3D_Dual_Extruder_Qidi

December 2, 2019

1 Ylab's Dual Extruder Qidi 1

Version 2 December 2019 - includes TPU

Ylab's main 3D printer is the Qidi 1, formerly known as the QidiTech 1.
It's a full-featured unit with dual extruders that can use

- ABS (higher quality, harder)
- PLA (lower cost) filament.

- TPU - flexible like silicon after printing

We're focusing on it because:

1.0.1 Relative to ORD unit:

- ORD only supports PLA
- Has been a little more prone to jamming
- Patchy vendor support
- Poor software support

1.0.2 Relative to big Ultimate unit:

- Also dual extruder
- Quality in our tests equal or better, depending on setting
- Lower cost of materils (Ultimate must be purchased in metered cartridges)
- Better software support
- Larger bed size
- This thing is HUGE! We're giving it back.

Tip: if you see an open 3D printer (i.e. without an enclosure), you can be pretty sure it only handles PLA. ABS requires higher ambient temperature so is generally only supported in closed unit.

Different material is not just a question of changing out the material. The melt temperature, bed temperature and flow rates all vary. As the printer is adding material, you want it to bond properly to the material that is already there, so the ambient temperature is critical.

1.0.3 Reviews and other web sites

This simple github page points to some reviews and quick setup configuration and sequence.

1.1 Why dual extruders? Colours, Support and Materials

Dual extruders are generally used for three reasons:

1. Multiple colours. But you must be using the same material type for both. Temperature issues. Usually requires you to build separate parts and combine them in the slicer. More complex than we will cover. * If separate and separable parts, best to print them separately. Faster. Less back-and-forth head movement.

2. Don't have to keep loading/unloading for different material

3. Support material. Think of it as a scaffolding that you use to build a complex part with an overhang or internal passages.

Support is provided in one of two ways:

- **Use the same material and extruder if it can easily be removed.** The software will attached it to the main stucture with tiny filaments that are easily broken off, and if reqquired, sanded away.
- **Use dissovable support material that is compatible with your primary material.** This is used for complex parts with internal passages.

Support material comes in a filament spool just like the material you are using to print.
The support material must be compatible with the primary filament

1.1.1 Advantages of using same material/single extruder for support

- **Better quality**. Pauses to switch material causes stringy "drips".
- **Faster print**.

  - 3D printing goes from the bottom up, so if you require both filament types at the same vertical levels, the printer will constantly be shifting from one material/extruder nozzle to the other.
  - Each change requires a position re-calibration, and usually leaves inconsistencies on the surface where each melt stop and restarted.
  - You can generally expect the process to take 2 to 4 times longer.

2 The 3D print process steps

1. **Get your design**

   Either make it yourself with a 3D CAD program, or pick one up off of the Internet
   The process usually generates one or more **.stl** files. This is the standard for 3D design output.

2. **Select your material and process/printer**

   Basic parameters/requirements * Strength * Temperature resistance
   Advanced parameters/requirements

   - Weight
   - Other environmentals - chemical/gas exposure
   - Wear if moving parts
   - Number of parts (speed required?)

- Multi-coloured parts - including transparency
- Material textures (soft? hard?)

Prototype vs final product

- Get the design right first with cheaper material
- Switch to more expensive material and different commercial shops/processes for final product
    - High strength nylon (powder/binder) - HP has printers
    - Metal - laser sintering; print with binder/de-binder/furnace sintering/shrink
    - Various liquid bases - right up to dental implant quality

3. Generate the production code for your process/machine
You do this using a slicer for the process/machine
The best will cover:

- Support material as required
- The full feature set of your machine
- Internal wall structures/density
- Various parameters for speed, quality, etc.

4. Load your material on your printer
5. Set up the printer parameters for your material
6. Load your design into the printer
7. Wait patiently.

3   No, we are not teaching you 3D CAD today

- Our parametric-focused 2D CAD class is the foundation you need
- We can't do it justice in an evening class. You will need to work at it.
- AutoCAD Fusion 360 available for free download
    - Can be a pain getting through the process. Keep working it and don't call us
    - Not the same as commercial AutoCAD
        * Interface has significant differences

- FreeCAD is a free open source 3D design package that is getting a huge amount of development effor. There are dozen of on-line video tutorials
- Many options and many great youtube tutorials
    - Read reviews at sites like this one
        * Beginner to pro recommendations
        * Links to tutorials
    - Go through a tutorial or a youtube video
    - Some have cool features like being able to scan a part from your phone into the CAD program!
    - **Different programs have different interfaces** and one or another may fit your experience or way of working.

4 Comparing Materials

Simplify3D's materials pages are excellent.
  Highly recommended reading for:

- Best use of materials
- Printer settings.

4.0.1 Let's have a look now

- ABS page
- PLA page
- TPU page

TPU is not officially supported on our Qidi printer. Ylab member Bryan has experimented extensively with it and has consistent success.
  Critical Items for TPU: * SOURCE OF MATERIAL IS CRITICAL. Some just don't work. * He's had most success with this one from SainSmart. * The settings further down in this document are for the SainSmart material.

4.0.2 Unidentified filament roll?

- Use a micrometer to check diameter (1.75 mm for our Qidi)

How to tell the difference? Bend and. . . * PLA breaks * ABS bends. . . but breaks eventually * TPU bends. . . and will not break

5 When to Use Separate Support Material

5.0.1 When do you need to use it?

- If support material is accessible on the final print, use same material/extruder as primary material
- If support material not accessible (i.e. internal passages you can't reach with fingers, etc), use separate dissovable support material

5.0.2 Avoid separate support material wherever possible

- Slicer links support material with extremely fine and usually breakable connections.
- Time to print goes up about 4X in our tests because of switching between extruders for every tiny layer

Pick the right one for the primary material you are using. Must be compatible with both the printer and the base material.

5.0.3   If you need it, which support material?

Ultimaker's page on support materials provides some great examples on how it's used.
   FOR ABS: HIPS (High Impact PolyStyrene) and more info here. Soluble in d-Limonene.
   FOR PLA: PVA (PolyVinyl Alcohol) is the material of choice. Dissolves in water!
   FOR TPU: At this time there is no clear recommendation. Some sites mention ABS because it adheres very well to TPU, but there is no solvent for ABS if the support is in internal passages. This site recommends PLA, but you will have the same solvent issue. For both materials, they recommend cutting away, not pulling off the suppport material.

5.0.4   Additional considerations for TPU

Because TPU is flexible after printing, you need to consider the layout of the print.
   Read these pages if you want to work with it:

- Best practices for TPU
- How to succeed with flexible material
- Cut away (don't pull off) the support material. It sticks too well.

6   Materials for our Qidi

6.0.1   No, ylab is not supplying you with material.

6.0.2   Amazon.ca might be your friend... and there are local suppliers

- Read the reviews. Quality can vary.
- Consistency of reviews is important. Seller may be jumping between suppliers

6.0.3   Our printer requirements: 1.75 mm filament. No exceptions.

- Applies to all filament types (PLA/PVA and ABS/HIPS)

7   Slicers

G-code is a numerical control programming language used to control lathes, CNC machines 3D printers and other devices. The slicer takes that design from the .stl file and generates the G-code to product it on the machine you are using.
   A 3D printer melts filament to build your design from the bottom-up, one very thin layer at a time.
   That's where it gets complicated. A lot of decisions can be made that will affect the quality of the part, the strength, the amount of material you will use, the finish, and the time to print.
   A good slicer will make a tremendous difference - and there are lots of different options. A given slicer program may not support the printer you are using or a feature you want - like, for example, dual filaments or support material.
   Slicers generally fall into 4 categories:

- **Machine-specific slicers.** These are often proprietary to the manufacturer. In some cases, they are based on customisation of an open-source product.

- **Commercial products**. These will support a variety of printers and come with support - but they cost money.
- **Open source products**. Free, but you get what you pay for. If your printer is directly supported, good for you. If not, you could spend more in time and effort than the cost of a commercial product.
- **Websites to generate G-code**. There are multiple website where you can upload your .stl file and generate the G-code. When we read the fine print, we found that most of them use an open source slicer as the back-end.

7.1 Slicers for our QIDI 1

7.1.1 Free: Default QIDI slicer

The QIDI web site has all the details on the printer.

The software page provides a slicer program for download. According to a message that shows up when you load it, the program is based on Cura - but there is no information on which version of Cura. See below for more info on Cura.

The software is available for Windows and Mac.

JUST USE IT * Has all the Qidi features supported - materials, dual extruders * Includes the .x3g converter which you have to load separately if you go with Cura * You can contact Qidi with support questions. They have a good reputation

8 Other slicer options:

8.0.1 Not Free: Simplify3D: The Gold Standard

This is a commercial product that directly supports our Qidi 1.

Simplify3D is widely respected for its quality output and support of the widest range of printers. It's affordable at US$149 and comes with a 2-week money-back satisfaction guarantee. If time is money, put a crowbar in your wallet and buy it.

Remember: even if you have all the time in the world, *the person you are going to for help to avoid paying the $149 might not*

8.0.2 Open Source: Ultimaker's Cura

Cura is considered by some to be the equal of the popular non-free Simplify3D.

If the Qidi software is based on Cura, why bother? Here are some good reasons:

- You may want to run it from Linux, which Qidi is not directly supporting.
- You may want to run the latest Cura version is always a little behind.

And here's the biggest reason not to:

- Support. If you use the Qidi software, you can contact them when you need help.

Cura can be downloaded here for Windows, Mac and Linux. It comes pre-configured for lots of 3D printers, but not the Qidi.

Since the Qidi software is based on Cura, you can configure Cura for the Qidi as follows:

- Download the Qidi slicer (see link above) and install it on one of the supported PC types (Mac or Windows)
- Read this page on how to add new machine profiles to Cura. It specifies in which directories the files need to be added. But where do you get the files?
- Pull the files from your Qidi install - same directory names! - and copy them to your Cura install.
- If you are installing Cura on Linux and your Qidi software is on Mac or Windows, you may need to run your conifg files through a quick converter to handle the line-ends properly.

If all this seems complicated to you... just use the Qidi software. Because that way you can always email Qidi for support.

Cura tutorial here

These sites provide samples of startup and shutdown gcode, and info on copying the gcode from the default Cura slicer.

From thingyverse's forums

From the Ultimaker forums

This 2017 site mentions parameters for free Cura 2.7.0, but recommends Simplify3D as best.

8.0.3 Slic3r

Slic3r is extremely sophisticated in its capabilities and has a hard-core following of developers and users. It's reported that many advanced features originated and continue to originate on Slic3r.

The interface isn't as friendly as Cura, and you will have to do more research for the setup and configuration. It expects you to load all your configuration information. If you check the information above on setting up Cura and the Qidi default slicer, you should be able to find all the parameters required in those config files.

8.0.4 ReplicatorG

ReplicatorG was previously the default slicer that came with the Qidi and may still be found on the SD card. It is limited in capability and does not support dual extruder or support material.

This example shows you how to use ReplicatorG with dual extruders... if you have separate .stl files generated for each filament type. It claims you can download some examples from ThingyVerse.

8.0.5 Octoprint - 3D printer controller

Octoprint provides a web interfce to your printer using a Raspberry Pi board. It front-ends Cura and allows you to load your project file remotely instead of slipping in an SD card. We haven't tested it yet with the Qidi 1.

It apparently will also work with a Slic3r plug-in.

This page deals with Octoprint and Qidi1.

8.0.6 Other reviews and opinions

All3DP has a good review of free and not-free options.

8.1 Loading your design:

9 SD CARD - BECAUSE USB CONNECTION IS BIGGER PAIN

- Design is **not** loaded over USB - it expects PC to trickle the print over USB and directly control
- Some software will handle it - but not our Cura/Qidi slicer
- So… your PC is stuck there.

OPTION: Running Octoprint from a Raspberry Pi as 3D print server… if someone wants to put the work into it * Ross has donated a Raspberry Pi to the cause

9.1 FULL SIZE SD CARD REQUIRED - or smaller chip with adapter.

9.2 BRING YOUR OWN WITH USB ADAPTER. Because they are delicate and easily lost.

10 Using the QIDI Slicer Program

Project file must be in .stl format. Supported by most CAD programs
Program is called Qidi Print (in Windows)

1. Start the program **Qidi Print**

2. Open the .stl file you want to print

   - __File -> File Open
   - Select the **.stl file**
   - Object will be loaded and shown on platform graphic

3. **Click on the graphic of the object**.

   - This enables menus on the left and right side of the screen
   - Note how colour of object matches extruder selection near top right of window

4. Select extruder by clicking on appropriate color box on left. If you change, note how your object changes colour

   - __IF YOU FILE HAS MULTIPLE OBJECTS, YOU CAN SELECT MULTIPLE EXTRUDERS.
     - Usually only useful if you are using two different colour filaments
     - Exception below for separate support filament

5. Set position on plate or rotate or scale if required

   - Buttons 2,3 and 4 from top on left menu

6. Select support requirement from left menu

   - Select either block support for no support, or custom support

- Custom support:
  - Extruder: Select same, or other extruder if using separate support material
  - Resolution: this is the gap between support risers. Leave default
  - Pattern: whatever you think will work. **Lines** seems to use least material. Pick something that works with your overall shape.
  - Density: leave default. 30% more than enough
  - Z distance: leave default. This usually thickness of layer. Will be adjusted automatically when we set resolution in later step/.

7. Select material settings from right menu

   - Select your extruder at top right
   - Select material (PLA or ABS; PVA for PLA support material; leave on ABS for HIPS support material)
   - Select resolution (Fine, High or extra fine). More fine =slower print. Think of that when building early prototypes
     - Height will adjust automatically
   - Infill is for thick walls. 15 % seems to work fine.
   - If support required, select **ADD SUPPORT**

8. Other settings on right menu

   - Extruder: leave default
   - Layer: leave default as set by resolution
   - Infill: leave default (was set at top), or play with patterns settings for fun
   - Additions: Leave default. Do some research yourself if you want to change
   - Speeds:
     - PLA: Print 100 mm/s, Travel: 120 mm/s
     - ABS: Print 60 mm/s, Travel: 80 mm/s
   - Temperature:
     - PLA: Print 220, Build Plate 40
     - ABS: Print 230, Built Plase 110
   - Print cooling: leave default (ON)
   - Support:
     - Hover over setting explanation of setting
     - Turn on and set as required as described in other settings
     - NOTE THAT CHANGING ON RIGHT CHANGES ON LEFT. OPPOSITE NOT TRUE.
   - Advanced: Don't go there unless you want to research yourself

9. Re-check all your filament and support settings

10. Generate the file - bottom right corner of window

    - Click the little edit button (pencil in a box) if you want to change the output file name
    - Click **Prepare** in bottom right corner. If you don't see the button, scroll down
    - Click **Save to file** on your PC, or directly on SD card.
      - **SAVE AS .x3g file!!!**

11 Preparing the Qidi for the print

File format: .x3g (generated by the Qidi slicer)

Load material - and note which extruder * Left or right, as seen from the front * The Qidi manual covers this. It's no longer on their website. We saved it here

If there is plastic crud left over on the printing surface:

- **Shame on the last person using the printer**
- Wait until the plate is pre-heated before attempting to remove. It will be a lot easier

11.0.1 The steps:

1. Power on

   - Plugged in?
   - Power bar on?
   - Power switch on back of unit

2. Top Cover and 3 sides

   - ON/closed for ABS
   - OFF/open for PLA

3. Settings - on printer front panel

   - Use Up/Down arrows and go to **UTILITIES**, then hit **OK**
   - Use Up/Down arrows and go to **Preheat Settings**, then hit **OK**
     - Select extruder (Right or Left).
     - **NOTE**: if using separate support filament, you will need to set for both extruders)
       * PLA: 220
       * ABS: 230
       * Use up/down arrows to set, press **OK** when done
     - If using support filament, same temperature as your primary material
     - Select and set **Platform** temperature
       * PLA: 40
       * ABS: 110
     - Hit **left arrow** to exit temperature menu
     - Hit **left arrow again** to exit utilities menu
   - For TPU:
     - Filament loading considerations:
       * When loading filament, feed the material directly and bypass the feeder tube. You want to minimize the pull on the filament.
       * Top cover OFF and sides open
     - Pre-heat settings:
       * Extruder: 230C
       * Bed: 60C
     - Enable retraction: False
     - Print Speed: 20 mm/s

- Travel Speed: 30 mm/s
- Printing Temperature: 230 C
- Build Plate Temperature: 60 C
- Build Initial Plate Temperature: 60 C

- Enable Print Cooling: True
- Under Expert Mode:
    * Flow = 101% (Recommended by they're site is 110% so should be okay to increase flow up to 110% but will increase stringiness since retraction is disabled. Increasing it improves bonding between layers. Shouldn't need to though) And I'm not even sure 101% is even necessary, I've not tried the default 100%.
- Removing after print:
    * Heat bed back up to 60C for removal. At or below 40C, it really sticks to the bed

4. Preheat

   - Select **Preheat** and hit **OK**
   - Ensure required extruder(s) are **ON** and platform is **ON**
   - Select **Start Preheating** and hit __OK
   - Watch and wait for preheat to complete. You will hear it beep a little tune.
   - Hit **left arrow** to exit the preheat menu
   - **IF THERE WAS PLASTIC CRUD LEFT OF THE PLATE, REMOVE IT NOW WITH PUTTY KNIFE IN DRAWER BELOW THE PRINTER**

5. Insert SD card and load program

   - Slot is behind the front panel
   - Card logo faces front/contacts face rear
       - **NOTE: IF YOU GET AN ERROR, BE SURE TO EXIT TO MAIN MENU (LEFT ARROW)** This seems to reset the printer's SD card reader
   - Select **Print from SD**
   - Scroll up/down until you reach file you want to print, and hit **OK**
   - Extruder(s) and platform may have cooled. It will complete reheating automatically
   - THE FOLLOWING STEPS HAPPEN WHILE WARMING UP OR EVEN AFTER PRINT STARTED
   - **FOR PLA:**
       - Press **left arrow** button to get to special print menu
       - Hit **OK** to toggle fan ON (when it's on, screen shows option to set fan off)
   - **FOR ABS:**
       - Toggle fan OFF. But is should be off by default.

   WHEN DONE, PLATFORM WILL LOWER AUTOMATICALLY
   REMOVE PROJECT NOW WHILE PLATE IS WARM AND MATERIAL EASY TO REMOVE. USE PUTTY KNIFE IF REQUIRED
   __Clean up any plastic bits you left on the pate
   # Halting the print
   Because it doesn't always work properly.

12 Long print jobs/multiple pieces

Separate pieces: print separately because faster for each piece.

But... what if you don't want to wait? Or don't want to keep reloading?

- You can set it up for multiple pieces - they go through slicer together
- **DO NOT START AND WALK AWAY**
- Stay at least 1 hour to ensure everything is working properly

  - Because that's when most problems happen
  - Material sticking properly to plate?
  - Quality of material is OK - especially if using a new roll or supplier?

13 Shutting down

1. Clean up any plastic bits left on the print plate
2. If you supplied your own material (you did, didn't you?), remove your spools.
3. Close all the covers and lids on the printer to keep dust/dirt out
4. Power off using the switch on the back.

14 Where to get cool stuff to print

Thingyverse

Do3D (because it has the letters DDO)

... **and hundreds of others.**

Search engines and video sites are your best friends!

15 HANDS ON!

- Loading material
- Loading design in slicer
- Configuring printer
- Starting a print

[ ]: