JetClay Academy

Introductory course.

3D ceramic printing

JETCLAY

Contents

CAP.

Introduction

Ceramic 3D printing offers a wide range of possibilities in artistic and design projects, allowing the creation of complex shapes, the personalisation of each piece and the optimisation of the material used by reducing waste. In order to master this technique you need to understand and control a few key factors (the viscosity of the ceramic paste, the pressure regulation, the extrusion speed...) which are essential to achieve a uniform print and avoid problems such as collapse or clogging.

DEFINITION AND BASICS

This process is based on additive manufacturing by extrusion, in which the head deposits successive layers of material with the right consistency to give the part its shape. Once the printing is finished, the piece must undergo a drying and firing process in a ceramic kiln to harden and reach its final state.

MAIN CHALLENGES

During the course, it will be essential to face some technical challenges, and mastering them will be key to achieve successful pieces and explore the full creative potential of 3D printing with ceramics:

→**Paste consistency:** Determining the optimal viscosity for a uniform and stable extrusion is essential.

→**Material flow:** The clay must move smoothly through the system, staying within a pressure range of 4 to 6 bar.

→**Print evaluation:** It is essential to recognise a well-executed print, to identify problems resulting from a shortage or excess of material and to apply the necessary corrections to optimise the result.

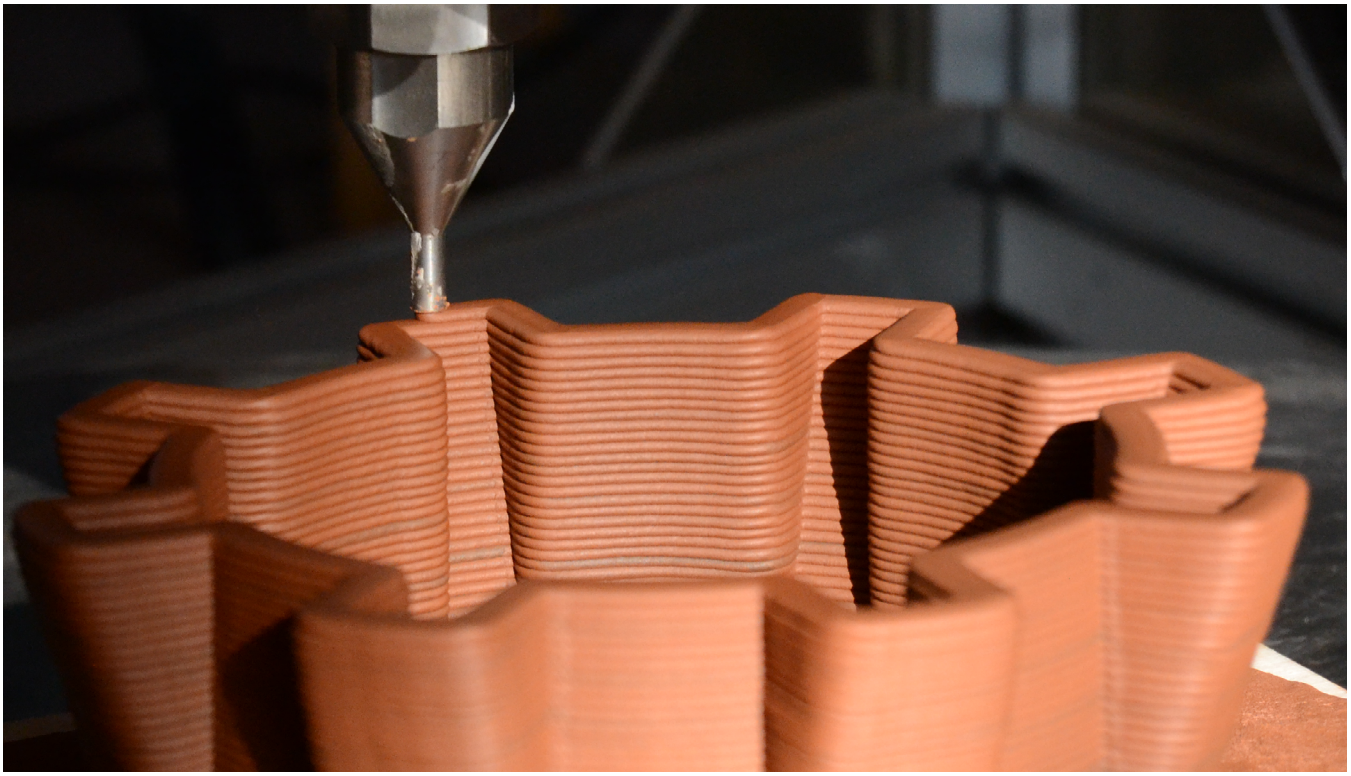
CAP.01

BASIC CONCEPTS

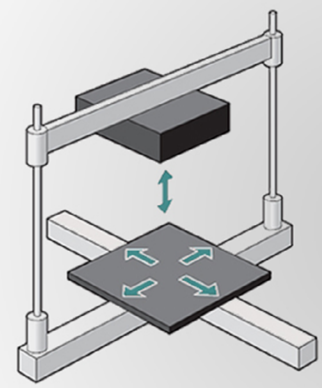
01.10 LDM printing

Liquid Deposition Modeling [*LDM*] technology refers to an extrusion 3D printing method designed for materials with a pasty or semi-liquid consistency, such as clays, porcelains and other ceramic or biomaterial mixtures. Unlike *FDM* [Fused Deposition Modeling] printing, which works with solid thermoplastic filaments, *LDM* uses a pump or pusher system [this can be a piston, air compressor or auger] that ejects the material directly in its fluid state onto the build base.

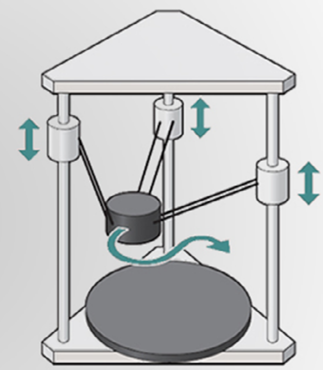
The key principle of *LDM* is to deposit successive layers of the paste with precise flow control, allowing three-dimensional objects to be formed with the same additive logic as in other 3D technologies. This approach is ideal for ceramics, as the material does not need to be melted by temperature, but is handled in its wet form and, after printing, goes through a drying and firing process.



01.20 Types of LDM printers



Cartesian style

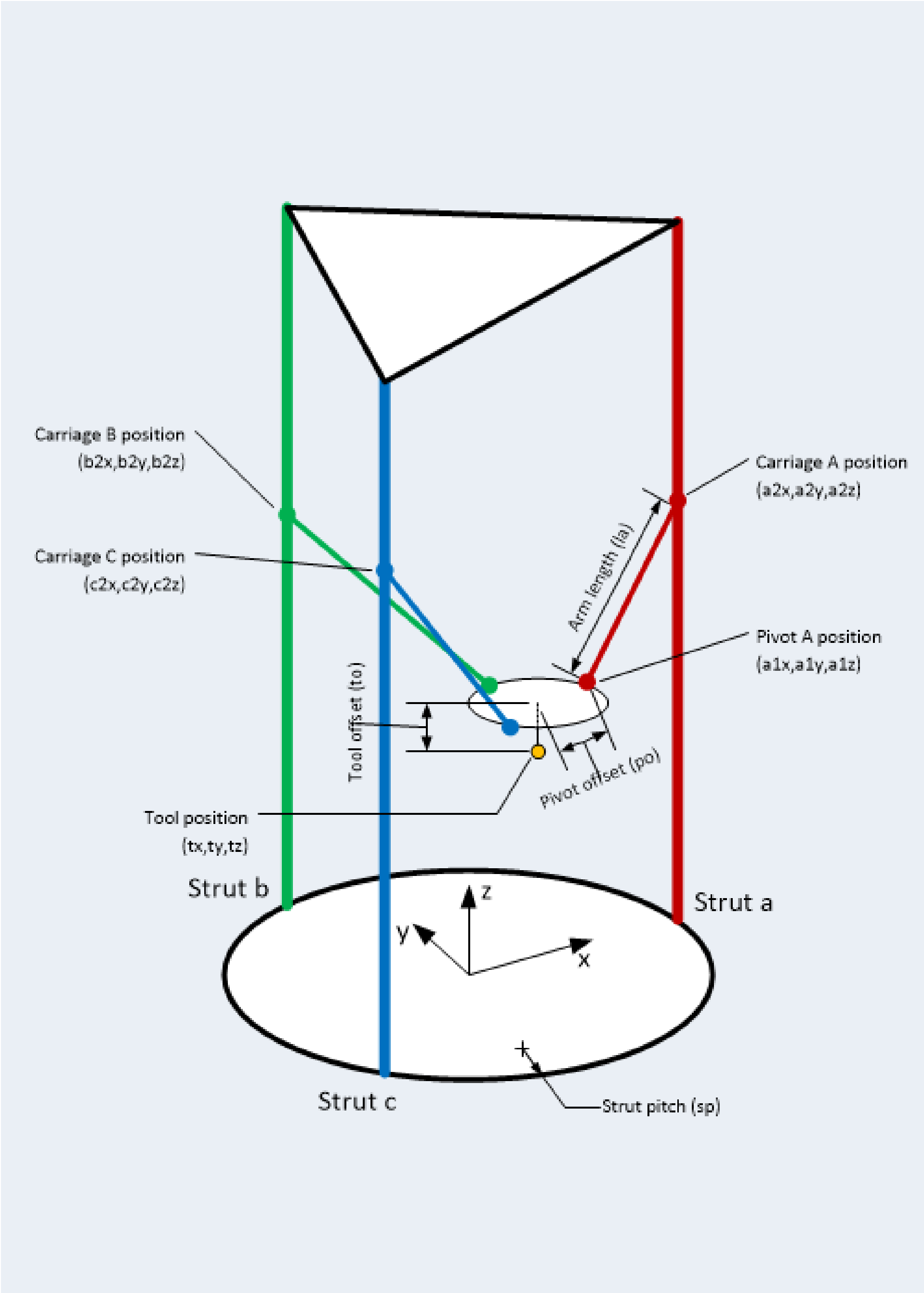


Delta style

/CARTESIAN PRINTER/ These machines are based on the movement of their axes [X, Y and Z] in a linear and orthogonal way, as their name indicates [Cartesian coordinate system]. They usually have a platform that moves in one or two axes while the extrusion head moves in the remaining axis. Their design is simple, easy to calibrate and very common in the 3D printing market. However, its stability and speed can sometimes be limited on very large prints or at very high speeds.

/DELTA PRINTER/ They work with three arms arranged around a triangular structure, all linked to a central extrusion head that moves in space by combining the displacements of each arm. This taller, slimmer design allows for fast, fluid movements, resulting in faster print speeds and smoother strokes. In addition, due to their configuration, they tend to have a better ratio between the occupied area and the available printing height, making them ideal for tall and slender parts.

01.21 Delta printer motion diagram



01.30 Printing systems

/RAM EXTRUDER/ *RAM Extruder* [*Reciprocating Action Machine*] is a system that uses a hydraulic or mechanical piston to exert pressure on the clay, forcing it out through the extrusion nozzle. Unlike the compressed air system, this method ensures a more uniform pressure and allows a higher material loading capacity. Among its main advantages, it offers greater control over the extrusion, reducing variability in the flow of material, and is capable of containing and processing large volumes of clay without the need for constant reloading. In addition, it is particularly suitable for working with denser or more viscous clays without the risk of clogging. However, it also has some disadvantages, as it is a more complex and costly system compared to compressed air, requires more maintenance and cleaning due to material build-up in the cylinder, and is less flexible for quick material or colour changes.

/COMPRESSED AIR AND TANK/ The compressed air and tank system uses an air compressor to pressurise a tank where the clay is stored in a paste-like state. The pressure generated pushes the clay through a tube into the extrusion nozzle, allowing precise control of the deposition of the material. This method offers a constant and regulated flow and is compatible with different types of clay by adjusting the pressure and viscosity. Its design is relatively simple and easy to maintain, making it an affordable option for ceramic 3D printing. However, it has certain limitations, as the capacity of the system is conditioned by the size of the tank, which requires frequent refills. It also relies on a continuous and well-regulated air supply, and can lead to variability in extrusion if the pressure is not properly adjusted.

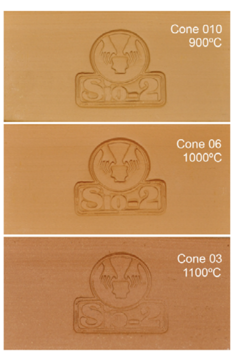
01.40 Ceramic materials and pastes: preparation and consistency

Clay is a versatile material used in ceramics and digital fabrication, with properties that change according to its composition and firing temperature. There are different types, such as red clay, stoneware and porcelain, each with unique characteristics in terms of plasticity, strength and water absorption. Choosing the right clay is key to obtaining optimal results in 3D printing and traditional ceramic techniques.

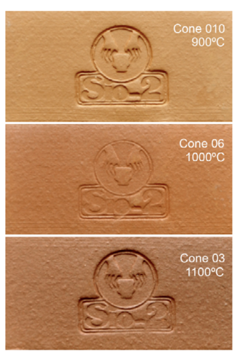
/HIGH TEMPERATURE CLAY/ It is usually fired above 1200 °C, reaching a vitrification point that makes it very dense and impermeable. Therefore, it is more resistant and suitable for pieces that are subject to intensive use or that require great durability. Some examples are stoneware and porcelain.

/LOW TEMPERATURE CLAY/ It is fired in a range of approximately 900 to 1100 °C and therefore retains a higher porosity. This may require the application of glazes or glazes to waterproof it and improve its strength. It is often used in traditional pottery or decorative ceramics.

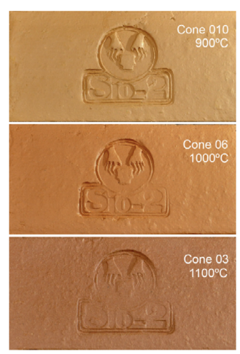
Example of some of the different types of low and high temperature clays that we can find, and with which we can print in our 3D ceramic printer ↓



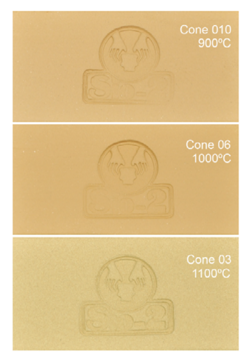
PF Red



PF/CHF Grogged Red 0-0.5

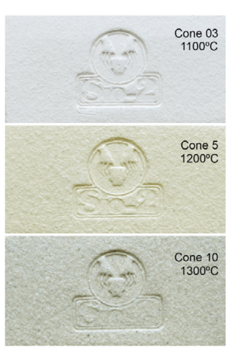


PF/CHM Grogged Red 0-1.5



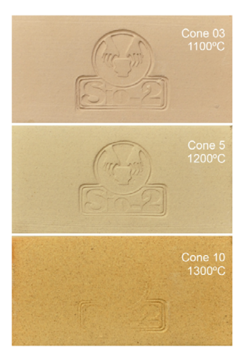
PT Terracotta

PRAI White Stoneware 0-0.2



PRAF White Sculpture 0-0.5

PRAM White Sculpture 0-1.5



PRGI Golden Brown Stoneware 0-0.2

↑ Clays to which different percentages of water have been added and in which different degrees of plasticity can be observed.

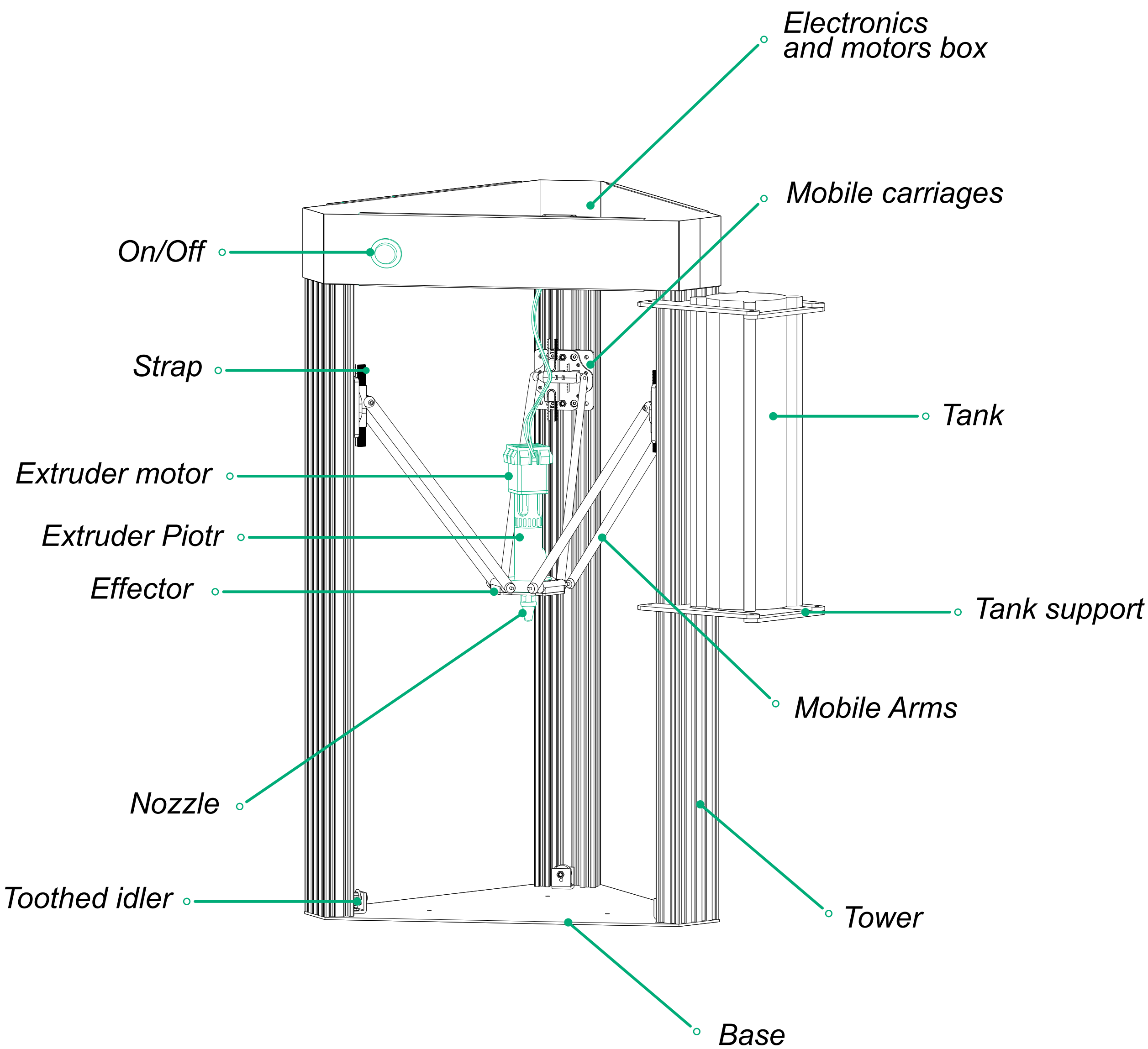
We can use a variety of ceramic pastes [porcelain, stoneware, clay or combinations of these] as long as we adjust the humidity to the right level to facilitate extrusion and prevent the piece from collapsing. A useful starting point is to add 5% water in relation to the weight of the clay [i.e. for every 1 kg of clay, about 50 mL of water], although this formula can change according to the quality of the material.

Some clays are marketed in 12-13 Kg tablets with a certain degree of initial moisture, while others come in powder form: in the latter case, it is safer to add the water little by little until the required consistency is reached. To check if the mixture is ready, simple tests are recommended, such as forming a churro with the clay and seeing if it holds its shape, or even carrying out small extrusion tests. The aim is to achieve the right viscosity so that the clay flows easily through the nozzle, but retains enough density not to deform. Regardless of the paste you choose, this balance between wetness and firmness is essential for successful ceramic 3D printing.

CAP.02

JETCLAY MINI

02.10 Parts of the JetCLay MINI



02.20 JetClay MINI printing system

As mentioned above, ceramic 3D printing is based on the controlled extrusion of a paste or fluid through a system made up of various components that must operate in total synchrony. The JetClay Mini uses a compressed air printing system, in which each of its parts plays a key role.

First, an air compressor is used to pressurise the clay tank; this compressed air pushes the paste into the extruder, where a worm regulates the amount of material coming out of the nozzle. Simultaneously, the machine moves the head following the trajectories defined in the printing software. The balance between the air pressure, the rotational speed of the worm and the movement of the printhead is crucial to ensure a continuous and uniform flow. In this way, successive layers of material are deposited with sufficient precision to form ceramic parts of different shapes and sizes.

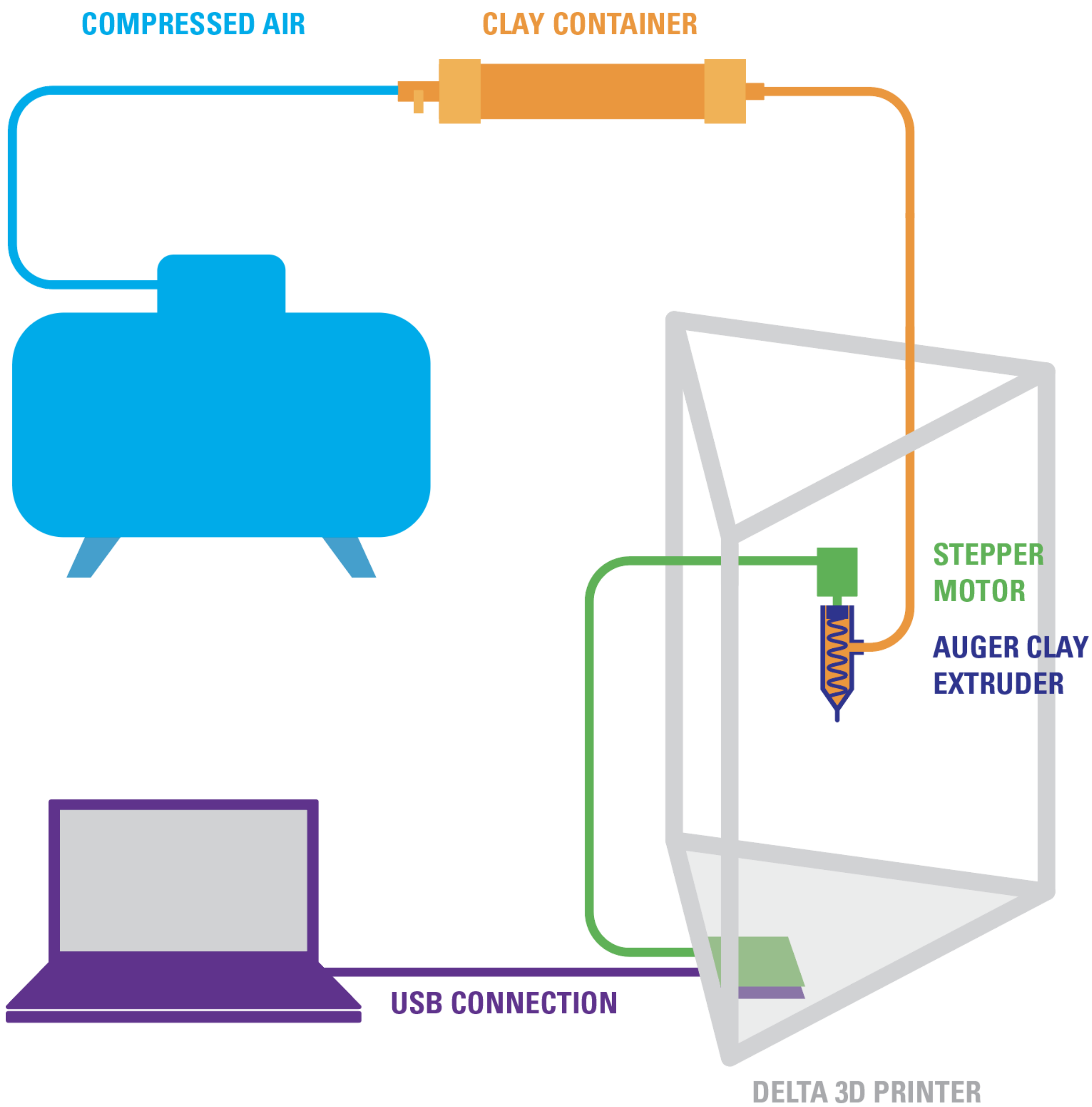
When all the elements [pressure, extruder, paste consistency and machine motion] are properly calibrated, the likelihood of successful prints with fewer defects is greatly increased.

/COMPRESSED AIR/
A compressor generates the necessary pressure to push the ceramic paste.

/CLAY TANK/
Contains the ceramic paste or fluid ready to be printed.

/EXTRUDER WITH ENDLESS SCREW/
Controls the flow of paste coming out of the nozzle.
Allows adjustment of the amount of material deposited in each layer.

/MACHINE MOTION/
Moves the head according to the trajectory defined in the printing software. The precision of these movements determines the final accuracy of the piece.



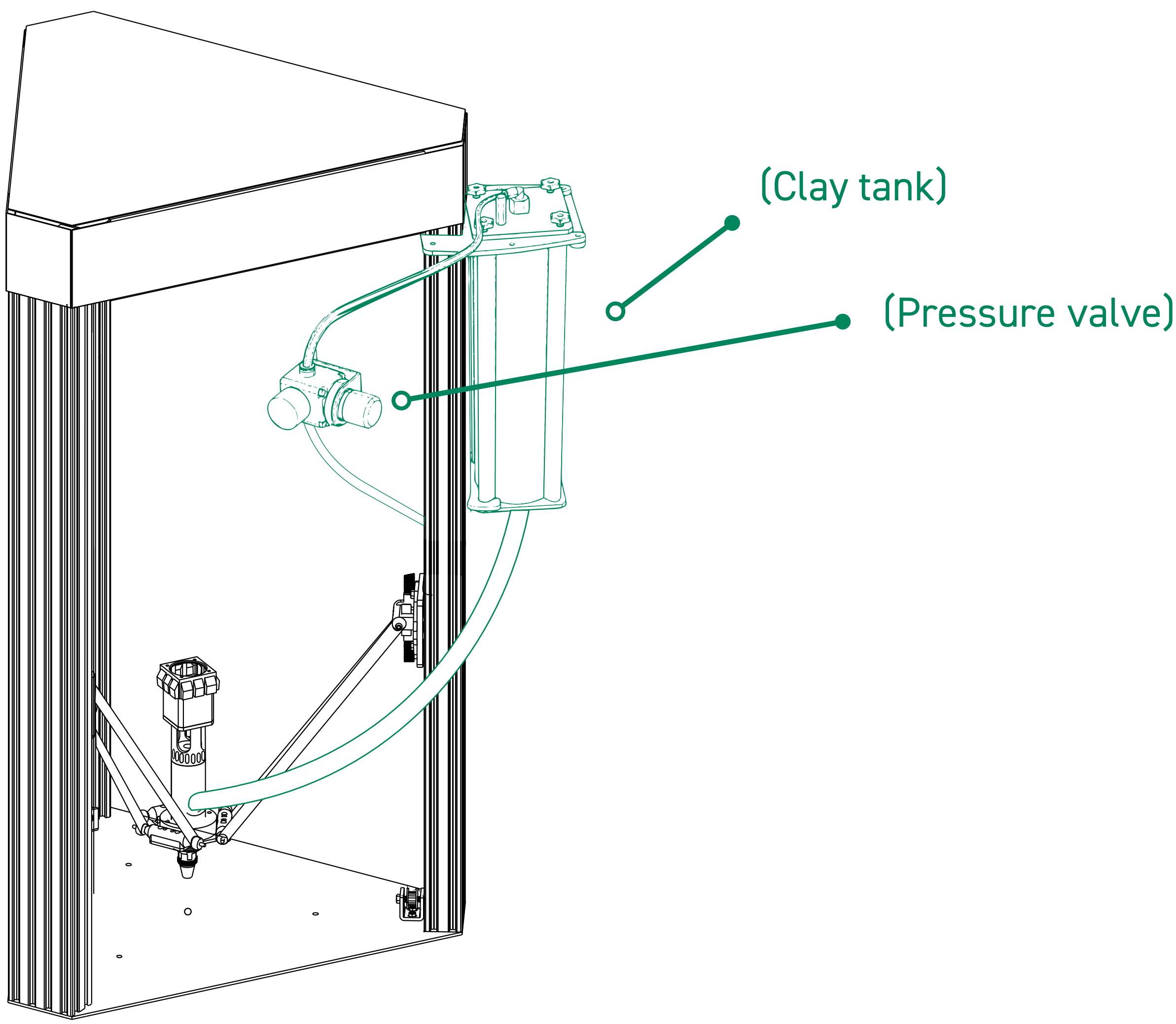
↑Diagram of the clay and air circulation system in the JetClay Mini printer, which is operated by compressed air and a charging tank.

02.21 Clay flow of the JetClay MINI:

The JetClay Mini employs a compressed air printing system, consisting of an air compressor and a clay loading tank.

The compressor must supply air to the tank at a pressure of 6 bar, ensuring that the clay flows evenly through the system within a range of 4 to 6 bar pressure.

The pressure exerted on the tank is regulated by a valve, which allows the bars to be increased or decreased as required. An increase in pressure increases both the amount of clay extruded and the speed at which it travels into the extruder, while a reduction decreases both factors.



Pressure adjustment through this valve is key to regulating the percentage of extruded material. Increasing the pressure is useful when there is insufficient material flow, resulting in brittle, cracked or thin layers. Reducing the pressure is necessary if there is too much material, too wide layers or bulges in the walls, indicating over-extrusion.

Changes in air pressure affect the extrusion flow in real time, without the need to stop the printer. However, this adjustment must be done with precision, as the valve is highly sensitive and a small twist can cause significant variations in the print.

The clay flow does not depend exclusively on the air pressure, but is regulated in conjunction with the extruder worm [or flow] control and the printing speed. These parameters, which can be adjusted in the ↗(Printing Factors), window, combine to ensure a constant flow and uniform deposition of the layers. This will be discussed in detail later in section 03.30.

02.30

JetCLay MINI controls

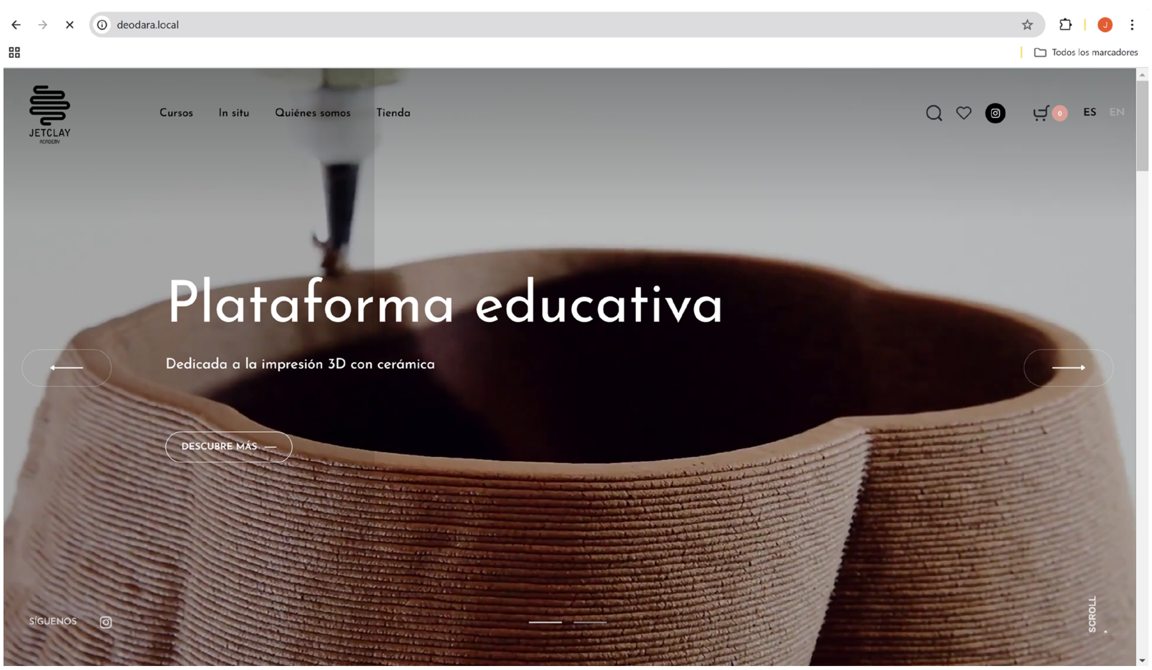
Jetclay MINI printers are operated and configured using *Duet Web Control (DWC)*, an intuitive interface that provides greater control over print dynamics. With this system, you can quickly upload files, instantly modify parameters and monitor the entire printing process in real time.

/ACCESS AND INITIAL CONNECTION TO DUET WEB CONTROL [DWC]/

→Open the browser:

↗Enter the IP address of your printer in the search bar [e.g. http://192.168.0.X] o or Printer name.local.

→The Duet Web Control main screen will appear.



↑Access to **DWC** can be done from any browser by entering the IP address or name followed by (.local)

/OVERVIEW OF THE DUET WEB CONTROL (DWC) INTERFACE/

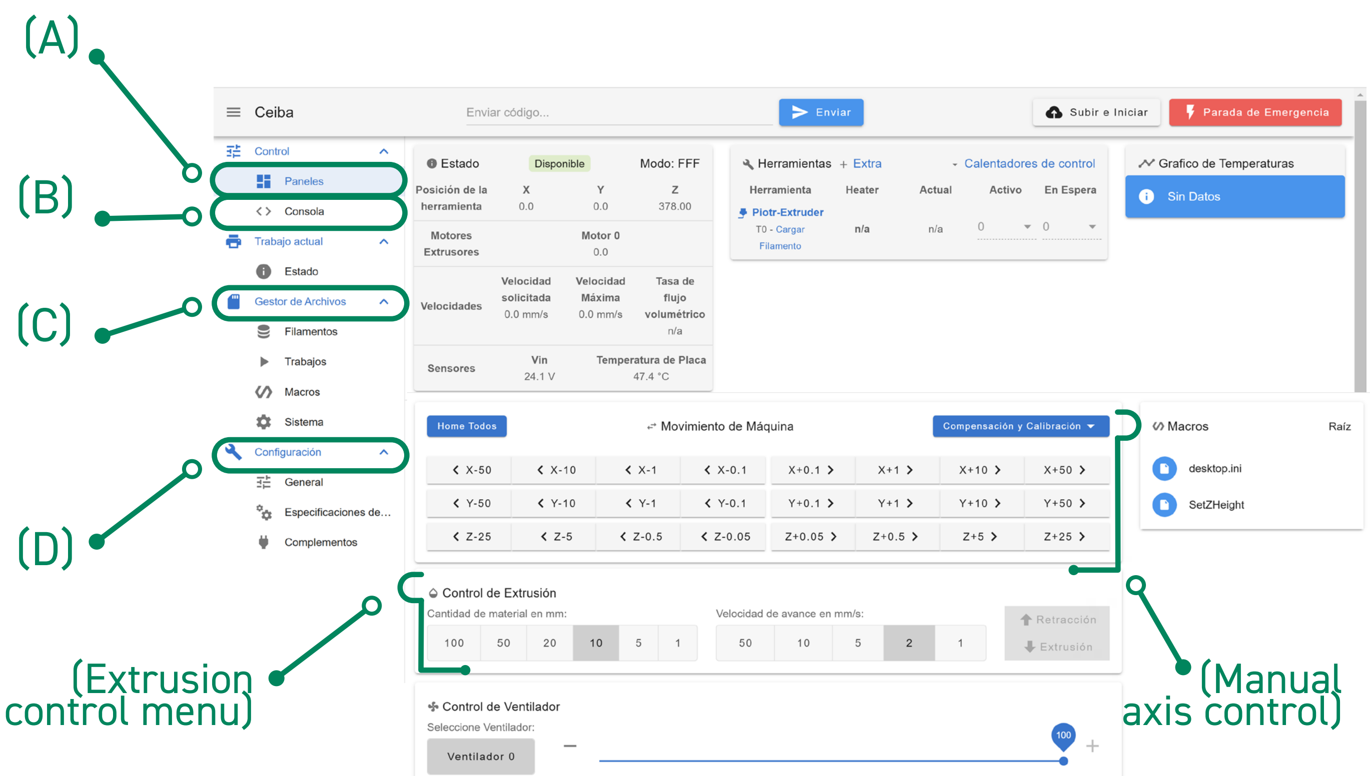
→When loading *Duet Web Control*, you will see several main sections:

A) (Dashboard): Displays bed and extruder temperatures, buttons for moving the axes, printer status, etc.

B) (Console): Allows you to send *G-Codes* commands and receive firmware messages.

C) (Files): Place where the *G-Codes* are uploaded and stored ready to print.

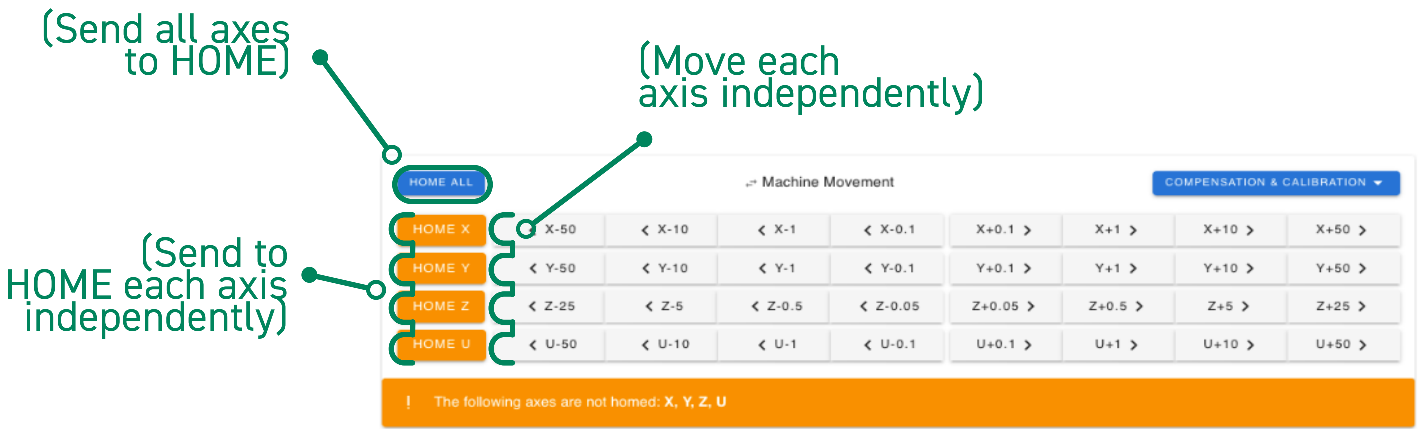
D) (Settings): Section for editing network parameters, firmware settings, macros, etc.



/BASIC MOVEMENTS AND CONTROLS/

→**Home [Homing]:** By pressing the ↗(Home All) button, the machine adjusts its axes to the home position automatically. It is also possible to move each axis separately by using the home buttons corresponding to each axis.

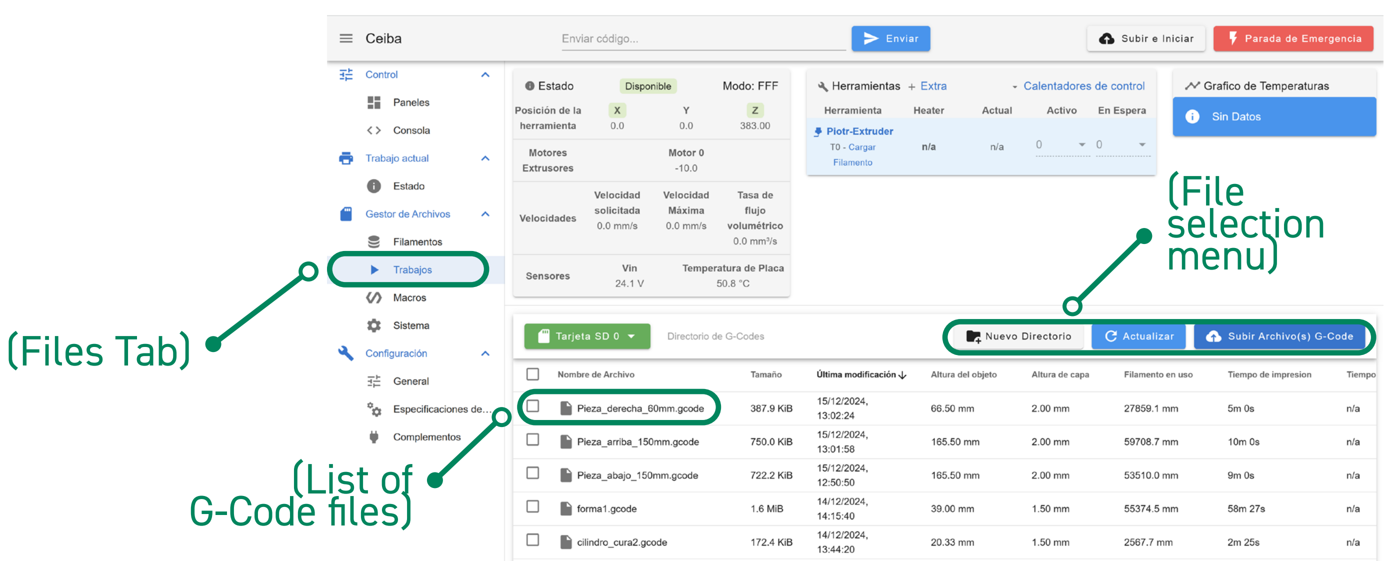
→**Manual axis control:** Use the ↗(move) buttons to move the extruder at preset distances [e.g. 1 mm, 10 mm, etc.].



/UPLOADING AND MANAGEMENT OF G-CODE FILES/

→**Upload files:**
Go to the ↗(Files) o ↗(Jobs) tab.
Click on ↗(Upload) and select the *G-Code* from your computer.

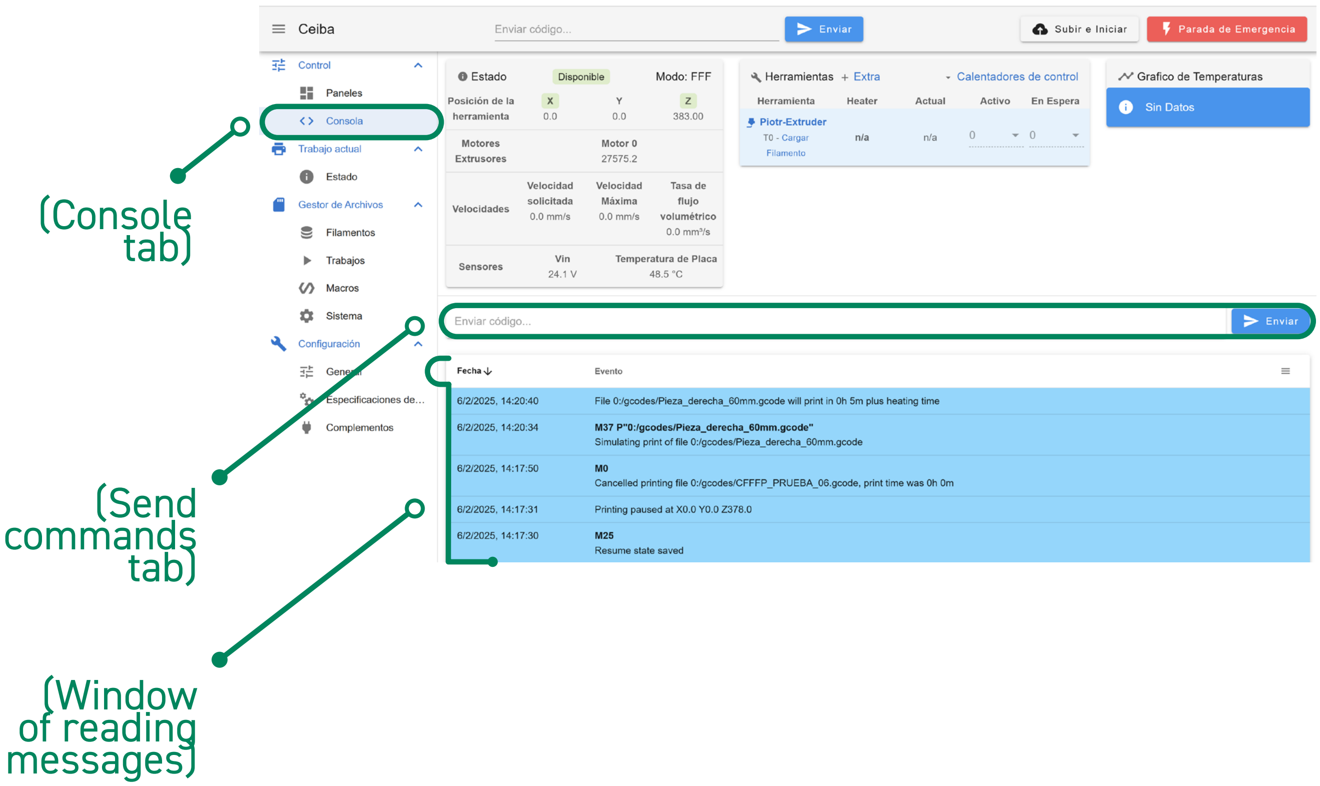
→**Print:**
Once uploaded, double-click on the ↗(file) or select the ↗(Print) option to start printing.



/CONSOLE/

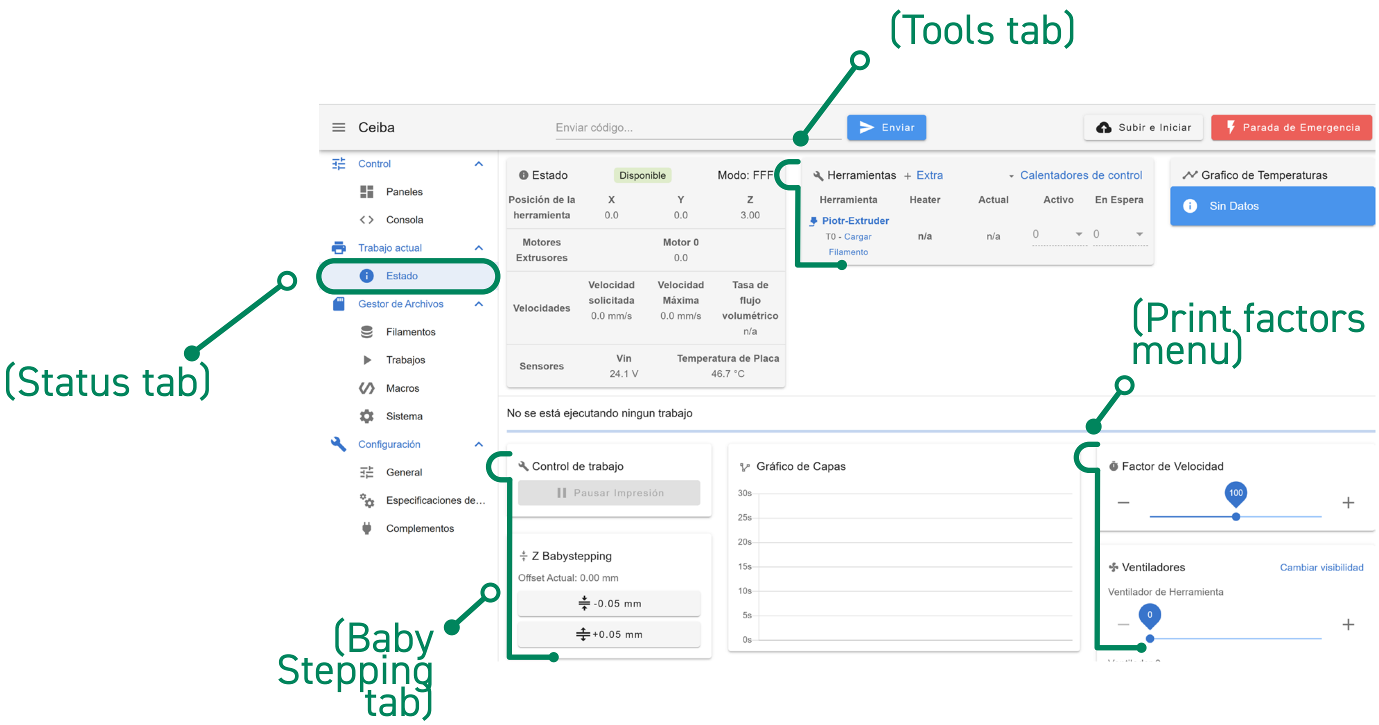
→**Send G-Code commands:**
From the ↗(Console) tab, commands can be typed in [such as M503 to view the current configuration, G1 X10 to move the X-axis, etc.

→**Read messages:**
Notifications, warnings and firmware confirmations [e.g. temperature sensor errors, end of stroke, etc.] are displayed here.



/STATUS TAB/

In this view, you will see the current status of the print and you will be able to adjust three fundamental parameters to optimise the quality of the part:

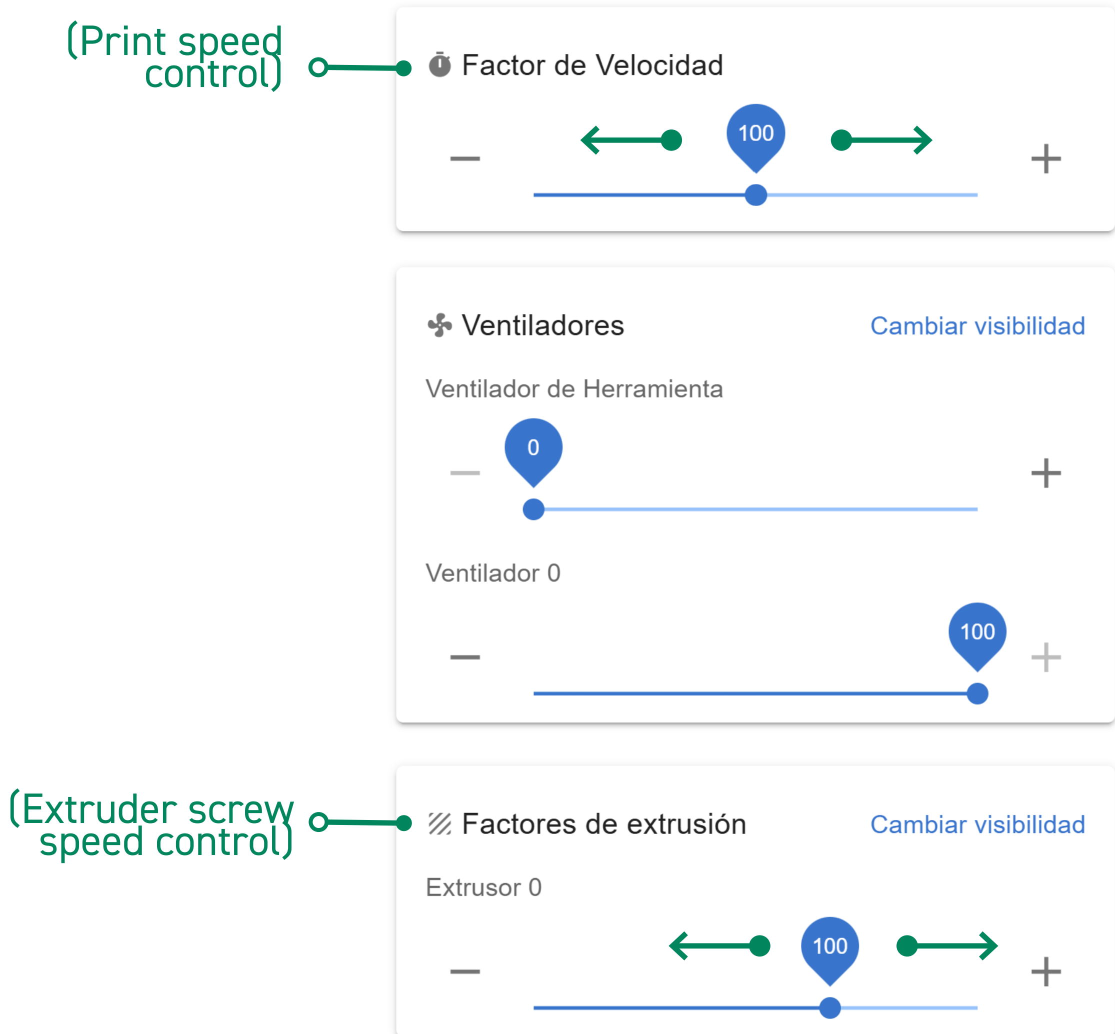


→**Speed Factor**
Controls the percentage of the print speed.

If you notice quality problems [e.g. the machine moves too fast and the layers are not deposited correctly], you can decrease this value to reduce the overall speed of all printer movements. On the other hand, if the printer is running stably and you want to speed up the process, increase this percentage.

→**Extrusion Factor or Flow**
Adjusts the percentage of extruded material.

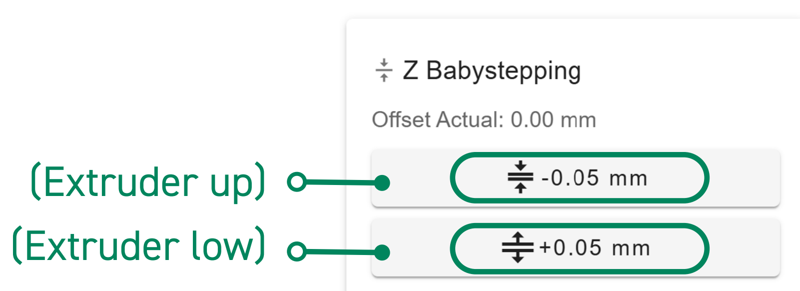
These changes are applied to the extrusion flow in real time, without having to stop printing. Increasing it can be useful if layers look hollow or the object appears to have low material density. Decreasing it can help if you notice excess material or certain bulges in the walls [overextrusion].



→**Baby Stepping**
Allows the nozzle height [Z-axis] to be adjusted in very small increments, without stopping or pausing printing.

Each increase or decrease slightly modifies the distance between nozzle and bed, improving adhesion and the initial quality of the part.

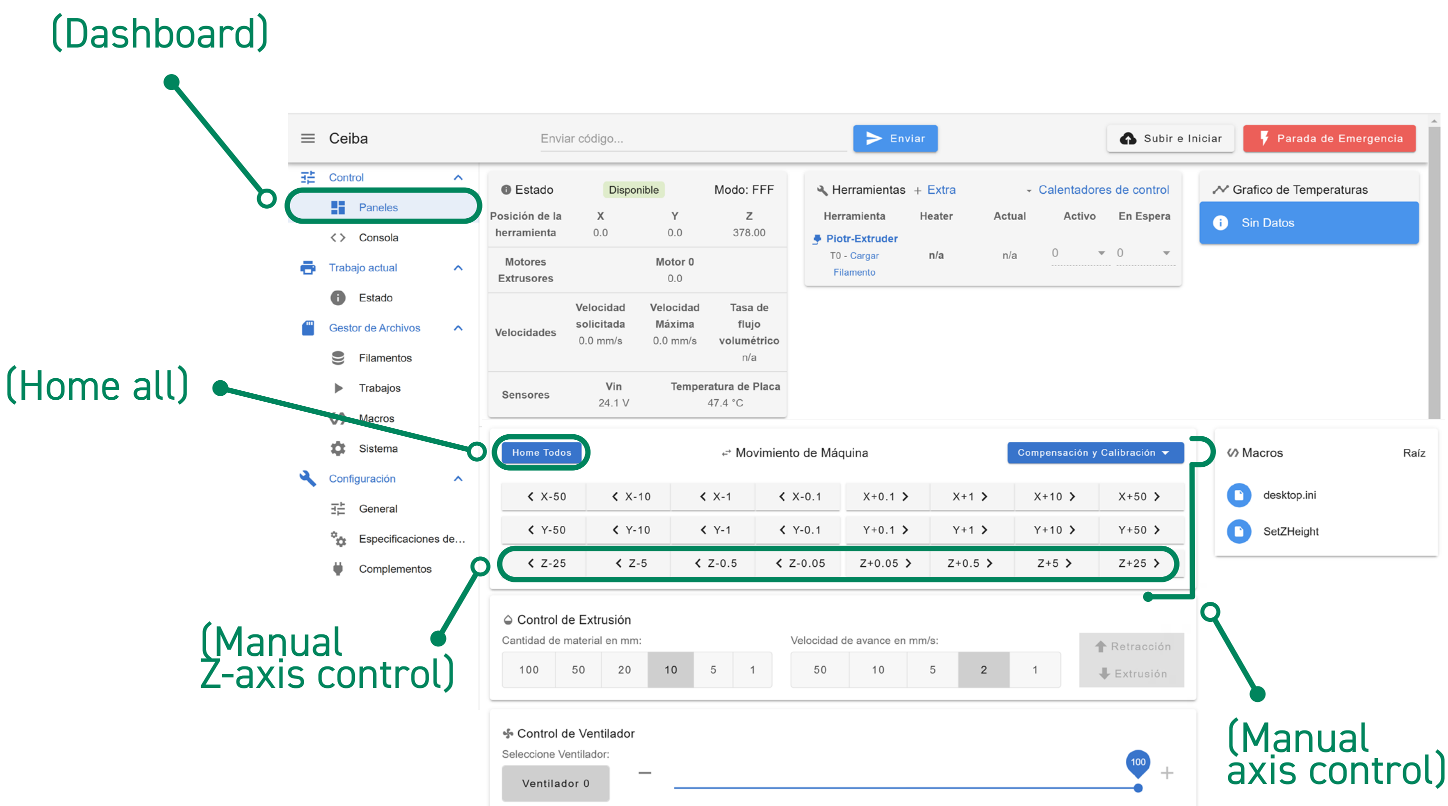
This is essential when you notice that the first layer is too flattened or, on the contrary, separated from the print bed.



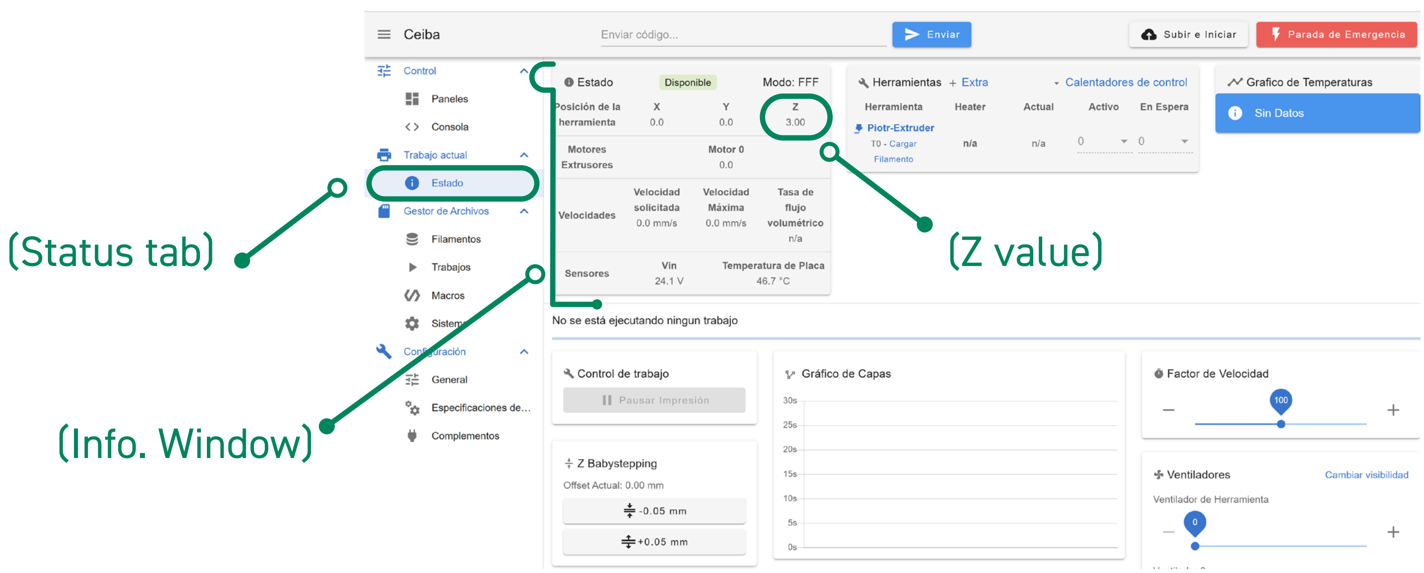
/Z=0/

Before starting to print on the JetClay MINI, it is necessary to place a build plate on the metal base of the printer. This plate, usually made of wood or plaster, acts as a support during printing and makes it easier to remove the finished part. The parameter Z=0 refers to the distance between the printer nozzle and the build plate. Setting it correctly is essential to ensure accurate alignment and good adhesion of the first layer.

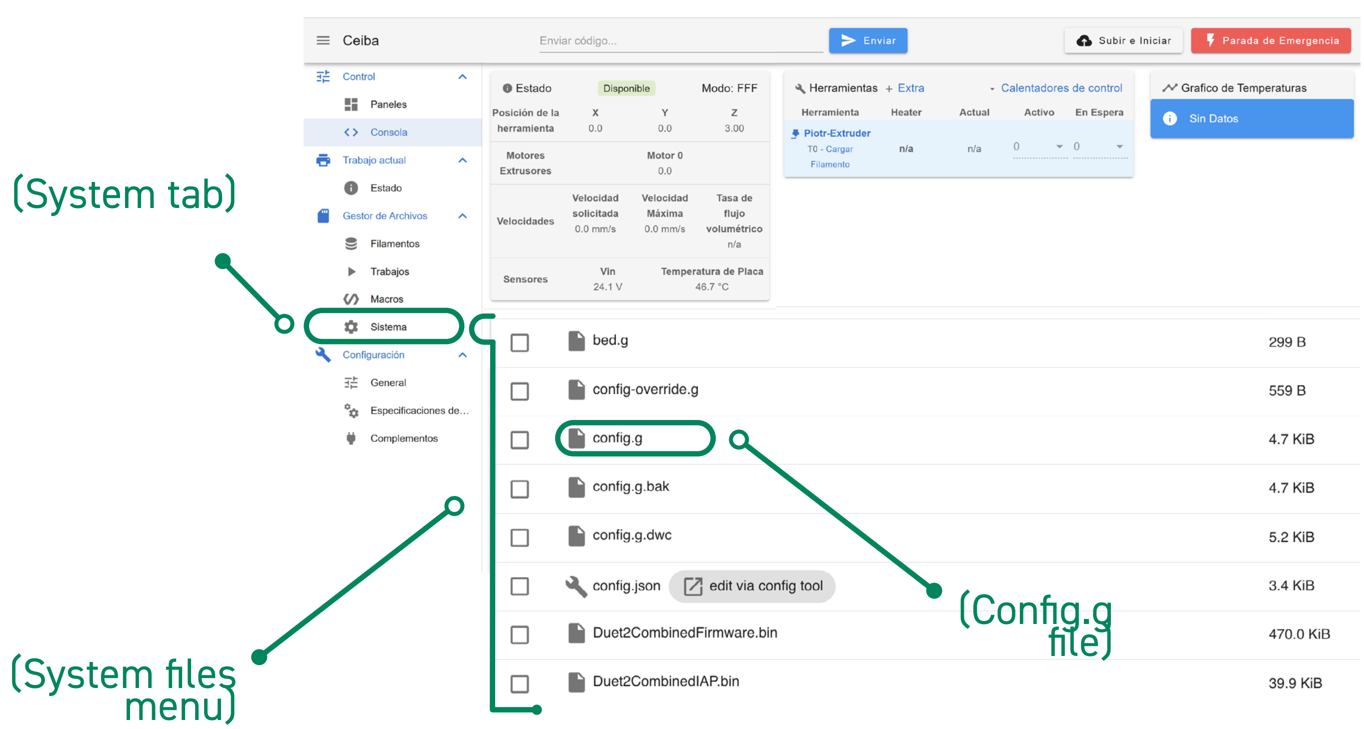
Once the build plate is selected and positioned, we perform a ↗*(home all)* to bring all the axes to their reference position. We then lower the extruder using the ↗(Z-axis hand controls), starting with wider offsets and reducing the range of movement as the nozzle approaches the base.



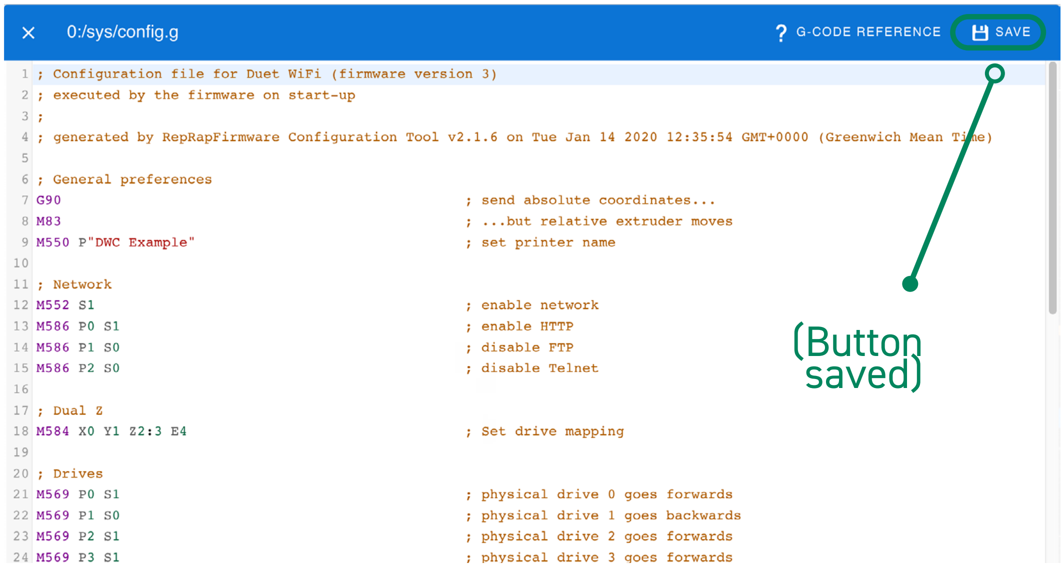
When the extruder nozzle touches the surface of the build plate, we should note the (Z value) that appears in the ↗(Information) window of the ↗(Status) tab.



Next, go to the tab ↗(System) and open the file ↗(config.g), where the printer configuration instructions can be found.



(Config.g file window)



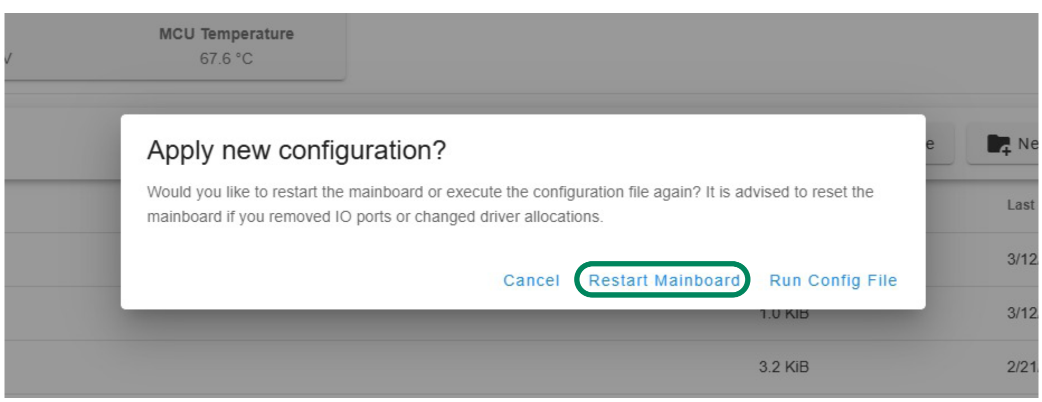
(Button saved)

```
M665 R186.511 L350 B100 H378.054 X -0.135° Y 0.135° Z 0.000°        ; Set delta radius, diagonal rod length, printable radius and homed height
M666 X0.18 Y0.18 Z-0.37                          ; put your endstop adjustments here, or let auto calibration find them
```

In this file, we must locate the value of *H* in the initial lines, which represents the height of the Z axis recorded in the last printout in relation to the construction base used at that time. To update it, we subtract the value of Z obtained in the information window from the value of *H* in the file ↗*(config.g)*, applying the following formula:

H' (updated) = H (value of H in config.g) - Z (value recorded when the nozzle touches the construction base).

After performing the calculation, we replace the old value of H in the file ↗*(config.g)*, by the new [result of the subtraction]. Once the change has been made, save the changes by clicking on the ↗(Save) button. When you do so, a window will appear asking you to restart the Duet system to apply the new configuration. It is important to remember that the modifications in ↗*(config.g) will not be activated until the Duet is restarted.*

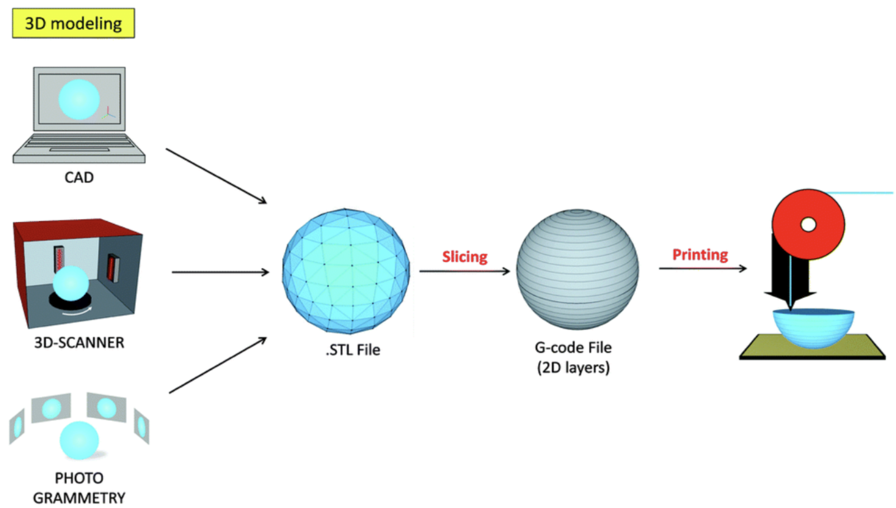


Once the system has been restarted, we perform a ↗*(Home all)* again to return the extruder to its set point with the new Z-axis height setting.

CAP.03

WORKFLOW

The 3D printing workflow follows a series of essential steps to ensure the accurate reproduction of digital models into physical objects →



03.10 3D Model

The first step is to obtain a digital model of the object to be reproduced. This model can be obtained in several different ways:

/MODELLING/
Using 3D software to create a three-dimensional model [e.g. Fusion 360, Blender or Rhino].

/SCANNING/
Capture the shape of a physical object through a scan, processing the data with specialised software such as Scan Viewer or CRStudio.

/PHOTOGRAMETRY/
Generate a 3D model of a physical object by capturing overlapping photographs from multiple angles, which are processed with specialised software such as RealityCapture [RC] or Agisoft PhotoScan.

/ONLINE LIBRARIES/
Download ready-made designs from Thingiverse repositories, MyMiniFactory, etc.].

03.20

STL File

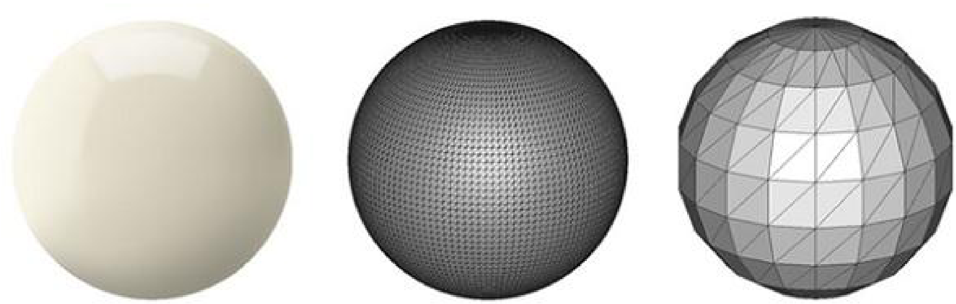
The *STL* [*STereoLithography*] format is one of the most widely used standards in 3D printing. It represents the external surface of an object in a mesh made up of triangles. Some important features::

/TRIANGLE-BASED GEOMETRY/

The resolution and level of detail of a model depend on the number of triangles that compose it. A greater number of triangles allows a better definition of the part, although it also increases the size of the file. Conversely, if the model has few polygons, the curves may appear faceted or angular.

/COMPATIBILIT/

The *STL* format is widely compatible with most slicers and 3D printers. Generally, 3D models are exported in this format from design software such as Fusion 360, Blender or Rhino, making them easy to prepare for printing.



↑ (Object) ↑ (CAD) Model ↑ (STL) Model

03.30 Slicer

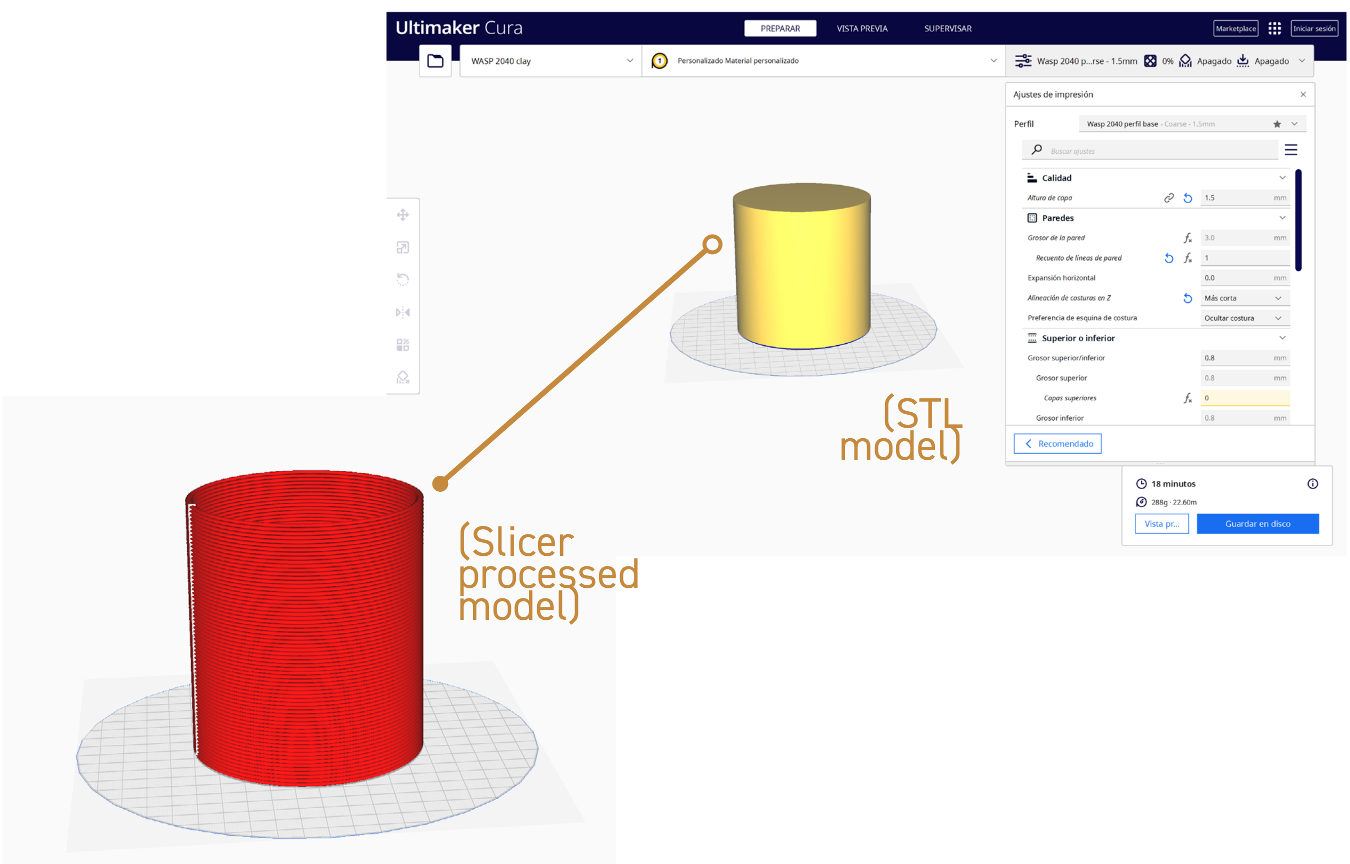
A slicer is the software in charge of 'slicing' or segmenting the 3D model [in *STL* or other formats] into horizontal layers. It also generates all the trajectories that the printer must follow to build the object. The main functions of a slicer are:

/LAYERING/
It calculates how the material will be deposited layer by layer.

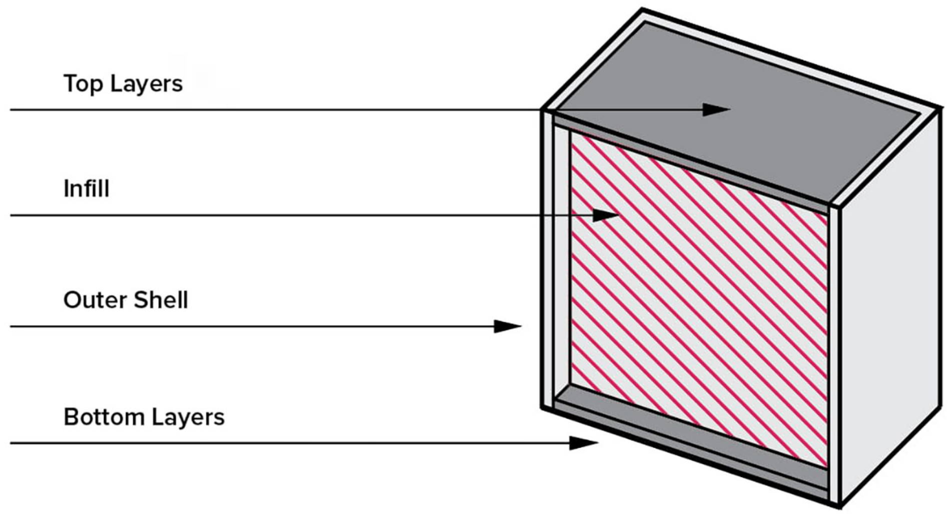
/PARAMETER CONFIGURATION/
Movement speed, layer height, extrusion flow, filling patterns, among others.

/GENERATION OF INSTRUCTIONS/
From these parameters, the slicer produces the final *G-Code* [see next section 03.31].

Cura is an example of slicing software widely used in *FDM* filament printers, but it can also be adjusted for clay printing by modifying or deactivating the temperature parameters. Although there are other options, such as Simplify3D and PrusaSlicer, in this course we will focus on Cura Slicer, as it will be the main working tool.



03.31 Slicer: Printing concepts



/TOP LAYER/

This refers to the top layer of the part, in case the design includes a closed surface on top. These are the last layers to be printed, sealing the structure on top.

/OUTER SHELL/

It is the outer layer or outline of the object, which, when printed, defines its external shape.

/INFILL/

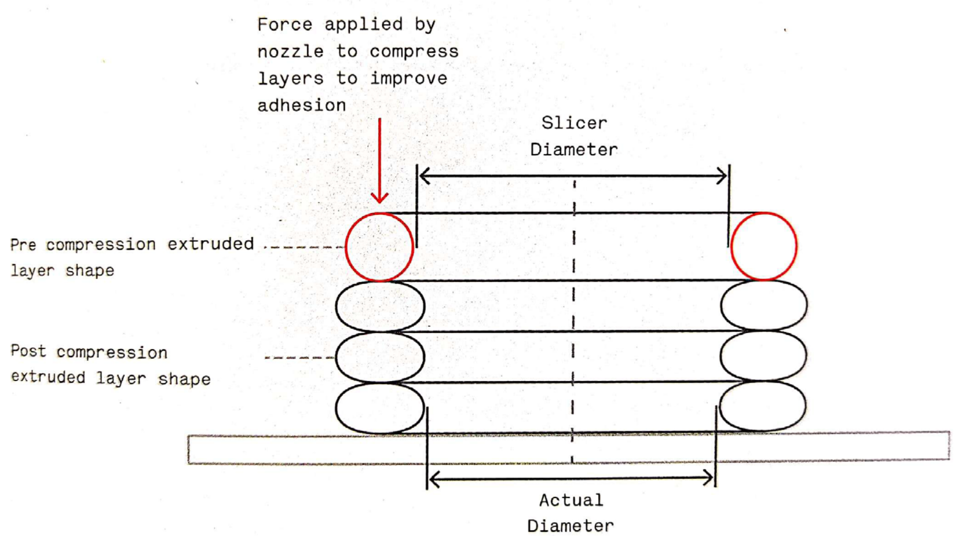
The infill refers to the infill of the part. This infill is printed in the form of a pattern and remains inside the contour layer of the part.

/BOTTOM LAYER/

This refers to the initial layer of the part when the design has a closed base. It is the first to be deposited on the printing platform and, although it can be composed of several layers, its correct adhesion is fundamental for the success of the printing process.

/LAYER HEIGHT/

This is the measurement of the height of each layer deposited during printing. In UltiMaker Cura Slicer, adjusting this parameter affects the resolution and the total printing time..



/LAYER WIDTH/

Refers to the thickness of the extruded material line in each layer. In UltiMaker Cura Slicer, changing this value can influence the dimensional accuracy and surface quality.

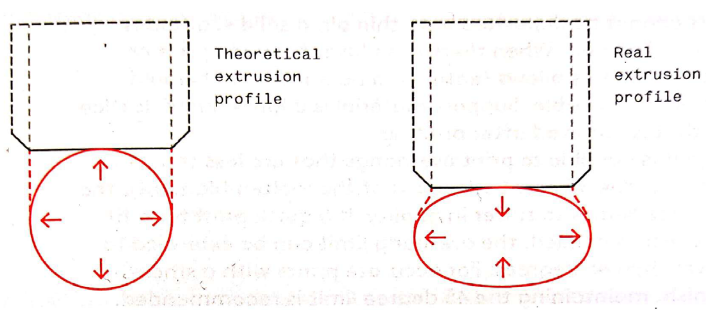
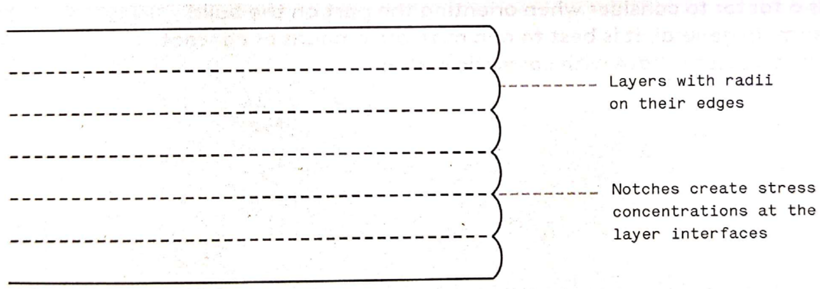
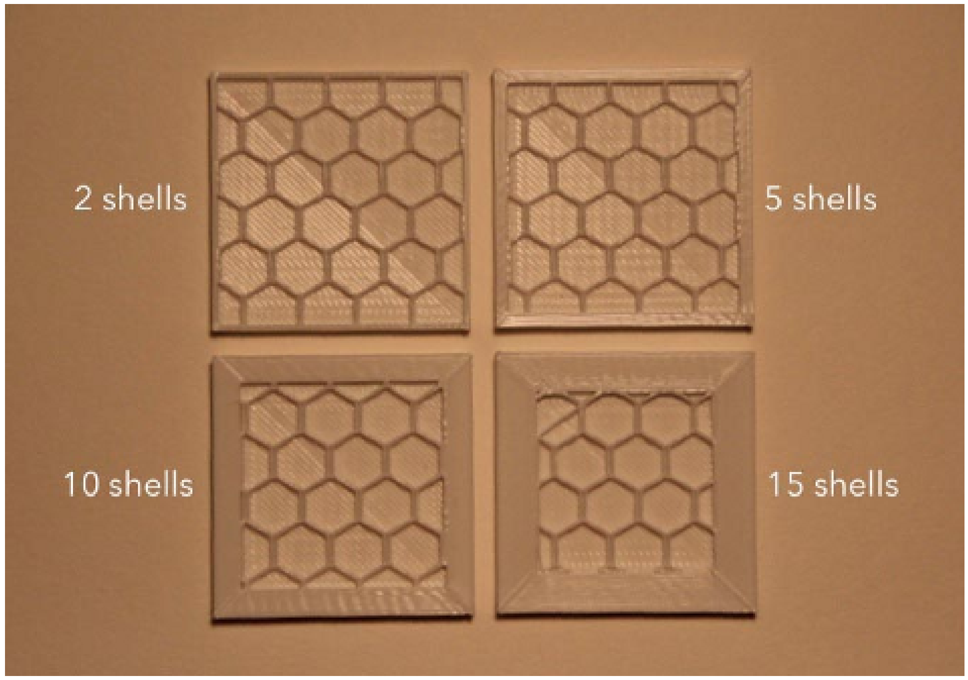


Figure 2.6 – FFF material extrusion profile



/OUTER SHELL THICKNESS/

As mentioned above, this layer corresponds to the outer contour of the printed object and determines its final shape. It can be composed of one or several layers of thickness, depending on the chosen settings. In UltiMaker Cura Slicer, setting the outer shell parameter correctly is essential to ensure a print with an optimal appearance and a resistant structure.

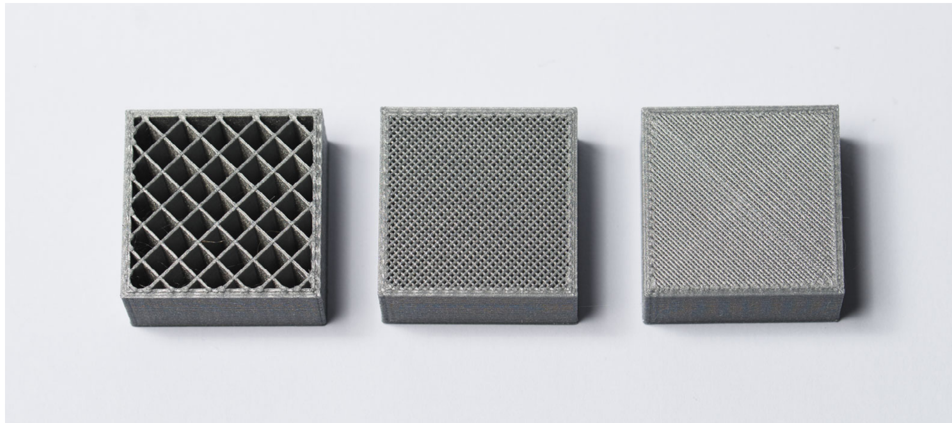


/SKIRT/

It is an initial contour line printed around the object that helps to stabilise the extrusion and detect adhesion problems before the actual printing starts. In UltiMaker Cura Slicer, the skirt is used as a preventive measure to ensure a constant material flow from the start of production.

/INFILLS AND INFILL PERCENTAGE/

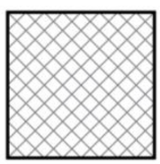
As mentioned above, the infill is the internal infill pattern that provides support and stability to the part, while its percentage determines the density of the infill. In UltiMaker Cura Slicer, these settings allow a balance to be found between weight, strength and printing time.



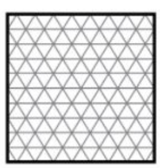
Printing of a model using the same pattern with different percentages. As can be seen, the higher the percentage, the higher the density →

/INFILL GEOMETRY/

The specific pattern or design used for the internal infill of the model. In UltiMaker Cura Slicer, the selection of a suitable infill geometry optimises stress distribution and reduces material consumption without compromising the integrity of the object.

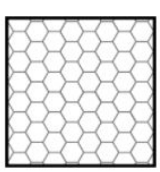


Rectangular - Standard infill pattern for FDM prints. Has strength in all directions and is reasonably fast to print. Requires the printer to do the least amount of bridging across the infill pattern.



Triangular or diagonal - Used when strength is needed in the direction of the walls. Triangles take a little longer to print.

Wiggle - Allows the model to be soft, to twist, or to compress. Can be a good choice particularly with a soft rubbery material or softer nylon.



Honeycomb - Popular infill. It is quick to print and is very strong, providing strength in all directions.

Printing a model using different types of infill patterns. Choose the infill according to the characteristics of the workpiece →

/MATERIAL FLOW/

It refers to the amount of material extruded during printing, thus controlling the accuracy and adhesion between layers. In UltiMaker Cura Slicer, adjusting the material flow is key to avoid problems such as under- or over-extrusion, ensuring print quality.

Printed part with too high material flow. The excess material causes the layers to become lumpy and uneven →

03.32 UltiMaker Cura Slicer Profile for JetClay MINI

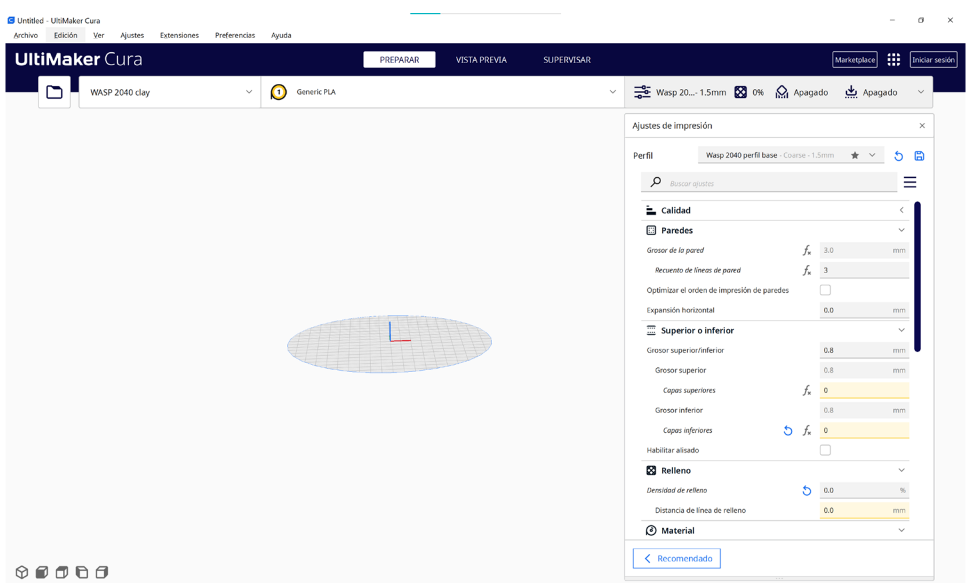
In order to generate *G-Code* files that guarantee quality printing, it is essential to correctly configure the slicer software according to the printer specifications and the requirements of the part to be printed.

In this case, we will use UltiMaker Cura Slicer and create a custom profile that fits the JetClay MINI printer's characteristics, including the essential parameters for optimal printing.

/OVERVIEW OF THE ULTIMAKER CURA INTERFACE/

→Download and install the latest version of UltiMaker Cura on your computer, which can be found on the official ↗UltiMaker.com > Software website.

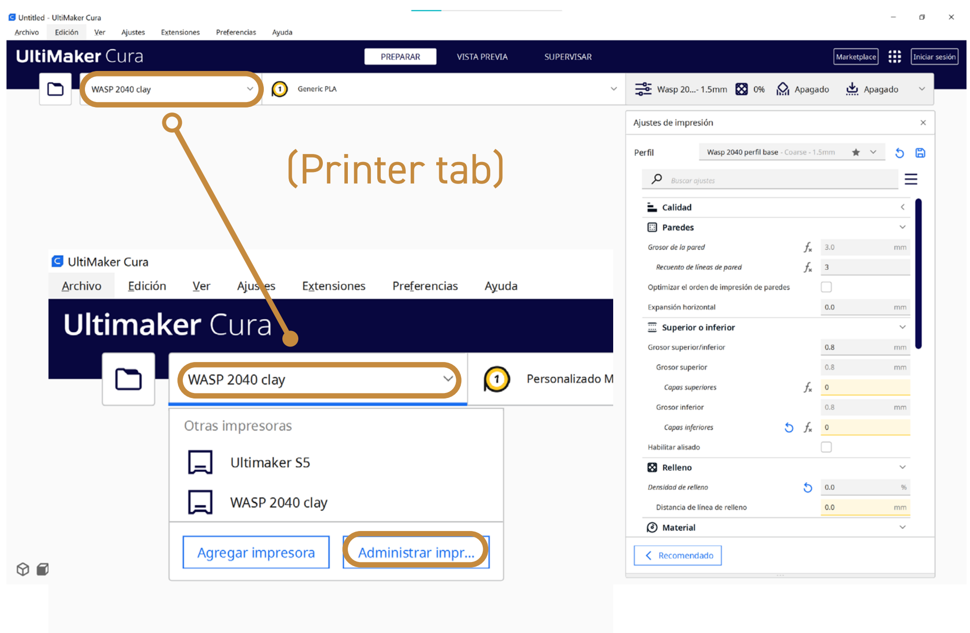
→Once the application is installed, you will be able to access the main screen of the Slicer.

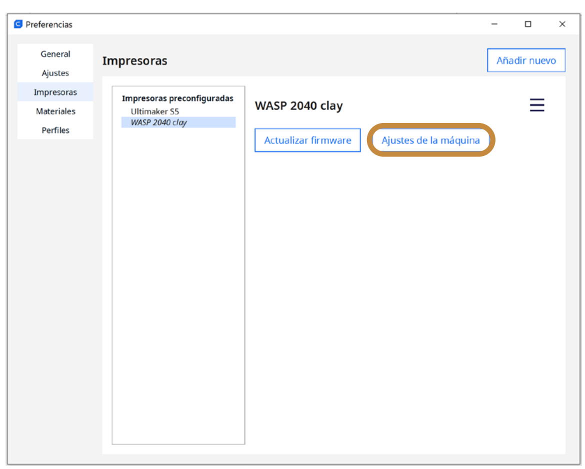


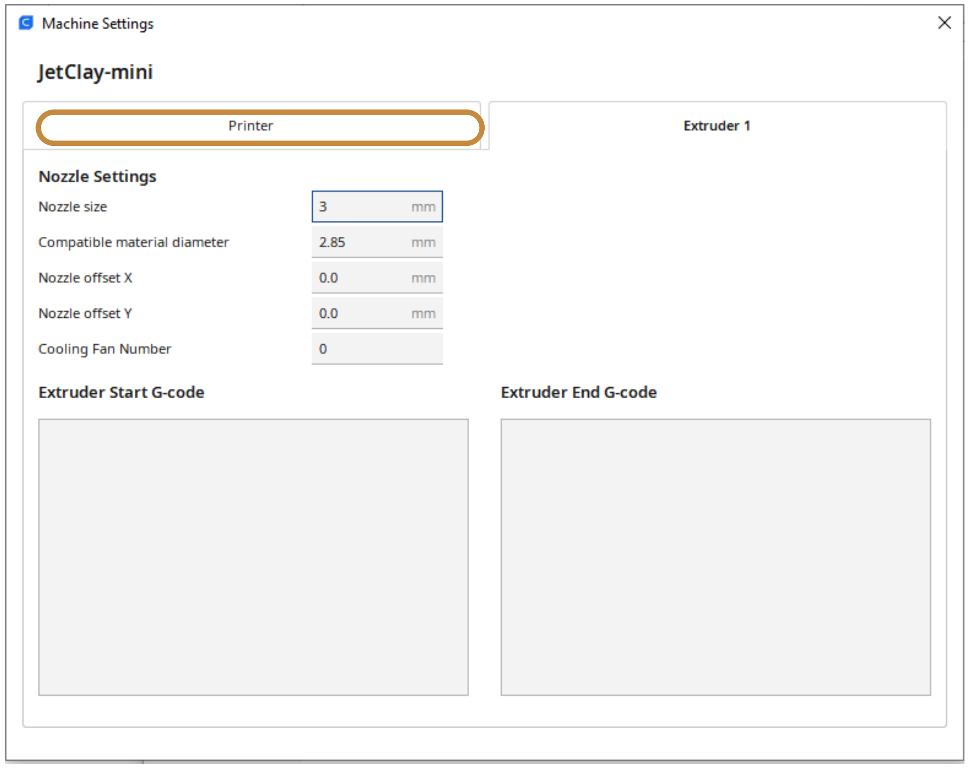
/PROFILE FOR JETCLAY MINI : PRINTER/

→ Printer settings:

→Access the ↗Printer Settings tab, add a new printer and configure its features according to the JetClay MINI specifications.

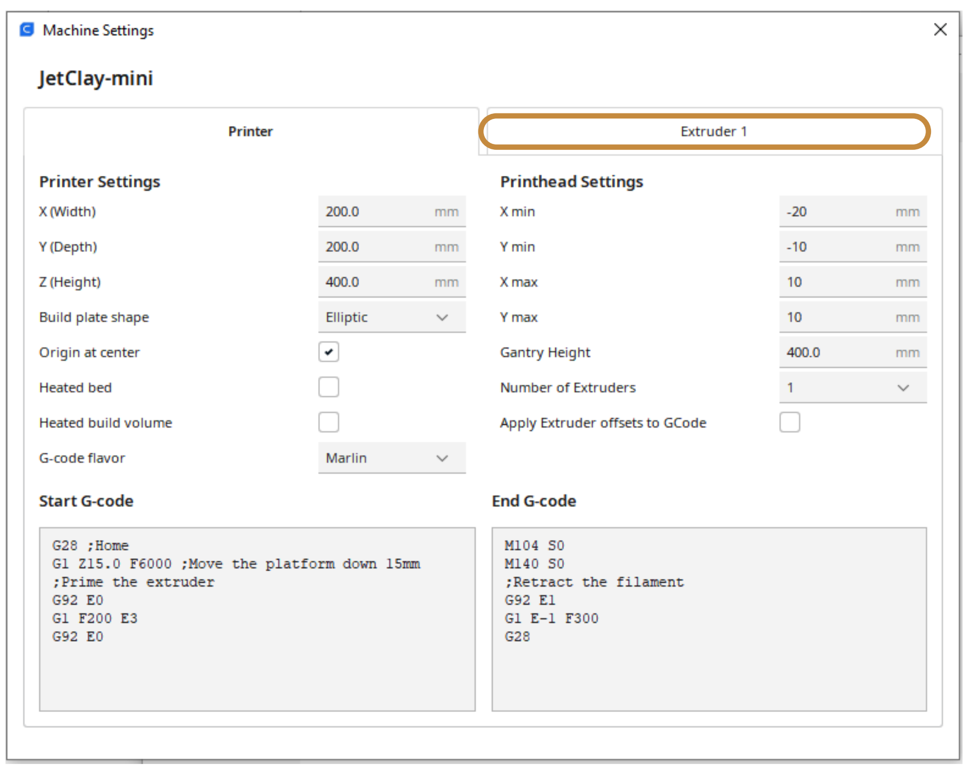






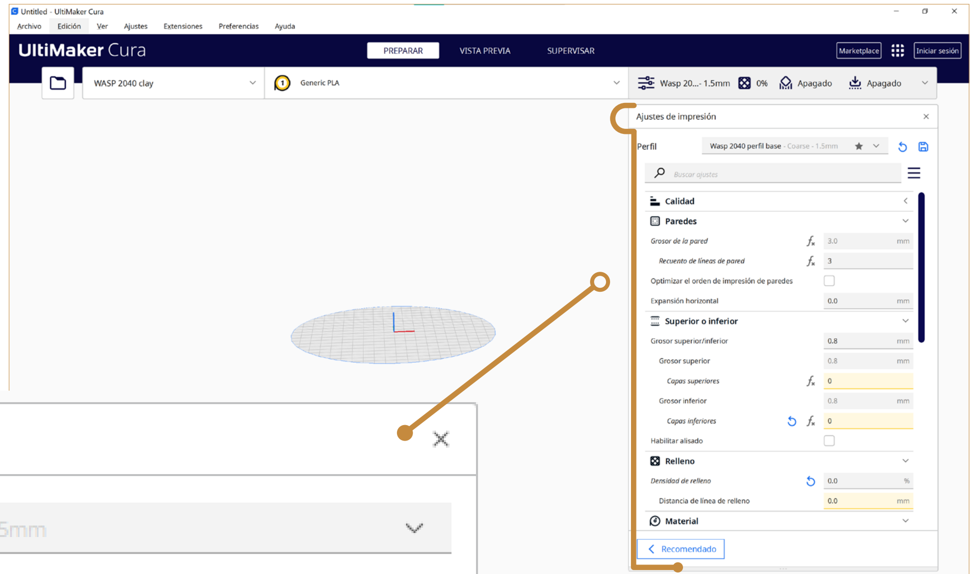
/PROFILE FOR JETCLAY MINI: EXTRUDER/

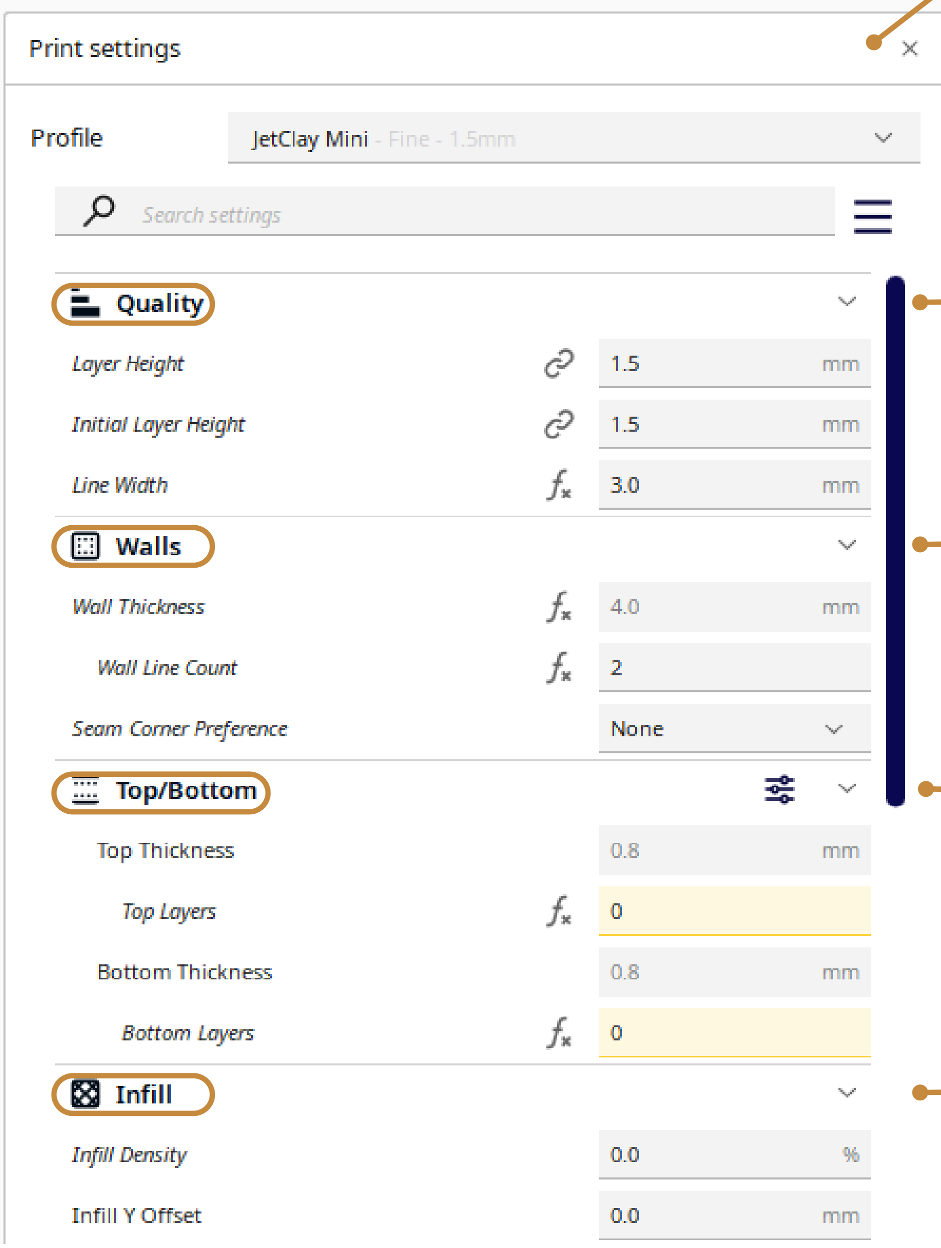
→Extruder settings:
→Switch to the ↗Extruder settings tab and configure its parameters according to the specifications of the Piotr extruder installed on the JetClay MINI.



/PROFILE FOR JETCLAY MINI: PRINT SETTINGS/

→**Setting print parameters:**
→Access the ↗Print settings menu and set the parameters according to the characteristics of the part and the expected final finish.



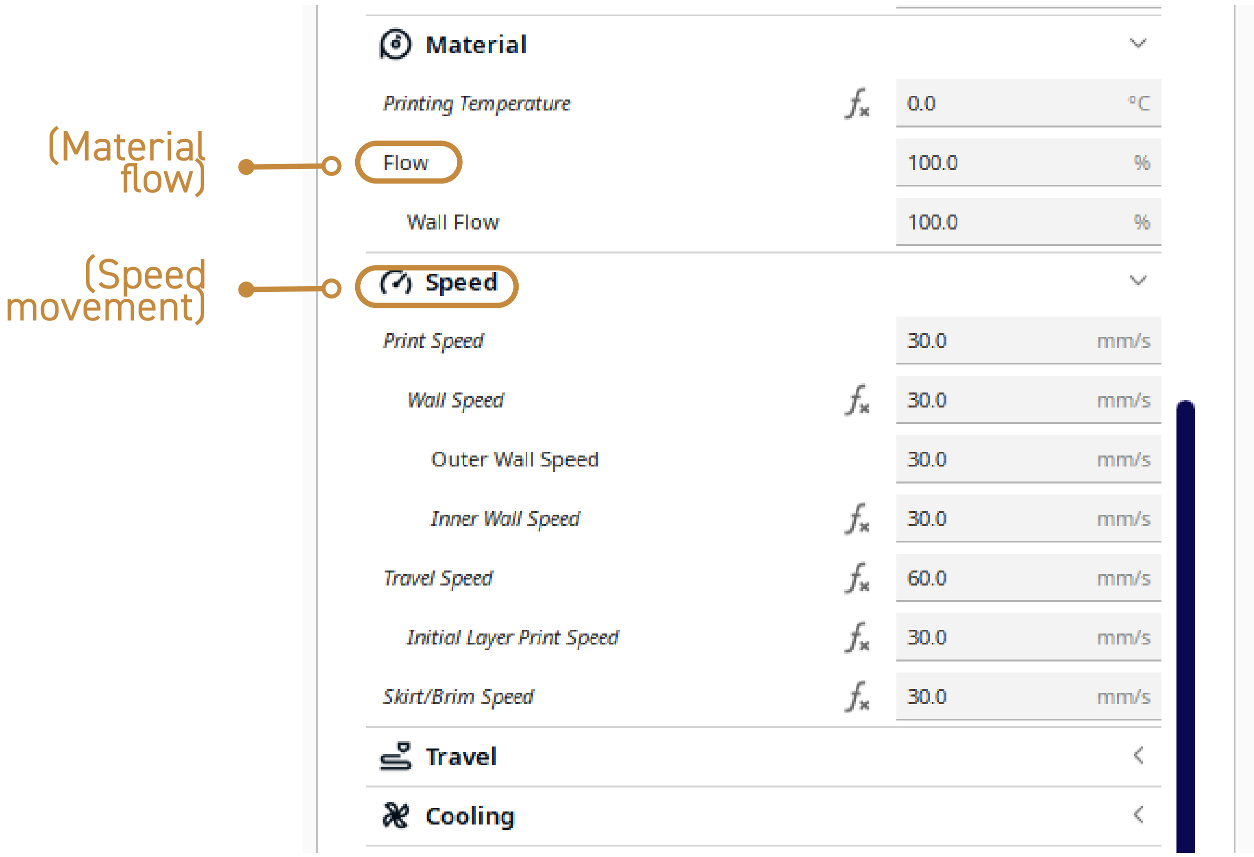


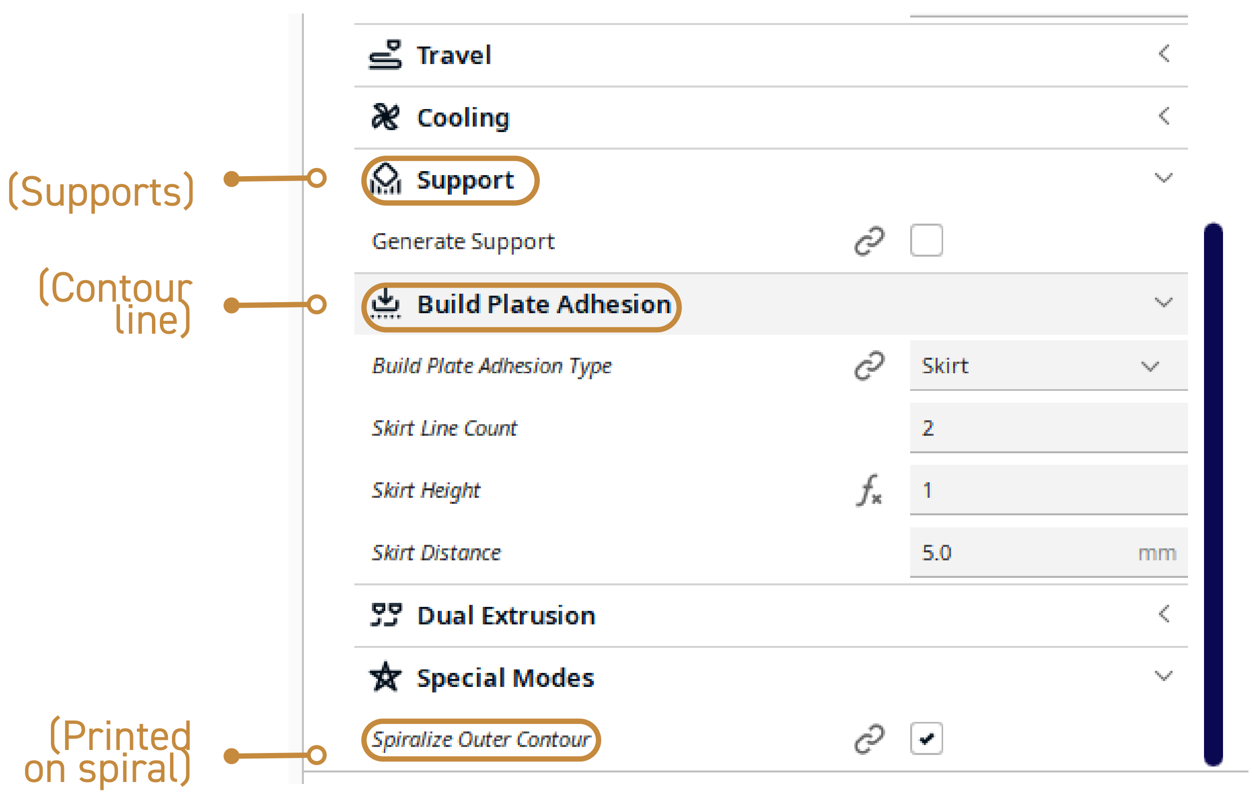
(Layer height)

(Number of walls)

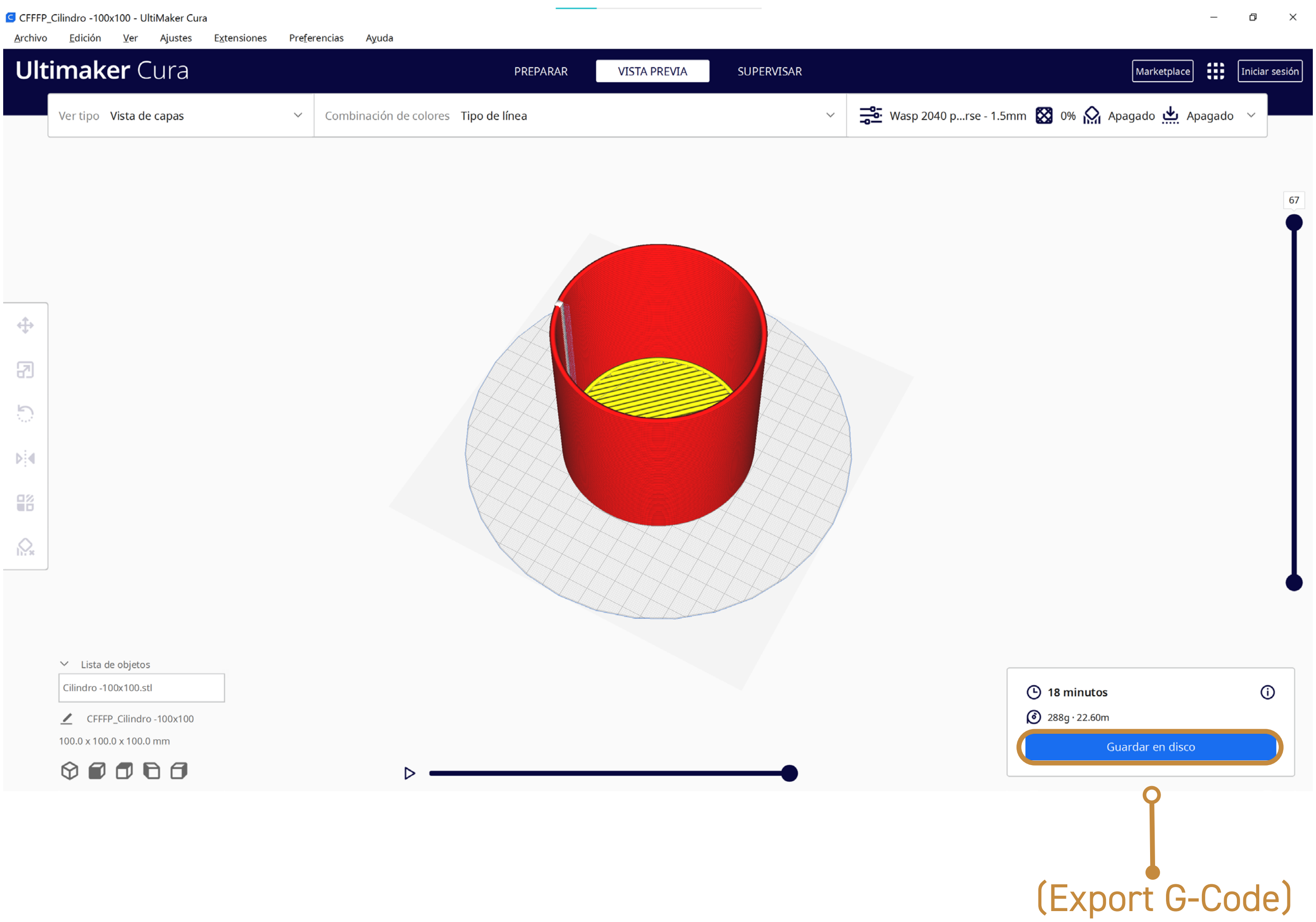
(Base / Top)

(Percentage and Infill geometry)


(Material flow)
(Speed movement)
Material
Printing Temperature
0.0 °C
Flow
100.0 %
Wall Flow
100.0 %
Speed
Print Speed
30.0 mm/s
Wall Speed
30.0 mm/s
Outer Wall Speed
30.0 mm/s
Inner Wall Speed
30.0 mm/s
Travel Speed
60.0 mm/s
Initial Layer Print Speed
30.0 mm/s
Skirt/Brim Speed
30.0 mm/s
Travel
Cooling


(Supports)
(Contour line)
(Printed on spiral)
Travel
Cooling
Support
Generate Support
Build Plate Adhesion
Build Plate Adhesion Type
Skirt
Skirt Line Count
2
Skirt Height
1
Skirt Distance
5.0 mm
Dual Extrusion
Special Modes
Spiralize Outer Contour

At this stage, it is essential to review all the parameters of the new profile created in Ultimaker Cura and verify that they match those indicated in this document, except those that have been intentionally modified to fit a specific part.



03.40

G-code generation

In the following sequence, it can be seen that the language used in the *G-Code* file is composed of several parameters. In order to interpret this type of code, it is essential to understand the alphanumeric values that make it up:

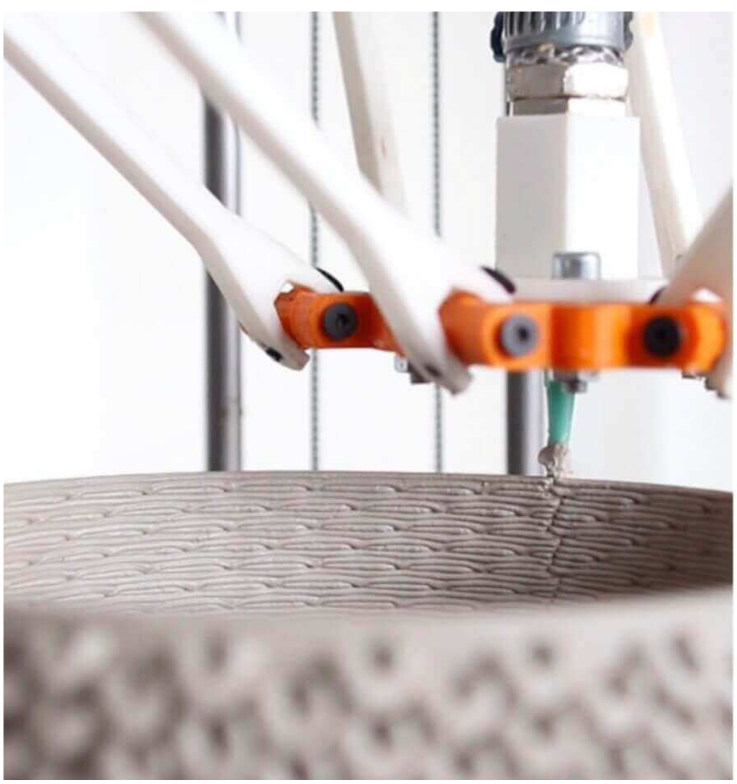
↗G1 F... X... Y... Z...E...

The letters X, Y and Z represent the three axes of the printer and define the coordinates of the movement. A value of X greater or less than 0 moves the printhead along the X-axis, while a positive or negative value of Y moves the printhead along the Y-axis. In the case of Z, any value above 0 raises the printhead.

In addition, the letter F indicates the speed at which the nozzle travels, and the letter E refers to the specific extrusion. For example:

↗G28: Performs homing of the axles.

Depending on the slicer and settings, extrusion speed, layer height, travel speed, etc. can be changed. In the case of clay printing, temperatures are ignored or set to zero, but material movements and dosing are retained.



```
;Layer count: 114
;LAYER:0
M107
G0 F3600 X96.354 Y97.539 Z0.300
;TYPE:SKIRT
G1 F1200 X99.570 Y97.539 E0.38592
G1 X99.743 Y97.444 E0.40960
G1 X100.304 Y97.219 E0.48214
G1 X101.011 Y97.004 E0.57081
G1 X101.151 Y96.991 E0.58769
G1 X101.340 Y96.950 E0.61089
G1 X101.876 Y96.878 E0.67579
G1 X102.069 Y96.869 E0.69898
G1 X102.069 Y91.739 E1.31458
G1 X109.740 Y91.739 E2.23510
G1 X113.169 Y93.365 E2.69049
G1 X113.169 Y96.819 E3.10497
G1 X123.696 Y96.783 E4.36822
G1 X124.248 Y94.339 E4.66889
G1 X128.551 Y94.339 E5.18525
G1 X130.147 Y94.739 E5.38269
G1 X131.276 Y94.739 E5.51817
G1 X131.657 Y95.118 E5.58266
G1 X133.234 Y95.513 E5.77775
G1 X133.083 Y96.535 E5.90172
G1 X135.508 Y98.947 E6.31215
```

03.50

Starting to print with the Jetclay MINI

1_Turn on the printer:

→Connect the Jetclay MINI to the power supply and turn it on.
→Wait a few seconds for the start-up sequence to finish and the printer to obtain or display its IP address.

2_Access Duet Web Control

→On your computer or mobile device, open the web browser and connect your printer to *DWC*.
→You will see the main interface ↗(Dashboard) of *Duet Web Control.*

3_Homing

→On the ↗(Dashboard) o ↗(Control) tab within *Duet Web Control*, press the ↗*(Home All)* button [or individually X, Y and Z if you prefer].

4_Z=0

→Place a printing base
→Set Z=0

5_Select the tool ("Piotr Extruder")

→In the control panel, locate the ↗(Tools) section.
Select the tool designated to extrude the clay, called ↗(Piotr Extruder) [or the name configured for your extruder].

6_Open the air pressure valve

→Raise the air pressure until the clay flows regularly, in a range between 4 and 6 bar.

7_Purge the extruder

→With the tool selected, go to the extruder control menu and find the option to extrude [e.g. 5 to 10 mm of material] at low speed.
→Watch the clay start to come out of the nozzle.

This step allows you to fill the extruder evenly and to avoid air bubbles, as well as to check that the consistency of the material is correct.

8_Setting up printing

→Upload your *G-Code* file on the tab ↗(Files) or ↗(Jobs).

9_Start printing

→Click the ↗(Print) button or open the file directly.
→Once printing is started, the *DWC* interface will automatically switch to the ↗(Status) tab, where the progress and control options during job execution will be displayed in real time.

10_(Status) tab in Duet Web Control:

→In this view, the current status of the print job is displayed and you can adjust the key parameters to optimise the quality of the part:

→**Baby Stepping**
Adjust the nozzle height [Z-axis] in very small intervals, if necessary. Very useful for adjusting the extrusion height of the first layers.

→**Speed Factor**
Control the percentage of the printing speed.

Solve quality problems in the gluing of the layers or the stability of the part by reducing the speed or speed up the process if you think the printing is too slow.

→**Extrusion Factor**
Adjusts the percentage of extruded material to solve problems of over extrusion or low material flow.

11_Pressure valve

→In the same way as the *DWC* window extrusion factors, turning the pressure valve adjusts the percentage of extruded material to solve problems of over or under flow of material.

03.50 Tips and recommendations

→**Apply adjustments sparingly:**
Changing too much at once could cause unexpected problems [sudden underextrusion or overextrusion, vibrations, etc.]. It is better to adjust gradually and observe the result.

→**Write down your settings:**
Keep track of the Speed Factor, Extrusion Factor [or flow] and Baby Stepping values that work best for you with different models or environmental conditions [clay moisture, nozzle size, etc.].

With these basic steps and settings, you can fine-tune your print in real time, ensuring that each layer is perfectly positioned and the extrusion is right for the part you are making. Using the ↗(Status) tab in *Duet Web Control* gives you immediate control over speed, material flow and nozzle height, which is essential for achieving good finishes and repeatability in your Jetclay MINI prints.