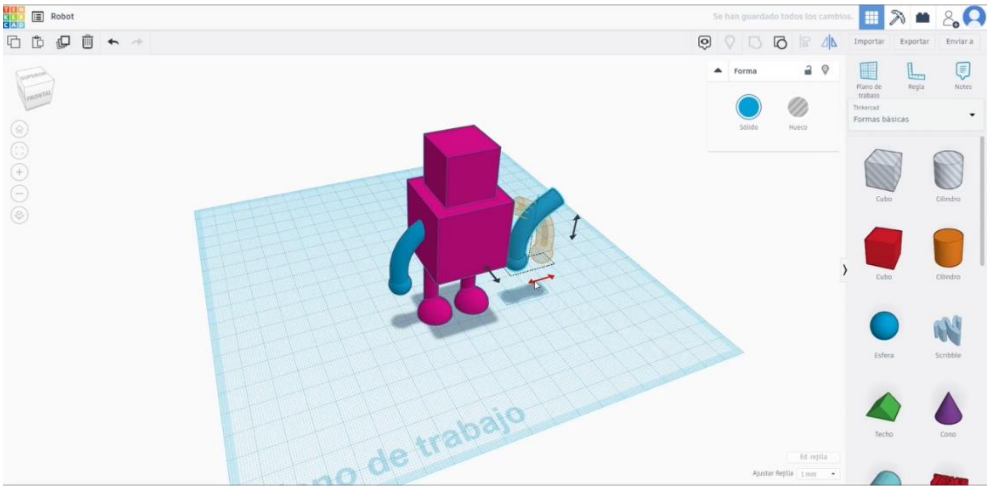
5. In Shapes generators list, find the curved tube. Set the values like in the picture below.



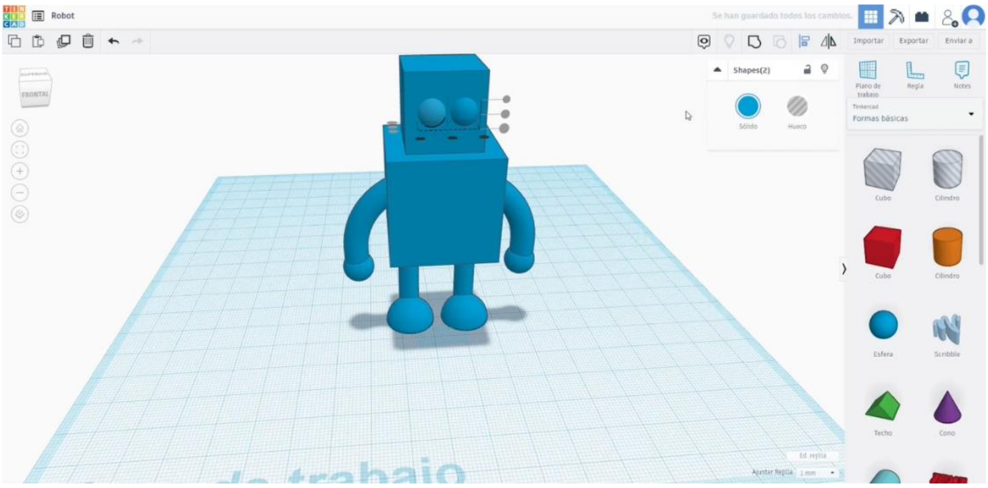
6. Built a 13 diameter sphere. Place it in the tip of the tube. Group the tube and the ball. Place it next to the bigger cube and scale it until it looks proportionate.



7. Duplicate the arm and make a symmetry. Place the new arm in the other side of the main cube. Align both arms. Then align the arms and the body. Select everything and press group button.

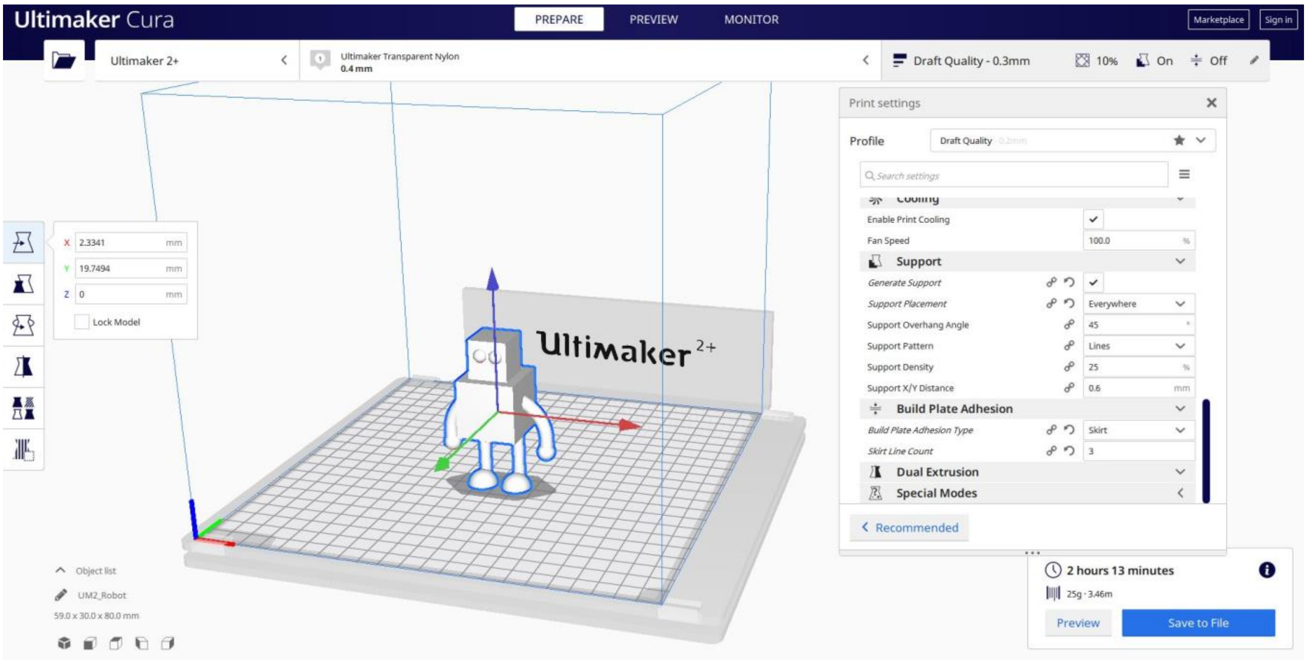


8. Use a couple of small spheres to create the eyes. Group them with the rest of pieces.

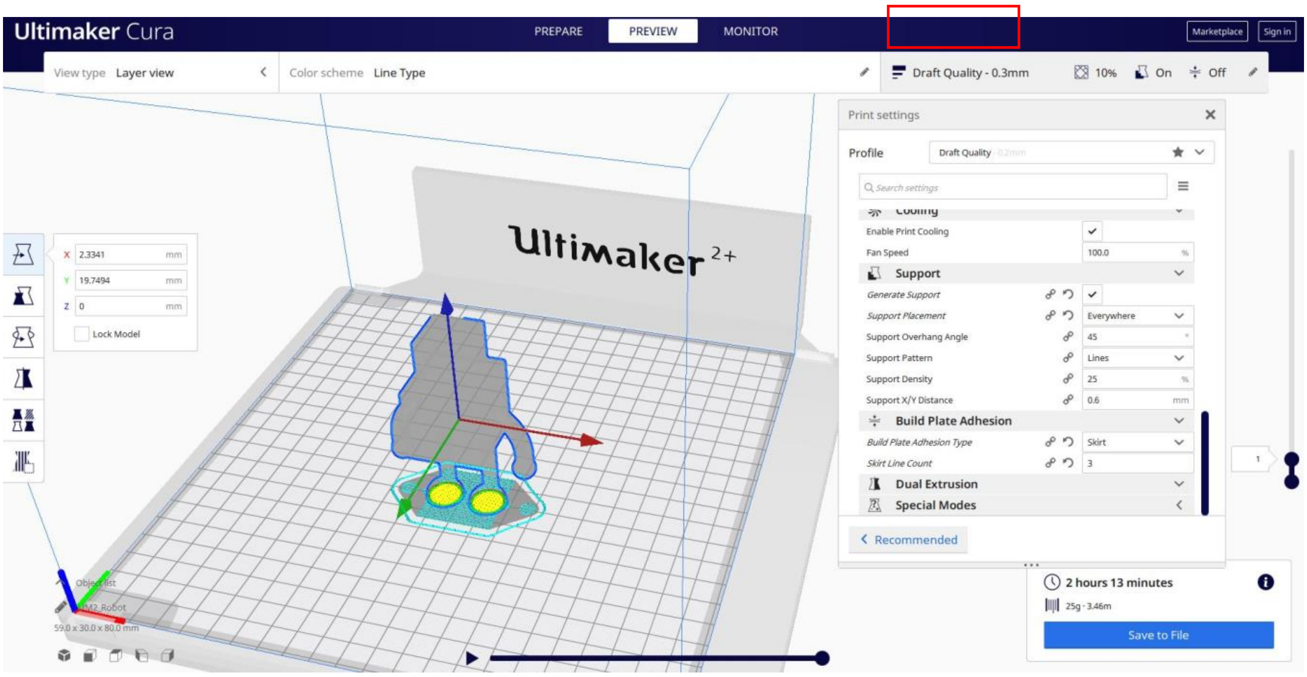


9.3.38.2 Robot 3D printing seetings

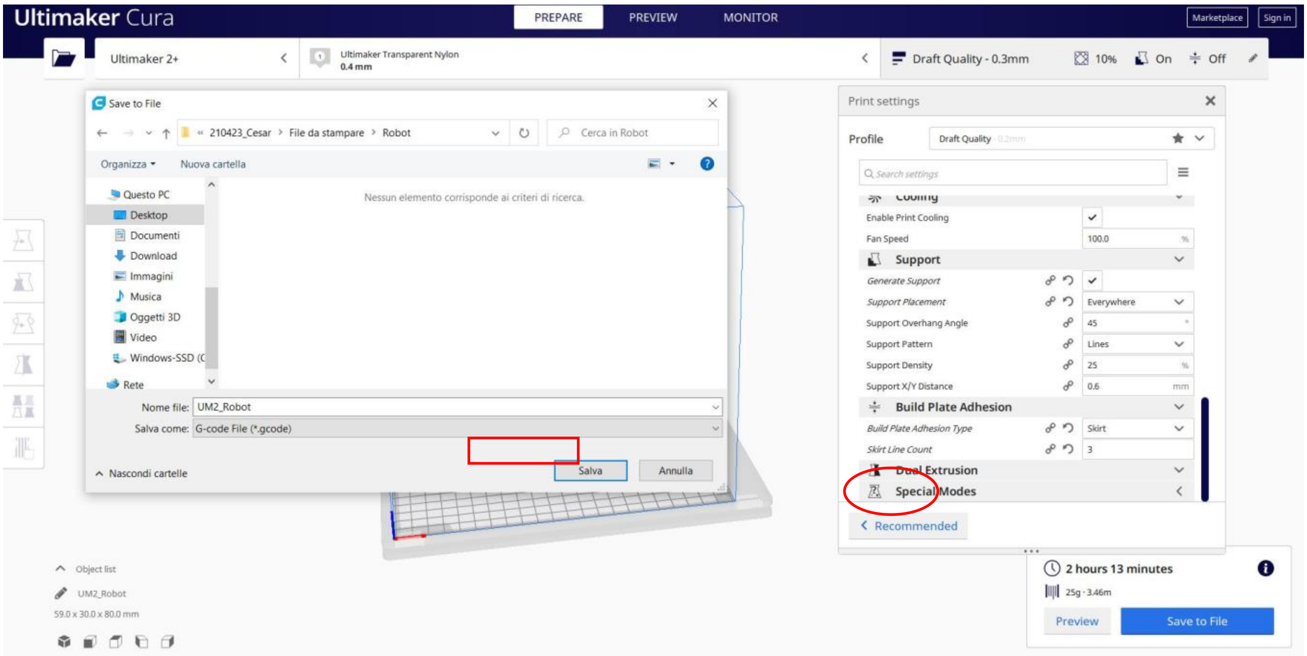
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. Enter the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



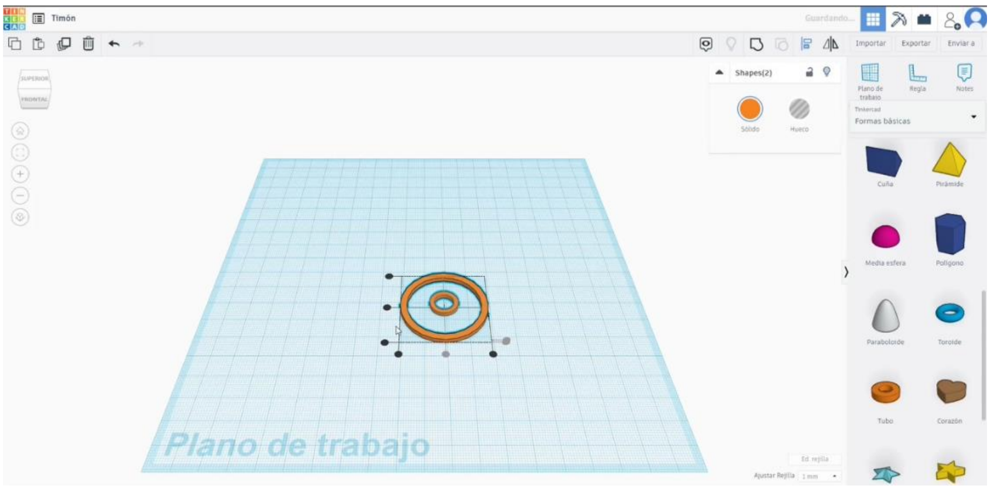
3. At this point I can save the ".Gcode" file to send to the machine.



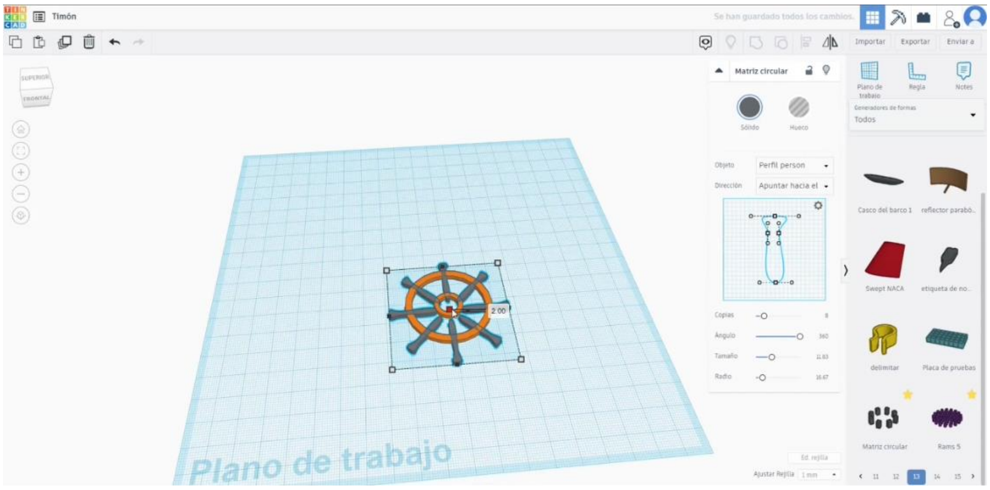
9.3.39 Part 39: Tiller

9.3.39.1 Tiller Design

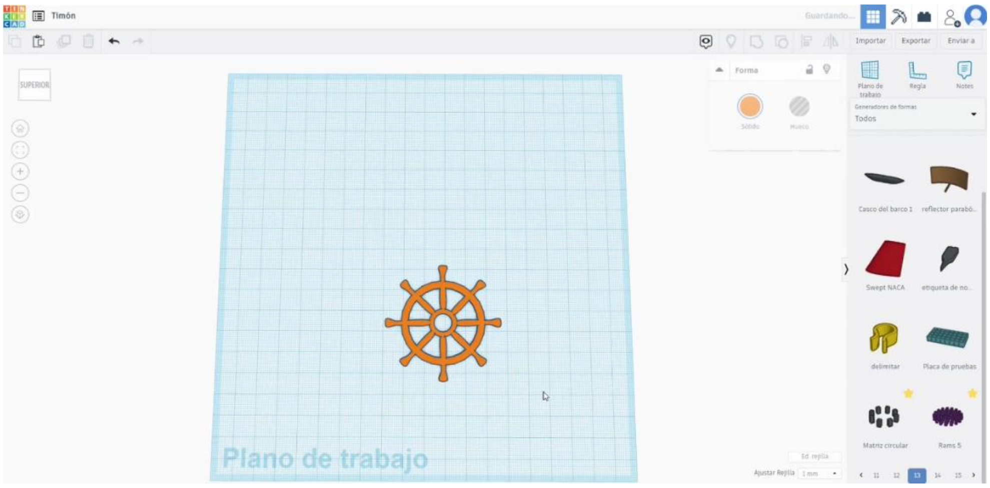
1. Build a 40x40x2 tube. Copy it and make it smaller. Align them.



2. Add a circular matrix. Choose the personalized profile and draw something like in the image below. Set the size and radius to get a proportionate design.

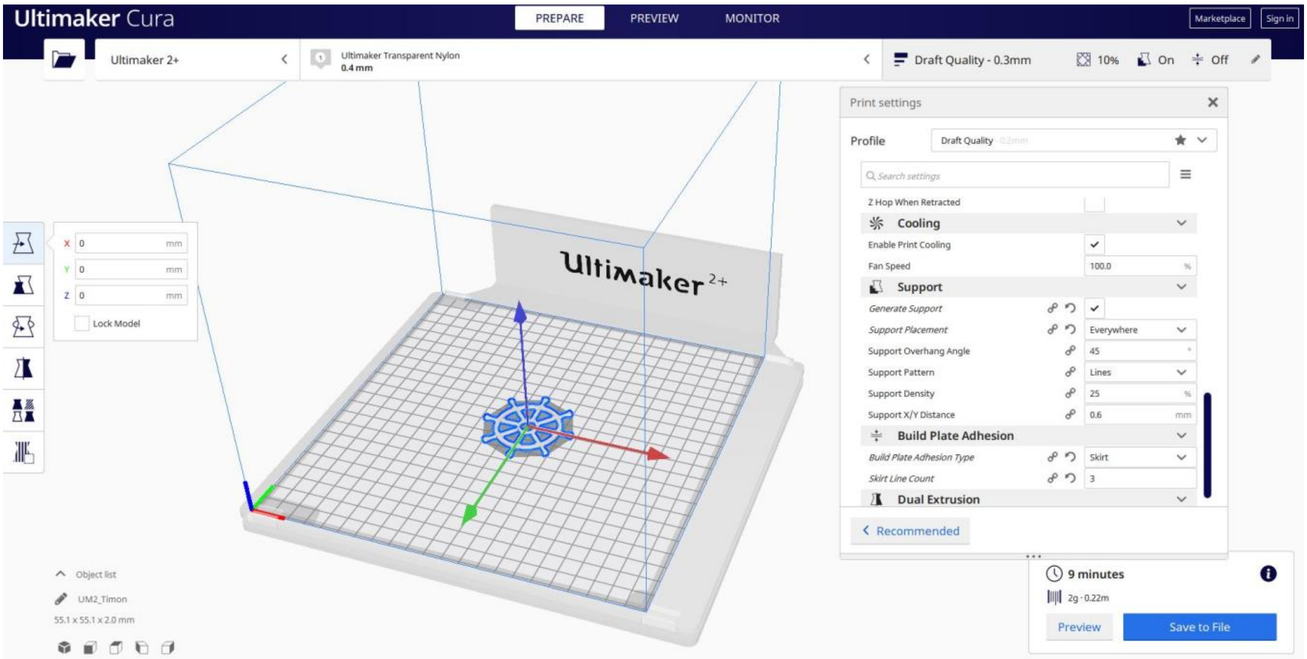


3. Select the matrix and the two tubes and pres the align button. Finally, press the group button.

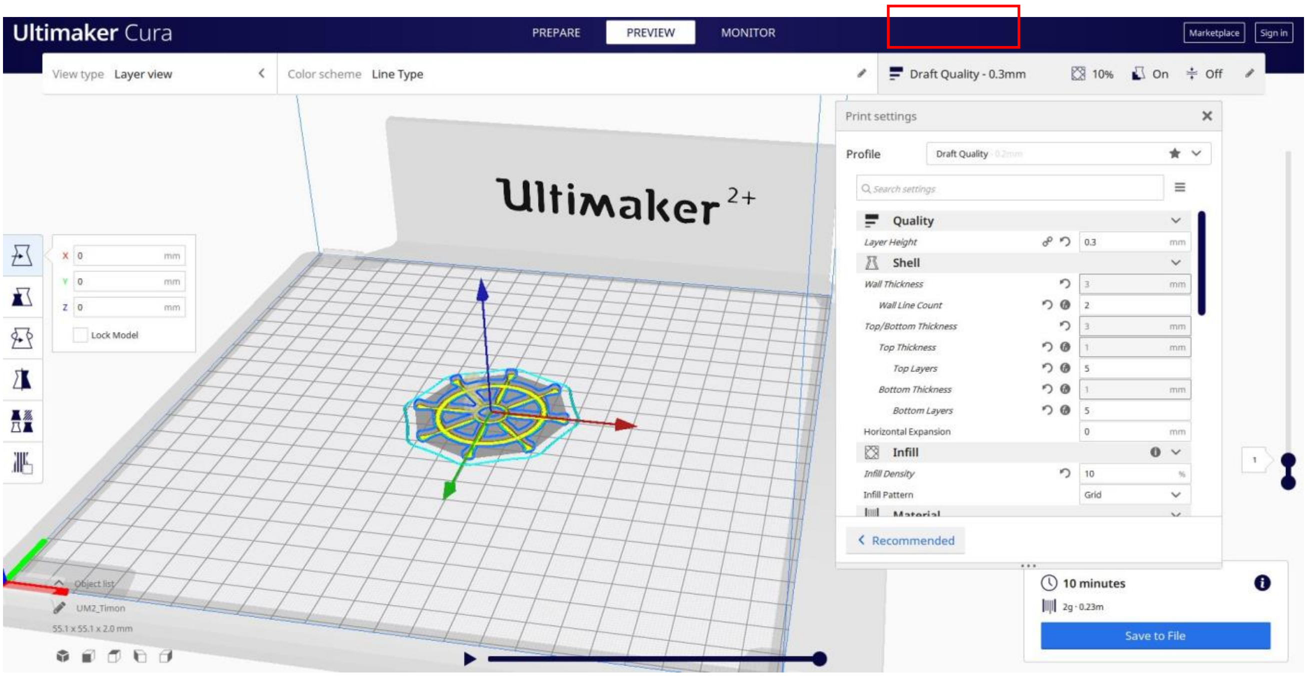


9.3.39.2 Tiller 3D printing seetings

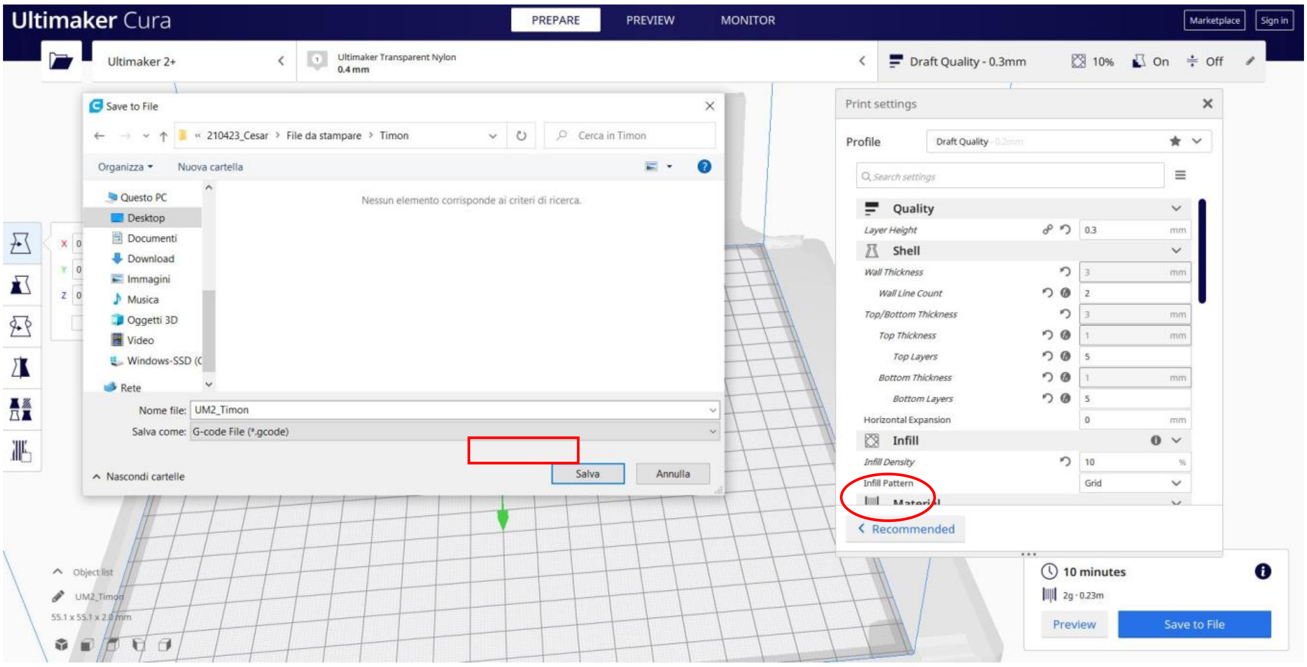
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



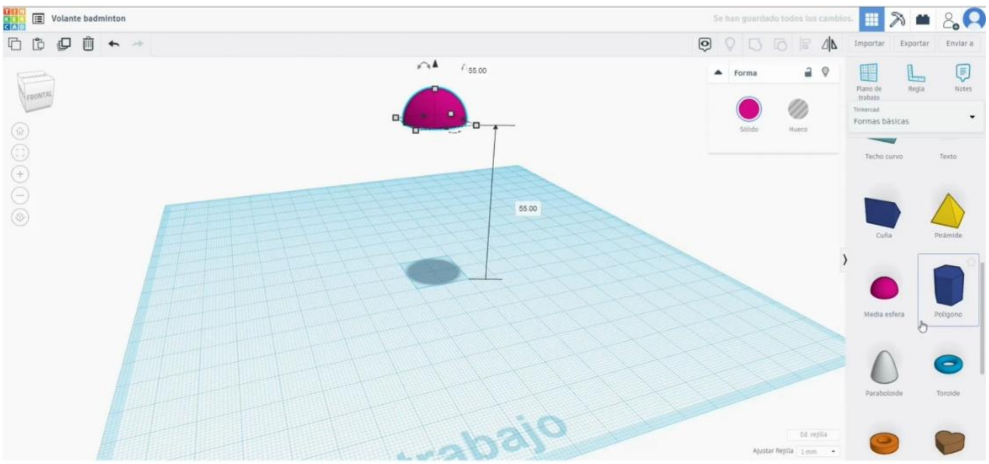
3. At this point I can save the ".Gcode" file to send to the machine.



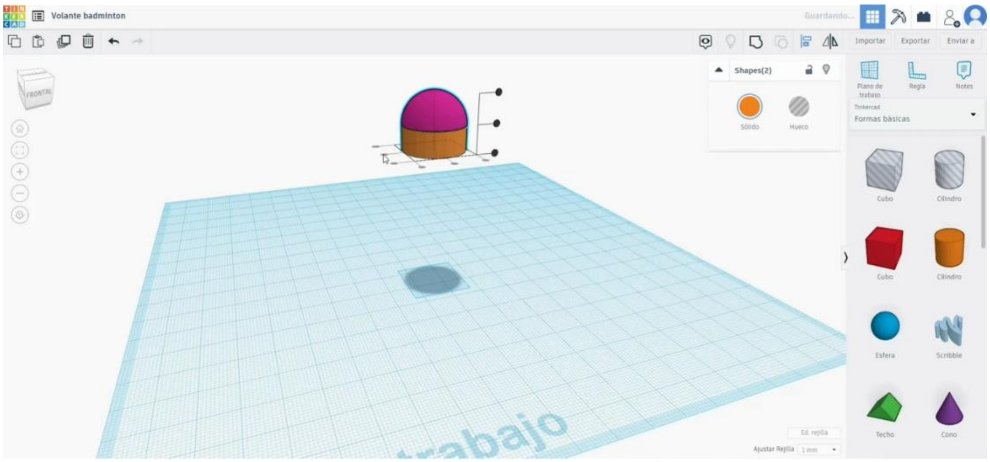
9.3.40 Part 40: Badminton shuttlecock

9.3.40.1 Badminton shuttlecock Design

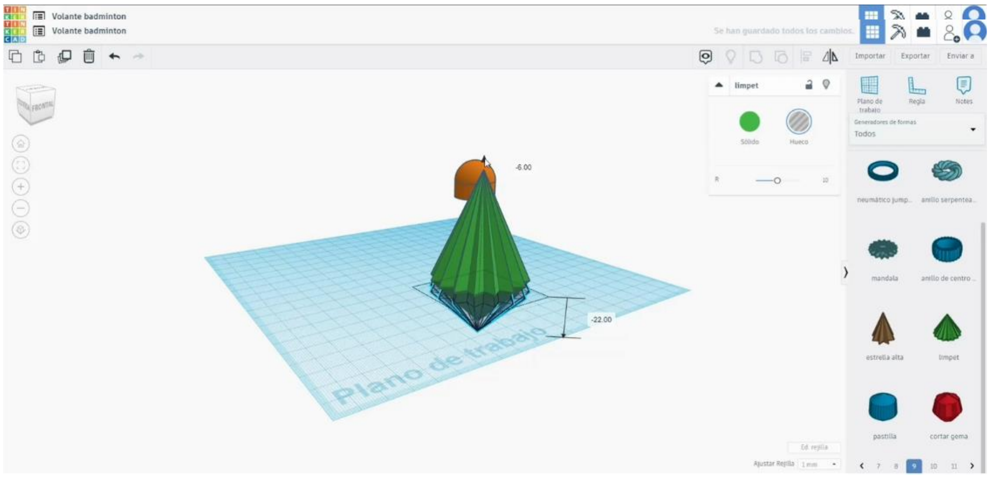
1. Build a 20x20x10 half sphere. Raise it to 55.



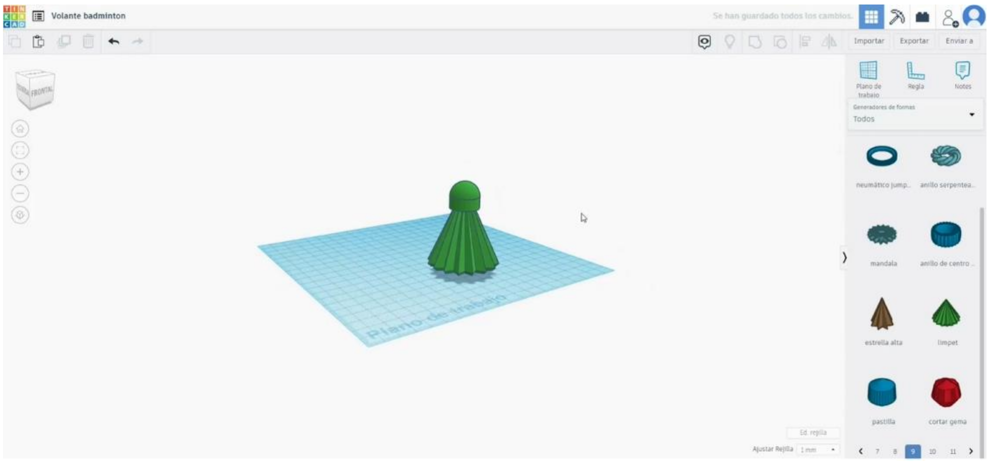
2. Add a 20x20x8 cylinder, raise it to 47. Align it with the half sphere and group them.



3. In the shapes generators, find the *Limpet*. Change the measurements to 50x50x78. Copy it and place the new copy aligned with the other but a little lower. Turn it into hole mode and group them.

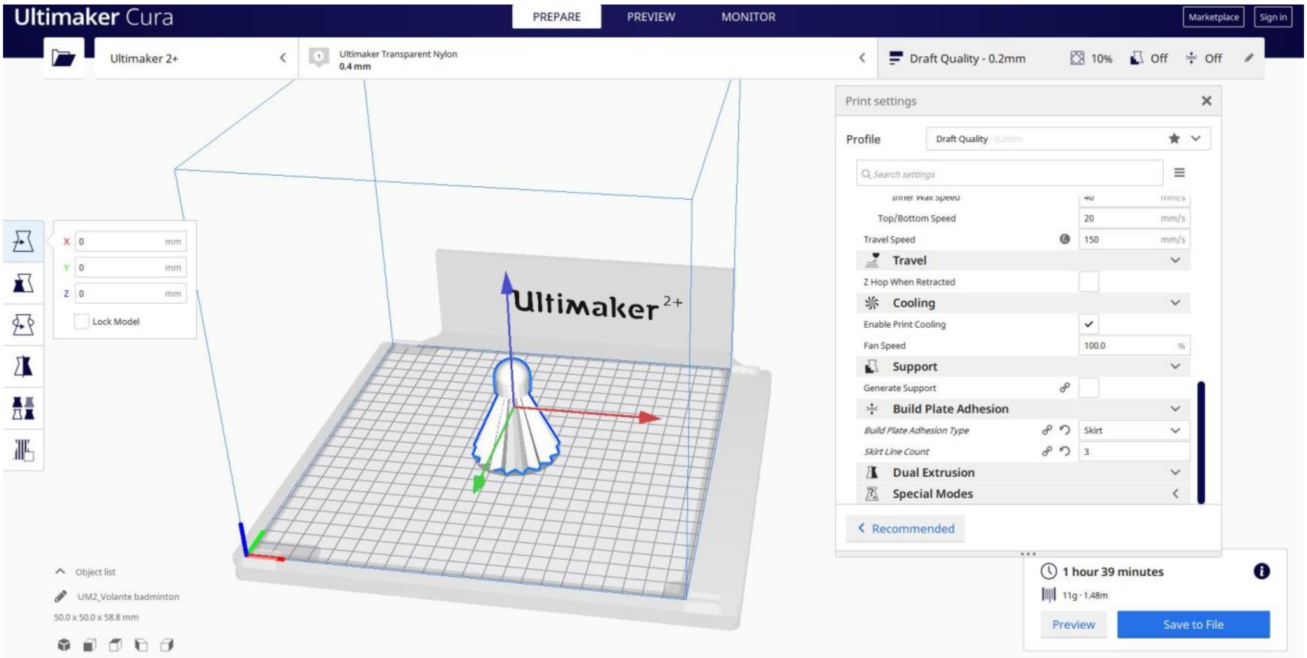


4. Align the result with the other shapes and group them.

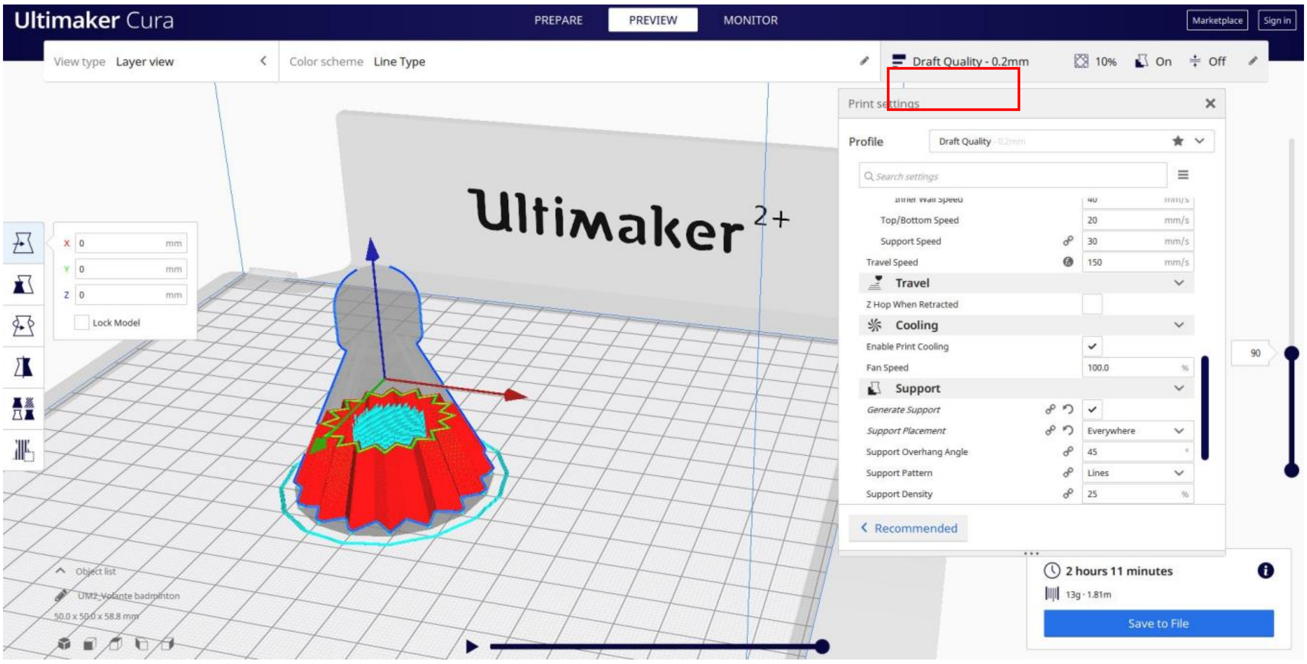


9.3.40.2 Badminton shuttlecock 3D printing seetings

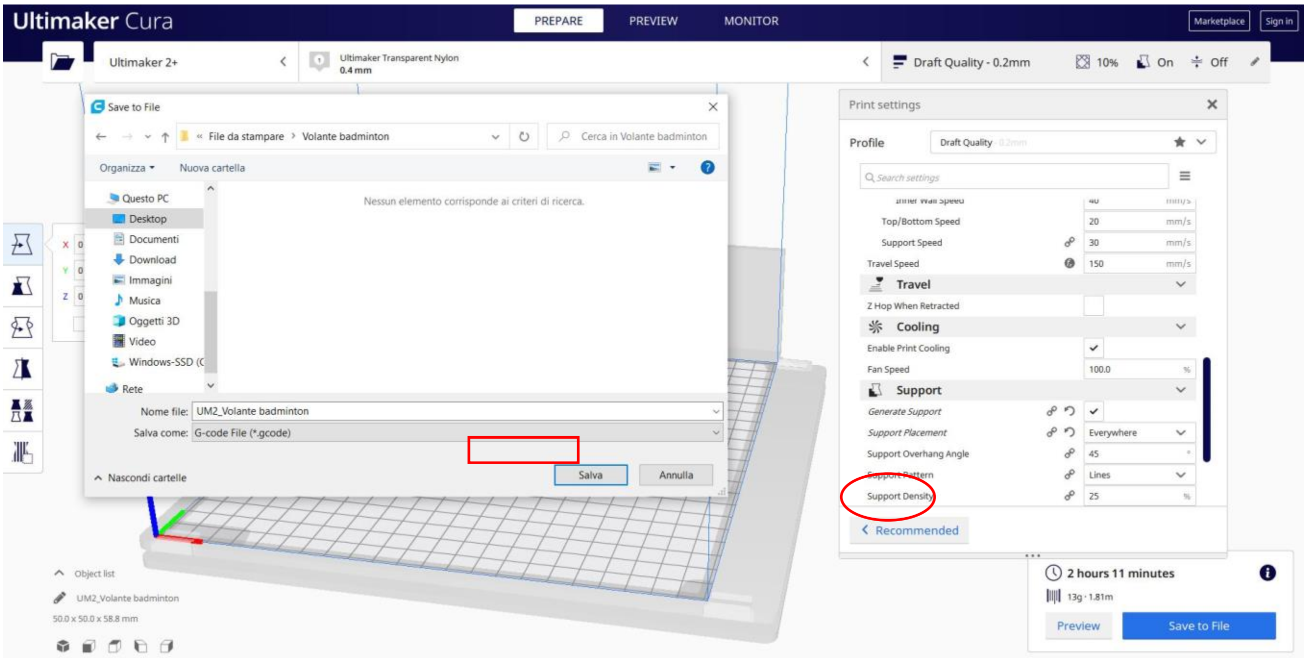
1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.



2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"



3. At this point I can save the ".Gcode" file to send to the machine.



Co-funded by the Erasmus+ Programme of the European Union

10 STL Parts & Videos & GCode & Downloads

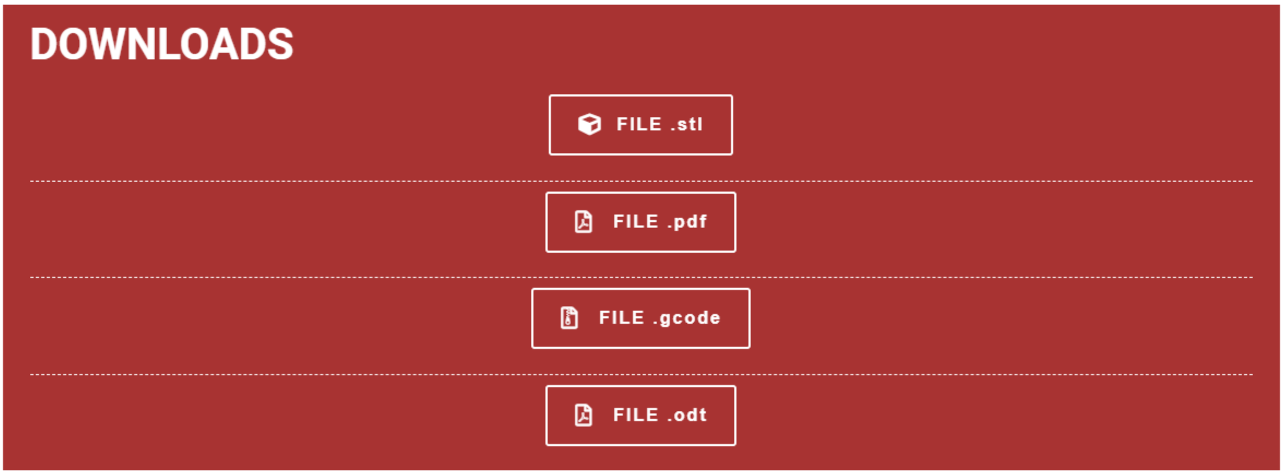
This manual is a compilation of all the educational materials created and which are collected both on the project website (www.robot3dp.eu) and on the ROBOT@3DP e-learning platform (https://elearning.robot3dp.eu/) where you can learn more about the technologies.

In the web section, in addition to viewing the videos of the design of each part, the settings for 3D printing and the video of how the part is printed, it is also possible to download all the related files: the 3D model in STL format, prepared for printing.

The file in pdf format with the appropriate printing parameters defined by the experts and tested in its manufacture.

There is also the GCode file which is used by the 3D printer and, finally, the design file of the model.

Download area: https://www.robot3dp.eu/training-3d-models/



In addition to the website and the e-learning platform, you can watch all the videos on the ROBOT@3DP YouTube Channel:

https://www.youtube.com/channel/UCot2SwGyRfuQWA8ZATg344g/videos

Video Lists:

- ROBOT@3DP Part Design Videos: https://www.youtube.com/watch?v=GAi-YLivYHo&list=PL-stpbVChp9KwbeZvKnkbLqT_p4ypNsFY
- ROBOT@3DP STL Printing Settings Videos: https://www.youtube.com/watch?v=cP7o51WNFRU&list=PL-stpbVChp9J3YbcH_BuliSnKzZCiF1-a
- ROBOT@3DP Part Printing Videos: https://www.youtube.com/watch?v=xVx4a3EB9JU&list=PL-stpbVChp9I5UrN0XnODS8VkzfJ601re