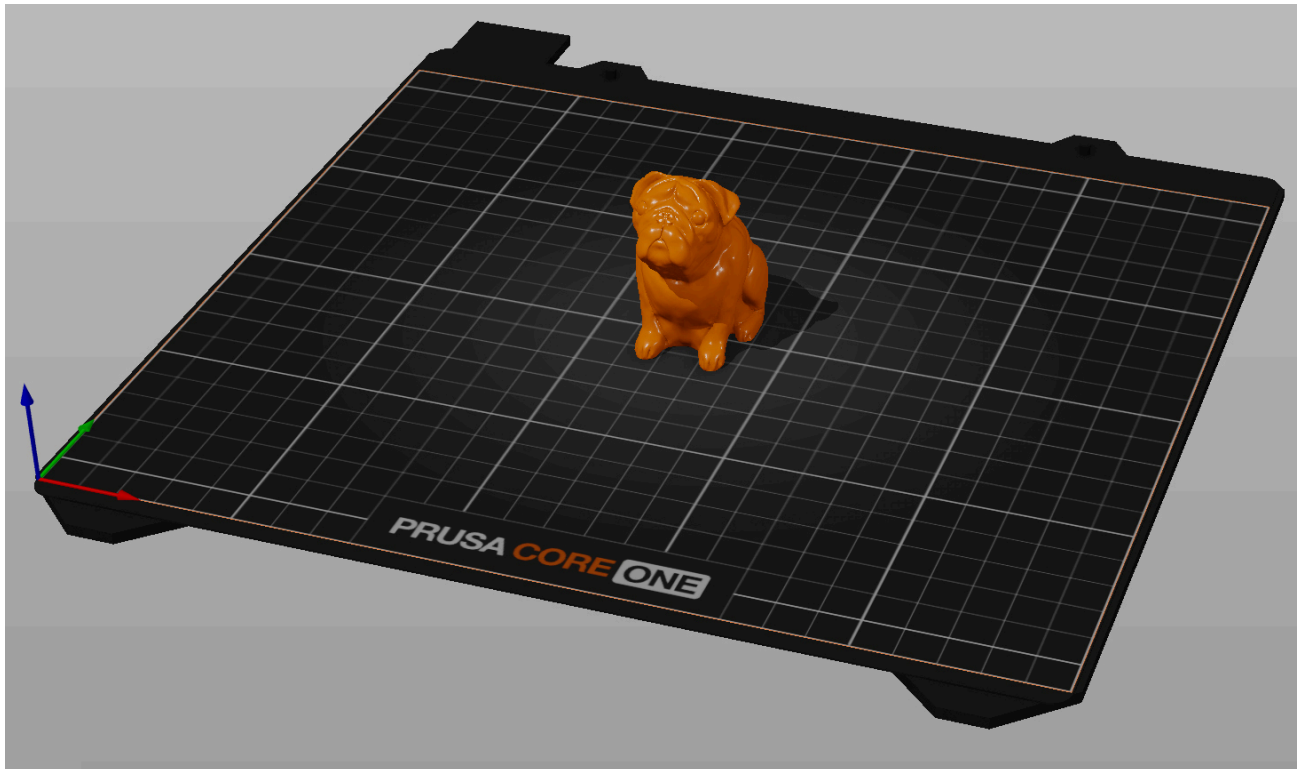


First print with PrusaSlicer (3.0.0)

 help.prusa3d.com/article/first-print-with-prusaslicer-3-0-0_1079761



Before printing your 3D file, it needs to be converted into a file type called *.gcode* or *.bgcode*. That is done by a program called a slicer. The best one for your Prusa or even non-Prusa printers is PrusaSlicer. This guide will lead you through using PrusaSlicer for the first time.

Before starting to use PrusaSlicer, make sure you know your Printer model (including the name and [nozzle size](#)) and the [filament material](#) you will print with.

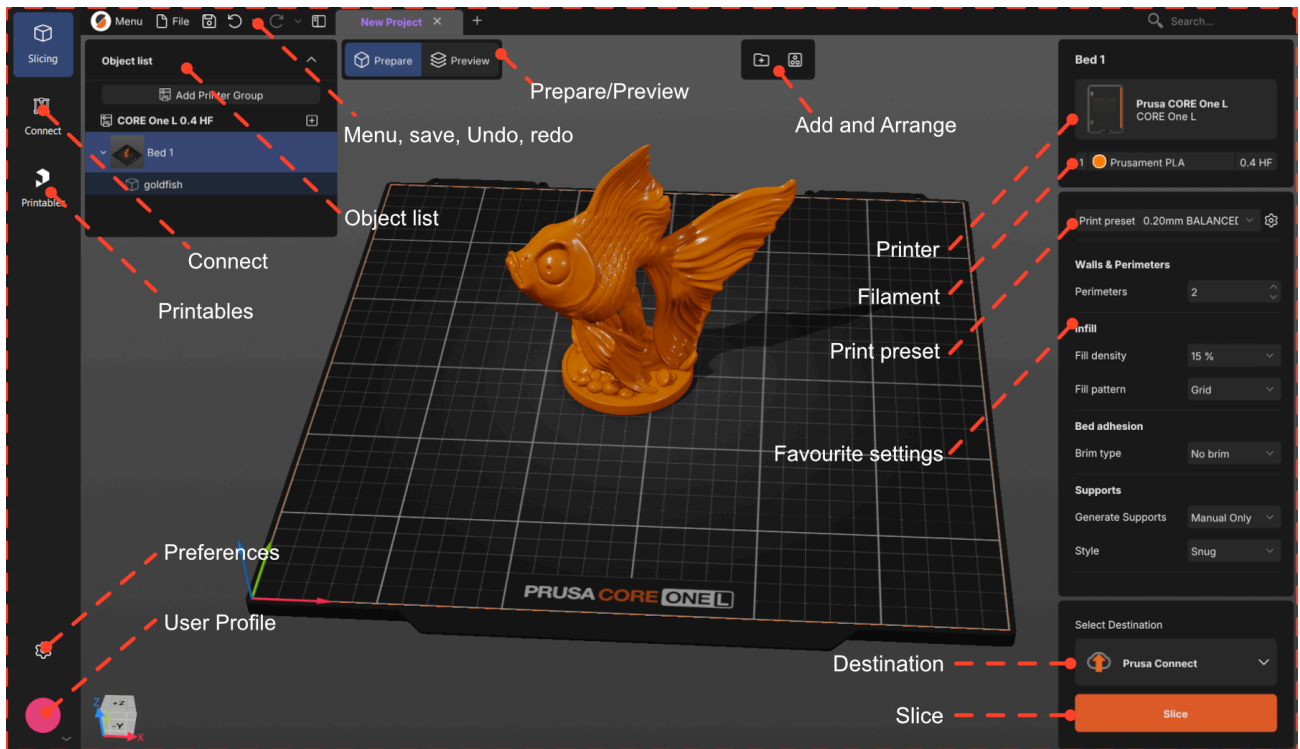
PrusaSlicer modes

PrusaSlicer has two work modes: **Prepare** and **Preview**. Most of the model preparation will be done in the Prepare mode.

In both modes, you can see the **Object list** at the top right of the workspace. That will contain all the items of your project, organized as *Printer -> Bed -> Object*.

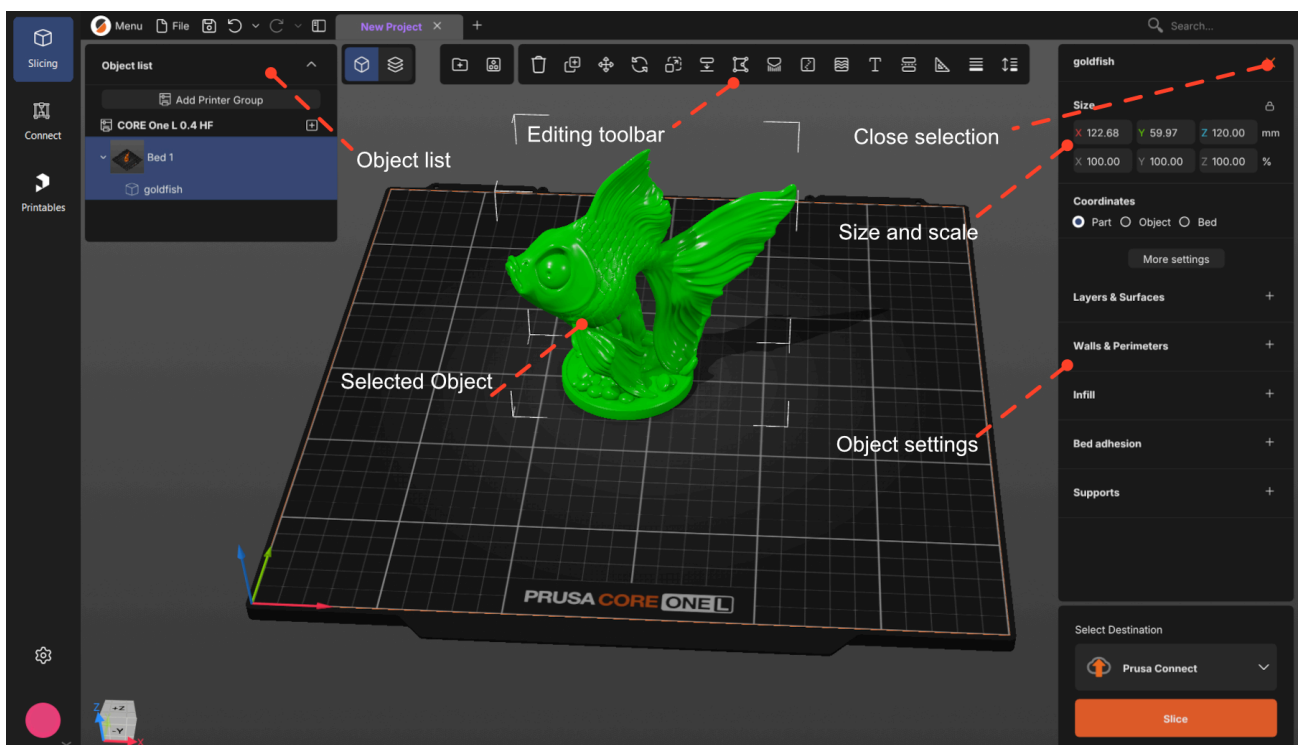
The Prepare mode is where you will arrange your model and choose your settings. There, you have the 3D scene, where you will add your model.

If no objects are selected, you will see the options **Add** and **Arrange** on the top menu, and the selected printer, filament, and print profile on the right-side menu.



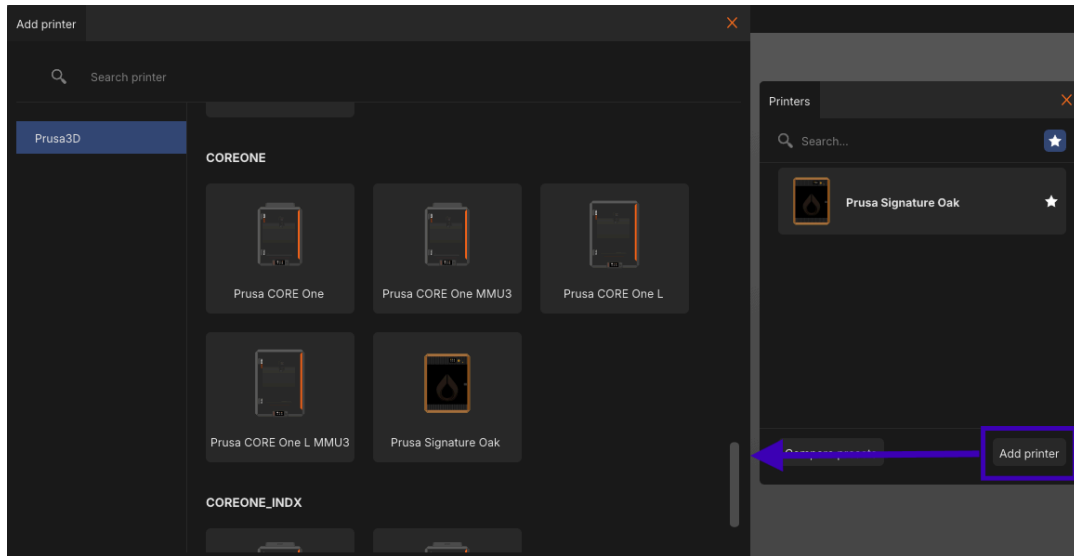
After an object is selected, you will see the full set of **editing tools** at the top menu, and the object's size and settings on the right-side menu.

To cancel selection, press Esc or click the close button.



Choose and set up your printer

Click to select the printer you will use. If your printer is not on the list, click *Add printer* at the bottom of that dialog and find your model in the catalog - there is a search box.

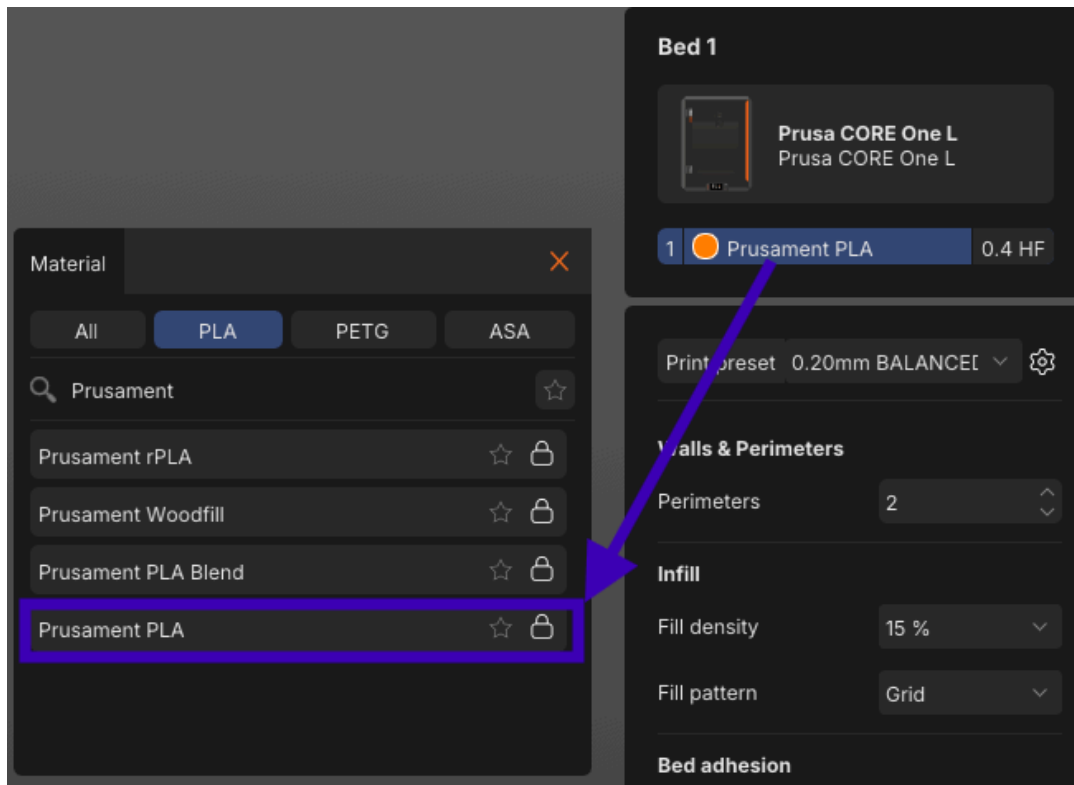


Set up the hardware. Always check the **nozzle** diameter and type, and the print **sheet**. The **gear** next to the printer name opens a menu with hardware setup

0:00 / 0:24

Choose your filament

Click the material name to see the list of available materials and brands. You can filter by material type — (the All / PLA / PETG / ASA) tabs — or search for the filament.

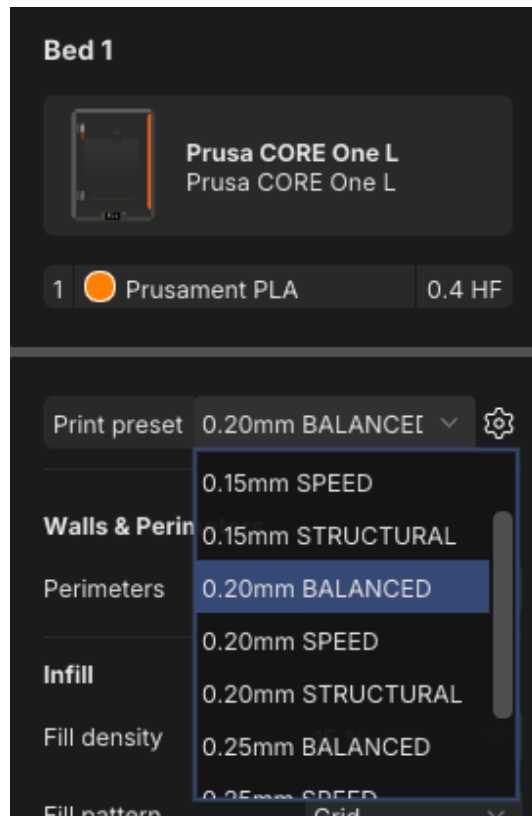


Choose a print profile

Each print profile comes with a predefined layer height — the main trade-off between quality and time. Thinner layers mean more detail and a longer print.

To start, use 0.15 or 0.20 mm. Both give very good quality at a reasonable print time.

Most layer heights also come in versions tuned for a purpose, like SPEED or STRUCTURAL. BALANCED is the neutral one you can start with.



Import your model

Any of these options work:

- **Drag and drop** the file onto the PrusaSlicer window
- Click **Add** above the plate or Ctrl+i
- **File** → **Import STL /3mf**

Supported 3D formats are: .3mf, .stl, .obj, .step / .stp, and SVG.

.3mf files can hold the whole project, not just models. Opening one may bring its own printer, filament, and print settings and replace yours. That is usually what you want when someone shares a ready-to-print project — just be aware it happened.

0:00 / 0:12



Orient your model

Your model may be oriented incorrectly after import. Four tools are very useful:

- **Place On Face F**: click one of the model's flat surfaces, and it snaps face-down onto the bed. The quickest fix for a model that imported at an angle.
- **Move M**: drag the model with the left mouse button at any time. The Move tool constrains the movement to a single axis.
- **Rotate R**: turn the model around the X, Y, or Z axis. Dragging the outer ring snaps in 45° steps, the inner ring in 5° steps.
- **Arrange A**: with several models on the plate, arrange them out on the bed without overlaps and with a safe gap between the parts.

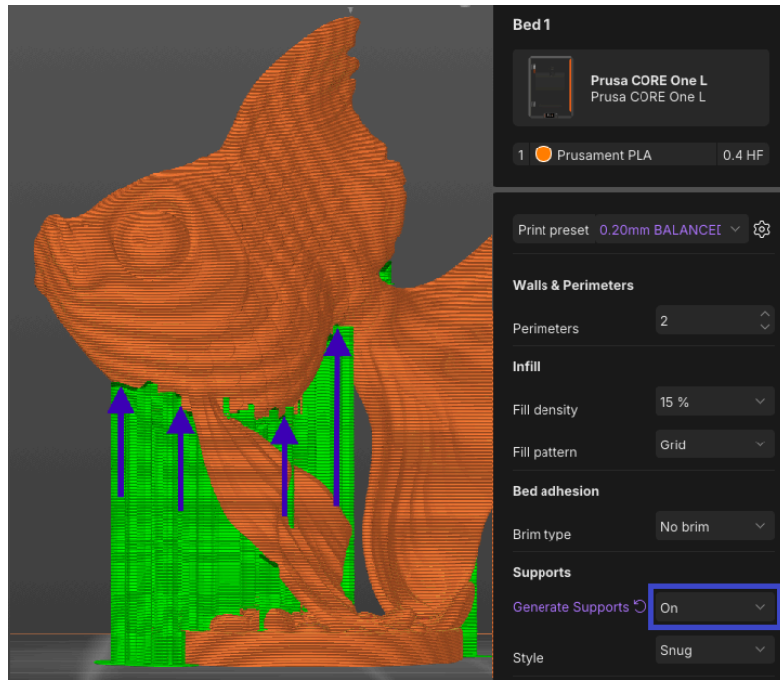
0:00 / 0:47



Support material

Printers build prints upward layer by layer, so anything that overhangs with nothing beneath it has nothing to print onto. It sags or falls, ruining the print. Supports are scaffolding that holds it up, and you break them off afterwards.

- **Manual Only (default):** only where you paint them on yourself, with the Paint-on supports tool L.
- **On:** added automatically wherever an overhang needs them. For most models, On is enough.
- **Off:** none at all.

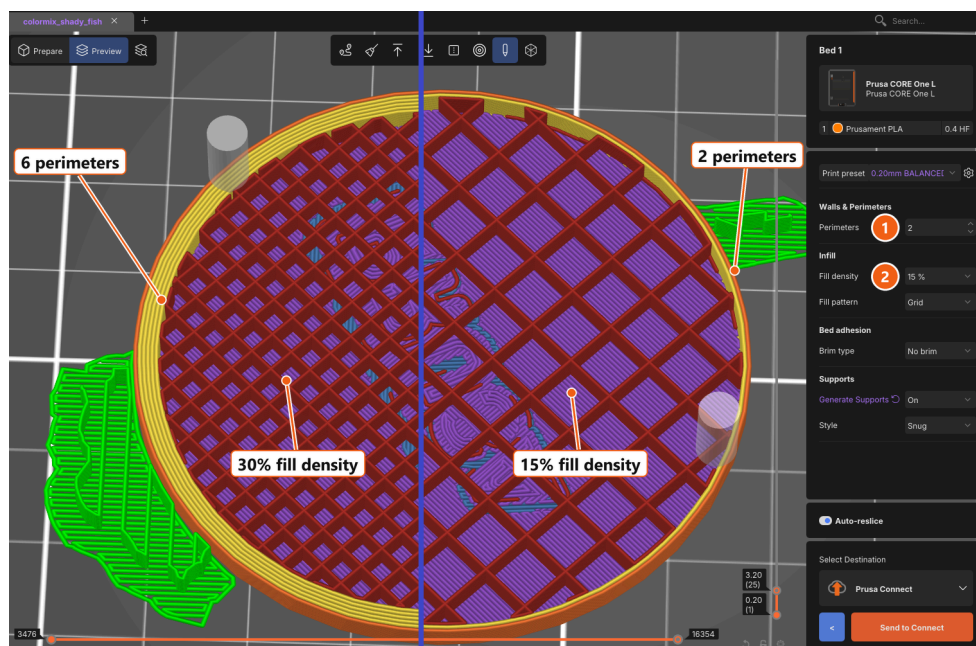


Perimeters and infill

Perimeters and infill are the two basic settings that control the structure of the finished print. Both can be easily found in the right panel: **(1) Perimeters**, **(2) Infill density**.

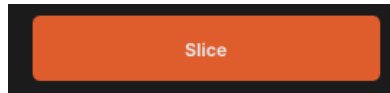
- Perimeters: the number of outer walls, which sets how thick the shell is.
- Infill: the internal structure. It makes the object rigid and supports the top surfaces.

Increasing either adds strength and rigidity and increases the print time. For a first print, the profile's defaults are fine.

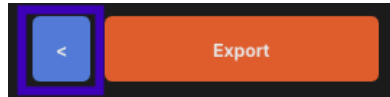


Slice

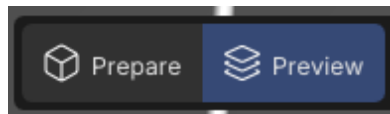
The **Slice** button is the most important in PrusaSlicer. Slicing takes the model geometry and all your settings, and generates the instructions your printer follows.



Clicking to Slice will bring you into **Preview** mode. To go back, use the < (back) button beside Export.



or the **Prepare / Preview** switch above the plate.



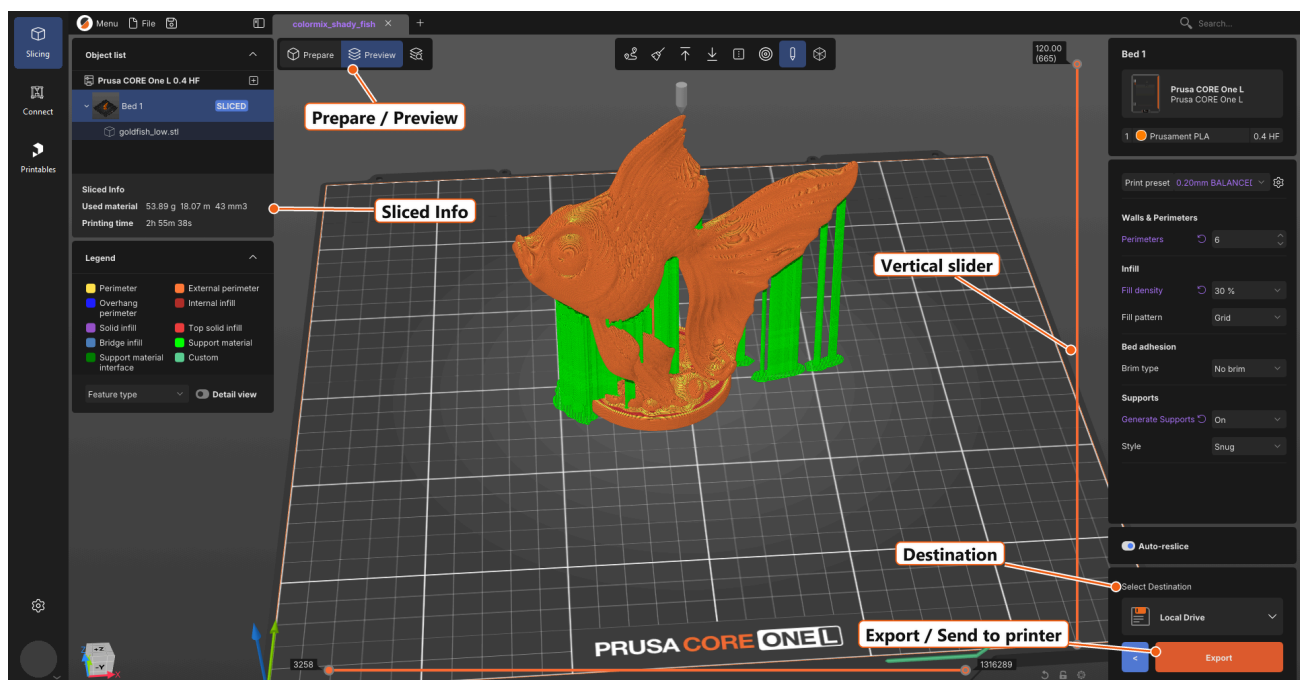
Check the result in Preview

You can see the **Used material** and the **Printing time** on the left panel.

Look at the print. The vertical slicer on the side allows you to check the print layer by layer by sliding it down. The three things to check are:

- **The shape:** At the top of the slider, it should look like your model.
- **Floating parts:** Every new piece of material should sit on something below it — the bed, the model, or a support.
- **First layer:** It should cover the bed. Drag the slider all the way down. This layer decides whether the print sticks.

If something looks wrong, switch back to **Prepare**, fix it, and slice again.



Export and print

The **select destination** block above the orange button controls where the file goes. Click it to choose.

- **A drive on your computer, or a USB stick/SD card:** The button reads **Export** and saves the file. Take the media to the printer.
- **Prusa Connect:** The button reads **Send to Connect** and uploads it to your Connect account.
- **A printer on your network:** The button reads **Send to Printer** and sends it straight to the machine.

Most first prints go out on removable media. If a USB stick or SD card is plugged in, PrusaSlicer offers it as the destination, and once the file is written, a notification appears with an **Eject** button — use it rather than pulling the card out.

A note on file types: you will get either a *.gcode* or a *.bgcode* file. *.bgcode* is the same instructions in a compact binary form, smaller and faster for the printer to read.

Then start the print from the printer's own menu. **Watch the first layer go down.** If it sticks, the rest usually follows.

Handy shortcuts

- Ctrl + I import
- Ctrl + S save project · Ctrl + F search settings ·
- Ctrl + Z UNDO
- Tab switch Prepare / Preview
- M move
- R rotate
- S scale
- F place on face
- A arrange

Can't find a setting? Press Ctrl + F and type its name. It works across every settings page.