



# QBee Workshop

## Using the Printer

### What is 3D Printing?

Easiest way to explain: Like normal printing. Imagine printing a square on a piece of paper, cutting the square out. Repeat the process and stack the additional copies over each other.

This additive process enables the creation of complex geometries that cannot be achieved through traditional manufacturing processes. It also allows for cost-effective rapid prototyping through the iterative design process. A wide selection of materials, including engineering grade plastics and exotic materials such as wood, bronze, copper, magnetic iron, etc., allows for a plethora of applications, limited only by imagination.

### How does it work?

Everything the printer does is contained in a file called a .gcode. This file is the output of the Slicer software (more on this later), which contains coordinates, temperatures, and various settings and parameters required to print a model.

Plastic filament is pulled off the spool into the Extruder and then pushed through the Cold End into the Hot End where it is melted. When extruding, molten plastic is pushed through the nozzle out onto the printer bed, along the coordinates contained in the .gcode file, creating the model layer by layer.

### Basic operation

The printer can be operated either through the LCD Panel at the front, or by using the web interface, accessible through any device that has a web browser. To use the web interface the printer has to be connected to the same Wi-Fi network as the device, or by putting the printer in AP (Access Point) mode, essentially making the printer a hotspot.

Showcase:

- Heating: Bed, Extruders (Active and Standby temps)
- Moving using jog controls
- Changing filament

## Preparation

Constraints – Thinking in 3D, three-dimensional workout for your mind.

Even though complexity is “free” in 3DP there are a few design constraints and best practices:





- Overhangs – as printing is done layer by layer going up on the Z axis the rule is that every layer must be deposited on a surface. Therefore, models which would have parts printed in the air or angles greater than 45 degrees need supports (printed scaffolding)
- Wall Thickness – if the model to be printed contains very fine details or is hollow, the thickness of any wall must be equal to, or greater than, the diameter of the nozzle. In case of a stock QBee that would be 0.4mm on Nozzle 1 and 1mm on Nozzle 2
- Print Size – Prints can only be as large as the printer's available build volume.

## Preparing models for printing

In some cases, the models created can have certain issues which would make them unprintable. One such issue would be “non-manifold surfaces”, this is when there is a hole in the surface. Models need to be “watertight” to be printed properly.

There are a number of ways and tools to fix this sort of issue.

1. 3D Builder – Windows 10's built in 3D viewer tools automatically detects and suggests repairing the model for you. It's not verbose and doesn't let you know what exactly it fixes.
2. Microsoft STL Repair/Netfabb – it is a free online tool provided by Microsoft. The model is uploaded to the server and after it is fixed in the cloud it is downloaded. As before it's not verbose and sometimes indiscriminate, removing features or closing intentional holes in the model.
3. MeshMixer – An open-source tool. It has an Inspector Tool which will highlight problem areas. The process is more involved, requiring manual fixing. It gives you the most control over how the problem is solved but it does require playing around with the software.

## Slicer & Printing

### Overview

The slicer software is used to prepare your 3D models for printing. This software has useful model manipulation functions such as scaling, rotation, duplication and a plethora of settings all affecting how the model looks and is printed.

So, what exactly is slicing? Slicing software converts your 3D models into printing instructions for your 3D printer. It cuts the model into horizontal slices, the layers, and generate paths for the extruder with a calculated amount of material to be extruded.

The multitude of different settings are grouped under their relevant headers to be found easier. There's also a search function if you know what setting you're looking for.

### Most important settings

The settings most touched upon when printing are going to be:





1. **Quality - Layer Height** – This is the main setting that will have the largest impact on both surface quality of the print as well as the time it takes for the print to be finished. The relationship is quite linear. For example, if a print would take 2 hours to be printed at a layer height of 0.2mm, the same print would take 1 hour on 0.3mm layers and it would take 4 hours on 0.1mm layers.
2. **Shells**
  - a. **Top/Bottom Thickness** – This sets the thickness/number of layers to be printed solid at the top and bottom of the model.
  - b. **Top/Bottom Pattern** – Mostly cosmetic, this changes how the top and bottom surfaces look.
3. **Infill**
  - a. **Infill Density** – This setting controls the amount of plastic on the inside of the model. Printing at 100% infill could lead to distortions on the model and is usually not recommended. For prototyping, between 10-30% is fine. For functional parts used in assembly, or exposed to stress, between 40-70% should be fine.
  - b. **Infill Pattern** – This setting determines the structure/geometry of the infill. There are different patterns which can reduce time, material use or even increase strength.
4. **Material**
  - a. **Printing Temperature** – This setting sets the temperature at which the hot end will be during printing. This setting varies from material to material or for different layer heights (mostly when going over the 0.3mm layer height mark) and speeds.
  - b. **Build Plate Temperature** – This setting sets the temperature at which the printing bed will be during printing. This setting varies from material to material but not as often as the printing temperature.
5. **Speed**
  - a. **Print Speed** – This is the general print speed. It is overwritten by other more specific settings below.
  - b. **Infill Speed** – This is the print speed when printing the infill. It can be set relatively high, as the aesthetic quality inside the print is not visible. Setting it too high can compromise the infill and the structural integrity of the print. A 70% - 85% value should work well.
  - c. **Outer & Inner Wall Speed** – These settings affect surface quality the most (outer wall especially). They should be set at 10-20mm/s increments of each other, with the Outer Wall not exceeding 30mm/s.
  - d. **Top/Bottom Speed (feature type Skin)** – This controls the speed of top surfaces of the print, or the bottom surface of a feature sitting atop a support structure. Depending on speed and temperature this can be set quite high without compromising quality.
  - e. **Initial Layer Print Speed** – One of the more important speed settings. The first layer is generally printed very slow regardless of material used. This ensures the best possible adhesion to the print bed. A value between 10-20 mm/s is recommended.
6. **Cooling – Fan Speed** – PLA loves air, so it is advised to keep this between 75-100%. On the other hand, ABS warps and shrinks when exposed to cooling so it should be turned off. Different materials will have different cooling requirements.



## 7. Support

- a. Support Overhang Angle – This sets the angle above which support structures will be placed. Usually anything over 45 can have supports, however, in some cases one can get away with higher angles. One way to find out is printing an overhang test and setting the angle according to the results
- b. Support Pattern – Different patterns use less material, less time or are easier to remove. Some work better with certain model. Experimentation is key.

## Advanced tinkering

As you've seen the number of settings you can edit is quite daunting, and to make the most out of your printer you can dive even deeper than what we've just gone through.

1. Shell – Z Seam Alignment – Layers all start printing in one point and end in another. If the start and end points are all aligned in the same spot they create a sort of a seam, or a scar, that travels up the model. You can try to hide that seam as much as possible either by using pre-defined settings or by selecting a general coordinate where the slicer will position all seams.
2. Infill
  - a. Infill and Skin Overlap – As 3D Printing bridges the digital with the physical, it can be quite analogue. The plastic is alive, so to speak, and potential inconsistencies can be resolved in the Slicer. These two settings extend the amount of printed Infill and Skin (top surface) so that there will be no gaps between the walls and the infill/skin.
  - b. Infill Layer Thickness – The infill layer height can be set in multiples of the layer height to reduce printing time. For example, if printing at 0.1mm layer height, you can set the infill layer height to 0.2 or even 0.3mm thus printing the infill lines every 2 or 3 layers, reducing printing time.
3. Material – Retraction Speed & Settings – If stringing or oozing is present, adjusting the length and speed of filament to be retracted can deal with the issue. Some materials are more viscous than others and can require these settings to be changed.
4. Travel – Combing – Combing reduces the number of retractions needed by keeping the nozzle over already printed areas. It is not advised to turn it off, however, setting it to No Skin can disable Combing over Top surfaces. This is particularly useful if you find that prints have marks on the top surface from traveling.
5. Build Plate Adhesion – Brims and Rafts – Some materials have a tendency to warp off the bed. To tackle these issues when they occur one can use brims and rafts.
6. Special Modes – Spiralise Outer Contours (Vase mode) – This setting hollows out the model and turns all walls into one wall equal to the nozzle width. Instead of printing each layer it prints continuously, gradually increasing in height, printing as the name suggests it, in a spiral. Very useful for beautiful and quick demonstration prints.



## Optimising

### Going for Speed

When trying to print an object as quickly as possible there are a number of things that can be done to achieve this. Quality will most definitely suffer in this process.

1. Increasing layer height. On extruder 1 the maximum is 0.3mm while on extruder 2 that is closer to 0.8mm. Printing at the extreme end of 0.8mm will require the speeds to be lowered and the temperature to be increased. There is only so much plastic that can be melted at any interval of time and by changing both speed and temperature we can stay within the limitations of the melting chamber.
2. Increasing speeds. For models with lots of flat top surfaces, most of the print time will go into printing the top solid layers. By increasing the speed at which the top layers are printed we can reduce the printing time considerably. We must also increase the temperature to keep up with the necessary flow paired with this speed increase.
3. Lower the number of top/bottom layers. If possible, printing less solid layers will decrease the amount of print time, especially at lower layer heights.

### Going for Quality

When time is not an issue, but having an impeccable surface finish is the goal, the changes are pretty straightforward.

1. Lower the layer height. Going down to 0.05mm layer height will almost make the layers disappear completely. However, print time will increase exponentially.
2. Lower the speeds. Printing slower will result in walls with less imperfections, less ringing (vibration artefacts), sharper corners and stronger layer bonding. Don't lower the speeds excessively as this, paired with inadequate cooling, can actually have negative effects on quality.
3. Position the model on the bed so that fine details are printed along the Z Axis. This has the highest resolution and doing so will have a considerable impact on quality.

### Iteration leads to perfection

Often, what needs to be done is changing one or two settings at a time, print a test piece, analyse the results, and then 'rinse and repeat'.

## Maintenance & Troubleshooting

### Unclogging

When switching between filaments with different printing temperatures, if the hot end is not purged or cleaned properly, the nozzle can become clogged and plastic will not be deposited. An example is printing with ABS which has a standard printing temperature of 240C and then switching to print with PLA which prints at around 200C. Any leftover ABS at that temperature will not be molten enough to be extruded and can end up covering the nozzle hole and blocking any PLA from coming out. To fix this issue one can do the following:



1. Using a needle. Heat up the extruder to above the temperature of the hottest printing material used, and insert the needle through the nozzle hole, clearing the blockage.
2. Doing a cold pull (atomic pull). Heat the extruder to 95C for PLA or 150C for ABS. Wait for the temperature to stabilise and then suddenly pull the filament out of the hot end. If done correctly this should result in the not quite molten plastic to be pulled out in the shape of the nozzle and cleaning any obstructions.

## Mesh levelling

While the print bed will be levelled before shipping, small imperfections can still be present. It is advised that if the printer is transported or anyone has tinkered with the levelling knobs, that the mesh levelling is done again. This can be done through the LCD screen or the Web interface.

## Belts

Every now and then check the belts' tension by gently striking them as you would a guitar chord. The noise should be a medium to high pitch. Belts can become loose over time and they might need to be tightened. Using an M3 Allen key, carefully unscrew the belt clip off the side of the X Carriage. Arrange the belts so that they have the same tension, while making sure the X-axis homed and aligned at both ends. Then, carefully screw the belt clip back in.

## Linear motion

Wipe the smooth rods with a clean cloth every now and then. Dust can gather in the bearings and impede smooth motion of the carriages. While lubricant should not be necessary, spraying some silicone lubricant (for plastics – do not use metal lubricants) over the rods and wiping it with a cloth will make sure that linear motion is at optimum.

## Glossary

Extruder – the assembly containing the Heat Sink/Cold End, Hot End, Nozzle.

Heat Sink/Cold End – part of the extruder which guides the filament in the Hot End. Needs to be cooled at all times.

Hot End – part of the extruder where the filament is melted.

Nozzle – part of the extruder where the molten filament is pushed out to be printed.

Printer Bed – the surface on which the plastic is deposited. Usually heated.

Oozing – when your heated nozzle drips filament without any intentional extrusion movement.

Stringing – the thin strings of plastic that may appear on a print, usually between columns.

