

FUSED FORM

---



# USER MANUAL

YOU'VE  
GOT THE  
POWER

# CONTENT



---

## 04

GENERAL SAFETY  
INFORMATION

---

## 05

MAIN COMPONENTS

---

## 07

FIRST STEPS

---

## 10

LOADING FILAMENT

---

## 12

3D PRINT

---

## 15

SLICER SOFTWARE

---

## 23

TECHNICAL SUPPORT

# Welcome !!!

Your machine has been designed and created by specialists and meets the highest quality standards. 🙌

The 3d printed tests were successful and we are sure that your prints will be incredible!

Our continuous improvement allows us to be leaders in the market and to be at the forefront of technology, always seeking to contribute to the industry.

For your best experience with our products, it is important that you make sure to complete the following step by step before printing:



1. Read the manual.
2. Watch the tutorial videos of the manual.
3. Make sure the power supply is stable (avoid voltage spikes).
4. Read the warranty policy.
6. Use the technical support email for questions.



*We are glad that you are  
part of our family*



## GENERAL SAFETY INFORMATION



Place the location where the 3D printer will be placed:

- Separated from direct air currents.
- Electric outlet nearby and regulated (if possible).
- Place it on a flat, rigid surface that prevents the machine shake
- Make sure the space has good air renewal

Use safety glasses when operating

Keep young children supervised when they are near the printer.

*DO NOT Touch the Nozzle or the Heated Bed during the printing process.*

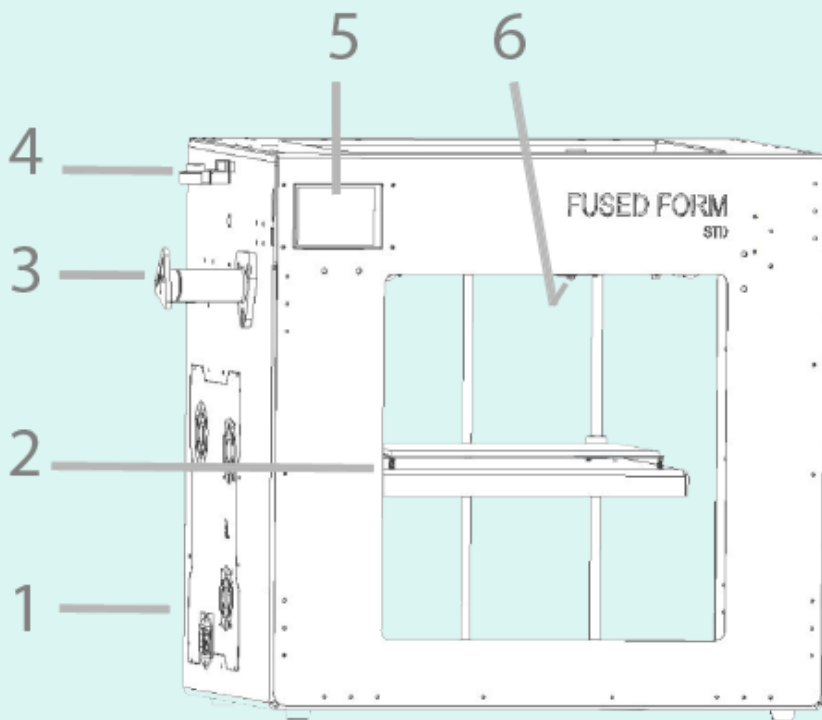
*These parts are at high temperatures and can cause burns.*

Check with the manufacturer first before removing or modifying any connections

Keep away liquids and flammable materials in electronic parts



# MAIN COMPONENTS

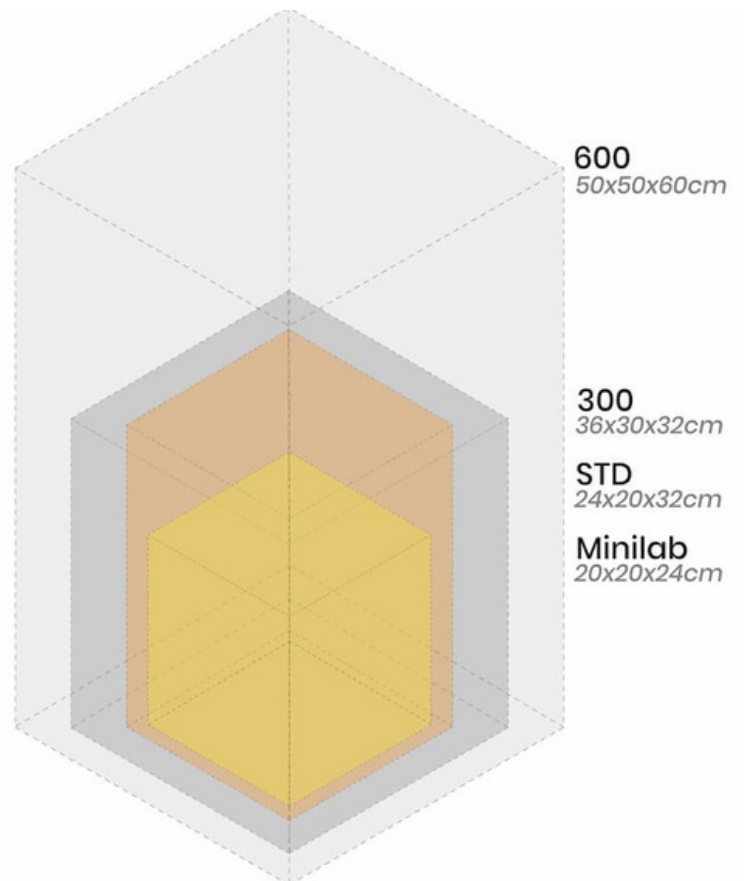


1. Power switch
2. Heated Bed
3. Spoolholder
4. Filament sensor
5. Touch screen
6. Nozzle

Voltage: 110V AC  
 Frequency: 50/60Hz  
 Máx consumption: 240W y 440W for FF600+

# PRINT VOLUME

MiniLab	20X20X24 cm
FF- STD	24X20X31 cm 20X20X31 cm Doppia
FF- 300	36X30X31 cm 32X30X31 cm Doppia
FF- 600	50X50X60 cm 46X50X60 cm Doppia
FF- 600+	60X60X60 cm 56X60X60 cm Doppia



# GENERAL ASPECTS

Here you will find a very simple guide of using your FUSED FORM 3D Printer

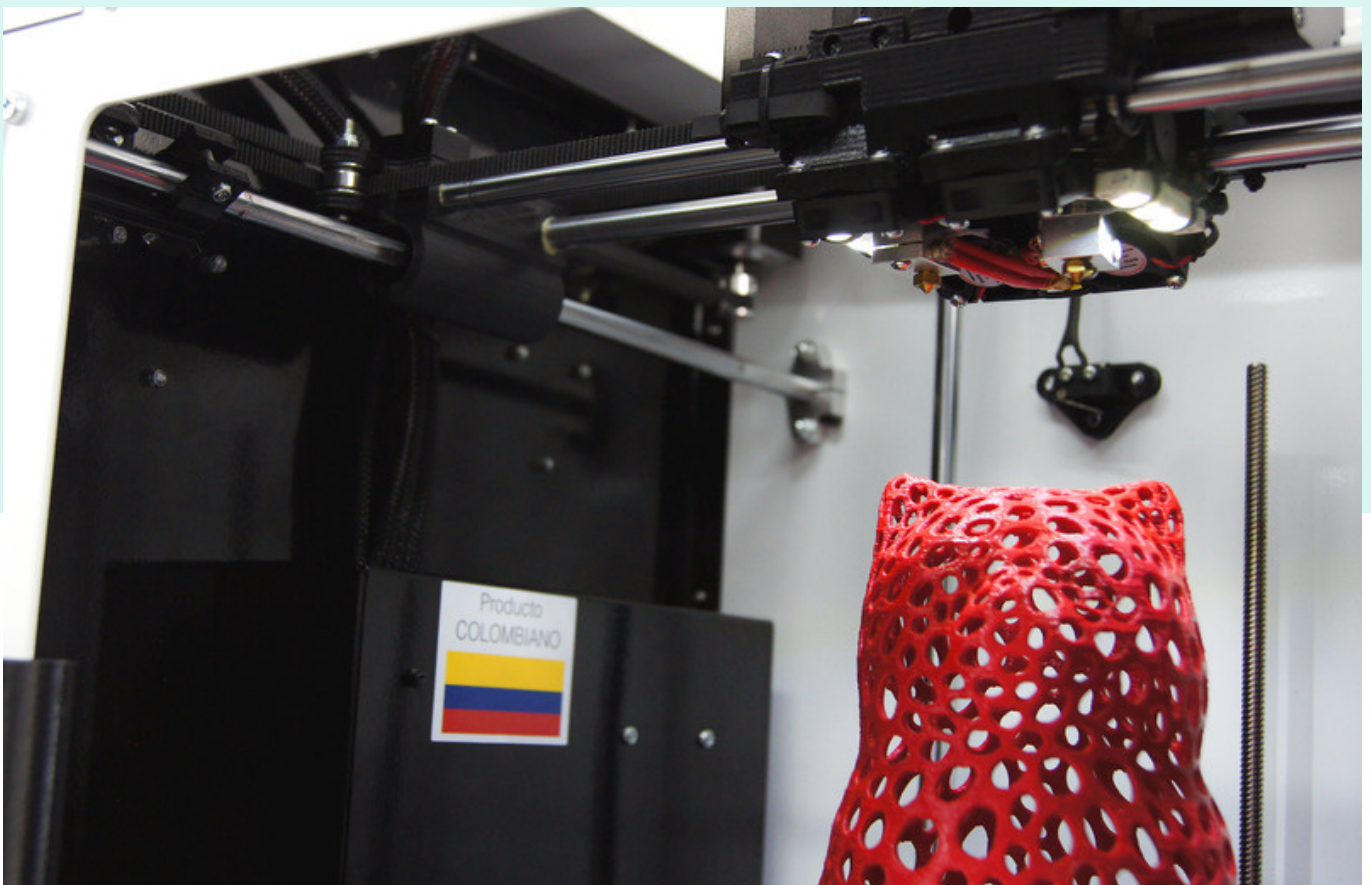
The first step of any 3d printing is to have a 3D design / model. Most programs such as: Rhino, Solidworks , Inventor, etc. allow you to export the file with a .stl or .obj extension.

In order to process and 3d print the object, the file must have one of the two extensions mentioned previously.

Once the .stl or .obj file is saved on your computer, you can proceed to use the slicer software, where we will convert the file into layers making 3D printer reads the file and 3d print it.

There, we will select how we want make the object, selecting what printing quality we want, how many infill and in what material.

Once the generated file has been saved in. gcode on the SD card, we can insert it into the 3D printer and 3d print it.



# FIRST STEPS

Here you will find a combination of videos and text. It is important that you read this manual **complete**

1

Here we show you the first steps of the process



Ctrl + Click  
on the video



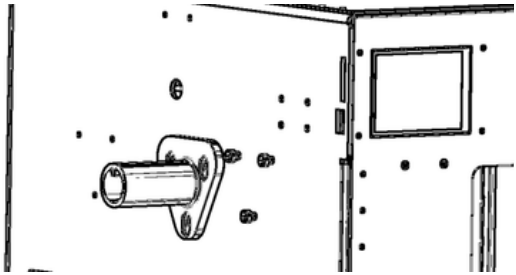
Ctrl + Click  
on the video

2

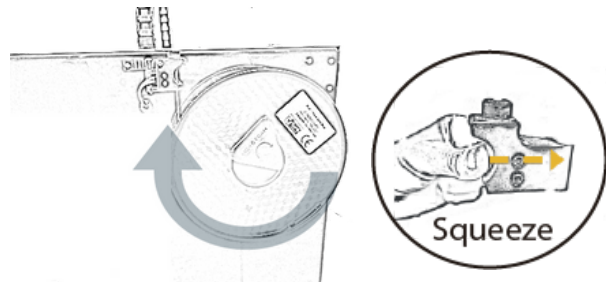
You can load the filament into the 3d printer like this

# LOADING THE FILAMENT

Install the spoolholder



The spool should be in the direction of the arrow:

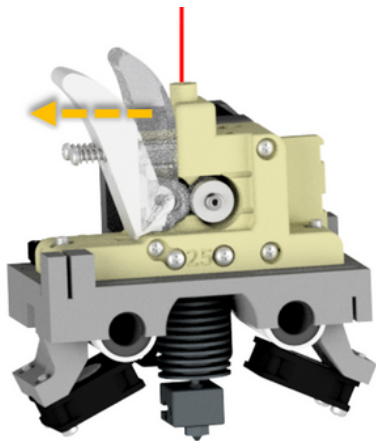


Heat the nozzle to the temperature of the material you are going to print:

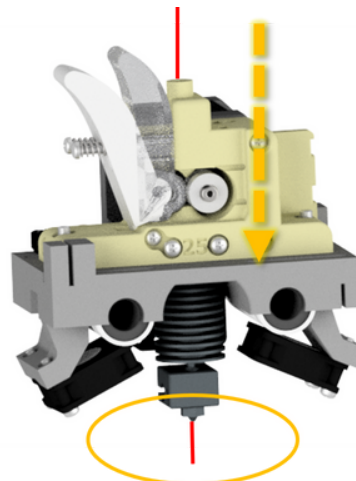


PLA: 210° ABS: 265° PETG: 235° TPU: 220°

1. Pull the lever



2. Feed the filament through the top hole until it comes out of the tip of the nozzle



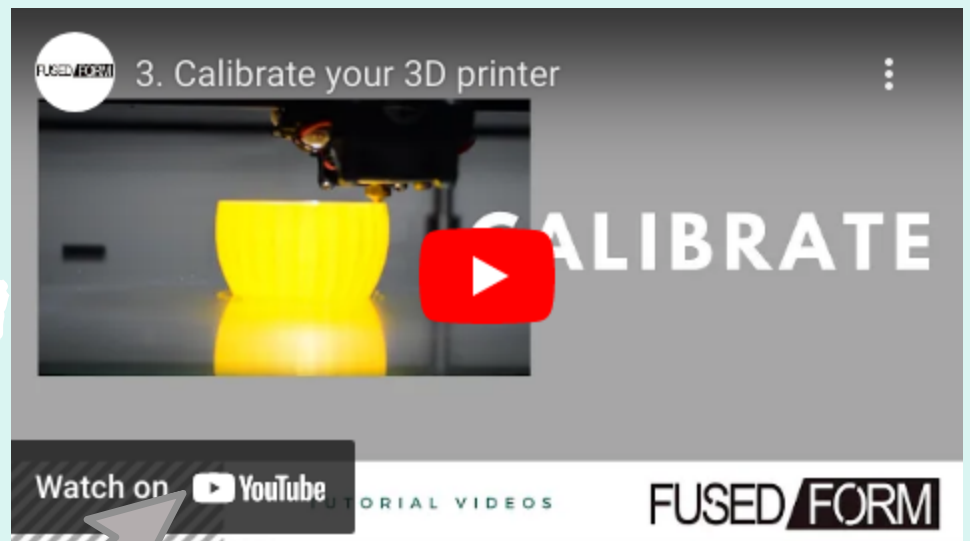
\*During this process you must have the level tilted back (step1)



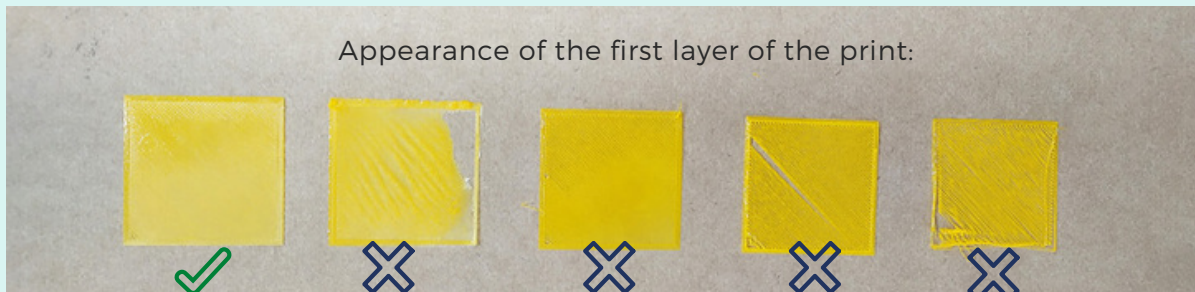
# CALIBRATION

Calibration is important and very simple at the same time.

See how to do it.



Ctrl + Click  
on the video



# BEFORE EACH 3D PRINTING



## ADHESIVE

Add lacquer or other adhesives at a distance of 20cm from the surface of the glass to generate greater adhesion of the print to the base.

Make sure to avoid spraying it on the rods. That will affect the bearings in the long run and affect your print quality.

You can also use special purpose glue for 3d printing.

\* The glass should be washed only with water \*



## OIL

Check that the rods smooth and threaded are oiled.

If they are not oiled, apply small drops on the rod surface.

Protect the glass in this procedure to prevent oil from getting dripped over the glass

# SET UP



- 1 When you take your filament out of the packaging it is recommended to break the tip of the filament that is at the center of the spool, to avoid tangling when it is running out, allowing the printer to correctly activate the filament sensor.



- 2 Check the nozzle: it must be clean and free of extruded material. If you have excess material, remove it with the tweezers to avoid burns

- 3 The filament spool has holes, these allow your filament not to get tangled. when you are going to store it you must enter the filament in one of these holes:



# READY TO 3D PRINT!



Choose any Gcode of PLA that is in the SD card that you want to print!

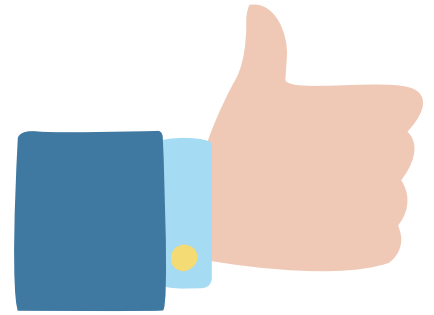
You can print any of the .gcode files whenever you have doubts about the performance of the 3d printer

A YouTube video player thumbnail. The title is "4. 3D print in your Fused Form 3D printer". The video shows a 3D printer printing a yellow cup. The text "3D PRINT" is overlaid on the right. There is a red play button icon in the center. The Fused Form logo is in the top left. A "Share" button is in the top right. At the bottom, there is a "Watch on YouTube" button, the text "TUTORIAL VIDEOS", and the "FUSED FORM" logo. A mouse cursor is pointing at the logo.

Crtl + Click  
en el video

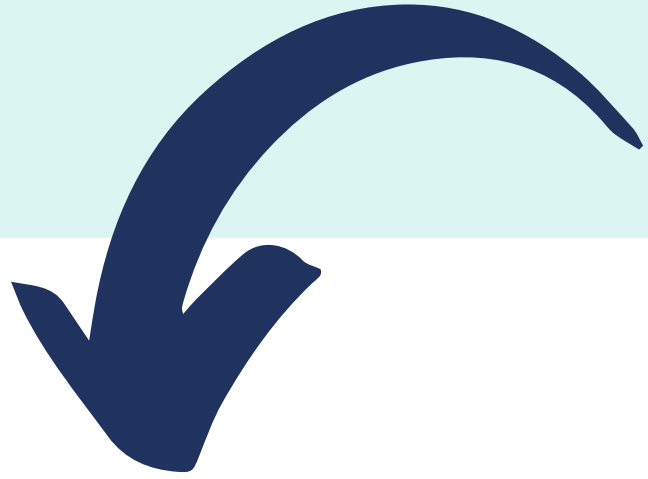
If you want to learn the slicing process  
go to the next section!

# BEST PRACTICES..



- If you are going to pre-heat the heated bed, avoid that the nozzle is touching the platform, this helps to avoid a possible clogging due to overheating
- If you have air conditioning, do not place the printer underneath, this will make it difficult for the heated bed to heat up
- The filament absorbs moisture, so it is important not to leave it installed in the 3D printer for more than 3 days.





**WWW.THINGIVERSE.COM**

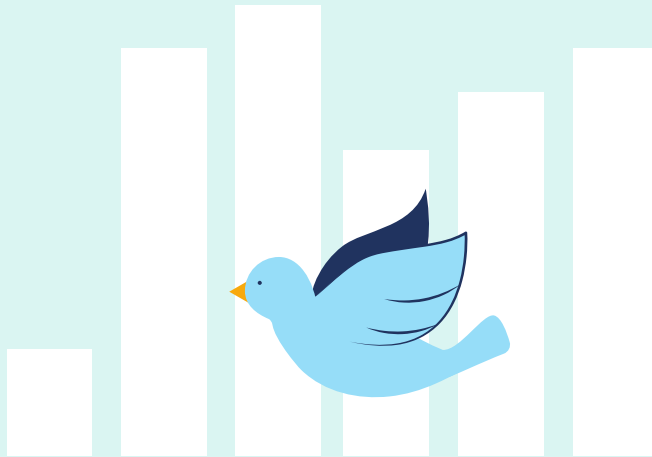
To get 3D files (with STL extension) you  
can download them at:  
[thingiverse.com](http://thingiverse.com).



## SLICING THE 3D FILE **FROM .STL TO .GCODE**

We will process the .STL file to convert it to the .GCODE (3d printer language) so that you can print your object successfully

# 3D SLICING SOFTWARE



There are different CAD slicer software available.

We provide you the parameters of:  
Cura and Simplify 3D

## CURA SOFTWARE

This software is free and can be downloaded from the internet.

The steps for downloading are as follows:

- 01** Search on Google: Cura Software
- 02** Download the latest version
- 03** Once installed, look for your Fused Form 3D printer

## SIMPLIFY 3D

This software is paid. You can buy it directly with your credit card.

The steps for downloading are as follows:

- 01** Search on Google: Simplify 3D Software
- 02** Once you have bought it you can upload the configuration that we have on our website in support/downloadable

# 4

## MOST COMMON SLICER PARAMETERS

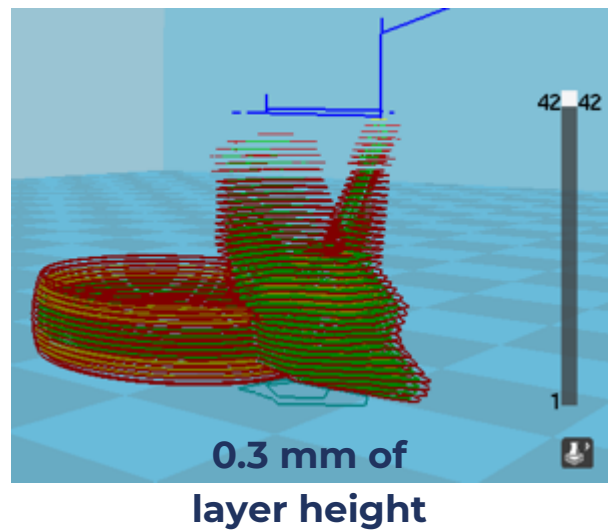
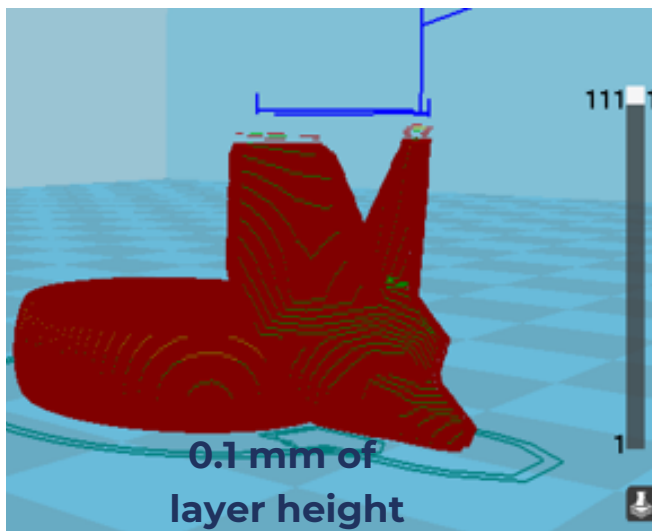
### **FOR 3D PRINTING**

# 1. QUALITY

Print quality is defined by layer height. The smaller the layer height, the better printing resolution you will have.

Layer height =  
Print quality

<b>Extra High</b> 0.08 mm	<b>High</b> 0.10 mm	<b>Normal</b> 0.15 mm	<b>Low</b> 0.2 mm
------------------------------	------------------------	--------------------------	----------------------

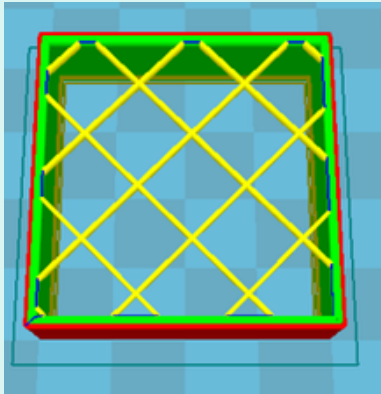


**Higher quality** 3d printing will take a little longer but sometimes it's worth it!

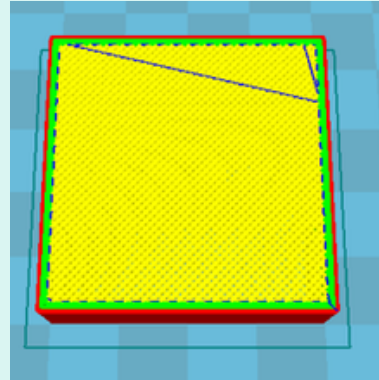
*good things take time*

# 2. INFILL

The infill let you define how the part will be internally: solid (solid), intermediate or hollow



**10% Infill**



**60% Infill**

**INFILL**

**%**

Empty

10% \*


Normal

15%

Solid

40% \_ 60%

\* Parts with lower infill may have bad quality on the surface

 **For mechanical parts it is better to increase the shell thickness**



# 3. FILAMENTS

Each material has its own characteristics. Its use will depend on the application you want to give the 3d printed object



## PLA

- Ease of 3d printing
- Dimensionally stable
- Faster 3d printing speed
- Low resistance to high temperatures
- Prints with the enclosure open

## ABS

- More temperature resistant than PLA
- Easy machining capacity: can be sanded, drilled, etc..
- It tends to shrink for large volume parts
- Enclosure necessary to avoid warping
- To compensate the warping you can scale the model to 106%

## PETG

- Good strength without high warping
- Low moisture absorption
- Its cost is higher than PLA and ABS
- Must print slow and with enclosure

## TPU

- Flexible
- The first layer should be set higher than the others materials
- It should print slow
- It should not be printed with layer heights greater than 0.2mm

If you are going to change the material it is important that you always **purge** with PLA.

Example: if you were printing ABS and now you are going to print TPU, you must remove the ABS from the nozzle, extrude PLA to purge and then load the new material

# MATERIAL TIPS



## Composite Materials

- for materials filled with: wood, metas: copper, aluminum, carbon fiber, etc. we recommend printing with a nozzle of 0.5mm or higher.
- For fiberglass and carbon fiber composites, additionally use 0.5mm stainless nozzle
- You must physically change the nozzle and in the software you must update the new nozzle diameter that you are going to use (all printers are delivered with a 0.4mm nozzle)

## Adhesives

- You can glue the **PLA** 3d printed objects with no more nails
- For **ABS** glue you can use ABS dissolvent or No more nails
- Remember to wear: glasses, gloves and mask

## Pre Dry

- It is recommended a pre drying of Nylon (PA), PC and other hygroscopic materials when are to be used for the first time, 6-8hrs in a dryer is recommended.
- While printing these materials it is important to keep them in an airtight container to prevent them from absorbing moisture. Ask us about our solutions for these materials

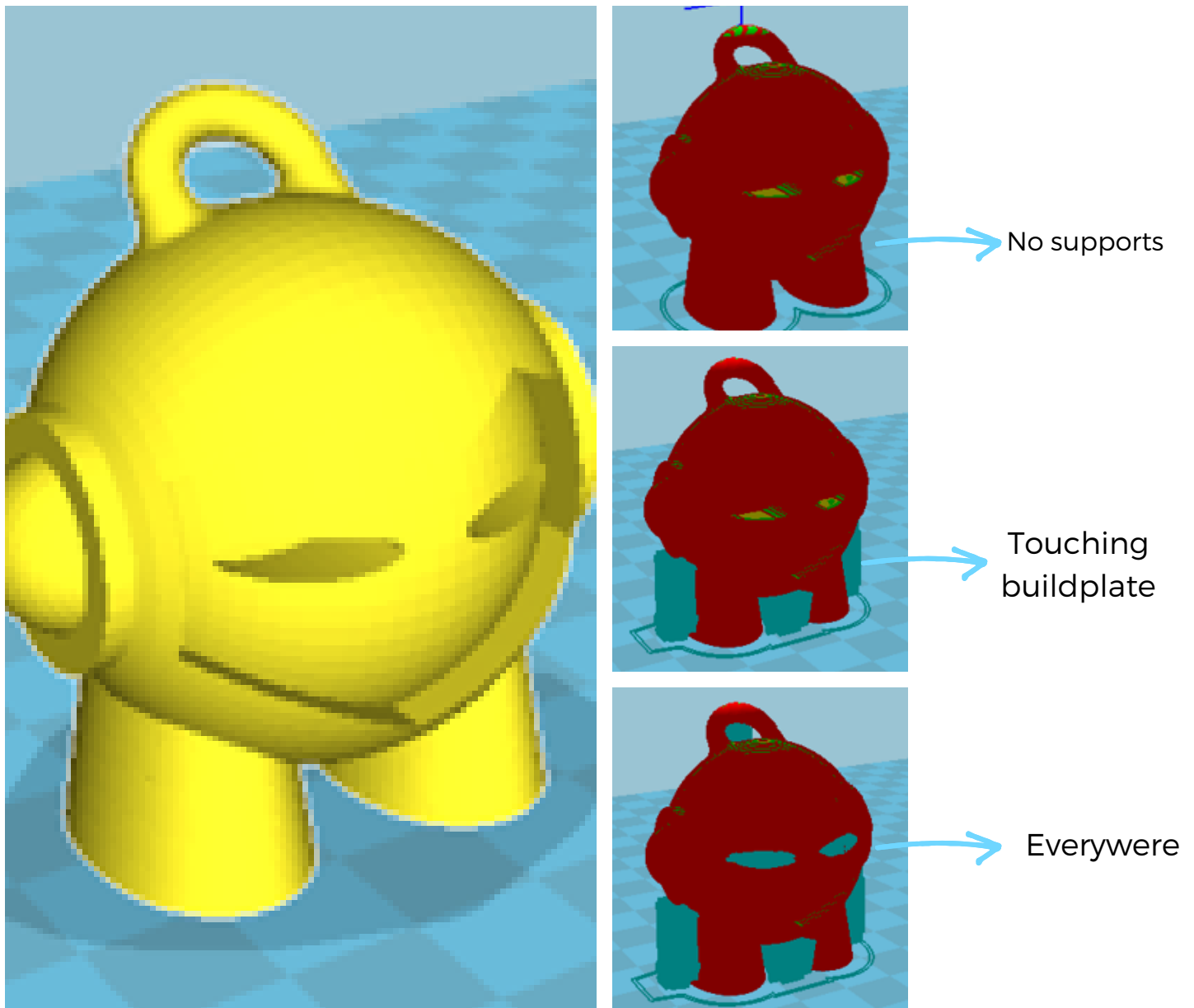
## Soluble Materials

- PVA and HIPS are used as soluble support material.
- PVA is used with PLA, TPU, Nylon among others. To dissolve it, it is enough to immerse it in water at room temperature for a few hours.
- HIPS is compatible with ABS. For easy removal it must be immersed in the chemical Limonene. You must wear glasses, gloves and a mask.

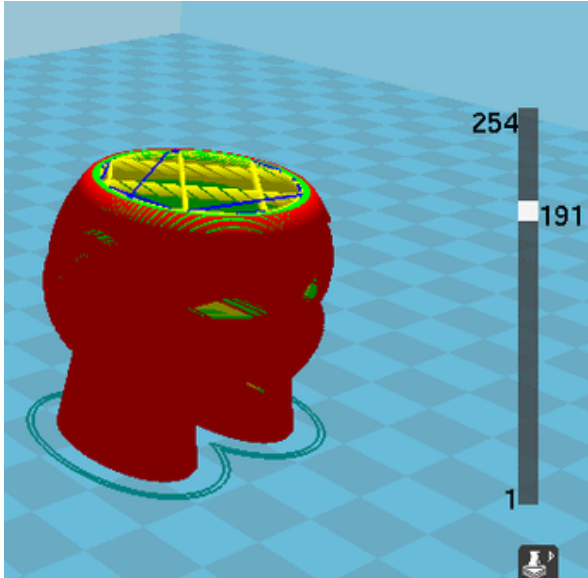
# 4. SUPPORT MATERIAL

The support material lets generate a removable material that works as a scaffold of the model. This is to prevent the cantilevered parts from being poorly finished. When you have a double extruder, usually the second nozzle is used for soluble support material (PVA or HIPS)

It is convenient to use it when you have angles greater than 60 °



# CHECK & SAVE



A layer by layer check is the most **fundamental** process, because: you can check:

- If the model is well built
- Check if the little details are ok
- General print check



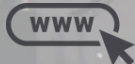
Remember to **check** that the parameters you selected are the indicated for your printing material

SAVE THE .GCODE ON YOUR SD CARD  
!! YOU ARE READY!!

DO  
Amazing  
THINGS!

# TECHNICAL SUPPORT

We can help you with this media sources:



[www.fusedformcorp.com/support/troubleshooting](http://www.fusedformcorp.com/support/troubleshooting)



[soporte@protolab3d.co](mailto:soporte@protolab3d.co) (Attach Photos)

# FUSED FORM

Empowering bright *ideas*