

INTRODUCTION

What is a 3D printer?

A 3D printer (see Figure 1) is a device that is able to create solid three-dimensional objects using a design produced on a computer. Currently, there are various technologies that make this possible. Your bq Witbox uses the method known as Fused Filament Fabrication (FFF), which builds objects by adding one layer of melted polymer filament on top of another until the item has been completed.

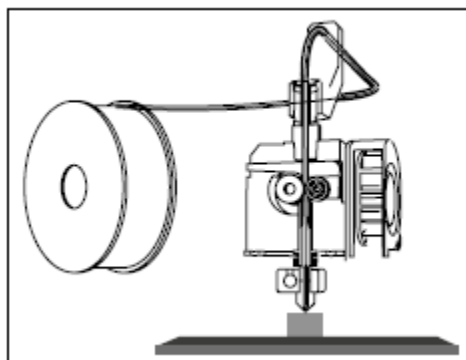


Figure 1: Diagram of a 3D printer

Online resources for the Witbox

You can find the most up-to-date version of this manual in electronic format on our website:

www.bq.com/gb/downloads-witbox.html

We've also provided a series of instructional videos that will help you get the most out of your Witbox desktop printer:

www.mibqyyo.com/videos/2014/01/16/witboxprimerospaso/

SETTING UP

Once you have plugged the Witbox in, it's ready to be switched on. The LCD screen will light up and display the name of the current firmware version.

Control panel

When you switch the Witbox on, the LCD screen will display information regarding the status of the device, such as its temperature, name, etc. Next to the LCD screen is a control wheel that you can press in order to display the main menu. Turn the wheel to left or right to scroll through the various options, and press it to select one of them.

Below are details of the options available via this menu:

Main information screen

The main information screen displays information on the status of the printer (Figure 2).

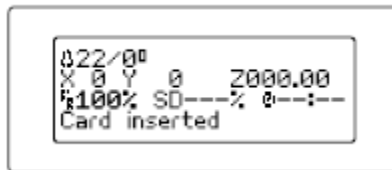


Figure 2: Main information screen

The first line of the screen shows information regarding the temperature. In this example, the figure of 22 °C corresponds to the current temperature of the Hot-End, while the figure of 0 °C corresponds to the temperature we wish to reach when printing begins (it is 0 °C because nothing is currently being printed).

The second line shows information regarding the position of the extruder in terms of the X, Y and Z coordinates. As nothing is being printed, the coordinates are all 0. If the printer was printing, the values for these coordinates would change as the extruder moved.

The third line shows information regarding several different parameters. The first figure shows us at which percentage of its maximum speed the printer is operating.

In this example, it is operating at 100% of its maximum speed. The second parameter (SD---%) shows us how much of the object (in percentage terms) has been printed, while the third parameter shows us how long the printing operation has been going on.

The last line on the display tells us that the printer is ready to be used: **bq Witbox Ready**.

Main menu

If you press the control wheel while you are on the main information screen, a new screen will appear showing the printer's two menus (Figure 3).

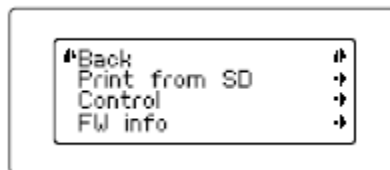


Figure 3: Main menu

The main menu contains three options: **Print from SD**, **Control** and **FW info**. This last option tells you which version of the firmware is installed.

Generally, when you access a sub-menu, the first option in the list will allow you to go back to the previous screen.

Below we will explain each menu in detail:

Menú Print from SD

The Print from SD option contains all the .gcode files that are ready to print. Press on any of them in order to begin printing (Figure 4).

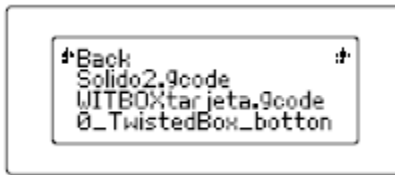


Figure 4: Print from SD menu

The importance of adhesion

The adherence of the polymer to the print base is another vital factor in determining the quality of the objects you print using your Witbox. It is therefore very important to ensure correct adhesion, particularly with regard to the first few layers of material.

Once you have levelled the print base correctly, we recommend that you do the following in order to improve adhesion:

- Make sure the print base is spotlessly clean. After you have cleaned it, try to place it inside the printer by grasping it from the sides, in order to avoid dirtying the area onto which the objects will be printed.
- Use lacquer, which will make it easier for the polymer to adhere to the print base (Figure 5).

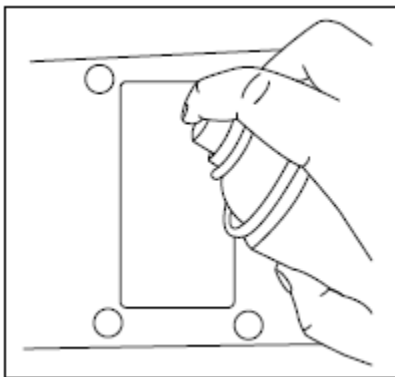


Figure 5: Apply lacquer

How to remove objects from the print base

Sometimes, objects may adhere too firmly to the print base and become difficult to remove. If you are still unable to remove the object after pulling on it firmly, try the following:

- Using a spatula, gently dig around the edges of the object in order to loosen it (Figure 6).
- Wetting the print base around the edges of the object.

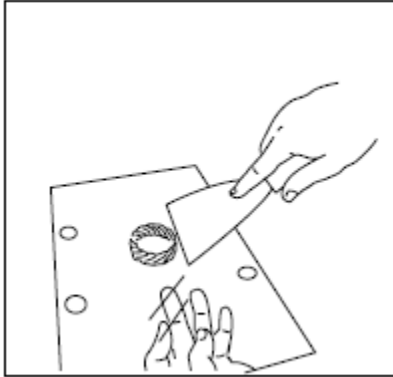


Figure 6: Remove the object using spatula

PRINTING WITH THE CURA PROGRAM

What is Cura?

Cura is a computer program designed by Ultimaker for printing objects using 3D printers. Its clear, intuitive interface lets users quickly and easily make decisions regarding the appearance and quality of the 3D object they wish to print. While users are setting the print parameters, Cura separates the design into layers so that it can be printed as quickly and easily as possible.

Installing and configuring the program

Compatible operating systems

Cura is compatible with the following operating systems:

Windows XP or later

Mac 10.6 (Snow Leopard) or later

Unix Ubuntu 12.10 or later

Minimum/recommended hardware requirements

- Minimum 512 MB RAM; recommended 2 GB RAM.
- Minimum 200 MB free hard-disk space; recommended 500 MB hard-disk space.
- Minimum Pentium 4 processor; recommended Intel Core 2 2.0 GHz processor.

Compatible file types

.STL: the file type supported by the majority of 3D design programs and the standard format for printing 3D objects.

.JPG or BMP: these formats allow for the creation of lithopanes.

.OBJ

.DAE

.AMF

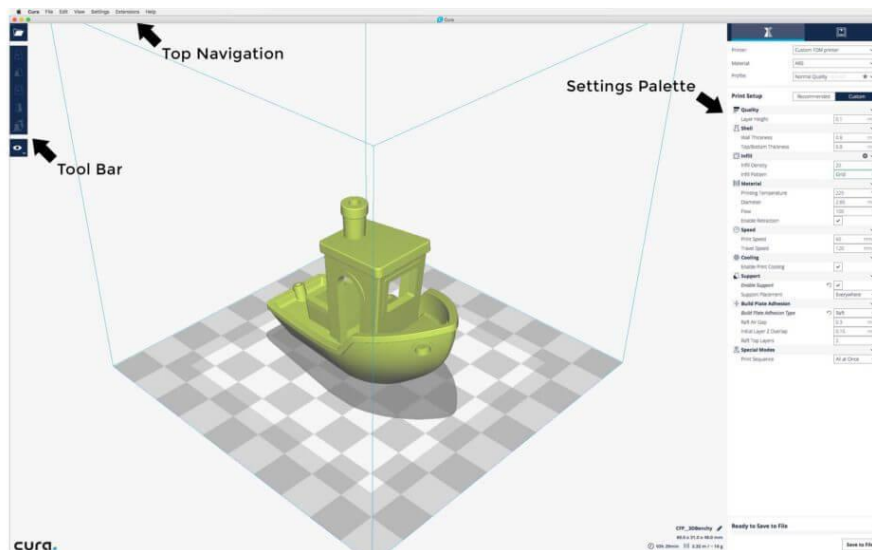
Downloading and installing the software

Cura is designed by Ultimaker. You can download it free of charge via the downloads section on bq's official website: www.bq.com/gb/products/witbox.html

Because it is a free program, Cura is constantly evolving; for this reason, we recommend that you download the most recent version for your operating system.

QUICK OVERVIEW OF THE CURA 3D INTERFACE

This screenshot shows the basic features of Cura 3D interface:



Cura 3D Top navigation

Open file: Use the button at the top left of the interface or the 'File > Open File' command to open your STL or OBJ file.

Edit: Use to select all models on the build plate, clear models and merge models. Basic commands but useful.

View Mode: This button lets you switch between Solid, X-Ray and Layers.

Settings: From here, you open the printer-specific and material settings.

Extensions: One for the advanced users, this enables you to install Cura 3D Extensions that enable manual G-code editing amongst many other functions.

Help: Get stuck then the Cura 3D help section is the place to start.

Cura 3D Tool bar (Left)

Open File: Enable you to load the STL or OBJ file you want to prepare for printing

Move: Once clicked you'll see arrows appear around any selected model enabling you to move it's position on the built plate

Scale: Increase or decrease the size of your 3D model

Rotate: Handy if you need to adjust the orientation of a model

Mirror. Once clicked a series of arrows will appear around the selected model, click to mirror the position of your model.

Select Settings. Handy features that enables you to add commonly used settings to the side bar such as wall thickness, infill and speed. These relate only to the model loaded.

View Mode: Solid X-Ray and Layers. If Layers is selected a slider will appear enabling you to quickly shift through the model layers.

Cura 3D Settings Palette (right)

Divided into two sections accessible by the material (Printing settings) or printer (Printer settings) icon at the top.

Printer Settings

This section gives you the complete low down on the material and print quality settings.

Printer. This is the printer that you selected in the first Step. If you have more than one printer then these can be setup and then selected from this dropdown.

Material. Quickly select the material that your printer is using. This option is used for printers that are tethered to the machine and will not appear if your machine doesn't have this feature.

Profile. Ultimaker's come with three print quality profiles; Low, Normal and High. These can be quickly selected from this dropdown. Once you become more familiar with the application, your printer and printing you'll be able to create your own custom profiles that are more tuned to your style of prints and modelling.

Print

Setup

Two options: Simple (Recommended (Mac)) and Advanced (*Custom (Mac)*)

Simple

(Recommended)

The Simple (Recommended) print options are calculated on the settings you input when initially configuring Cura 3D for your printer. This option is a great choice when you're just starting out and even if more experienced printers can use these presets when first using a new printer just to see how the software and printer communicate. Options are limited under the Simple (Recommended) header, but you can quickly adjust the infill, plate adhesion and basic support structures.

Advanced

(Custom)

This is where the fun really starts and will enable you to adjust the print settings, from quality through to speed. We'll look at this section and the options a little later.

Handling your model in Cura 3D

Now you're familiar with the Cura 3D interface it's time to get started and the first step is to load up a 3D file. You can either use one that you've created yourself and exported as an STL or OBJ file or download one.

Load up a 3D file in Cura 3D:

Click the open icon in the top left of the interface or use File > Open File and select the STL or OBJ file you want to use. For this tutorial I've used [3DBenchy](#) as it's a great model to use when you're testing out a new printer or software.

Now that the model is loaded you can take a look to see if the position is correct and without any adjustments the model will print. If all is OK then the model will be highlighted in yellow. If not it will be highlighted in grey and Cura 3D will notify you with a warning as to why the model won't print.

The first few steps are to check the print and make sure that it fits on the print platform. It's also a good idea to familiarize yourself with the views and how to navigate around the window in Cura 3D.

Navigate the Cura 3D build area: Hold down shift and left click to move the build area around the screen. This is often useful if you zoom into the model to check out some of the finer detail only to find that the detail you want to look at is hidden off screen due to the scale.

Rotate around the Cura 3D build area: Hold down CTRL and left click to rotate around the build area. This is handy for checking the model from every angle.

Zoom the build area: Use the mouse's middle scroll wheel if you have one to zoom in and out of the model. If you don't have a mouse with a scroll wheel I would highly recommend getting one.

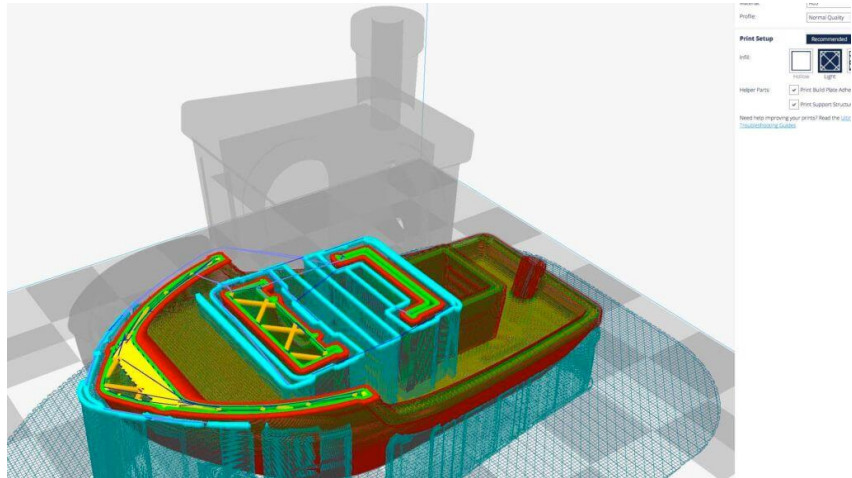
Cura 3D Tutorial: Different ways to view the model

In Cura 3D there are three basic ways to view the model and each is useful for different reasons, especially when a problem arises with your prints.

Solid. The 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. Generally if all looks OK and you've used the shortcuts just to navigate around the model then hopefully all will turn out just fine.

X-Ray. This feature is great for when prints go wrong and quickly enables you to see parts of the internal structure of your print. Where it really comes in handy is if your print is suffering from manifold edges — edges that intersect other edges. The X-Ray enables you to see what needs to be reworked.

Layers. 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 Layers. An accurate way to do this is with the arrow keys, alternatively there's a slider for quickly looking through all of the layers that build up your print. As you get more advanced with the application this feature is handy for pinpointing layers where you want to change settings in the G-code, such as increase fan speed, layer height or flow.



Adjusting your model for the Cura 3D platform. If your model needs adjusting all you need to do is click on the model so that it is highlighted and then select one of the options from the tools on the left. Here you can quickly move, rotate and scale the model.

When you click on any of the Tool options in Cura 3D you'll see the arrows appear around the model. Just grab an arrow or hoop to make the change in the direction that you want. If you go wrong then you can just right click and select reset.

If you want to print more than one model. With the model selected right click and select duplicate. Cura 3D will automatically reposition the models. If there's enough space to print the two or more then all models on the platform will be yellow. If there isn't enough space then the model out of the print area will be shaded grey.

Simple (Recommended) Print settings

The easiest way to produce a print is to let Cura 3D handle things with the 'Simple (Recommended)' options. Although pretty much everything is handled for you, you can still select the quality and support options.

Printer. Make sure you have your printer selected

Material. If you're printing tethered select the Material

Profile. Select the quality at which you want to print

Infill. Hollow will print quickly but offers no internal support, Solid will produce the strongest model but take the most time and use the most material. Light is a good starting point.

Helper Parts. These are your support and Adhesion settings. If this is your first print then switch both on. As a rule if your model has plenty of contact with the print platform then switch off 'Print Build Paste Adhesion'. If your model has no overhangs switch off 'Print Support Structure'.

Generate a G-code file

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

Save the 3D print file. In Cura 3D click either 'Save to file', 'Save to SD' or 'Send to Printer'

Estimate of time for 3D print. Cura 3D will give you a rough estimate on the length of time it will take for your printer to print the piece.

Start the 3D print. If tethered sit back and wait for the printer to fire up and start printing. If you save to SD then eject the SD card from your computer and transfer to your printer. Select print, select the file and go.

Cura Software Settings

Using the Simple (Recommended) Cura 3D settings is just the start and will produce fantastic prints, but as you progress you'll need a little more flexibility over those settings. The more advanced options can be found under the Advanced (Custom) tab. Here's a brief breakdown of what to expect.

In the print settings panel on the right of the interface select Advanced (Custom).

Quality. This refers to Layer Height. This setting will vary depending on your printer the smaller the layer height the better the quality, alternatively increase the layer height for a faster print. Layer Height is set by default 0.1 mm for the Ultimaker 2 Extended.

Shell. This refers to the thickness of any walls that your model might have, as well as the thickness of the bottom and top layer. The Shell has to be an integer multiple (1x, 2x, 3x, etc.) of the nozzle diameter. A thickness setting of 0.8 mm means that walls will be 2 lines wide (as the nozzle of our Ultimaker 2 printer has a diameter of 0.4 mm).

Infill. Used as the support structure for your models, infill is contained within your model. The higher the density the stronger and more support your model will have.

Material. Not all materials are the same and sometimes needs a little fine tuning and this is the place to do it. Usually, the only option you'll need to alter are the retraction to avoid 'Stringing'. As a recommendation always use branded filament to avoid most common print problems.

Speed. This refers to the print and travel speed. Generally the slower these are the better the print.

Cooling. An important feature of any printer and definitely worth making sure it's switched on. Fans generally don't fully kick in until the 5th layer of the print has been laid down.

Support. If your model has overhangs then you need to ensure this option is switched on, you have a choice of supports everywhere or just touching the build plate.

Build plate Adhesion. Not all models have enough surface area to sufficiently grip to the build plate. If this is the case then using these settings will help your model adhere.

Special Modes. If you're printing more than one model then this option will enable you to specify if they're all printer together or one after the other.

Quality settings

Quality: Every printer has a maximum and minimum layer height. In this box type in the value you want for the Layer Height. 0.02mm would be high quality and require a slow print, whereas 0.6mm would be low quality and can be printed with relative speed.

Shell: Select the Wall thickness. A thicker wall will create a stronger print and will generally get a better finish. If using a 0.4mm nozzle then the wall thickness has to be a multiple of that; 0.4mm, 0.8mm, 1.2mm etc.

Top/Bottom Thickness: is again usually set to a multiple of the nozzle size. Starting at twice nozzle diameter is always a good place to start. This helps to avoid common issues such as pillowing.

Material settings

Material. Every material has its own specifications and requirements and these are usually handled by the printer hardware rather than the software. The only time you should have to adjust these settings is if you are experiencing a print problem such as under extrusion.

Material	
Printing Temperature	210 °C
Diameter	1.75 mm
Flow	100 %
Enable Retraction	<input checked="" type="checkbox"/>

Enable Retraction. In almost all cases make sure that this setting is switched on in Cura 3D. Retraction pulls filament back into the nozzle as the print head moves and helps to avoid unsightly stringing.

Speed settings

Print Speed. This is the overall speed of printing as the filament flows. The slower the speed generally the better print. You can increase the speed (**the recommended is 60 mm/s**). If you do increase the print speed then it's a good idea to increase the Printing Temperature at the same time.

Travel Speed. This is the speed at which the head travels when not printing. On the Ultimaker 2 this is set to 120 mm/s. The travel speed is linked to the print speed and should be a multiple. This makes accelerating and decelerating smoother when changing e.g. from infill to shell printing.

Infill settings

Infill Density. This is the amount of material that is printed as the support structure within the model. The higher the density the higher the strength. This is usually set to 20%.

Cooling

Enable fan Cooling. Turning off the Enable Cooling Fan setting is not recommended as it will result in warping and other ugly surface effects.

Support settings

It's generally a good idea to keep the Enable Support option on unless you're sure that there are no floating or overhanging parts.

The Placement option determines where supports are attached: Touching Build plate means the supports are based only on the build plate; Everywhere means the support structures can also be built on the model or inside the model.

Platform Adhesion settings

Having prints that won't stick to the print platform is a common issue and the Build Plate Adhesion is designed to enhance the adhesion of your prints by adding a Skirt, Brim or raft.

Skirt: It's always a good idea to use at least the Skirt, this is a line around the print on the first layer that just helps to prime the extruder.

Brim: This is just a few additional lines attached directly to the object's bottom to increase the surface area and also helps to minimize any effects of warping.

Raft: This sits under the print and can be removed at the end. It's a thick grid designed to maximize your prints adhesion to the print platform.

Each option in Cura 3D has a variety of additional settings that enables you to tailor them to your model.

Generate a G-code file

Once you've finished tweaking the settings the model is ready to print. All that's needed is to export the file from Cura 3D to either a card or send it directly to the printer. Cura 3D will now handle everything converting the 3D STL or OBJ into the G-code file required by the printer in the same ways as using the Simple (Recommended) settings.

In Cura 3D click either 'Save to file', 'Save to SD' or 'Send to Printer'

Cura 3D will give you a rough estimate on the length of time it will take for your printer to print the piece.

If tethered sit back and wait for the printer to fire up and start printing. If you save to SD then eject the SD card from your computer and transfer to your printer. Select print, select the file and go.

HOW TO CREATE YOUR OWN 3D MODEL

To make your 3d design you can use any 3D modelling software.

If you do not have previous experience with any software we recommend using tinkercad for simple designs. Tinkercad is an easy to use, browser based, 3D modelling tool. It is free and because it is browser based, there is no software to download.

<https://www.tinkercad.com/>