

Orca Slicer Quick Start Guide

A walkthrough of setting up Orca Slicer for FDM Printing

[Orca Slicer](#) is an open source (read: free & community-built!) slicing software for 3D printing. Using Orca Slicer, users can convert 3D models into printable `.gcode` instructions for use on a variety of FDM printers, regardless of brand. For setting up [Chitubox](#) for [resin printing](#), please see our [Chitubox Quick Start Guide](#).

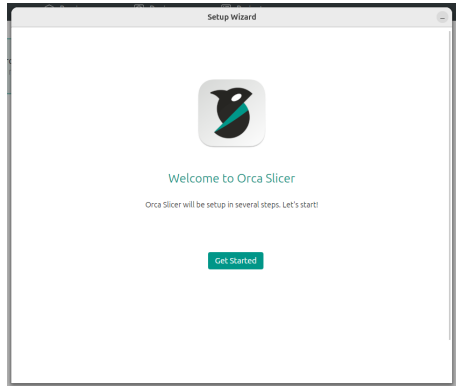
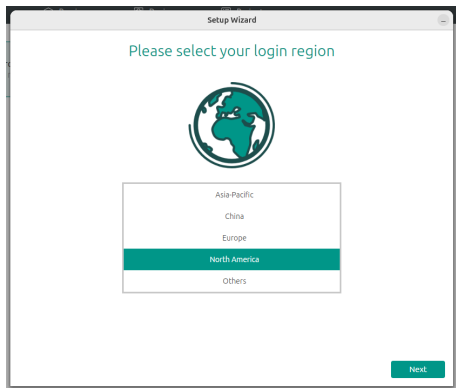
Orca Slicer is compatible with [Mac](#), [Windows](#), and [Linux](#) devices.

As of the creation of this guide, the most recent version is [2.3.2](#).

Upon the first installation, Orca Slicer walks you through how to set up your environment (printers, filament types, etc.) using their Setup Wizard.

[Click here](#) if you have previously installed Orca Slicer.

First Installation - through the Setup Wizard

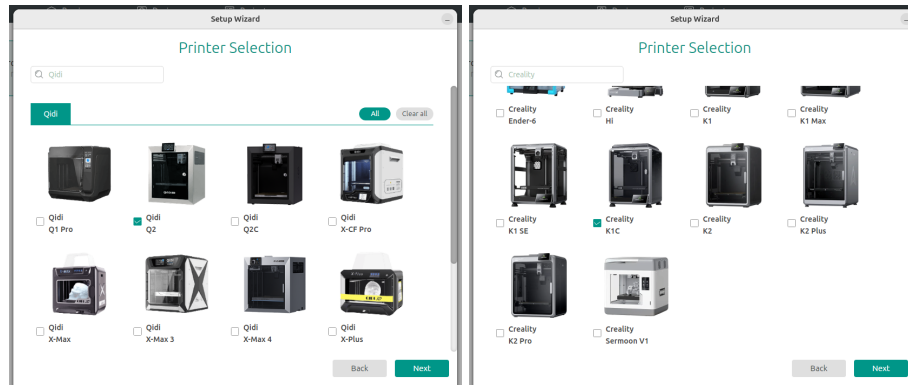
1. Welcome Splash Screen Click "Get Started."	 The image shows the 'Setup Wizard' window for Orca Slicer. It features the Orca Slicer logo at the top, followed by the text 'Welcome to Orca Slicer' and 'Orca Slicer will be setup in several steps. Let's start!'. At the bottom, there is a green button labeled 'Get Started'.
2. Region Selection Select the region from which you will be using the software (likely, North America).	 The image shows the 'Setup Wizard' window for Orca Slicer, specifically the 'Please select your login region' screen. It features a globe icon at the top. Below it, there is a list of regions: 'Asia-Pacific', 'China', 'Europe', 'North America', and 'Others'. The 'North America' option is highlighted with a green bar. At the bottom right, there is a green button labeled 'Next'.

3. Printer Selection

Click “Clear All” and select the printer(s) you will be using.

At the TTL Makerspace, we have a **Qidi Q2** and a **Creality K1C**. Be sure to select the right version - select the Creality **K1C** and *not* the Creality **K1**!

Tip: use the search bar to find the models quicker!



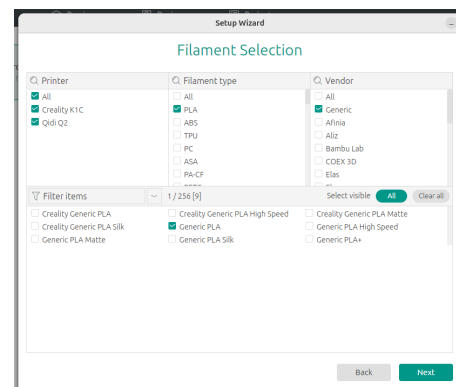
Example Printer Selection based on available printers at the TTL Makerspace.

4. Filament Selection

Click “Clear All” and select the filament(s) you will be using.

The most common filament both in general and in the Makerspace is PLA. The “**Generic PLA**” filament is the most simple & reliable, good for most PLA printing use cases!

If you are using a specific kind of PLA (or a different kind of filament entirely, such as PETG, ABS, etc.), be sure to select/add the corresponding filament.

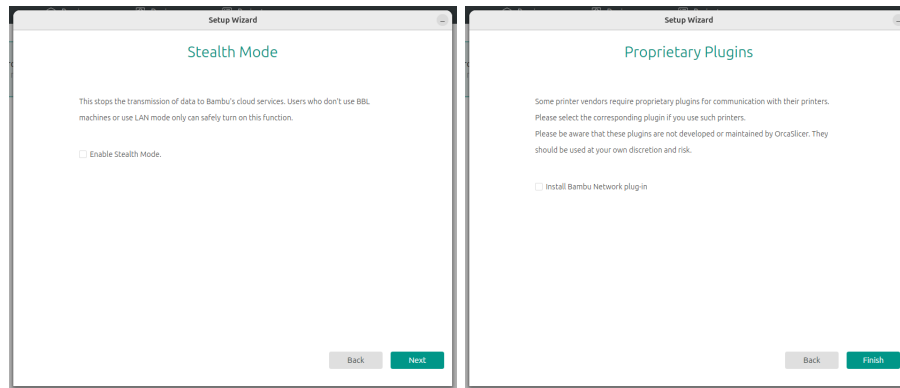


Example Filament Selection using the provided filtering system.

Printer: **All**
Filament Type: **PLA**
Vendor: **Generic**

5. Optional Further Steps

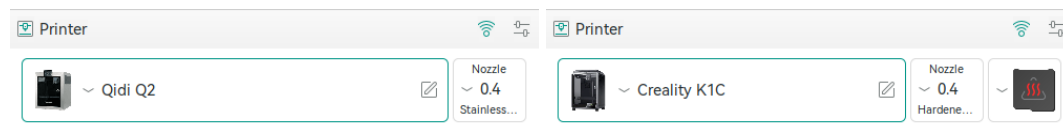
One of the benefits of Orca Slicer is that it can be used with a variety of printers. The following dialog boxes are Bambu-specific and, unless you are looking to print on a Bambu printer, **these settings can be ignored**.



6. Finalize Printer Settings

After the Setup Wizard, Orca Slicer opens an empty project for you.

For each printer you are using, make sure that the **Nozzle is set to 0.4** (or the appropriate dimension for your printer).



Now you're ready to go!

Check out our additional 3D printing resources including video tutorials, an in-depth OrcaSlicer guide, online tools, and more @ code.ttlmakerspace.com!

Continue below to see the “Previously Installed” section.

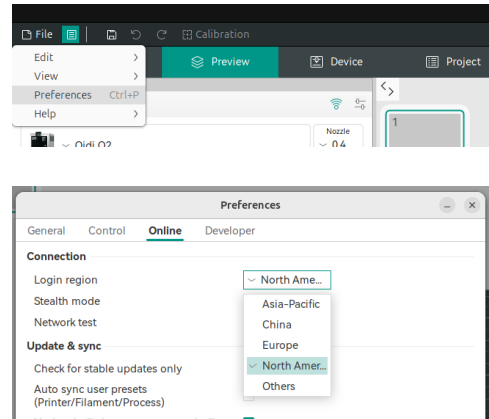
Previously Installed

If you have previously installed Orca Slicer, there are some presets & settings to double check before slicing and printing your models at the TTL Makerspace.

1. Region Selection (opt.)

Along the top bar, click the dropdown menu icon (☰) and click “**Preferences.**” Navigate to the “**Online**” tab and beside “**Login Region,**” select “**North America**” (or whichever region you will be logging in from!).

This ensures that you will be connected w/ the proper support channels when needed, that you'll have access to the appropriate cloud services, and more.



2. Printer Selection

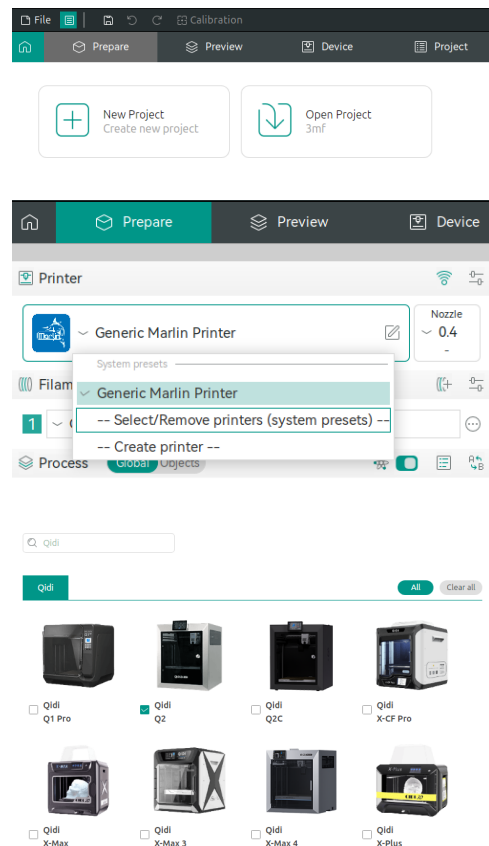
From the Home screen, click the “**Prepare**” or “**New Project**” button.

On the lefthand side, open the dropdown menu in the “**Printer**” tab and click “**Select/Remove printers.**”

Find and select the TTL Makerspace's two printers: the **Qidi Q2** and a **Creality K1C**.

Be sure to select the right version - select the Creality **K1C** and *not* the Creality **K1**!

Tip: use the search bar to find the models quicker!

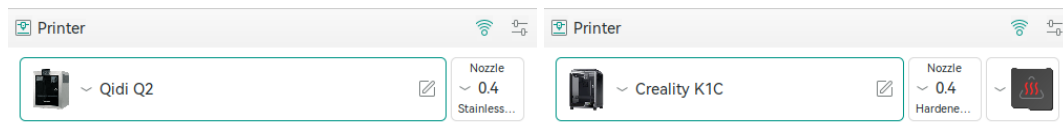


Example Printer Selection of the TTL's Qidi Q2 printer.

3. Finalize Printer Settings

After confirming your printer selection, be sure to double check the **nozzle diameter** (dropdown menu next to the printer).

For printing at the TTL Makerspace, for each printer you are using, make sure that the **Nozzle is set to 0.4**. Check the manual/online resources for any of your own machines to find the nozzle diameter & ensure proper print settings.

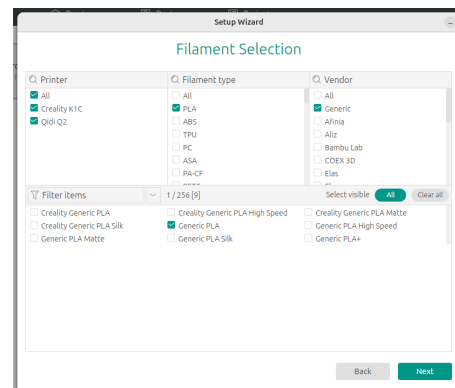
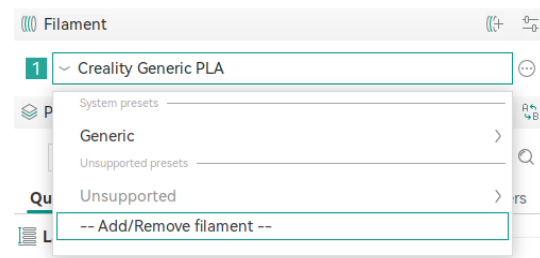


4. Filament Selection

Under “Printer,” you’ll find the “Filament” section. Click the dropdown menu and navigate to “**Add/Remove filament.**”

Your selections in the following window should **match the material that you intend to print with**. If you are using a specific kind of filament, be sure to select the corresponding filament.

The most common filament both in general and in the Makerspace is **PLA**. The “**Generic PLA**” filament is the most simple & reliable, good for most PLA printing use cases!



Example Filament Selection using the provided filtering system.

Printer: **All**

Filament Type: **PLA**

Vendor: **Generic**

Now you're ready to go!

Check out our additional 3D printing resources including video tutorials, an in-depth OrcaSlicer guide, online tools, and more @ code.ttlmakerspace.com!

