

# PLA

**3D PRINTING MANUAL  
WITH FILAMENTS**

[ILARO.ORG](http://ILARO.ORG)

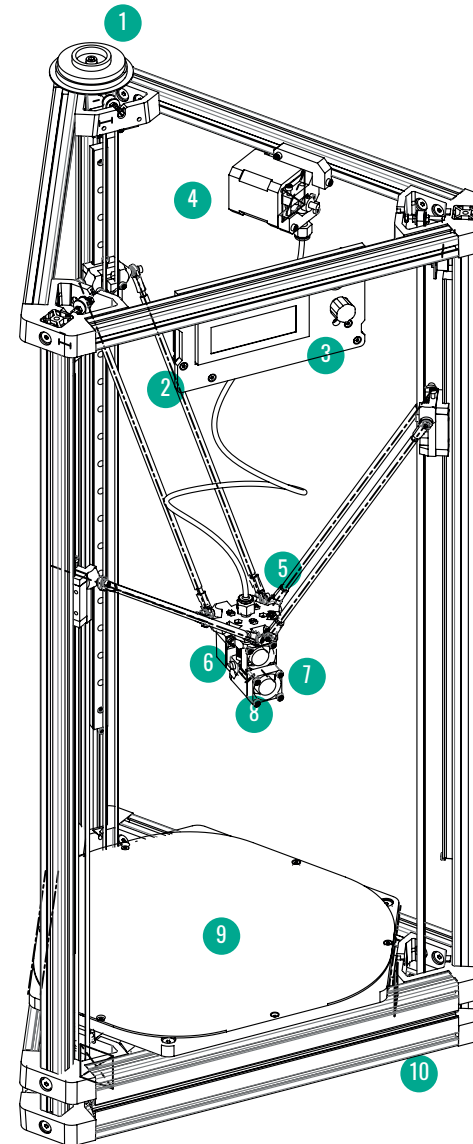
# Additive (3D printing)

Additive manufacturing (more commonly known as 3D printing) is the process of building additive part, layer by layer. There is a wide range of 3D printing technologies, each with its own benefits and limitations, and each is capable of printing parts of different materials.

Almost any geometry can be produced, it is one of the strengths of 3D printing (although there are still rules that we must take into account depending on the technology). In addition, 3D printing is not based on expensive tools, which has the advantage of rapid verification and development of prototypes and low-volume production parts. One of the major limitations of 3D printing is the inability to produce parts with material properties equivalent to those performed by subtractive or formative techniques.

3D printing also has limitations on repeatability, which means that parts often have slight variations due to differential cooling or deformation during curing.

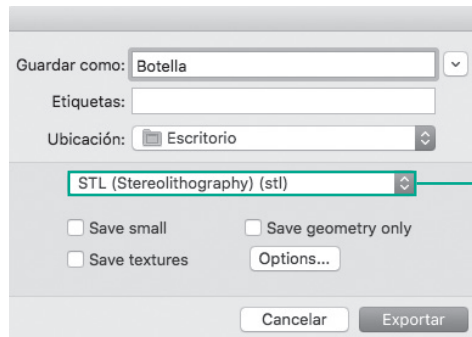
# Calypso Delta



- 1 - Pla Filament
- 2 - SD Card
- 3 - Shield LCD
- 4 - Pull filament
- 5 - Push fit
- 6 - Heatsink
- 7 - Fan
- 8 - Nozzle
- 9 - Hot bed
- 10 - Power On/Off

## STEP 1

## Export the file



To print with the 3d printer we need a STL model. We can get the file previously modeling the object with any 3D modeling software (Sketchup, Rhinoceros, SolidWorks, etc..).

Select "save as" and save with the extension .stl (Stereolithography)

## STEP 2

## Laminate Configuration

To print .STL file we will use CURA, which is an "Open Source" program developed by ultimaker for the lamination of 3D objects. From this program you can do all the necessary steps to move from a 3D model to a real object.

Next we will explain the most important parameters to take into account to print our piece.

**Layer height (mm)**  
layer height between 0.1 to 0.4.

**Shell thickness (mm)**  
This parameter determines the width of the edge of the object

**Enable retraction**  
activates retraction

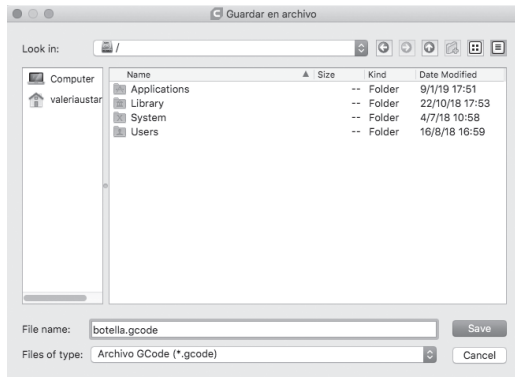
**Fill Density (%)**  
this value indicates the filling that the figure is going to have. 10% for artistic pieces, 30% for pieces that have to support weight or efforts.

**Printing temperature (C)**  
temperature of the hotEnd at which we are going to print. For the PLA between 200°-205°.

**Support type**  
creates brackets where necessary on parts that are airborne and cannot be suspended.

**Filament diameter (mm)**  
Set the diameter of the filament we're using. In our case it will always be 1.75 mm.

## STEP 3 Export the .gcode file



Then we will export as name.gcode and save it on an SD card



Insert SD card into the screen slot on the left side



Search our .gcode file in the card menu by turning the metal button

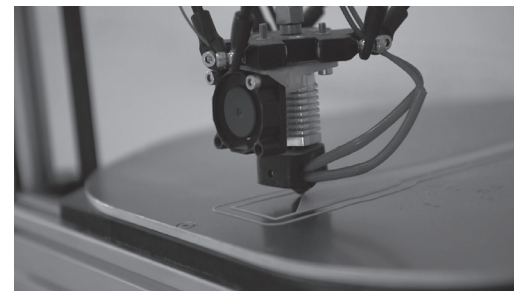
## STEP 4 print



Select by pressing the metal button.



Printing



**MANUAL BY**  
**Valeria Ustárez**

**2019 | [ilaro.org](http://ilaro.org)**

**LICENCIA GPL.V3.**