Notes: This document is temporary until the next update of ORCA which hopefully will include our printers, natively.
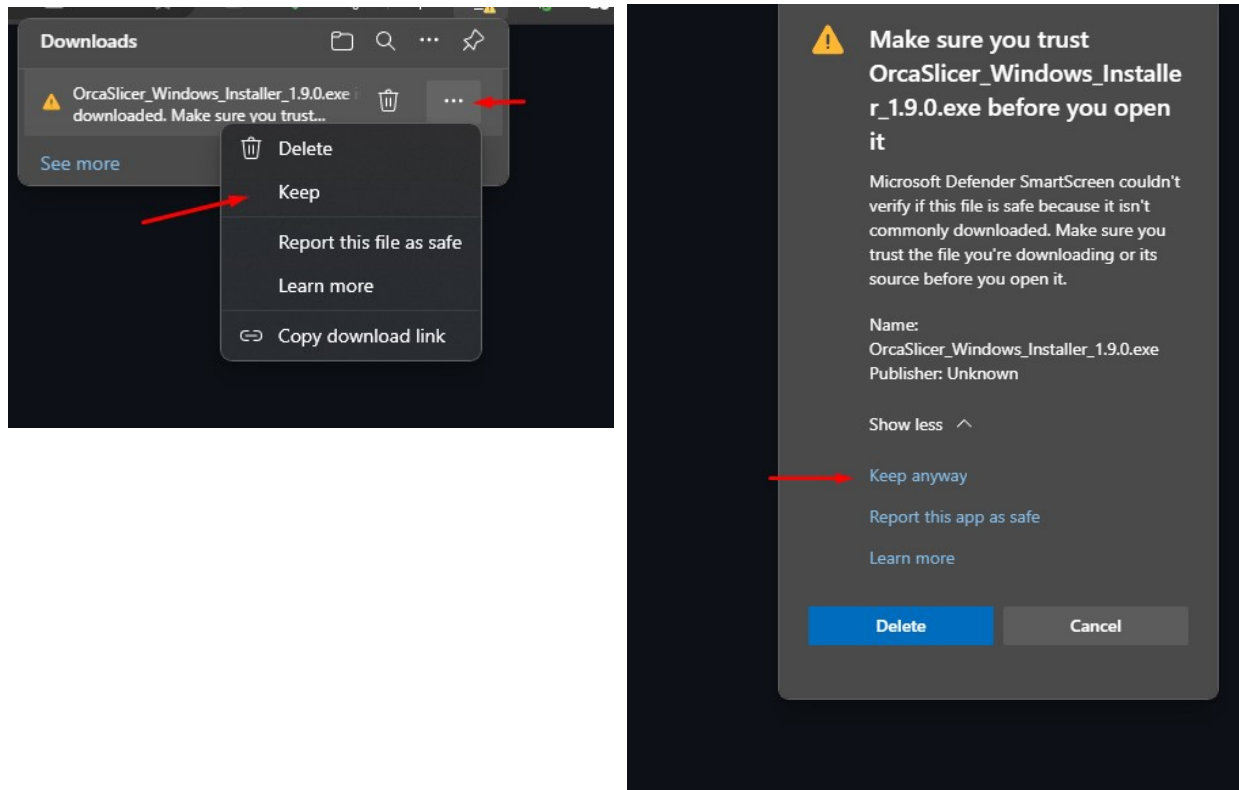
Step 1:

For Windows Users, download Orca Slicer:
https://github.com/SoftFever/OrcaSlicer/releases/download/v1.9.0/OrcaSlicer_Windows_Installer_1.9.0.exe

Other versions of OrcaSlicer can be found here:
https://github.com/SoftFever/OrcaSlicer/releases/tag/v1.9.0

If the install does not automatically start, find the option to "keep" (see arrows on examples given below).
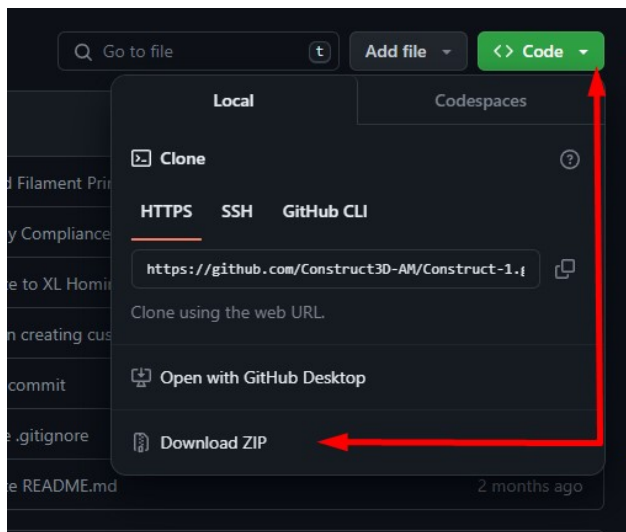

Step 2:

Run the installer.

Step 3:

Open Orca Slicer

Step 4:

Download the profiles: You can find the configs for Orca Slicer here:
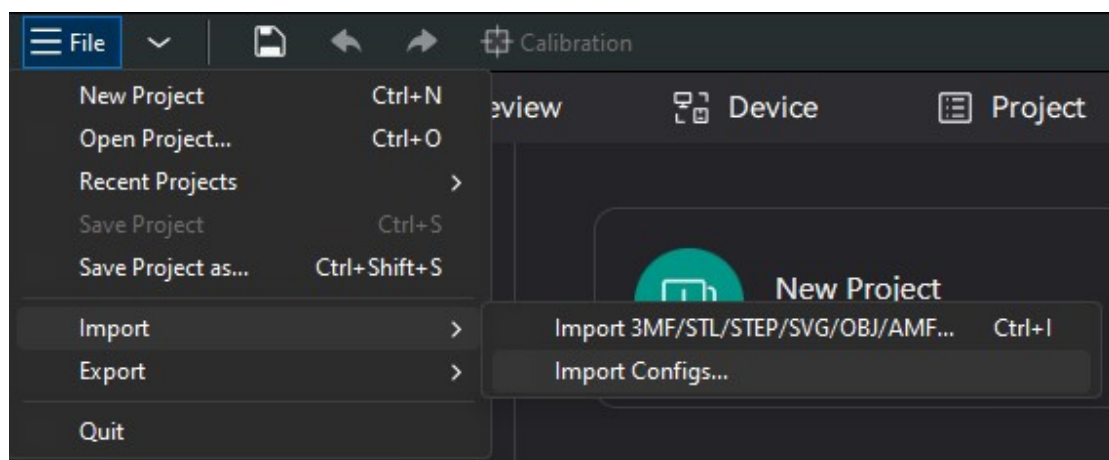https://github.com/Construct3D-AM/Construct-1/tree/master

To download the configs (+ all supporting files), click the green "code" button and then the Download Zip option.
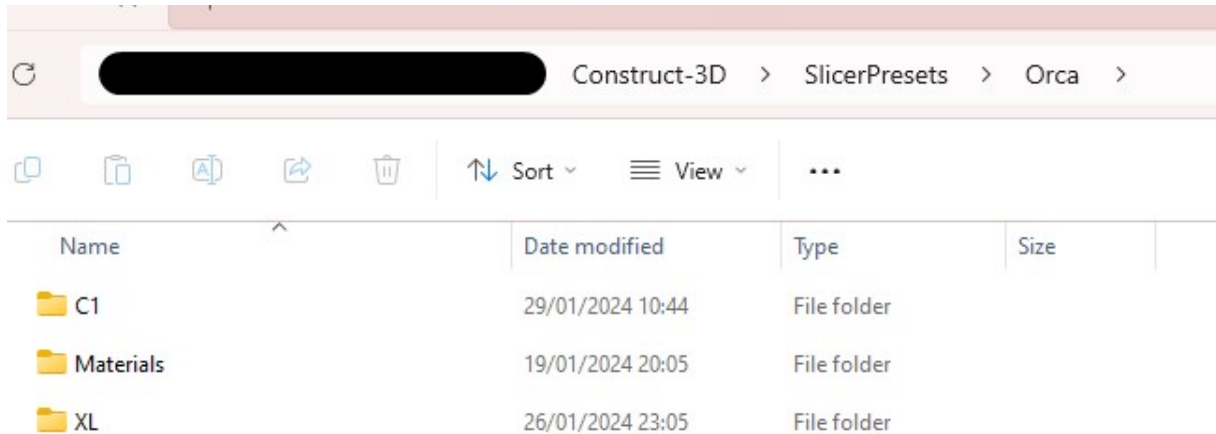

Proceed to Unzip the file.

Step 5:

Add the profiles: in the top right, click the File hamburger button. Navigate to import ->

Import Configs

Navigate to your newly downloaded folder and follow the folder path:
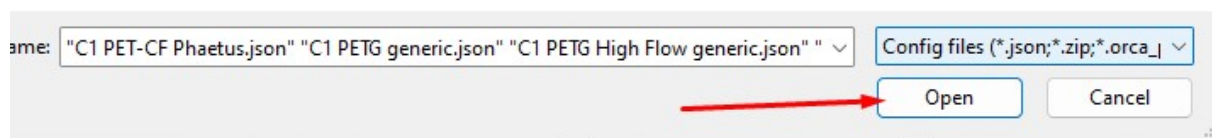
Construct-3D > SlicerPresets > Orca

Import materials: open the Material folder and navigate to the correct nozzle type for your machine. [CHT Vanadium Profiles will follow shortly]

Select any materials which you want to use – you can select multiple at once – and then click the "Open" button to import them to Orca Slicer.
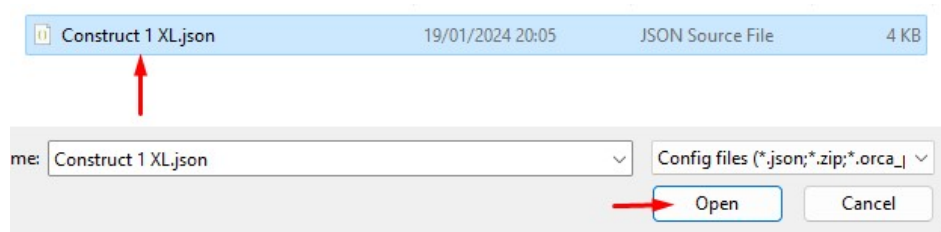
Add the printer: go back to the Orca base folder and navigate to the correct folder for your machine. (See example using an XL printer)
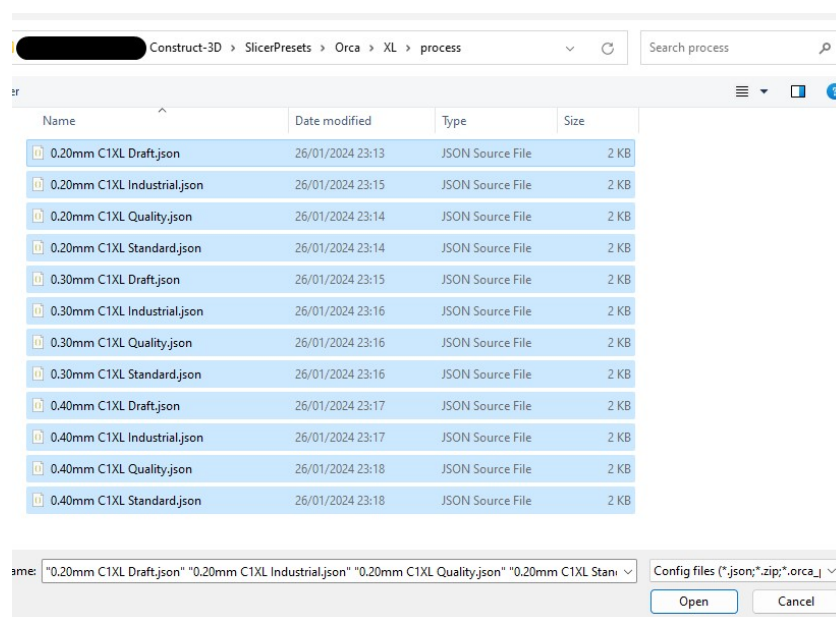

To add the printer to Orca Slicer, open the "Printer" folder, select the "Construct 1 XL.json" file and open it.

The printer has now been successfully added to the slicer.

Next we can add the print profiles, these tell the slicer how to create printer files that best suit the machine to achieve higher quality than the default profiles provided.

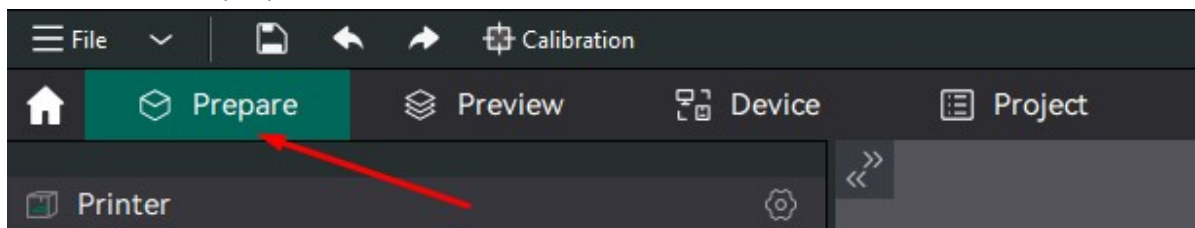Navigate to the "Process" folder (Construct-3D > SlicerPresets > Orca > XL > Process

Once all the files have been selected you can click "Open". This will import all the print profiles to Orca Slicer

Step 6:

Orca Slicer now has all the relevant information for CONSTRUCT3D printers. Next, set them up as the default.
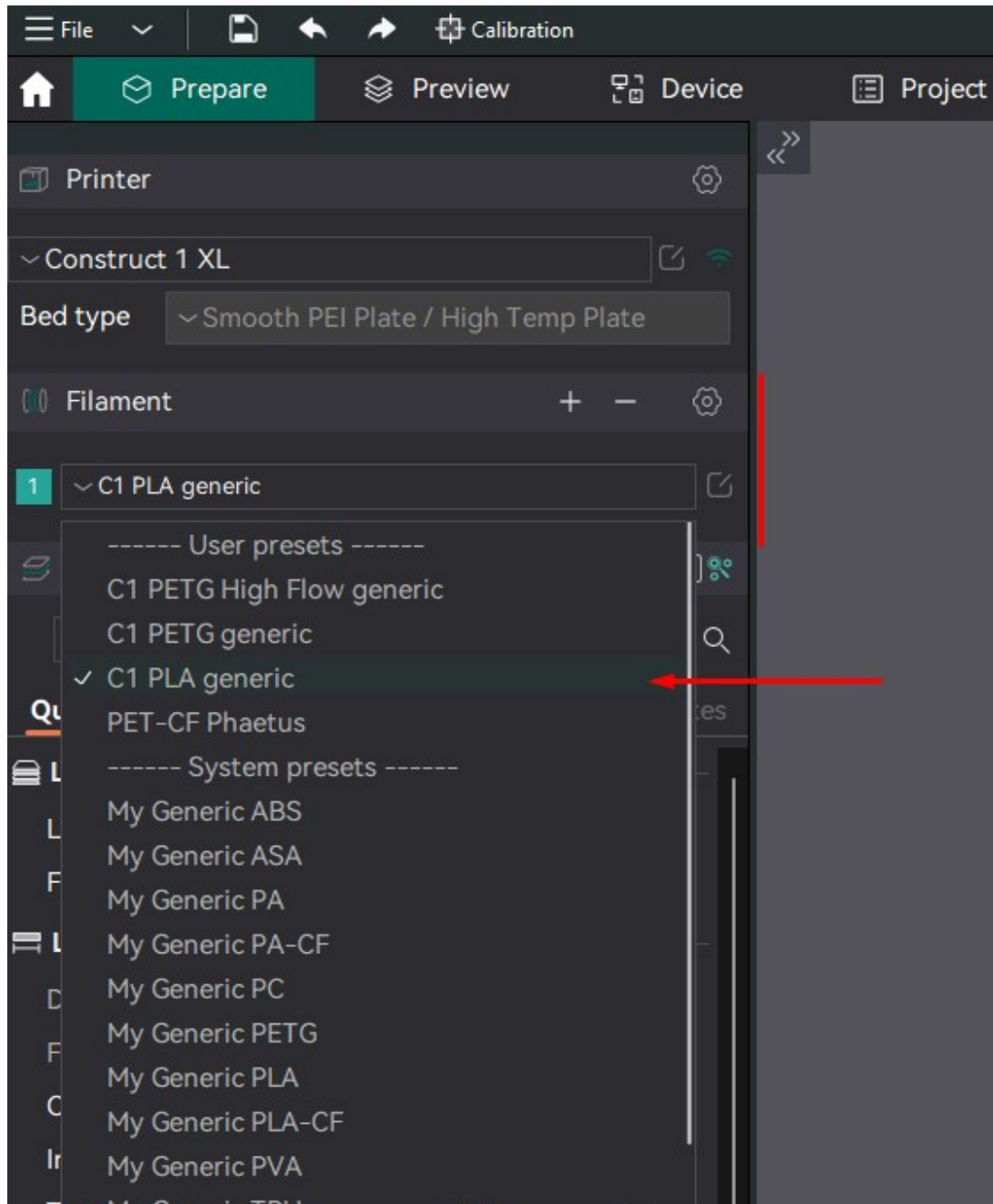
Proceed to the "prepare" tab in Orca

Under the "printer segment" there will be a drop down box. This may currently say "MyMarlin". Click the drop down box and select your printer. (See example below)
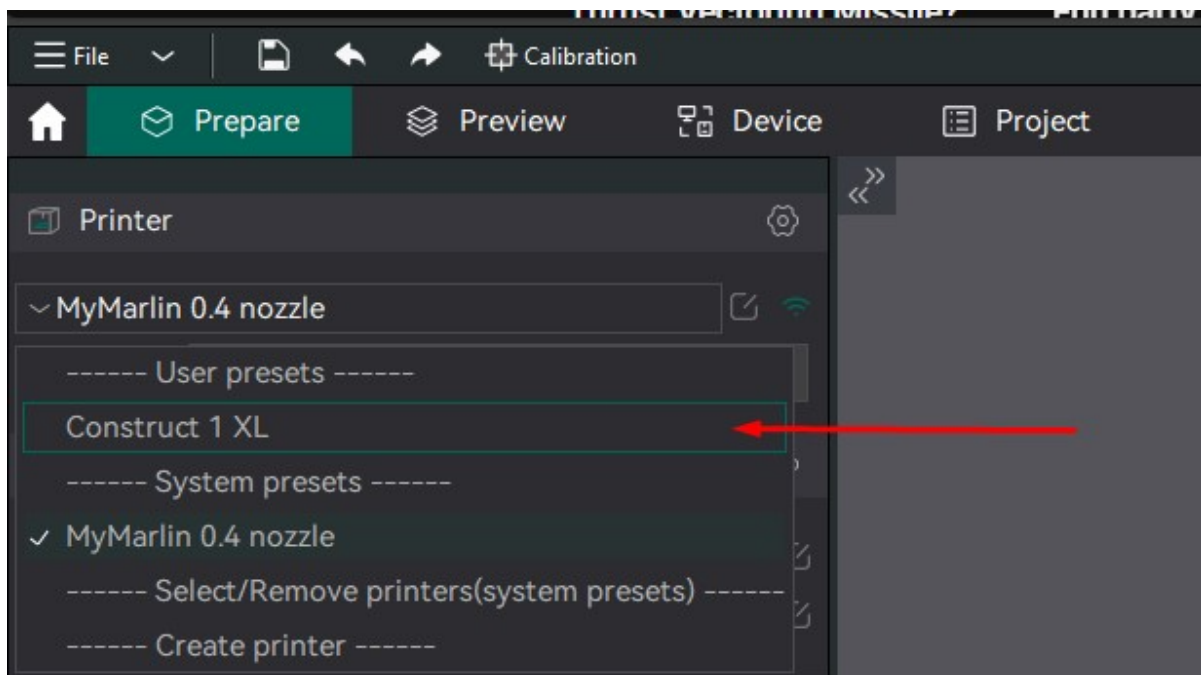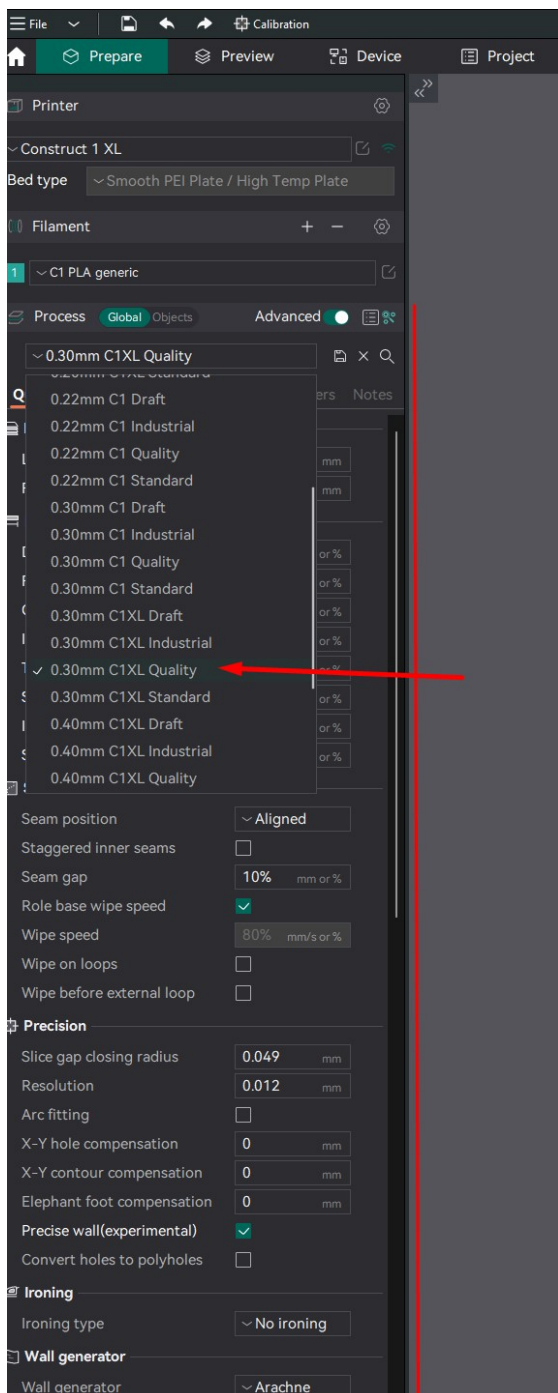
Select filament: Below the "Printer" tab, there is a "Filament" tab.

Click its corresponding dropdown and select one of the preset filaments. The example below is for PLA.

Select a print profile.

The example below is for a 0.3mm profile for the XL

Behind the dropdown for the preset are the inidividual settings that are included in the profile.

The presets automatically update these settings so depending on how much granular control over the slicing process, you are free to modify these.
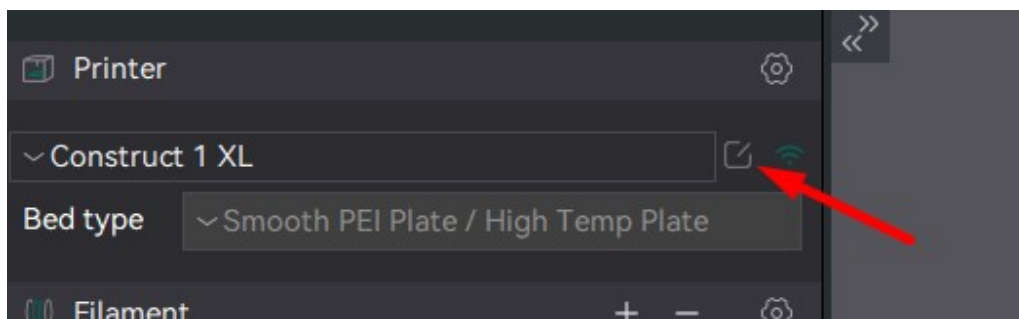
Step 7:

    i)   Importing the 3D model for the build platform.

    ii)   Telling the slicer the printer's IP for remote print sending.
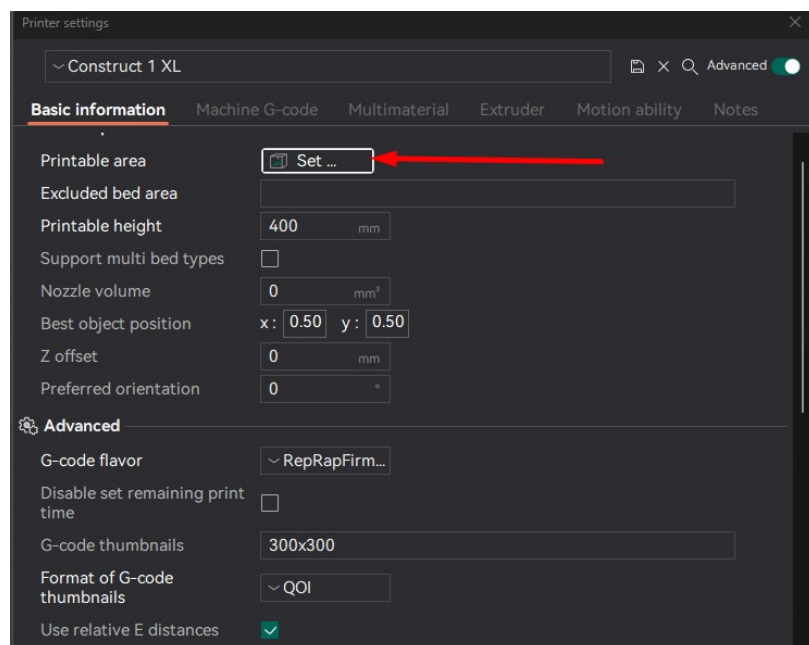
i) Orca should be displaying the current build volume as a grid in the centre of the screen.

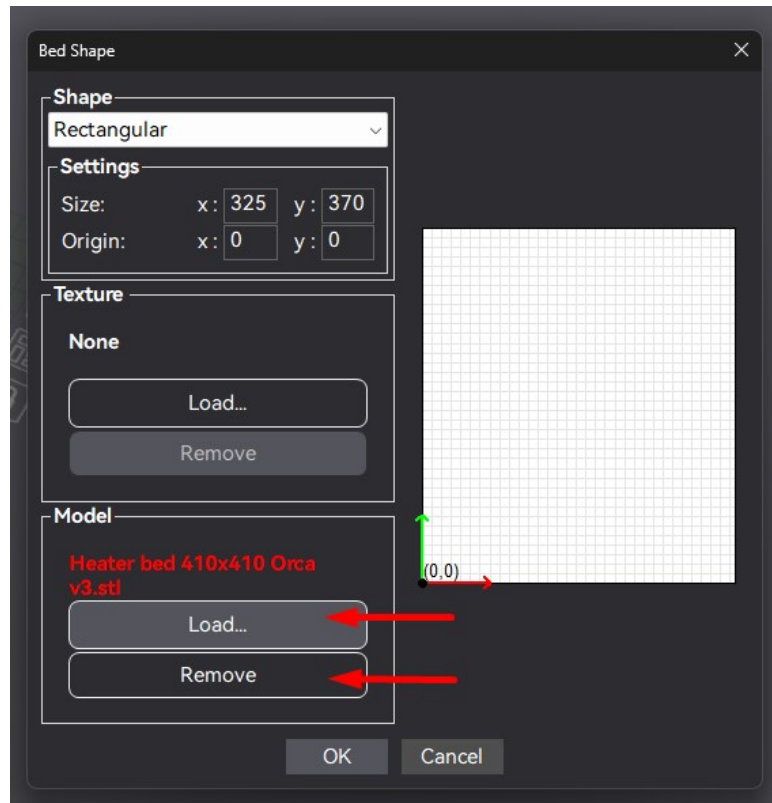To update the build volume to accurately represent the printer bed:

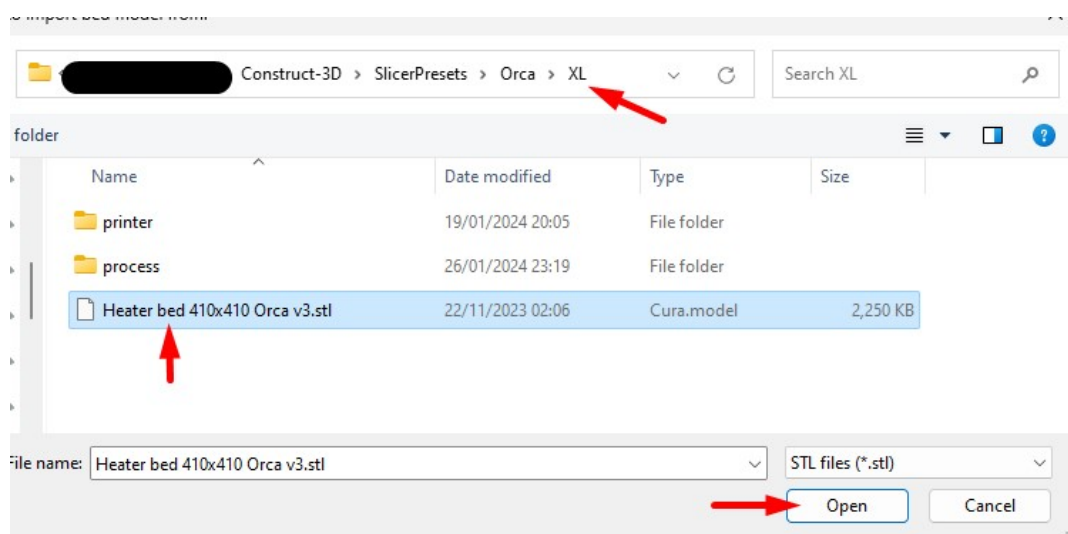   a) click the options button next to the printer's name.

    b) click the "Set printable area" button in the new dialog box

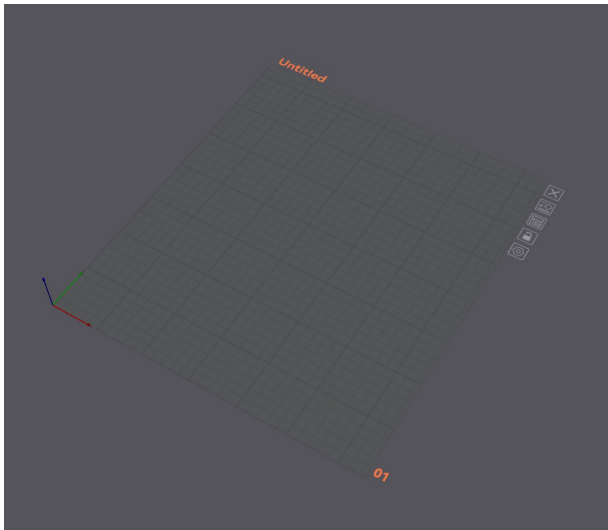c) click the "Remove" button under the model tab, and then click the "load" button

d) navigate to the main directory for your printer, select the required Heater bed .stl and open it.
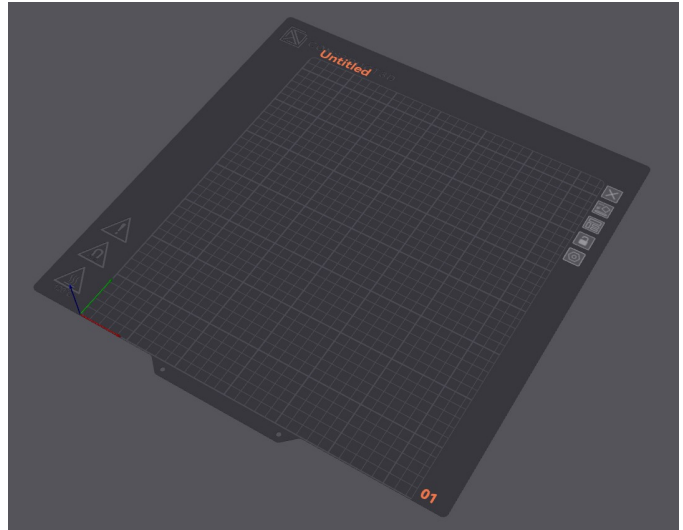
This will now update the grid layout to accurately represent the build plate of your printer.
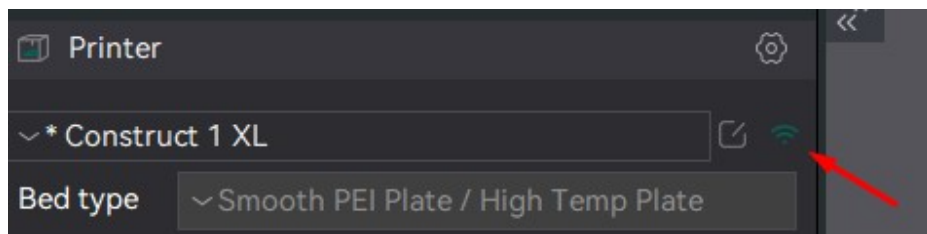
Before   After


ii)   Enabling the remote print functionality of the slicer.

   a)   First note the IP address of the printer (refer to the printer manual).

   b)   Click the Wi-Fi symbol next to the printer name.

   c) Add the IP in the following text field, and click the test button.

If the test is successful a dialog box will appear saying 'success'.

Congratulations! Your slicer is now 100% set up and ready to be used.
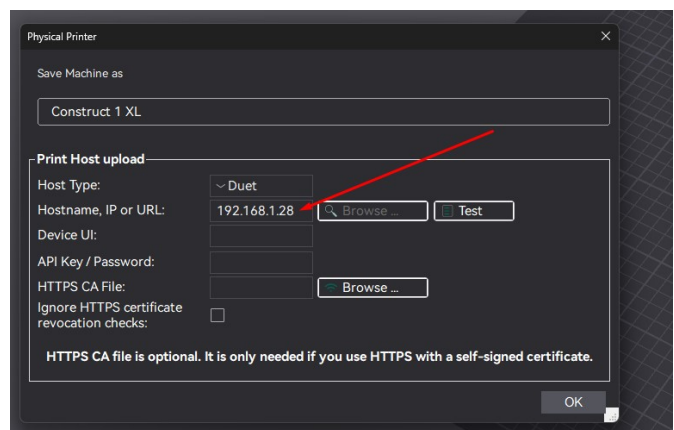
NOTE:

Please remember that the print profiles provided are intentionally generic to be a great starting point for almost any material or model. These printers can perform even better when you have a good understanding of what material you are using and what your model requires for optimum slicer settings.

Once you are comfortable with how the printer works, feel free to play around with the slicer settings. Happy printing!