

3D PRINTING HANDBOOK

USER MANUAL FOR 3D PRINTERS:

- ORIGINAL PRUSA SL1 KIT
- ORIGINAL PRUSA SL1



PRUSA
RESEARCH
by JOSEF PRUSA

 ENG

ORIGINAL INSTRUCTIONS: Basic info



Please always refer to the <http://www.prusa3d.com/drivers/> for an updated version of this 3D printing handbook (PDF download).

Translated versions of the handbook are available at:

Czech: www.prusa3d.cz/ovladace/

French: www.prusa3d.fr/drivers/

German: www.prusa3d.de/treiber/

Polish: <https://www.prusa3d.pl/sterowniki/>

Italian: www.prusa3d.it/driver/

Spanish: www.prusa3d.es/drivers-y-manuales/

QUICK GUIDE TO THE FIRST PRINT

1. Read the safety instructions carefully
2. Place the printer on a flat and stable surface
3. Download and install updated firmware and software packages
4. **Calibrate the printer** by following the on-screen calibration flow / wizard
5. Inspect the FEP film (bottom of the tank) for any kind of damage
6. Pour resin into the tank
7. Use the Print menu file browser to select a sample object to print



Important notice, tip, hint or information that helps you print with ease.



Read carefully! This part of the text has the greatest importance - either for user safety or for proper printer service.



This symbol indicates text related to a printer kit only.



How to contact Prusa Research tech support:

First, check the last chapters of this handbook for troubleshooting guides, or visit <https://help.prusa3d.com> for a complete list of common issues, print quality troubleshooting and other guides. If your problem is not listed there, or the solution does not work, please send an e-mail to support@prusa3d.com or use the Live chat at <https://shop.prusa3d.com>. Try to explain your problem as thoroughly as possible.

About the author

Josef Prusa (born Feb 23rd, 1990) became interested in the 3D printing phenomenon before joining Prague's University of Economics in 2009 - at first it was a hobby, a new technology open to changes and improvements. The hobby soon became a passion and Josef grew into one of the leading developers of Adrien Bowyer's international, open source, RepRap project. Today, you can see the Prusa design in different versions all around the world, it is one of the most popular printers and thanks to it, knowledge about the 3D printing technology significantly increased among the public.

Jo's work on self-replicating printers (you can print the other printer parts with your printer) still ongoing and the latest model, Prusa i3 MK3S, is currently on the market - it's an improved version of the original 3D printer's third iteration. Prusa Research keeps expanding into new territories and the Original Prusa SL1 you just purchased is just one of the examples.

Josef Prusa also organizes workshops for the public, participates in professional conferences dedicated to the popularization of 3D printing. For example, he lectured at the TEDx conference in Prague and Vienna, at World Maker Faire in New York, Maker Faire in Rome or at the Open Hardware Summit hosted by MIT. Josef also teaches Arduino at Charles University and was also a lecturer at the Academy of Arts in Prague.

In his own words, he imagines 3D printers will be available in every home in the not-too-distant future. "If anything is needed, you can simply print it. In this field, you just push the boundaries every day... We're glad you're part of it with us!"



Josef Prusa

Table of contents

Product details	6
1 Introduction	8
2 Glossary	9
3 Quick-start information	11
4 Your first print	14
4.1 Unpacking	14
4.2 First start and Calibration	14
4.3 Navigating the menus	17
4.4 Updating the firmware - recommended	18
4.5 Network connection - recommended	18
4.5.1 Internet connection - Ethernet cable	18
4.5.2 Internet connection - Wireless connection	19
4.6 Time and time zone configuration (optional)	19
4.7 Pouring resin into the tank	19
4.8 Running out of resin / Adding resin mid-print	21
4.9 Starting the First print	22
4.10 Removing the print	23
4.11 Removing a print stuck to the bottom of the tank	24
4.12 Removing supports	24
4.13 Bundled 3D models	25
5 After-print checks and actions	25
5.1 Washing / Curing / Drying models	26
5.1.1 Washing	26
5.1.2 Drying and curing	26
5.2 Disposing of resins	27
5.3 Menu structure and advanced functions	28
6 Printing your own models	29
6.1 Obtaining a printable model	29
6.2 Creating your own models	30
6.3 What are SL1 files?	30
6.4 PrusaSlicer	31
6.5 Slicing objects for the SL1	32
6.6 Importing an object into PrusaSlicer	33
6.7 Positioning an object	34
6.8 How to generate supports for SLA prints	34
6.9 Pad, supports, layer height and exposure times	37
6.9.1 Pad	37
6.9.2 Supports	38
6.9.3 Layer height explained	38
6.9.4 Exposure time explained	39
6.10 Slicing, exporting and printing custom models	40
6.11 Object orientation and slicing examples - Reims Cathedral, Eiffel tower, Jewelry, Characters	41

6.12 How to recognize problematic areas	43
6.13 Printing large and heavy objects	44
6.14 Hollowing objects in Meshmixer	44
6.15 Hollowing objects in Windows 3D Builder	46
7 Connecting PrusaSlicer to the SL1 over the network	47
8 Updating the printer's firmware	48
9 Material Guide	49
9.1 Important information about resins	49
9.2 Which resins to buy?	50
9.3 Types of resins	50
10 Software/driver package	51
11 Controlling the printer via a web browser	51
12 Optional Accessories	52
12.1 Original Prusa Curing and Washing Machine - CW1	52
12.2 FEP Films - Original and Aftermarket	52
12.3 Additional accessories	54
13 Regular Printer Maintenance	54
13.1 Keeping the printer clean	54
13.2 FEP film / Tank maintenance	55
13.3 Printing platform maintenance	56
13.4 Exposure screen maintenance	56
13.5 Acrylic lid	56
13.6 UV light and potential health risks	57
13.7 Fan cleaning / Removing dust	57
13.8 Carbon filter replacement	58
13.9 Tower maintenance	58
13.10 Tilt maintenance	58
14 Hardware troubleshooting	59
14.1 FEP film / Tank bottom replacement	59
14.2 Spilled resin	61
15 Print quality troubleshooting	61
15.1 Removing an object stuck to the bottom of the tank	62
15.2 First layer doesn't stick (Non-Adherence)	62
15.3 Printed object has missing parts	64
15.4 Printed object is missing small details	66
15.5 Printed object detached from supports	66
15.6 Broken supports	67
15.7 Printed object is skewed / deformed	68
15.8 Print doesn't appear / resin does not solidify	68
15.9 Printed object has very visible layers	68
15.10 Delamination / Separated layers	69
15.11 Warped layers	70
15.12 Large holes in printed objects	70
16 FAQ - Frequently Asked Questions	71

About this handbook

This handbook is designed to help you start printing with the Original Prusa SL1 as fast as possible. It's divided into several main parts - **pay close attention to chapters about correct usage, health risks, calibration and first print.**

- Basic information about the printer
- Unpacking
- Mandatory calibration
- Your first 3D print and post-printing actions
- Printing your own models - Slicing, Hollowing...
- Advanced techniques - Material Guide, Accessories
- Maintenance, Troubleshooting, FAQ

Product details

Title: Original Prusa SL-1 / Original Prusa SL-1 (kit)

Manufacturer: Prusa Research s.r.o., Partyzánská 188/7A, Prague, 170 00, Czech Republic

Contacts: phone +420 222 263 718, e-mail: info@prusa3d.com

EEE group: 3 (IT and/or telecommunication equipment)

Device use: indoor only

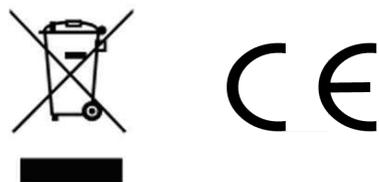
Power supply: 100-240 VAC, 1.8 A (50-60 Hz)

Working temperature range: 18 °C - 32 °C, indoor use only

Working humidity: 85 % or less

Compatible resins: 405nm wavelength

Declaration of conformity can be downloaded from <http://shop.prusa3d.com>



Assembled printer weight (gross / net): 14.5 kg / 10.4 kg. The serial number is located on the back of the printer, in the printer's support menu and also on the packaging.

Original Prusa SL1 is equipped with a wireless network card. This wireless module complies with IEEE 802.11 b/g/n standard and it can achieve a speed of up to 72.2Mbps with single stream in 802.11n draft, 54Mbps as specified in IEEE 802.11g, or 11Mbps for IEEE 802.11b to connect to the wireless LAN. The integrated module provides SDIO interface for Wi-Fi.

Licenses

The Original Prusa SL1 is published under a GNU GPL v3 license (www.gnu.org/licenses/gpl-3.0.en.html). If you improve, or alter any part of, the printer, and you want to sell them, then you have to publish the source code under the same license.

Source codes can be found at <https://www.github.com/prusa3d>

Please be very cautious during any interaction with the printer. This printer is an electrical device with moving parts and UV light emission.

1. The device is for indoor use only. Do not expose the printer to rain or snow. Always keep the printer in a dry environment at a minimum distance of 30 cm from other objects. Make sure none of the vents/fan outlets are blocked.
2. Always place the printer on a stable place, where it cannot fall or tip over.
3. The printer supply is household power outlet 230 VAC, 50 Hz or 110 VAC / 60 Hz. Never connect the printer to a different power supply; it may cause malfunction or damage to the printer.
4. Place the power cord so you cannot stumble over it, step on it, or otherwise expose it to any potential damage. Also make sure that the power cord is not mechanically or otherwise damaged. If so, stop using the damaged power cord immediately and replace it.
5. When you disconnect the power cord from the socket, pull the plug rather than the cord to reduce the risk of damage to the plug or to the AC outlet.
6. Never disassemble the printer's power supply; it does not contain any parts that could be repaired by an unskilled worker. All repairs must be performed by a qualified technician.
8. Do not reach inside the printer while it is still in operation. An injury may be caused by its moving parts or electrical current.
9. Prevent children from unsupervised access to the printer even when the printer is not printing.
10. Do not leave the printer unattended while it's still on!
11. Do not print with a disassembled printer or with a printer that has an open lid or any parts missing.
12. Never look directly into the UV light.
13. Keep in mind that resins are liquids which produce odors. Make sure the printer is set in a ventilated place. Use the bundled plastic tray to shield the machine from drops of liquid.
14. Wear gloves and consider using a respirator and protective goggles when handling resins
15. Some people may be allergic to resins - either to skin contact or even the vapors. If you experience any kind of discomfort when working with resins (like itching), stop immediately and consult your doctor

1 Introduction

Thank you for purchasing the Original Prusa SL1 printer! No matter which version you've bought, an assembled unit or a kit, your purchase supports us in the **further development** of our machines. Please, read this handbook carefully, since all chapters contain valuable information regarding the **use, maintenance, and troubleshooting** of the SL1 printer.

The SL1 is our first SLA printer, but the team behind it has years of experience in the field, which enabled us to bring you an easy-to-use machine with plenty of useful features. The SL1 has a number of smart sensors, calibration wizard, high-resolution display combined with a powerful UV panel, and it's completely open-source!

You can find more information about the SL1 and other printers on our official website <https://www.prusa3d.com> or on our **community hub** <https://www.prusaprinters.org>, where you can download models for printing, chat with other users or create local printing communities!



① How does SLA 3D printing work?

The Original Prusa SL1 is an SLA, or more specifically, an MSLA 3D printer. SLA stands for Stereolithography Apparatus and the “M” at the beginning means “Masked”. SLA printers use UV light to cure (solidify) photosensitive liquid resins.

The SL1 has a built-in high-resolution LCD screen which displays a mask in the shape of a single layer. Then, the UV LED panel shines UV light through the mask into the resin tank, which has a transparent bottom. UV light cures (solidifies) a single layer of the printed object, which adheres to the printing platform. After a layer is cured, the printing platform moves by a small increment upwards, creating space for another layer. SLA is thus an additive printing method.



2 Glossary

Printing platform - Printing platform is made of a single-block of aluminum and it is connected via a cantilever to the motorized tower. The large flat part is the area to which the printed object is attached to. The printing platform can be easily removed by loosening the **Knob** without affecting the calibration

Knob - Large black knob at the top of the cantilever is designed for an easy mounting/dismounting of the printing platform.

Tank - The container for liquid resin. It has a transparent bottom that allows UV light to pass through. The bottom of the tank is an FEP film, a consumable that will degrade after some time. The FEP film is replaceable (see chapter on Printer Maintenance). Use only warm water and dishwashing soap to clean the FEP.

Tilt - The general name for the whole bed with the motorized tilt function. The tilting platform is one of the biggest advantages of this machine. Thanks to this solution, smaller forces are required to pull the object from the

bottom of the tank, which results in better-looking models and more reliable printing.

Tower / Z-Axis - The mechanism that raises and lowers the cantilever with the printing platform. The cantilever is secured with a **ball screw**.

Lid - The orange-colored lid is made of acrylic and its layers protect the liquid resin from UV light (such as sunlight), which would otherwise cure it. Still, it's not a 100% shield and the printer should not be placed in direct sunlight. It must be closed during printing.

Liquid resin - Liquid materials used for printing in SLA 3D printers. It requires special care during handling - see chapter Resins

Curing / solidifying - Curing is the process of solidifying liquid resin using UV light. Even sunlight contains UV light and it can cause the liquid resin to solidify. The recommended curing time per single layer is usually between 5-12 seconds depending on the resin used - there may be exceptions. Layer height, object

size, and exposure (curing) time are the three factors that have a major impact on the length of the printing process and quality of the printed object.

Touchscreen Display / Touch screen - The main control interface of the SL1. The capacitive touchscreen is used to navigate through the menus, start prints, adjust settings, or calibrate the printer.

Power Button - Turns the printer on or off. This button won't disconnect the power supply. To cut off power completely, always disconnect the power cord from the back of the printer.

2K screen / Print display / Exposition display - This display is part of the tilt assembly. It shows a mask (shape) of a single layer at a time, allowing UV light to pass

through and solidify a single layer. The fixed resolution of the display means that the X/Y resolution of the printed object is also fixed.

UV LEDs / UV panel - The UV LED panel emits UV light through the mask shown on the print display and solidifies (cures) the resin in the tank, creating the desired object layer by layer.

Ethernet port - Can be used to connect the printer to the local network. It is located on the back of the device.

USB port - Used for reading SL1 print files from a USB drive

SL1 file - File containing printing instructions compatible with the SL1. SL1 files are created by slicing a 3D object using PrusaSlicer

2.1 Accessories - bundled and optional

There's a number of accessories and tools you will find in the package. They will help you to use the printer more effectively and safely.

- **Gloves** - for safer handling of resins
- **Protective tray** - to shield the printer from resins during object removal
- **Sharp pliers** - for support removal
- **Funnel with a filter** - for filtering resin
- **Plastic spatula** - the only spatula you can use to carefully clean the FEP film
- **Metal spatula** - for removing objects from the printing platform only
- **Allen and torx keys** - for maintenance
- **Orange resin** - 0.5l bottle
- **FEP film** - spare FEP films (not pictured)



Optional accessories you should consider buying:

- **Paper towels** - for cleaning up the printer, drying printed models...
- **Isopropyl alcohol** - for cleaning the prints
- **Plastic tablecloth** - to contain resin spillage and to make cleaning up easier
- **Pipette** - for removing larger resin drops/stains

- Respirator / protective glasses

3 Quick-start information

The SL1 requires **maintenance and regular care**. It doesn't take a lot of time and it's not complicated, however, there are several parts on the 3D printer that can be potentially damaged as a result of incorrect use. Please keep that in mind - **failure to follow the instructions in this manual may lead to severe damage to your printer and/or to a loss of warranty**. Here is a quick run-down of things you should and shouldn't do with your Original Prusa SL1.



SL1 3D printer

- ! Place in a **well-ventilated room** with ambient temperatures between 18-32°C
- ! Connect to the internet and download the **latest firmware** updates
- ! **Calibrate** the printer before the first use
- ! **Keep clean** at all times
- ! Use the bundled **drip tray** when filling the tank or removing prints

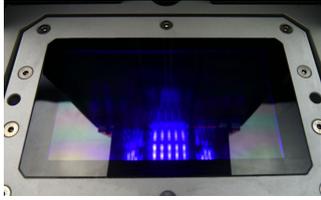
- ✗ **Do not** place the printer in **direct sunlight**
- ✗ **Do not** keep the **lid open** while printing

Resin tank



- ! **Empty and clean** the tank after every print because tiny bits of cured resin might remain in the tank
- ! Clean only with **warm water** and a bit of dishwashing soap or regular soap
- ! **Replace the FEP film** with a new one in case it's damaged

- ✗ **Do not** clean the FEP foil with **IPA** or similar chemicals
- ✗ **Do not** use **sharp objects** to remove prints (e.g. metal spatula)
- ✗ **Do not** print with **damaged** (punctured) FEP foil
- ✗ **Do not pour more than the recommended amount of resin (max 200 ml)**



Print display and UV LED panel

! Use a **soft paper towel** or cloth and an LCD screen cleaning spray to clean the protective glass

! You can **take a picture** of the UV LED panel with your smartphone to check if it works correctly

✗ **Do not look directly in UV light**, use a camera to check the UV panel

✗ **Do not leave spilled resin** on the print screen

✗ **Do not use sharp objects** to clean the screen

Resins



! Wear **gloves** when handling resins, also consider wearing a simple **respirator** and protective goggles

! Store at **room temperature** in **opaque bottles**, away from kids or animals

! Treat as **dangerous liquids**

! Follow your country's laws for disposing of dangerous liquids

✗ **Avoid skin contact**

✗ **Do not drink**

✗ **Do not mix** different resins

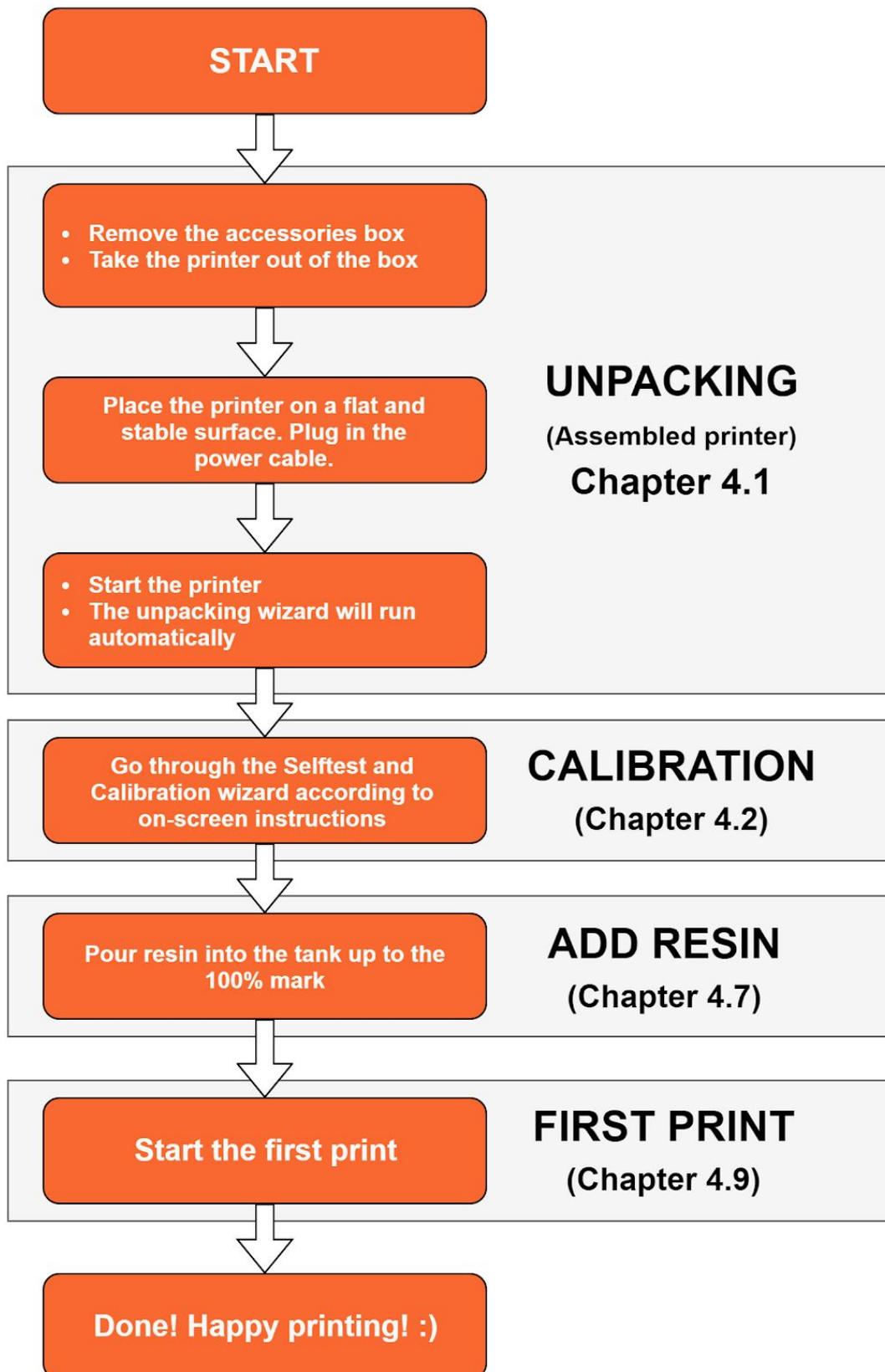
✗ **Do not mix** used and unused resins

✗ **Do not flush** resins down the drain / into sewage

✗ **Do not leave** spilled resin anywhere - clean it immediately

ⓘ **Setting up the printer is quick and easy, don't skip it!**

Once you receive the printer, the whole process of unpacking, calibration and first-time setup takes about 10-20 minutes. Follow the instructions in the next chapters and you'll be ready to print in no time!



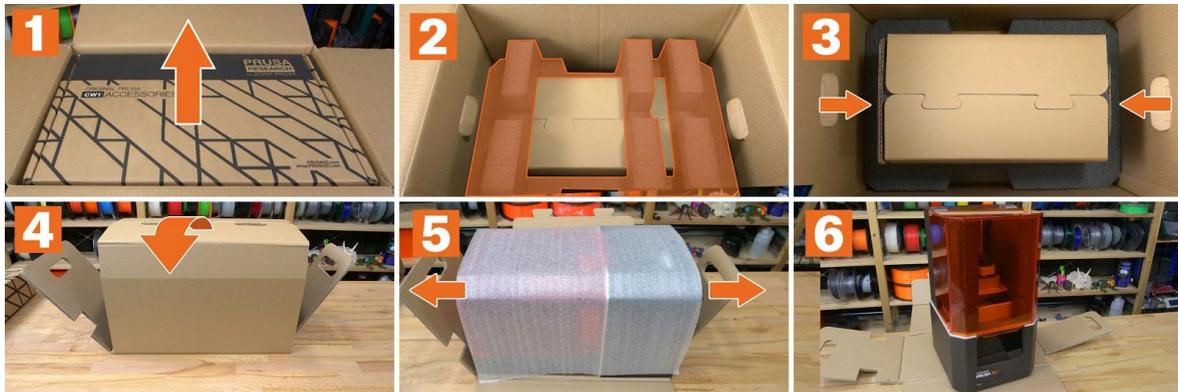
4 YOUR FIRST PRINT

We know that you are eager to start printing immediately, however, there are **a couple of steps** users need to take before they can start printing. Read these chapters carefully.

4.1 Unpacking

After you open the box, remove the accessories case first (1), then remove the top foam (2). Grab the box using the two handles on the side and pull it up (3). Open the cardboard locks (4) and remove the protective wrap (5). Finally, take the unpacked SL1 and place it on a stable surface (6). **This is all for now! Do not open the cover yet!**

If the printer was kept in low temperature environment for an extended period of time, **let it warm up** in ambient room temperature for at least one hour **before you start it**. Otherwise, you may get errors related to low temperatures.



Start the printer and run the initial wizard. Continue in chapter [4.2 First start and calibration](#).

4.2 First start and Calibration

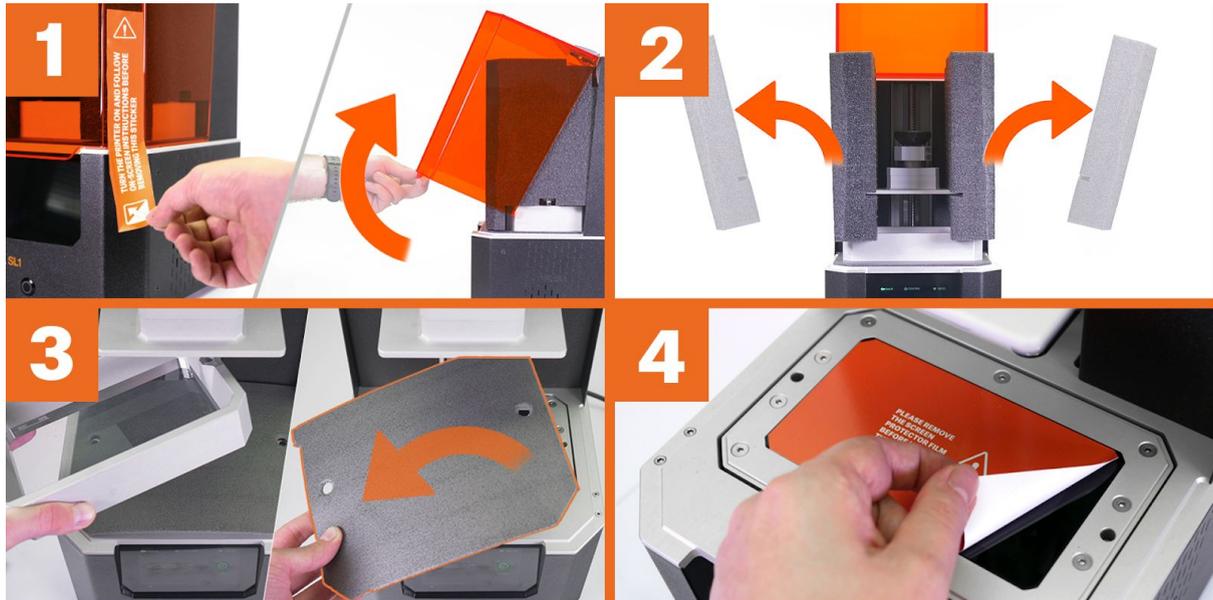
The calibration is essential for correct operation

Without a properly calibrated tilt and printing platform, the printed object won't stick to the platform, or even worse - you may damage your printer.

Connect the power cord and turn on the printer using the **switch on the back** of the machine, located directly next to the power plug. Then push the **round power button** on the front side. The start of the booting sequence is signaled by a slow-flashing light around the power button. After a few seconds the display will show the Prusa Research logo and then the **Wizard** appears. Use the touchscreen to navigate through the wizard. **Take your time, read all instructions carefully** and remember you scan **swipe left/right** to reveal more information about the current step.

Each described step has to be confirmed by pressing the on-screen Continue button!

The first is the **Unpacking Wizard**, which will help you to remove the remaining parts of the packaging:



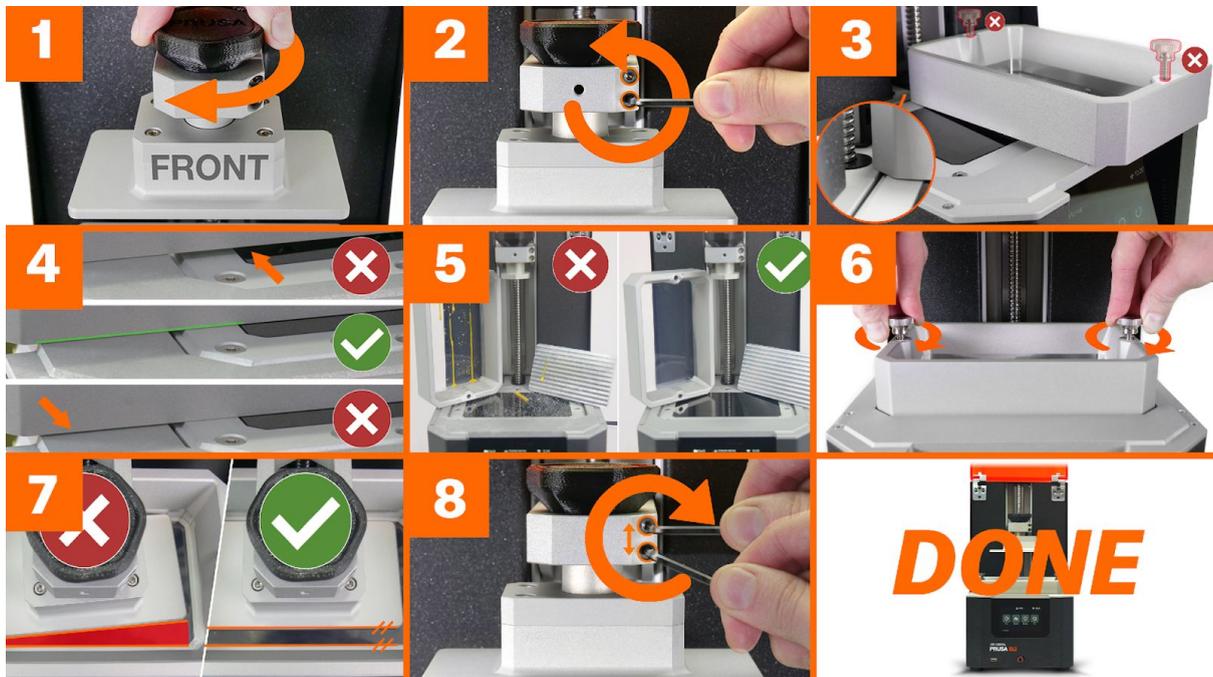
1. Peel off the sticker on the right side of the printer and open the cover
2. Remove the foam protectors
3. Remove the tank and the foam layer underneath it
4. Carefully peel off the sticker from the display

Once the 3D printer is fully unpacked, the **Selftest** starts. This procedure will make sure your printer arrived in a good shape:



1. Insert the resin tank and tighten both screws
2. Remove the printing platform
3. The printer will perform an automated test of the tower axis and fans
4. Close the lid
5. Look into the printer through the top of the lid. You should see the Prusa logo
6. Insert the printing platform **rotated by 60°** and tighten the black knob

Finally, there's the **Calibration wizard**:



1. Make sure the print platform is clean, slide it onto the platform holder on the cantilever and tighten the black knob.
2. Loosen the screws on the cantilever using an Allen key. The platform should move quite freely, but the knob on top has to remain tightened
3. Take the empty resin tank, remove both large screws, rotate it by 90° and place it across the silver printer frame as depicted in the picture. Make sure the tank rests flat and level on the printer frame and **not on the display**
4. Use the on-screen controls to move the tilt until the display and frame are in direct contact with the resin tank. The tank must not be lifted above the frame. See the picture below for a reference. Tap or hold the on-screen buttons to move the platform.
5. Make sure that both the platform and the tank are perfectly clean.
6. Place the resin tank on the tilt bed and secure it with both screws. Tighten them evenly and with the same amount of force, otherwise, the tank may end up in a skewed position

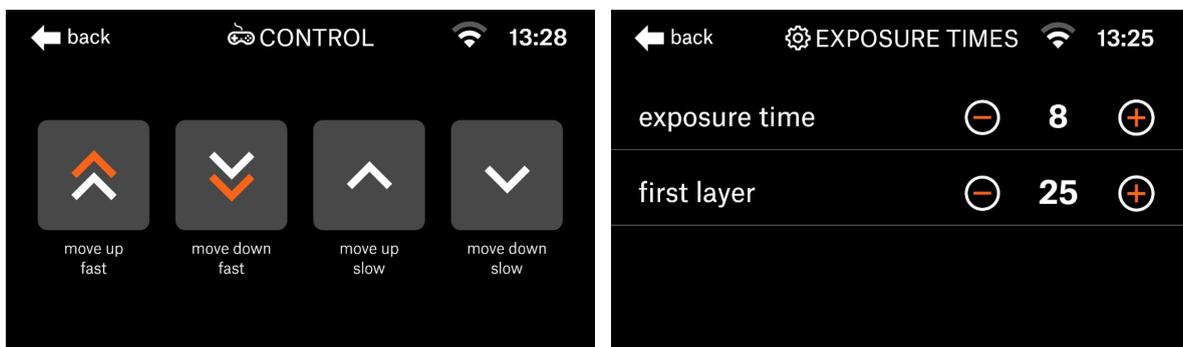
7. The platform will now go all the way down to the bottom of the tank. Rotate the platform so it's parallel to the side of the tank
8. This is the **most important part** of the whole calibration: tighten both screws on the cantilever with an Allen key as evenly as possible! Tighten one screw by a few degrees, then tighten the other one, then the first screw again... This prevents the platform from undesirable rotation.

That's it! The printer is now **calibrated** and doesn't have to be calibrated again unless you transport it to a new location or replace certain parts of the printer (e.g. the FEP foil). The printing platform is secured by the black knob at the top of the cantilever and removing it from the printer won't affect the calibration.

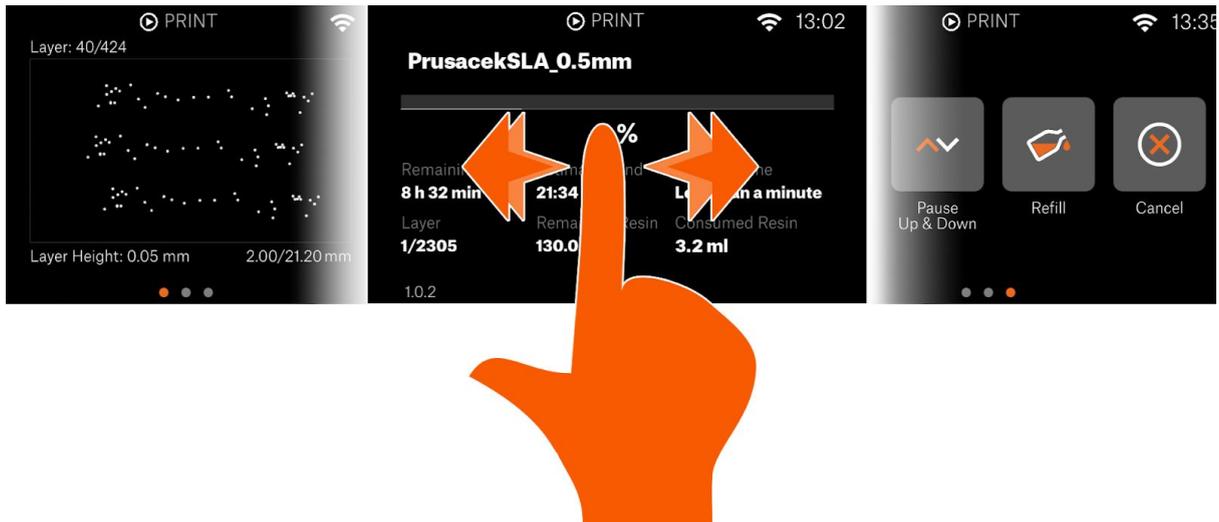
Once the calibration is done, the Main Menu is displayed. Please see chapters [4.3 Navigating the menus](#) and [5.3 Menu structure](#) for more information.

4.3 Navigating the menus

SL1 uses a **touchscreen display** for navigation and for configuration of basic and advanced features. To enter a sub-menu (e.g. Print, Settings...), press the desired icon on the screen. To **go back** to the previous screen, press the **Back on-screen button**. To **change values** in various sub-menus, use the on-screen symbols (such as + and - or arrows up and down). Pressing and holding a symbol results in a **continuous increase or decrease** of the associated value. Some screens can be swiped to reveal additional information - these screens are marked with little dots at the bottom of the page.



Use touchscreen controls to navigate the menus. Use short or long button presses to change the on-screen values



Menus with dots at the bottom can be swiped left/right to reveal additional information

4.4 Updating the firmware - recommended

Once your printer is connected to the internet (see the next chapter), it can check whether a **newer firmware is available online**. Please follow the on-screen instructions to update the firmware.

It's a good practice to **keep the firmware up-to-date** all the time because newer versions come with new features, bugfixes, and improvements.

If you don't have the printer connected to the internet, you can **update the firmware using a USB drive**. Download the firmware file from <https://prusa3d.com/drivers>, place it on a USB drive and plug it into the printer. The system will recognize the file on the drive and initiate the update process automatically.

4.5 Network connection - recommended

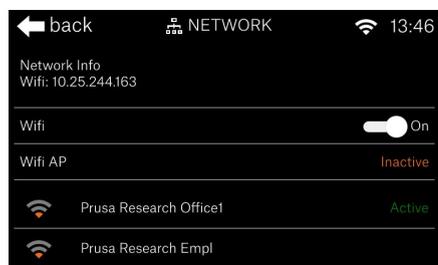
Before you start the calibration process, it is advised to **connect the printer to the internet**, so that it can download updated firmware files (if they are available). There are **two ways** how to connect the printer to the internet.

4.5.1 Internet connection - Ethernet cable

You can use a standard **RJ-45 ethernet cable**. Connect it to your router, and the printer should obtain an IP address automatically - however, this depends on your network configuration. Since every network is different, please refer to your router's instructions or your provider's tech support in case you run into trouble. To check whether the connection has been successfully created, go to *Main Menu - Settings - Network*.

4.5.2 Internet connection - Wireless connection

The SL1 3D printer has a **built-in Wi-fi network card** compatible with 2.4GHz networks. To open the **Wi-Fi network browser**, go to Settings - Network. A list of **all available networks** will be displayed. Select your Wi-Fi network and connect to it by typing your password using the on-screen touch keyboard. You can connect a USB keyboard to the front USB port in case the on-screen keyboard is too small.



i Network troubleshooting

Since every network is different, we cannot provide exact instructions on how to connect to your wireless/wired network. If you encounter any trouble, please refer to your router's manual, or contact your internet provider's tech support. Quality of your wireless connection depends on many factors - such as the type/configuration of the router, distance of the printer from the router or wireless interference in your area.

You can recognize a successful connection by looking at the Wi-Fi icon - it should now display connection strength. To see more details about the connection, please navigate to *Main Menu - Settings - Network - Network Info*.

4.6 Time and time zone configuration (optional)

You can configure the printer's time zone and current time in Settings - Advanced Settings - Time Settings. If you have the printer connected to the internet, you can use Automatic time settings to have the time updated automatically depending on your time zone. Next, you should go to Timezone menu and select your continent and nearest town/city. If you don't have the printer connected to the internet, please set the time manually using Date and Time options.

i *Setting the correct time is useful during printing - the screen will show you when the print job is going to finish.*

4.7 Pouring resin into the tank

Before you start printing, always make sure the FEP film on the bottom is clear (transparent), clean and has **no holes, dents or deep scratches**. Check the tilt platform with the display and make sure it is completely clean and there is no debris anywhere.

⚠ Wear gloves when handling resins!

Place the tank on the tilt platform. There is a **small groove** running around the tilt bed that will help you to position the tank properly. You may hear a little “click” as it falls into place. Use the two resin tank screws to secure it to the tilt bed. Tighten screws **simultaneously and with the same amount of force**, otherwise the tank may end up in an uneven position.



Tighten both screws simultaneously and with the same amount of force. Pour the recommended amount of resin - there is a max level mark in the tank.

Once the tank is secure, **pour the resin in it**. Notice that there is a **max level mark** in the tank that will help you measure the optimal volume. The SL1 features a built-in resin level sensor, so in case there's too little or too much resin, the printer will pause and display an on-screen info message. Please follow the on-screen instructions to fix the issue.



The max resin level mark in the tank.

⚠ Do not overfill the tank!

Any time you are asked to refill the resin to the max, it always means adding it up to the 100% mark, not to the edge of the tank!

4.8 Running out of resin / Adding resin mid-print

The SL1 features a built-in resin level sensor, which measures the amount of resin in the tank before every print. Some models might be **too large** and require **more than the maximum amount** of 200 ml resin in the tank. The printer can recognize when the resin starts running low and **pauses the print**. An on-screen message is displayed, asking you to **fill resin** back to the max level mark. Once you refill the resin, you can press the on-screen **Continue button** to resume the print.

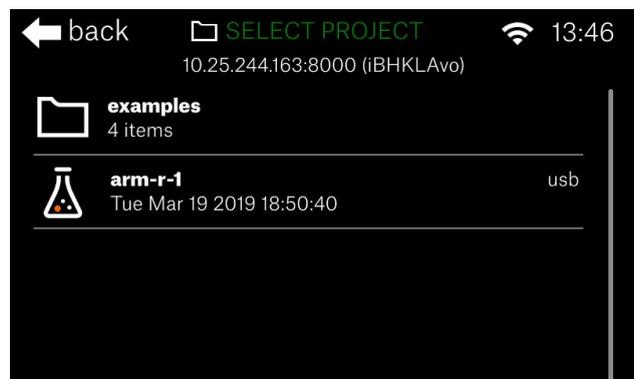
ⓘ Keep in mind that in certain cases (especially if the print has been paused for a long time), the new layer may not adhere perfectly to the rest of the object - or not adhere at all.

4.9 Starting the First print

One last check before you start the first print:

- ? Is the printer updated to the latest firmware version?
- ? Did you calibrate the printer using the wizard?
- ? Have you checked the FEP film in the tank?
- ? Have you poured a correct amount of resin in the tank?
- ? Is the lid closed?

Go to *Main Menu - Print - Examples*. A list of available print files will show up on the screen. **Swipe the screen up and down** to scroll through multiple entries. Start with a sample model - these models are ideal for the first print because they will show clearly, whether the printer has been **calibrated correctly**. Tap the model to select one of them and then confirm the selection by pressing **Continue**.



The printer will perform a **quick self-test** and start printing. Keep the lid closed and do not interrupt the print. Since the printing platform is **submerged in the resin**, you can't see whether the first layer attached correctly. Try listening for the **typical noise** that comes with the layer separation - it sounds a bit similar to the sound of a sticky tape being removed. Don't panic if you don't hear it clearly, though.

You can use the *Up & Down function* to check whether the first few layers attached properly (e.g. the pad and beginning of supports). To activate this feature, **swipe** the screen right during printing and press the *Up & Down* button. **However, the *Up & Down function* may create visible lines on the printed object.**

Even if the first layer won't stick to the printing platform correctly, there will be **no permanent damage to the printer**.

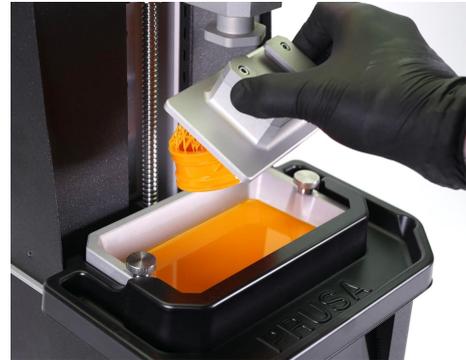
4.10 Removing the print

⚠ Safety instructions - wear gloves!

When handling resins, always wear protective gloves and always clean up any drops of resin around the printer. In case the resin gets into contact with your skin, wash it off immediately with water and soap. If you experience discomfort (such as itching) or the resin gets in your eyes, seek out a doctor immediately.

After the printing process finishes, open the lid and **place the black plastic tray on the printer** as depicted in the picture.

Use a **plastic spatula or paper towels** to wipe the top part of the **printing platform** clean. Then, **unscrew the black knob** at the top of the tower to unlock the printing platform. Carefully slide the printing platform out, and turn it sideways, so the **remaining resin goes back to the tank/tray**.



Use the bundled **metal spatula** to remove the print from the printing platform. To prevent injuries, **do NOT use your fingernails to remove the prints** since the plastic can easily get under your nails and break off.



Removing a printed object with a metal spatula

Inspect the printed sample object. Check for any **inaccuracies, missing parts, visible layers/lines, dents, blobs, and other defects**. If your print has visible defects, check the [last chapters of this handbook](#) for print quality troubleshooting.

You should **clean the printing platform thoroughly** after every print - drops of liquid resin can solidify in sunlight, which would make them difficult to remove. Clean the channel at the top of the printing platform thoroughly. Use isopropyl alcohol to clean the platform.



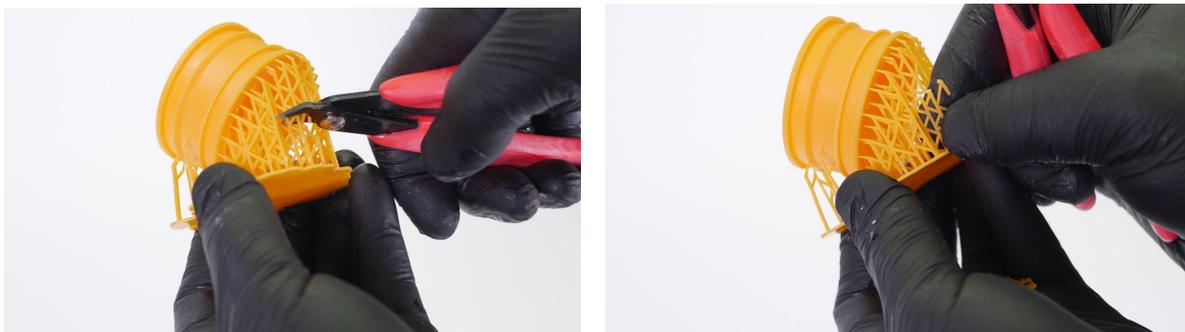
Cleaning the print platform

4.11 Removing a print stuck to the bottom of the tank

In case the first layer did not stick to the printing platform, it remained **stuck at the bottom of the tank**. No matter how big or how small the stuck layer is, there's a **quick safe way** that will help you **remove it from the FEP film** without damaging it - see chapter [15.1 - Removing an object stuck to the bottom of the tank](#).

4.12 Removing supports

Supports for SLA printers are designed to be **thin, yet strong enough to support the printed material**. The end of support is called a **support head** and they are usually even thinner than the rest of the support, which makes them **easy to remove**, leaving little to no trace on the object itself. However, heavier or larger objects may require **stronger (thicker) supports**.



To remove supports from 3D printed objects, there's a couple of things to keep in mind:

- It's easier to remove the supports **directly after** the model is 3D printed and washed in isopropyl alcohol. Curing the model with supports still attached can make the supports more difficult to remove (although this is often resin-specific)
- **Don't use excessive force.** Instead, use the bundled pliers to carefully remove the supports.
- To achieve **better results** (especially with tiny objects), try cutting the supports **in the middle** first, then proceed to the ends. Another good option is to rinse the supports in hot water for a short while. Supports will become softer and will be easier to remove
- Removed supports should be treated as **plastic waste**

4.13 Bundled 3D models

The Original Prusa SL1 comes with several bundled models. These models are pre-sliced and their purpose is to **provide a benchmark for end users**. By including complex geometry, extremely thin lines and miniature patterns, these models will give you a good overview of whether your printer has been **calibrated correctly**. The models are pre-sliced, so if they don't print correctly, you can be sure the problem is not in slicing settings.

To delete the example objects from the printer's internal memory, or your own projects from the **USB drive**, go to the **Select Project** menu (*Main Menu - Print*), slide any of the projects to the left to reveal a waste bin icon. Tap this icon to immediately delete the project.

i You can find the sample models in the printer's menu. Go to **Print - Examples** to display a list of sample models. All samples are designed to be printed with the Prusa Orange resin. If you decide to use a different resin, modify the default values!

5 After-print checks and actions

The SL1 needs to be **cleaned up after every print** to avoid potential damage to the printer. Failure to maintain your printer properly may lead to severe damage and loss of warranty.

- **Never leave the resin in the tank** after a print finishes. There may be tiny bits of solidified resin left in the tank, which could potentially damage the FEP foil or the display the next time the printing platform would go all the way down. Also, resin could solidify directly in the tank if you keep it in for extended periods of time.
- After you pour resin out of the tank, **take a very soft paper towel and wipe the tank clean**. Make sure no resin has been left on the outer sides of the tank or on the underside.
- If the FEP film has a **greyish tint** (it's not completely clear), you can restore it to the original look by cleaning it with warm water combined with a bit of dishwasher soap or regular soap. Dry it thoroughly afterward. **Do not use IPA to clean the FEP.**

- Make sure there are **no visible dents, scratches or holes** in the bottom of the tank. Check the display on the tilt platform and make sure there is no dirt, stains or cracks on the display. Then insert the tank back into the printer and secure it with resin tank screws.
- **Clean the printing platform** - use paper towels to wipe the platform clean or wash it carefully in isopropyl alcohol. Read instructions on the IPA bottle before use.

5.1 Washing / Curing / Drying models

⚠ Until the models are completely cured, there may be trace amounts of liquid resin or IPA on their surface. Wear gloves when handling uncured objects! This applies especially to the following chapters.

Prints made of solidified resin tend to be **slightly soft and sticky**. To reach optimal surface properties, it is necessary to **wash, dry and cure** the print. We recommend using our own **Curing and Washing Machine** (CW1 for short). It's a 3-in-1 device: it can wash the object in isopropyl alcohol, dry it using hot air and cure it with UV light - all in a **matter of minutes** and without any extra hassle.

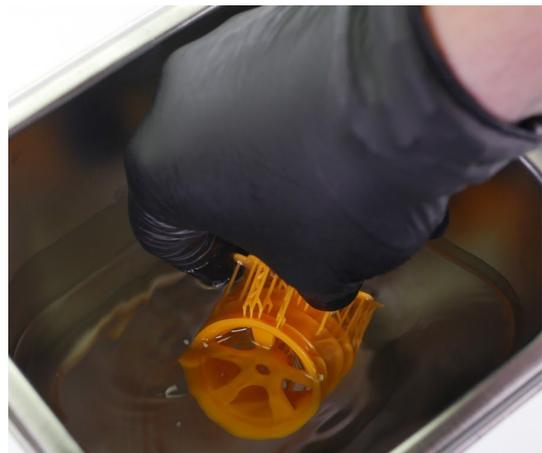
The **Curing and Washing Machine** features a rotating platform and a lid with a reflective surface that improves the overall performance, reducing the total time needed to perform all three actions to less than ten minutes.

Without the Curing and Washing Machine, you have several options:

5.1.1 Washing

Resin-based prints should be **always washed in isopropyl alcohol (IPA)**. Exercise caution when dealing with this liquid - it's flammable and it shouldn't get in contact with your skin or eyes. **Always read the instructions** on the bottle before using it.

When you wash prints in IPA, try moving them around or use a small and very soft brush/toothbrush to properly clean all parts of the object. The print can usually stay submerged in IPA for several minutes without any noticeable decrease in surface/material quality. **Don't keep the object submerged in IPA for extended periods of time** (hours or days). If the print develops various cracks (even after 24 hours), it's usually because the print has been kept in IPA for too long.



5.1.2 Drying and curing

Use a **soft paper towel** to dry your print and/or put it on the windowsill - ideally in **direct sunlight**.

Curing the model without a strong UV light source can **take a long time**. One of the common practices is to place the model in **direct sunlight for an extended period of time** (e.g. 24-48 hours) depending on the intensity of sunlight. The CW1 can cure an object within 3-5 minutes.

Protip: You can use a **UV LED nail lamp dryer** (a machine used for curing nail polish). They run around 30-40 USD and they can cure the surface of a resin-based print quickly and efficiently. Just make sure to get a product with a chamber large enough to accommodate the prints.

5.2 Disposing of resins

⚠ Resins are dangerous chemicals!

Used resins should be treated as dangerous liquids (same as motor oils etc.). See your country's laws that cover the disposal of dangerous waste / liquids / chemicals to learn more about this matter. Liquid resin must not be, under any circumstances, poured into drainage or sewerage (e.g. into a sink).

After you finish a print, we recommend **pouring the leftover resin into a new bottle**, so it doesn't mix with unused clean resin. Used resin may contain tiny solidified bits that may affect the quality of the next print or even damage your printer. Also, since the resin has been exposed to UV light, its properties might have changed slightly.

We recommend curing the paper towels soaked with liquid resin in a curing chamber (either our Curing and Washing Machine or a UV LED nail lamp dryer). Another option is to leave them in direct sunlight to cure the resin, then treat them as plastic waste.



Use the bundled funnel with a filter to pour used resin into a bottle (don't mix old and new resins). Clean the tank thoroughly after every print.

If you decide to use the resin again anyway, always make sure the liquid is **properly filtered** and does not contain any solidified leftovers from previous prints. If the print quality is lower with reused material, consider switching to an unused resin. Clean the filter often to prevent clogging.

5.3 Menu structure and advanced functions

- **Print**
 - **Project browser**
 - **Print settings**
- **Control**
 - **Home Platform** - performs the homing procedure
 - **Home Tank** - performs the homing procedure
 - **Disable Steppers** - cuts off the power to stepper motors
- **Settings**
 - **Network** - Wi-fi connection configuration / Network information
 - **Recalibration** - Re-runs the Calibration wizard
 - **Advanced Settings**
 - **Move Platform** - Manual control of the printing platform
 - **Move Resin Tank** - Manual control of the tilt
 - **Time Settings** - Configuration of time settings
 - **Hostname** - Configuration of the SL1's network name
 - **Login Credentials** - Configuration of login credentials for network access
 - **Wizard** - Re-runs the Selftest
 - **Display Test** - Displays a testing pattern on the print display (keep the lid closed!)
 - **Update Firmware** - Over-the-air firmware update
 - **Save Logs to USB** - save logs onto a USB drive
 - **Platform Axis Sensitivity**
 - **Tank Axis Sensitivity**
 - **Limit for Fast Tilt**
 - **Tower Offset**
 - **Rear Fan Speed**
 - **Auto Power Off**
 - **Cover Check**
 - **Resin Sensor**
 - **Factory Reset**
 - **Support**
- **Off**

6 Printing your own models

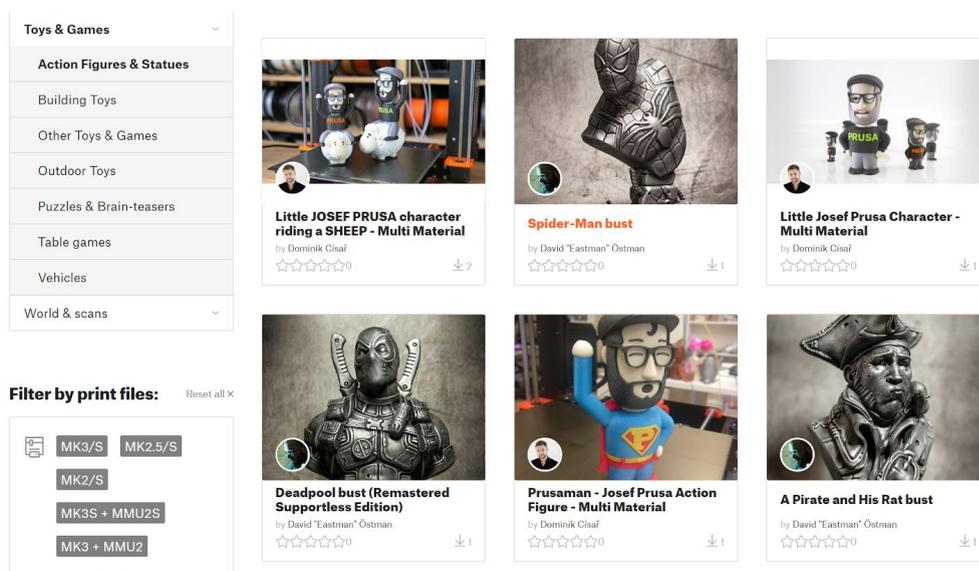
Once the printer is fully calibrated (see the previous chapter) and the sample models are looking good, you will probably want to **print your own model**. There are various ways how to do that.

i Slicing procedure overview in 6 steps

1. Import an object into Slicer □ 2. Position the object, configure printing parameters and slice the object □ 3. Check the preview and adjust slicing parameters (e.g. supports) if needed □ 4. Export the SL1 file onto a USB drive and insert it into the printer □ 5. Open the Print menu and select the object for printing □ 6. Confirm the settings and start the print

6.1 Obtaining a printable model

The easiest way how to get started with your own 3D printing is to find **models on the internet** - they are usually in .3mf, .stl or .obj formats. Fortunately, 3D printing has many fans, so there are sites from which you can download a large variety of ready-made 3D models for free. These range from a simple playing dice to a detailed figure from a favorite movie, game or TV series. Be sure to check out our own **community website PrusaPrinters.org** - it's a brand new community hub for all Prusa 3D printer owners! It has a ton of cool features and **plenty of amazing models** for you to print.



PrusaPrinters.org

3D models are **very often free to download** under the **Creative Commons - Attribution - Non Commercial** (Models not to be used commercially, you must always include the name

of the author) or for a small fee. We have selected the most interesting sites:

- <http://www.prusaprinters.org>
- <http://www.thingiverse.com/>
- <http://www.myminifactory.com>
- <https://pinshape.com/>
- <https://www.youmagine.com/>
- <http://www.shapeways.com/>
- <http://www.gambody.com>

However, models in .stl, .obj or similar formats **cannot be 3D printed directly**. First, they need to be “**sliced**” (**converted**) into a SL1 file, which is then placed onto a **USB drive**. Connect the USB drive with a sliced project into the printer and select the model to print from the **Print** menu. Please see the chapters [6.3 What are SL1 files?](#) and [6.4 PrusaSlicer](#) for more information.

6.2 Creating your own models

To create a **3D model**, you need a dedicated program - a **3D editor**. There are a number of various 3D programs and your choice will usually depend on what type of model you wish to create.

The **easiest place to start** is **TinkerCad** (www.tinkercad.com) - it’s an online editor that runs in your browser’s window, so no installation is required. It’s free, easy to navigate, and you will find plenty of tutorials online. However, TinkerCad is mostly focused on creating **less-detailed and larger** (e.g. mechanical) parts, often ideal for FFF/FDM printing. Another popular tool is **Autodesk Fusion 360** (www.autodesk.com/products/fusion-360) for PC, Mac, and iPad. The website provides a quick guide along with detailed video tutorials so it is an ideal choice both for beginning enthusiasts and professionals.

i *To fully utilize the precision of the SL1, you should focus on programs such as **Blender**, **Zbrush**, **Maya**, **SculptGL**, **MakeHuman**, **SketchUp**, **Clara.io**, **Moment of Inspiration** or **3D Studio Max**. These applications are better for creating organic shapes (such as characters or character parts), but only a few of them are free.*

6.3 What are SL1 files?

Before you can 3D print a downloaded or created model, you need to **convert it** into an SL1 file first. This is typically done by a process called “**slicing**”, which requires a special software - **PrusaSlicer**.

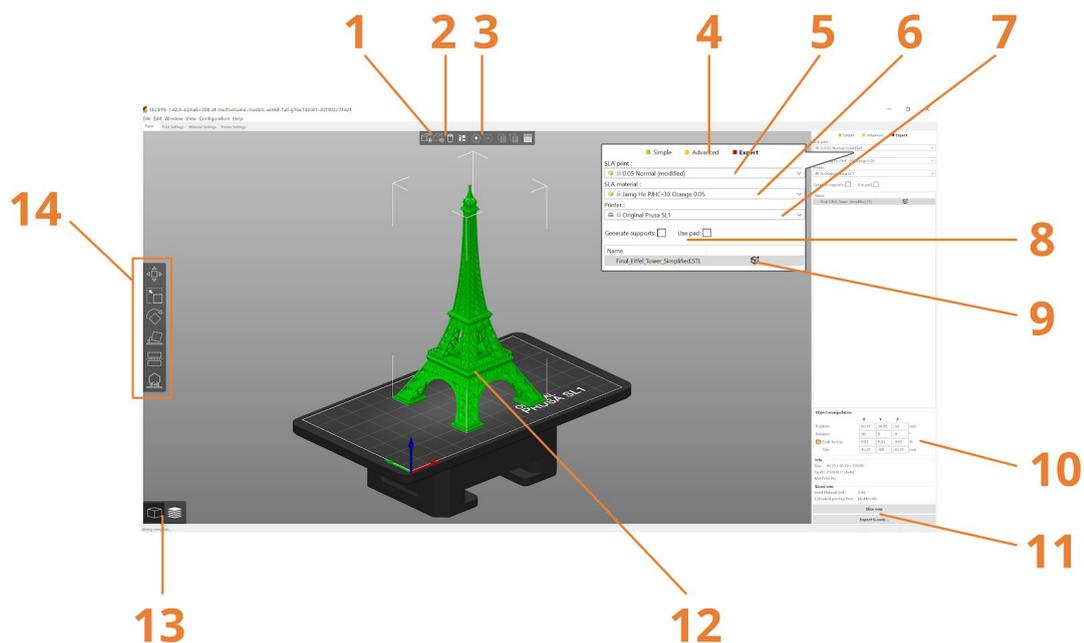
Essentially, the slicer takes a 3D model and **cuts it into thin layers**. Each layer is saved as a separate PNG file. These files, along with a configuration INI file, are stored in a

compressed ZIP file, which is automatically renamed to SL1. If you rename SL1 back to ZIP, you can actually see the content of the file.

6.4 PrusaSlicer

PrusaSlicer is our own open-source tool for slicing models. It's the only slicer you will ever need for your Original Prusa printers! It contains print profiles and recommended settings for the entire range of Original Prusa devices.

You can download **the latest version of PrusaSlicer** from www.prusa3d.com/drivers (stable releases) as a part of the **Apps & Drivers package**.



1. Add object
2. Remove object / Remove All
3. Add / Remove an instance of an object
4. Switch between Simple / Advanced / Expert mode
5. Layer height
6. Liquid resin type
7. Printer selection
8. Generate supports / Use pad checkboxes
9. Object modifiers
10. Object size / scale information
11. Slice Now and Export G-code buttons
12. 3D object preview
13. 3D editor view / Preview window switching
14. Move, scale, rotate, place on face, cut and support tools - explained in following chapters

Basic importing, slicing and exporting 3D objects is simple. However, PrusaSlicer has advanced menus and settings for experienced users, allowing for a higher level of control.

📘 Want to learn more about PrusaSlicer?

Visit our Blog at <https://www.prusaprinters.org> where we publish tips, tricks and extensive guides that will turn you into a master of slicing!

6.5 Slicing objects for the SL1

Every 3D model is different, so an **individual approach is needed** for every single object. However, we can still offer several tips, tricks and hints that will make the slicing and printing processes easier. If you already have experience with FFF/FDM printing, you will notice that the **preparation of models for SLA printers is slightly different**. These chapters will explain all the steps required for a successful preparation of a model for printing - this covers support generation, explanation of terms such as pad, exposure or layer height, and many other things.

📘 Learn about SLA supports!

SLA has a completely different type of supports and different object orientation compared to FFF/FDM 3D printers. If you already own one of our Original Prusa i3 3D printers, you should check out the main differences below.

There are several goals you should try to meet as closely as possible when preparing an object for slicing/printing.

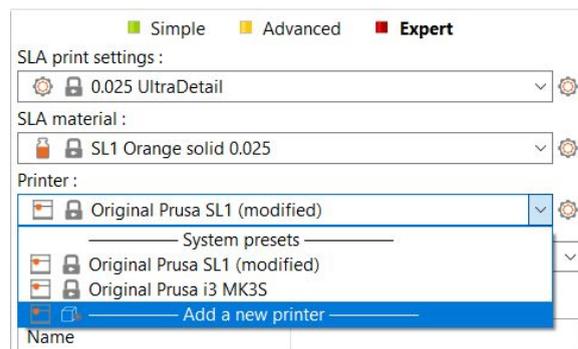
- **SLA is an additive manufacturing method** - objects are created layer-by-layer, which means that layers are added on top of existing ones and they cannot be created mid-air. Such parts **must be supported with supports**, a scaffolding-like structure. Choosing a good object orientation minimizes the amount of supports needed. Supports for SLA are different compared to FFF/FDM 3D printers.
- **Decide when to use the pad and/or supports** - for some objects, using the pad and supports is not mandatory. One of such examples is the model of Eiffel tower. It has four reasonably sized contact points, which can be placed directly on the printing platform. Since the tower rises at an optimal angle, no supports are required.
- **Tilted models have better results** - when you have an object with a large flat base, it's a good practice to rotate it, so it raises under an angle of around 45° in two axes depending on the model's shape. Tilted models require lesser peeling force during layer separation process
- **Decide between quality and "safety"** - even though supports for SLA printers are tiny compared to FFF/FDM printers, they can still leave small marks on the object's

surface. If you rotate the object so the important parts are facing away from the printing platform, it will help you to conserve details on the print. However, sometimes it's better to add a higher number of supports for a "safer" print - especially when printing larger objects that may detach due to their weight. Thicker supports are an option, too.

- **Object orientation affects the print time** - tall objects will take longer to print than shorter ones. Consider rotating the object to save time.
- **Consider making the printed object hollow** - SLA prints in general can sometimes fail due to excessive suction: basically, when you print an object, the printer needs to separate it from the tank bottom. If the print has a large flat area or a cavity, the suction can be so strong, the object will detach from the supports / printing platform. Consider using Meshmixer to hollow the object and create two holes in it - this has several benefits: the object will be lighter, you will use less material, and the chance the print will fail will be minimal.
- **Using "bridges" is not recommended** - FFF/FDM 3D printer users know a method called bridging. It means the printer can extrude and cool down the string of filament in mid-air to overcome a gap between two points even without using supports. However, this is not recommended for SLA printers. The first layer of the "bridge" is usually too thin and flexible, so the rest of the layers won't adhere perfectly and little cracks and gaps can appear. Add supports or find a better orientation for the object.
- **ALWAYS** carefully check the sliced object in the **Preview window** and make sure that there are no unsupported parts of the model or other issues that would cause the print to fail

6.6 Importing an object into PrusaSlicer

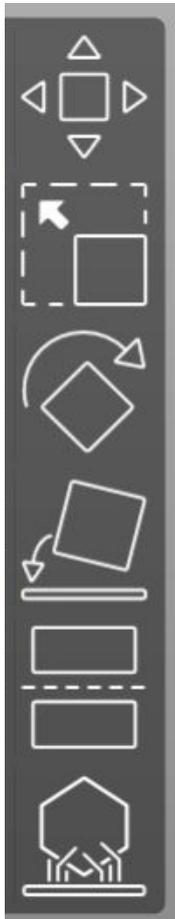
Once you start PrusaSlicer, select the **Original Prusa SL1** in the *Printer menu* on the right side of the window. If it's not listed, you can add it either by selecting *Add a new printer* option in the same menu, or by going to *Configuration -> Configuration Wizard*. Then, select the **layer height** in the *SLA print menu* and finally the **resin** you want to use. Please note that built-in resin profiles have pre-defined and tested settings. If you select a wrong profile, it may affect your print due to incompatible exposure times.



PrusaSlicer allows you to import **STL, OBJ, AMF and 3MF** object files - these are the most common types of 3D files you can find on the internet. You can either drag-and-drop them in the *Plater* window, or you can use the **Add...** button in the top toolbar.

Next, use the tools in the left toolbar to **Move, Scale and Rotate** the model. If the object is blue, it means it's too big for the printing platform. See the next chapter ([6.8 Positioning an object](#)) for more details. **Every object is different**, so there's no default orientation that would work for every object. However, you can find a couple of examples in the next chapters that will give you a better idea how to rotate and support certain types of objects.

6.7 Positioning an object



Move tool (M) - moves the object in XYZ axes. Click and drag the object to move it in XY axes, use the blue arrow to move it in Z axis (up and down).

Scale tool (S) - scales the object, either uniformly or in a single axis. You can adjust the values in the right-side menu by entering numbers in Scale factors section. Entering 100 resets the object to original size.

Rotate tool (R) - rotates the object in X, Y or Z plane. Click and drag one of the rotation gizmos to rotate the object smoothly or in steps (by dragging it along the circular guide).

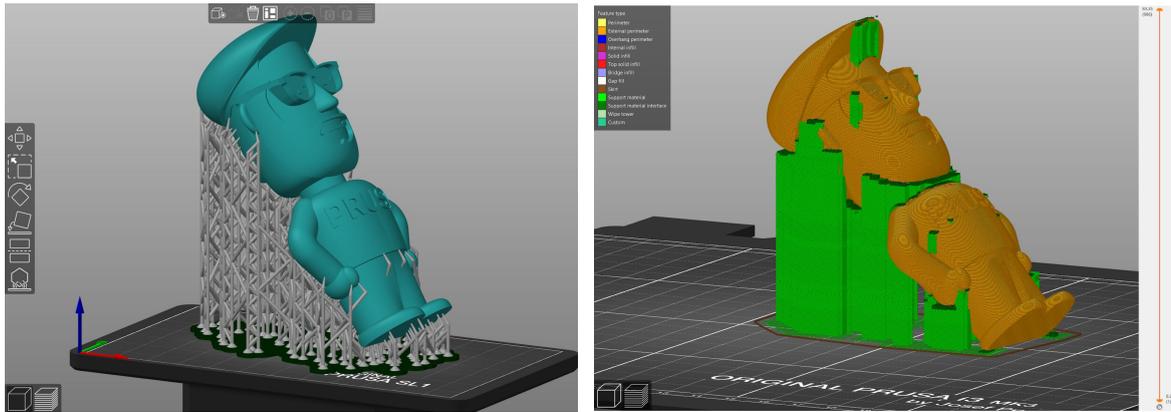
Place on Face (F) - displays selectable "faces" (sides) on the object. Click on a face to automatically position the object so it lays down flat on the selected face. Useful for objects with a flat base, or if you want to rotate an object quickly.

Cut tool (C) - For splitting an object. In case you have a model that has an unwanted part (e.g. a base under a figure), you can use the cut tool to cut the model in half. You can decide whether to keep both parts of the model or delete one of them. Cuts can be made only in plane.

SLA Support points (L) - Only available for Original Prusa SL1. See the next chapter for detailed explanation.

6.8 How to generate supports for SLA prints

Supports are an extremely important tool - it's rarely possible to print an object completely without supports. Compared to FFF/FDM 3D printers, resin-based printers usually **require more supports**, because the models should be printed lifted above the printing platform and under an angle to achieve better results. However, **SLA supports are thinner and easier to remove**. PrusaSlicer is fully equipped for automatic support generation, as well as manual editing.

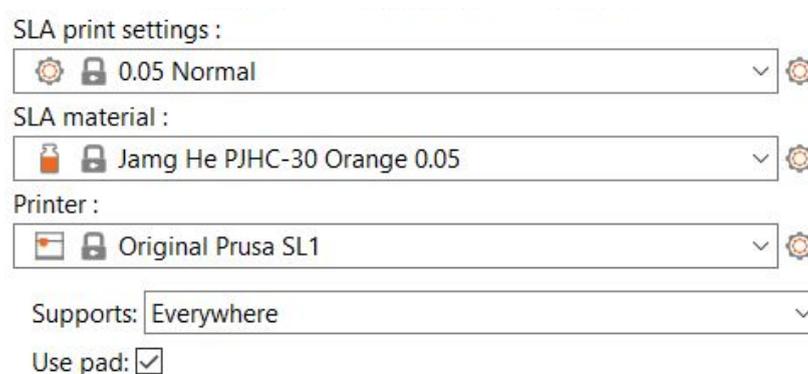


Comparison of SLA supports (left) and FFF supports (right)

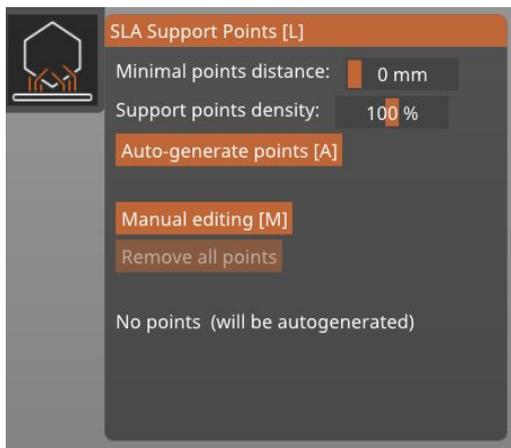
Each print will require a **slightly different approach** based on the shape, size and details present on the model. In this chapter, we'll explain the basics of support creation in general. For more examples, see next chapters.

Decide whether the object you are slicing requires a **pad**. The pad (see chapter [6.3. Pad](#)) is a **base for the model and supports**, which makes printing and removing the object easier. Some objects with a moderately sized flat base don't need a pad in order to be printed successfully.

In the **right-side menu**, you can choose to **turn on/off the pad** and also select the **type of supports** that will be generated: **None** (no supports will be generated), **On build plate only** (the supports will be generated only on the printing platform), or **Everywhere** (supports will be generated everywhere, that means even between various parts of the model). You can then move onto support generation, but you can always come back and change these options.



Turning the pad on/off in the right-side menu



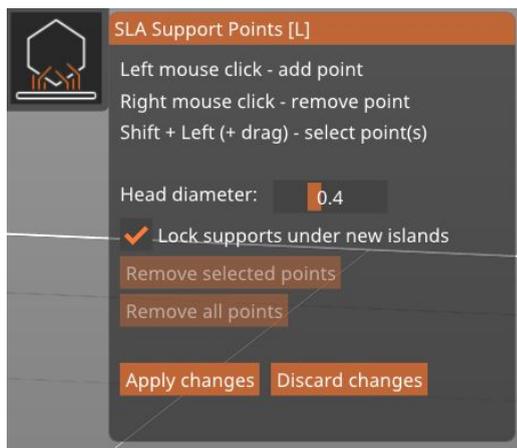
In the picture on the left is the **SLA Support Points** menu, which can be found in the left-side toolbar. If you can't open it, it usually means the model you have imported is too large for the print bed. Move, rotate and scale the object so it fits - the object should be displayed in green colour.

Minimal points distance: specifies how far apart the attaching points for support heads should be generated. The default value is 1 mm.

Support points density: specifies how many supports will be generated. Larger and heavier objects will require more supports. Support points density takes the Minimal points distance into account.

Auto-generate points - starts generating the supports based on configured parameters. You can re-generate the supports if you are not satisfied with the result and decide to change the values.

Manual editing



Clicking the Manual editing button opens a new menu and a new set of tools, which enable you to manually add, remove and modify support points on the object. Use Left mouse click to add a support, right mouse click to remove a support, and combine Shift+Left mouse to drag a selection box to select more supports. You can left-click and drag a support point to reposition it.

Manual editing of supports is useful when the automatic generation doesn't deliver a perfect result. Once you have sliced and printed a few objects, you will be able to recognize which parts of the model may require extra supports - those are usually places we call "islands", basically the critical parts that start mid-air. It's a good practice to add a few extra supports to these places to ensure that the model will withstand the peeling forces.

- **Head diameter** - specifies the **size of the support head**. The maximum value is 1 mm by default - this can be changed by modifying the support diameter in *Print settings* -> *Supports* -> *Support pillar diameter*. Increasing this value may require increasing the *Object elevation* value as well. The default value for support head diameter is **0.4 mm** and it's suitable for most objects. Larger and heavier objects **may**

require stronger connections (higher values). The support point gizmo has a **small cone-shaped part** that shows the direction in which the support will be attached and its size.

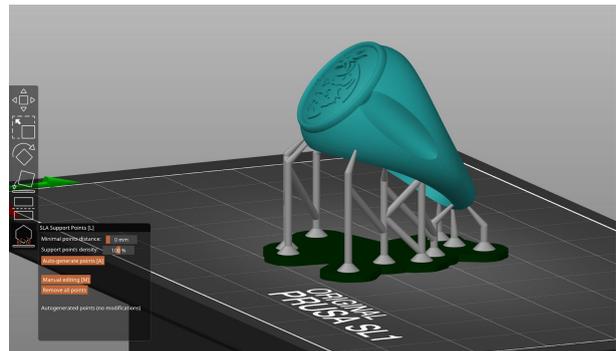
- **Lock supports under new islands** - these are the critical parts of the object that often start mid-air. Automatic generation of supports can recognize these parts and creates supports for them accordingly. If you decide to manually edit the supports, you can “lock” these critical points so you don’t accidentally delete them
- **Remove selected points** - removes the points that you have selected (e.g. by shift+left clicking them). Selected points are marked red. Locked points won’t be deleted.
- **Remove all points** - removes all points, including automatically generated and even locked points
- **Apply changes** - confirms the changes and returns to the previous menu
- **Discard changes** - discards manually made changes and returns to the previous state

6.9 Pad, supports, layer height and exposure times

There are several tools and settings that will help you print **even very complex objects** without a hitch. **Pad and supports** are useful for achieving a good object positioning on the printing platform, while the **layer height and exposure times** affect the number of details on the object, as well as the overall printing time. However, the print time is mainly affected by the total height of the object.

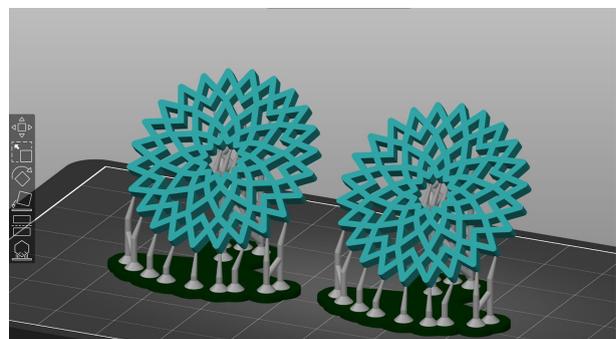
6.9.1 Pad

The pad is a structure generated **under the print** - it’s the base for supports. You can easily **detach the pad** along with the print from the printing platform with a spatula. The **pad is recommended**, however, it’s also possible to print certain objects without it, depending on the object’s shape. The pad can be turned on/off in PrusaSlicer. If you want to **print with supports**, using a pad is strongly recommended



6.9.2 Supports

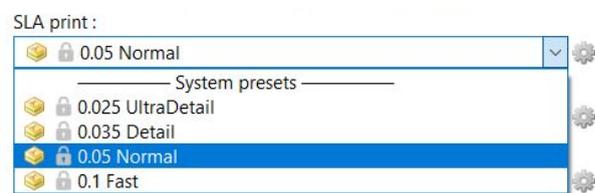
SLA printing is an **additive manufacturing method**: every new layer needs to have something below it, so there’s something to build upon. It’s **not possible to start a new layer mid-air**. You can either rotate the object until you reach optimal angles, so the



overhangs are small enough, or you can use **supports**. Supports are **scaffolding-like structures** that are automatically or manually generated in PrusaSlicer. Supports for SLA printers are very thin and they have pointed ends, so the contact area with the model is as small as possible. Supports have **various settings** (thickness, support head diameter, object elevation...) and you should change these settings based on the object you want to print. Heavier/larger objects require more (or thicker) supports.

6.9.3 Layer height explained

Compared to FFF/FDM printers, MSLA 3D printers have a fixed XY resolution (determined by the resolution of the LCD screen), so the only axis, that dynamically affects the **looks of the model**, is the Z-axis. How much the Z-axis (the tower with the printing platform) moves between layers is determined by the **layer height settings**.



The technology behind the SL1 enables it to reach a layer height of 0.01 mm. However, this value is not recommended for regular use, since there's no real benefit in using it. Instead, we're offering **our presets with 0.025, 0.035, 0.05 and 0.1mm layer height**. With the SL1 printer, 0.05 mm is considered the "normal" quality and we recommend sticking to it - it offers a good balance between speed (time) and good surface finish with almost invisible layers. **Lower values** will result in **better surface finish**, but also in **longer print times**. Please note that certain resins are limited to a narrower layer height range.

i Using lower layer height will lead to more detailed objects, but the print will take much longer to finish. To see an estimation of how long a print will take with various layer heights, check the PrusaSlicer window after you slice an object.

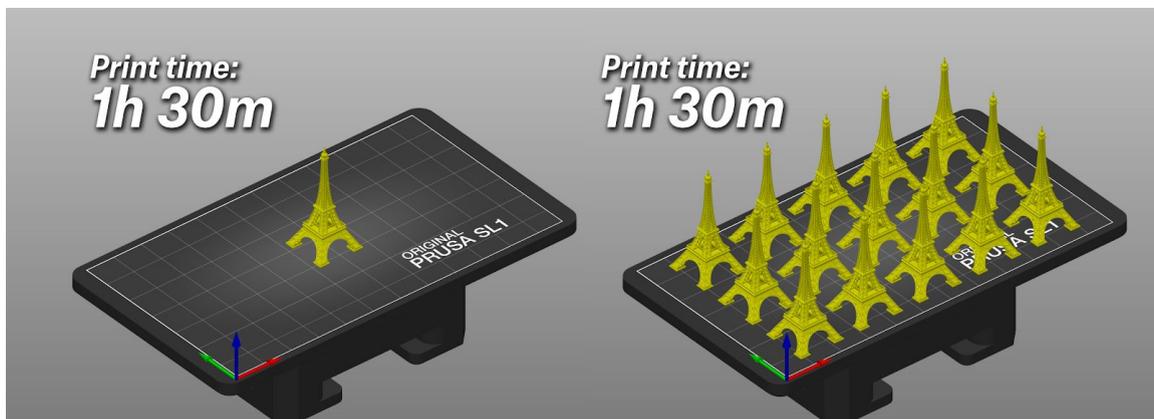
Info	
Size:	61.82 x 52.06 x 44.01
Facets:	43992 (1 shells)
Manifold:	Yes

Sliced Info	
Used Material (ml) :	5.19
- object(s)	2.31
- supports and pad	2.88
Estimated printing time :	3h 33m 16s

PrusaSlicer can estimate the printing time and amount of resin needed

6.9.4 Exposure time explained

PrusaSlicer allows you to **change the exposure time** (the number of seconds for which a single layer is cured) for standard layers and first layers separately. There are no “universally usable” numbers, so we **recommend using the preconfigured PrusaSlicer profiles**. We have tested over a hundred of different resins, and prepared tested profiles for each of them. Using our built-in profiles is recommended. If the resin you bought is not listed in PrusaSlicer, you can **tweak the values yourself** - sticking to 6-10 seconds for standard layers and 25-35 seconds for first layers is a good practice.



Both prints will finish in 1 hour and 30 minutes

The SL1 features a **high-performance UV LED panel**, which has enough power to reach very short exposure times while retaining good surface properties of the printed model. Going under 5 seconds is possible, however, most resins **won't solidify in such a short amount of time**. The prints will start to lose detail or won't have enough time to solidify completely, and won't stick to the rest of the object, resulting in a failed print. However, some resins on the market can be cured with as low as 3-second exposure. The best option is to use resins with **pre-made presets in PrusaSlicer**, which also feature correct exposure times.

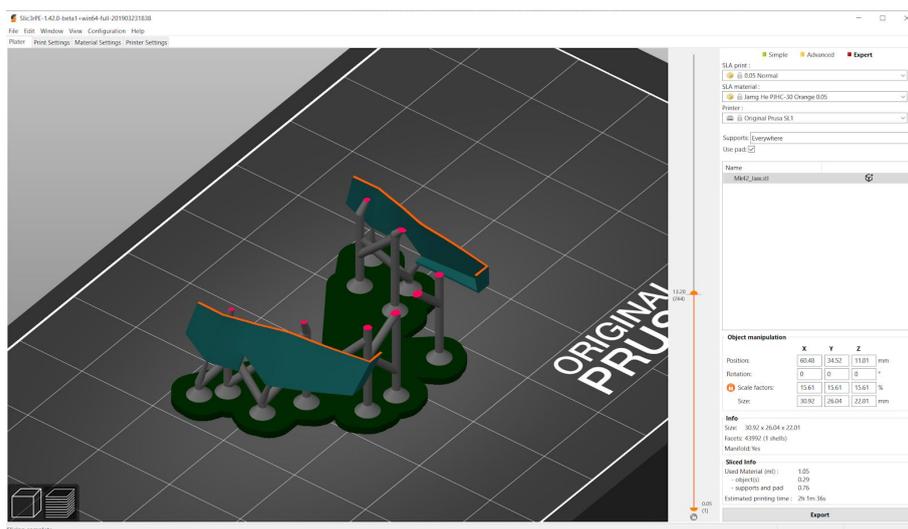
It is possible to change the exposure time before, or even during the print by going into *Settings menu -> Change exposure time*. However, changing exposure during print may result in visible lines across the printed object. **Extremely low or high numbers will almost always result in a failed print.**

i Important information about print times

*Keep in mind that no matter how many objects (or how big) you place on the printing platform, the exposure time remains **constant**, so each layer will be finished in the **same amount of time**. In other words, printing a single 10 cm tall object or a dozen of 10 cm tall objects placed next to each other will always take the same amount of time. This is a major difference when compared to FFF/FDM printers. Take advantage of it!*

6.10 Slicing, exporting and printing custom models

Once your object is placed on the printing bed correctly and supports are generated, you can press the **Slice now** button. Slice will analyze and process the 3D model and turn it into a number of separate layers.



Inspecting the object in Preview mode before the export

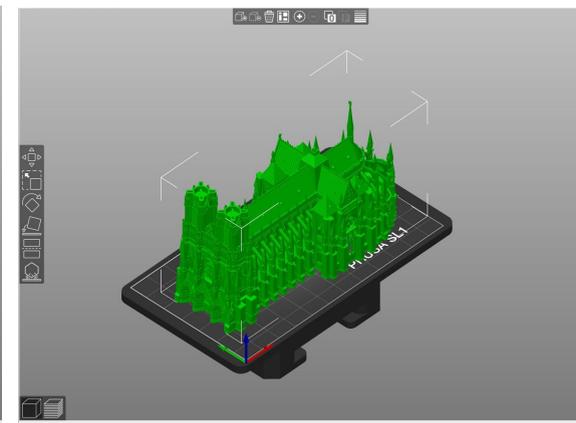
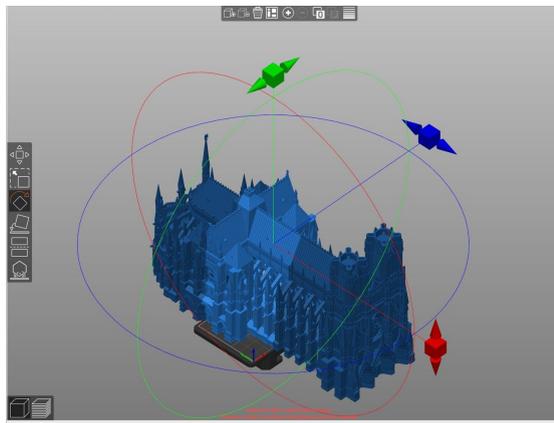
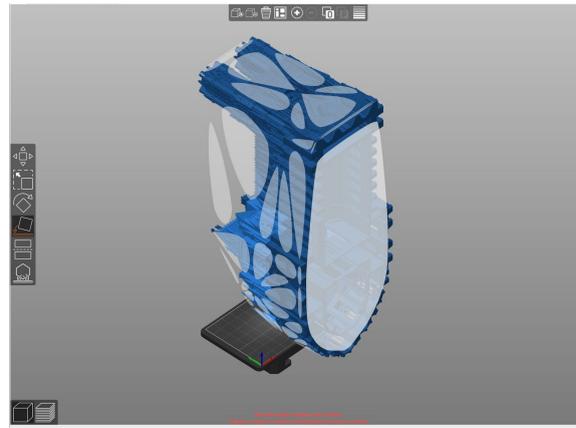
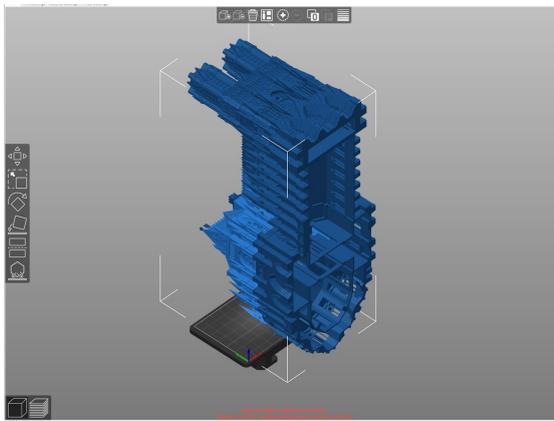
Inspect the object in the **Preview window** using the slider on the right side of the window. If everything seems to be ok, press the **Export** button to generate an .SL1 file. Place this file on a USB drive and connect it into the printer. Tap the **Print** icon to display a list of compatible projects on the drive. Select the desired model and confirm it with the Continue button. You can also upload files directly into the SL1's internal storage. See chapter [7 Connecting PrusaSlicer to SL1 over the network](#).

6.11 Object orientation and slicing examples

There are different kinds of objects you can print, so it's nearly impossible to give exact instructions for every type of model out there. On the following pages, we're offering a selection of popular models and offer some tips on how to position them before printing.

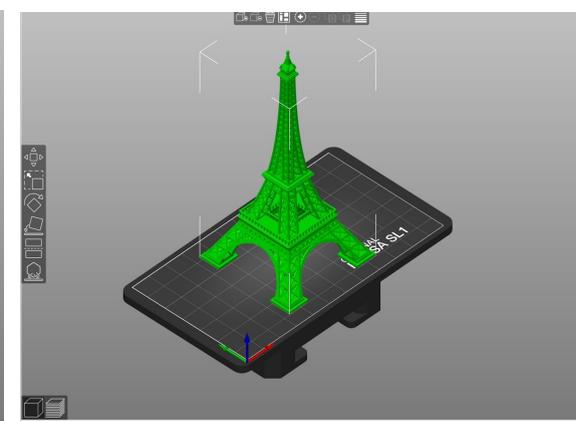
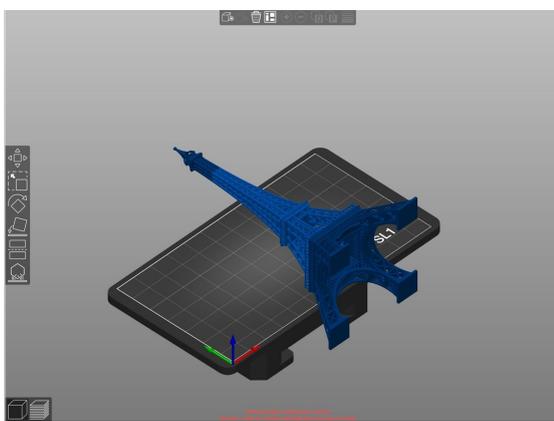
6.11.1 Reims Cathedral

This is a very complex model which is actually very easy to slice and print. Disable supports and pad and use the place on face function to quickly rotate the object. Then use Scale, Move and Rotate tools to adjust the final placement before you slice the model.



6.11.2 Eiffel tower

This is another model that is very easy to slice. Disable the **Pad and Supports** and use the **Place on face** function along with move/rotate/scale tools to position the object until it's green (meaning it fits the printing platform).

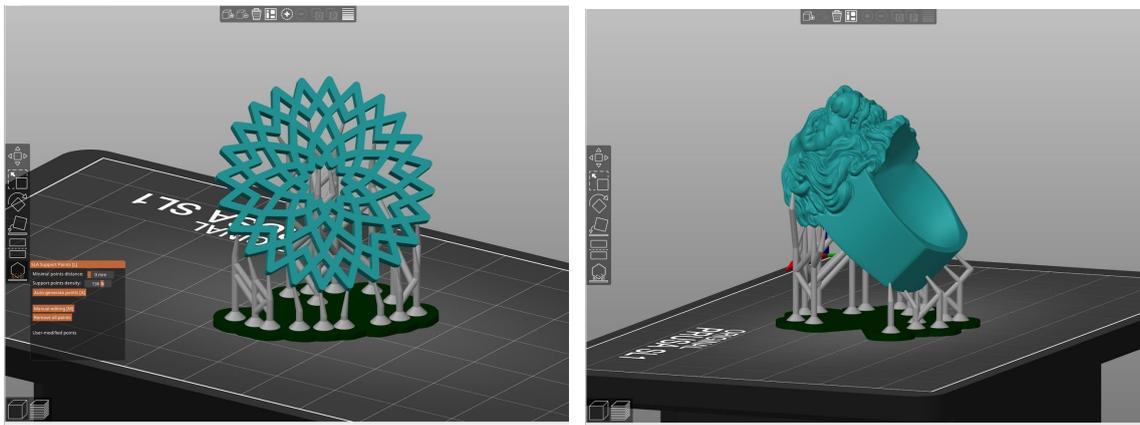


6.11.3 Jewelry

The Original Prusa SL1 is **perfect for printing tiny, highly-detailed models**. There's a couple of things to avoid in order to achieve a great looking surface. The most important thing to keep in mind is the fact that supports can leave **tiny marks** on the object's surface, so if your model has parts that should be printed perfectly, make sure there are no supports touching them.

You will need to use a combination of the **Pad, supports, Rotate/Move/Scale tool**. Also, configure the supports correctly in case the automatic generation won't deliver the desired result. Jewelry and similar objects are very small, so you don't usually need a ton of supports.

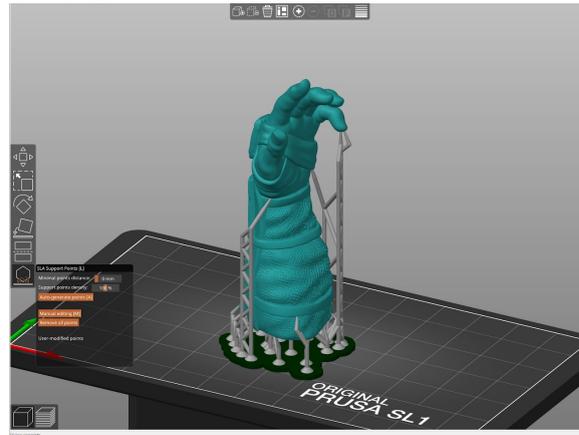
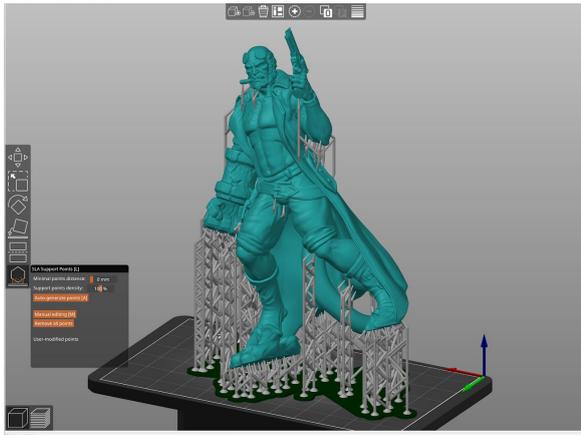
You can check the examples below to see how we positioned them before printing.



6.11.4 Characters and props

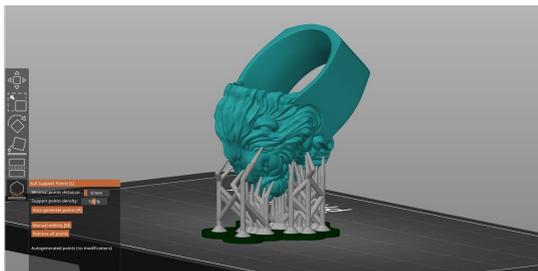
Characters and props are often some of the **most complex objects** you can find. The Original Prusa SL1 can handle the details on character models without any issue, so you can produce high-quality figures or props from your favorite movie or game.

Usually, you will need to use the pad and supports, and adjust the **density of supports** (or add extra supports manually) in case you decide to print a more complex object. Also, keep in mind that objects consisting of several parts, like body, arms, head and legs, can be a bit challenging to assemble in case the original model creator did not leave enough tolerance for joints or pegs. The only option is to **adjust the problematic parts** with a fine sanding paper. Larger models should be **hollowed** first to reduce weight - hollowing helps to save material and hollowed objects require less supports. See chapter [6.12 Hollowing objects](#)

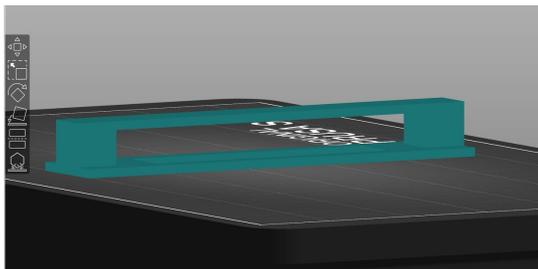


6.12 How to recognize problematic areas

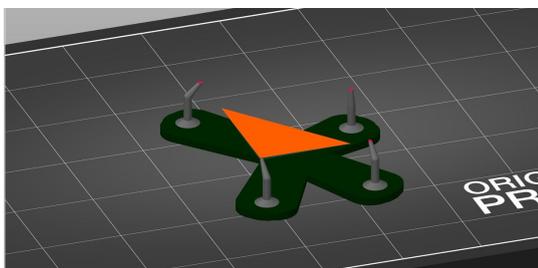
Each time you slice an object, inspect it carefully in the Preview window. Never start a print without checking the model layer-by-layer first. The Preview mode will help you to discover problematic areas that could lead to a failed print. These are usually parts that aren't properly supported, broken model files, incorrectly placed holes (with the hollowing method) and similar issues.



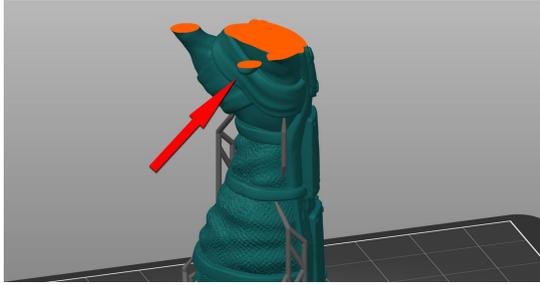
Incorrectly rotated object - the SL1 will print it without issue, however, removing the supports may leave small marks on the detailed (and most important) part of the object.



Bridging doesn't work the same way as with FFF/FDM 3D printers. The SL1 cures a single layer at a time, which in this case would most likely lead to layer separation due to peeling forces. Tilt the object by 45° degrees and generate supports to print this object without issues.



Insufficient amount of supports - the orange colour shows a single layer that the printer is going to cure and then attempt to separate from the bottom of the tank. The single support can't withstand the peeling forces. This usually happens if you set the support density to extremely low numbers.



Part of an object starting mid-air. PrusaSlicer can recognize these issues almost every time - these errors usually appear if you manually delete an important support point. You can fix it by re-generating the supports or by adding support points manually.

6.13 Printing large and heavy objects

When printing large and heavy objects, you may find out that the **standard supports** are **not strong enough** to support the weight of a large object.

There are two ways how to deal with large and heavy objects:

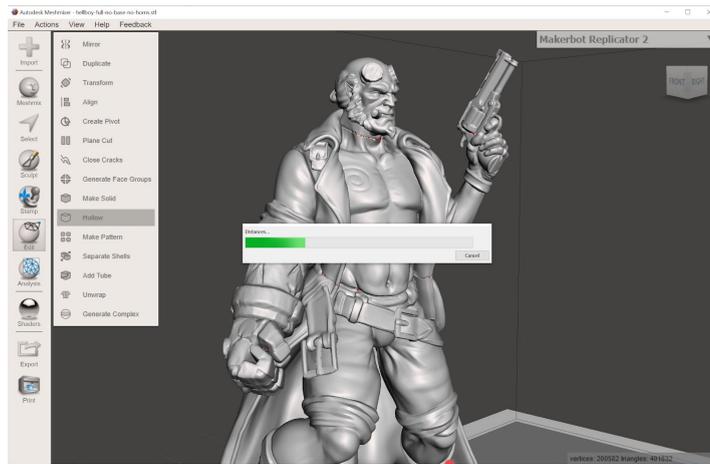
- 1) **Change the settings for supports**, make them thicker and sturdier
- 2) **Hollow the object in Meshmixer** - see the next chapter

You can easily modify the support parameters by opening the Print Settings tab in PrusaSlicer and selecting Supports. There are three values you should try modifying:

- 1) **Support pillar diameter** - changes the diameter of supports
- 2) **Support head front diameter** - changes the size of the support head
- 3) **Support head penetration** - changes how far into the model will the support head go

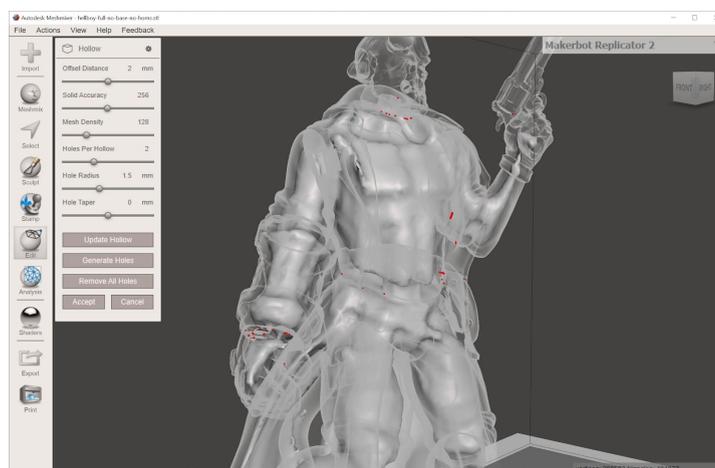
6.14 Hollowing objects in Meshmixer

In many cases, you don't need the model to be completely solid. Before you start slicing a model, you can **hollow it first**. To produce a hollowed model, you will need a free tool called **Meshmixer**. Download it from the [official website](#), install it and run it. Once the application starts, click the Import button to load the desired 3D model.



In the menu on the left side select Edit -> Hollow.

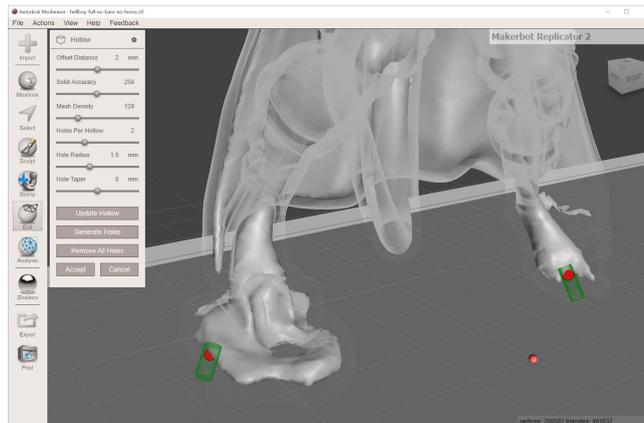
The model will be analyzed and finally, a semi-transparent preview will be displayed. Default settings are usually perfectly fine, however, if you want to modify the shape of the hollowed part, you can use sliders to modify it.



A hollowed object in Meshmixer

A good idea is to place **two draining holes** somewhere in the object. Depending on how you print your model, the resin can actually get trapped inside it, increasing the weight of the object. Also, in case you accidentally **break the model**, the uncured **resin will leak out**.

In the Hollow menu, click the Generate Holes button. This will place **two holes** at the bottom of the object. You can use the left mouse button to drag them to a different place. Try to position the holes somewhere, where they won't be too visible.

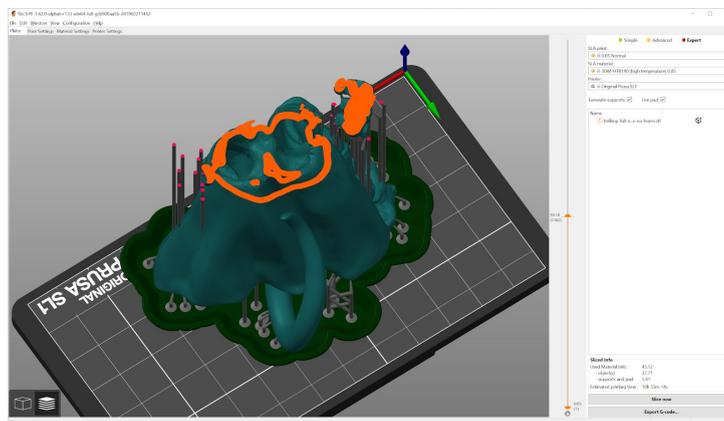


Positioning drain holes - place them somewhere, where they won't affect the look of the model - e.g. on the underside of the feet

After you are done hollowing the model and creating holes, click the **Accept** button

Check the model for potentially broken geometry and if everything looks good, select **File -> Export** and save the model as a 3MF or STL file.

Then import the file into PrusaSlicer and slice it as usual. After slicing, **inspect the model** layer-by-layer in the Preview window. If you notice any defects, go through the Meshmixer procedure again and fix potential issues.



Hollowed object displayed in PrusaSlicer's preview window

6.15 Hollowing objects in Windows 3D Builder

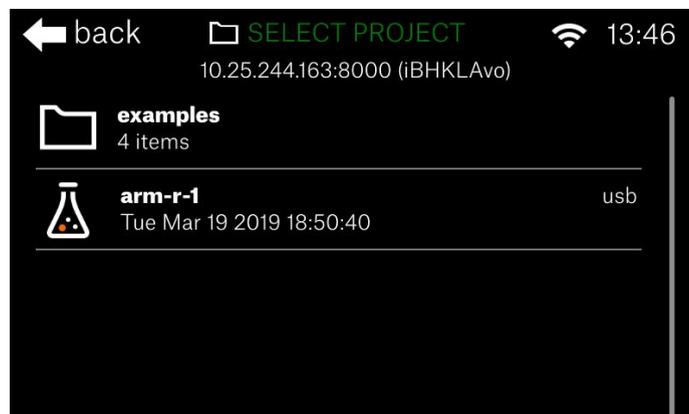
If you have Windows 10, you can use Microsoft's simple 3D editor called 3D Builder to hollow your model. Simply import the model into the application and use the Hollow tool to create a cavity inside the model.

You can then use the cylinder tool in combination with the Cut tool to create holes in the model.

7 Connecting PrusaSlicer to the SL1 over the network

PrusaSlicer can be **connected to the SL1 3D printer directly**, either via ethernet or Wi-fi connection, enabling you to **send sliced models** into the printer's internal storage over the network.

Open PrusaSlicer and make sure the **Original Prusa SL1** printer is selected in the **Printer** menu on the right. Go to the **Printer Settings** tab, navigate to **General** category and look for **Print host upload** section. Click the **Browse** button to show a list of detected devices on your network. Select your printer and click **OK** to confirm the selection. If the printer is not detected, try inserting its IP address manually in the text field called Hostname, IP or URL. The printer must be turned on in order to be recognized.



The API key is shown in the Print menu in the brackets next to the IP address (iBHKLAvo). Keep in mind that every printer has a different API key.

Enter the API key into the text field in Slicer (Printer Settings -> General -> Print Host upload).



Connecting to the SL1 with PrusaSlicer

Press the **Test** button to check the connection. If everything is configured correctly, the test will be successful.

Go back to PrusaSlicer's main window with the 3D view. There should be a **new button labeled "Send to printer"**. Once the model is sliced, you can press this button to export the SL1 file via the network to your printer. You can check the upload progress in *Window -> Print Host Upload Queue*. Go to your printer, open the Print menu and the new file will be at the top of the list.

⚠ Warning! We strongly suggest not to leave the printer unattended. Always prepare the printer for the next print according to the instructions in this handbook. The purpose of this function is to eliminate the need for a USB stick and improve the convenience of use. It is not possible to start the print remotely.

8 Updating the printer's firmware

We recommend keeping the printer **updated at all times**. Updated firmware packages bring various benefits, such as new features, bug fixes and various improvements. There are two ways to update the printer's firmware:

OTA firmware update

OTA (Over The Air) firmware updates are delivered via **ethernet or Wi-fi network**. Connect the printer to the **internet** as described in the chapter [4.4 Network connection](#). Go to *Settings - Advanced Settings - Firmware Update* to initiate the update. The printer will check for a new firmware and when an update is detected, you will receive an on-screen prompt asking you to confirm the installation.

USB firmware update

If your printer is not connected to the internet, you can **download the firmware file** from the official website www.prusa3d.com/drivers. Download the ZIP file to your computer, unpack it and place the update file on a **USB drive**. Then insert the USB drive in the printer's USB slot. Then go to *Settings - Advanced Settings - Update Firmware* and select the firmware file from the list.

WARNING! Never turn off or reset the printer during the firmware flashing process!

9 Material Guide

Liquid resins are **photosensitive materials** used for printing on SLA 3D printers. Resins are also known as photopolymers and they are **cured (solidified) by UV light** - this also includes sunlight. Compared to filaments for FFF/FDM printers, the major difference is that there are not various materials such as PLA, PETG or ASA. There's just THE resin, a base material that can be further enhanced with **added colors or additives**. Typical effects of additives would be **increased hardness or toughness** of the printed object.

i *The Original Prusa SL1 3D printer is compatible with 405nm resins.*

Resins usually consist of three basic components:

- **The core** of the resin (monomers and oligomers)
- **Photoinitiators** - molecules reacting to UV light, which initiate the solidifying process
- **Additives** - admixtures that change the color and properties of the resin



Prusa Orange resin

9.1 Important information about resins

There's a couple of things to keep in mind when working with resins:

- **Wear gloves** to avoid direct contact of the resin with your skin
- A simple **respirator** is recommended
- Keep out of the reach of children and pets
- **Shake the bottle** well before every use for around 10 seconds (make sure the lid is securely tightened before doing so)

- **Wash with water and soap** in case the resin gets in contact with skin/eyes
- If you experience itching or any kind of discomfort after skin contact, **seek out medical assistance**
- **Keep away from light**, store in opaque bottles
- **Keep at room temperature** (18°-32°C), lower temperature changes the viscosity of the resin
- Do not store in dusty areas
- Resin produces fumes - keep the room **ventilated** during printing
- **Some people may be allergic to resins** - in case you experience any kind of discomfort (itching, rash), stop using the machine and contact a doctor

 **Do not throw resin bottles (empty or full) into regular waste.**

Liquid resins should be treated as dangerous chemicals and you should follow your country's laws about disposing of dangerous chemicals. There are usually special containers (e.g. oily waste cans) available for public located near gas stations etc.

9.2 Which resins to buy?

Once you start looking for resins to purchase, always check the **wavelength of the resin** - it determines at which wavelength the solidifying process starts. The Original Prusa SL1 printer is **calibrated for 405nm resins** and has no brand restrictions. Since it's an open-source machine, you can choose any brand of resin you like, as long as it is **compatible with the 405nm wavelength**. However, always use **recommended exposure times** for the resin of your choice - extremely incompatible times may even lead to your printer getting damaged (e.g. setting 30 second curing time for a resin that solidifies in 6 seconds).

Resins should be stored in their **original containers and properly sealed**, ideally in a dark room with a stable temperature. If the original container is damaged, use a **non-transparent plastic bottle** and the bundled funnel that came with the printer to pour the resin in it. Never store resin in bottles from drinks - if you have to, label it clearly. **Do not store resins in cold places as it may lead to gelation of the resin.**

9.3 Types of resins

Material type	Properties
Standard resin	<ul style="list-style-type: none"> + Smooth surface, lots of details - Fragile - Not suitable for mechanical parts
Clear resin	<ul style="list-style-type: none"> + Semi-transparent - Can be turned nearly fully

	transparent through post-processing
Casting resin	<ul style="list-style-type: none"> + Lots of details + Great for preparation of casting forms + Little to no remnants after burning the resin
Hard and resistant resins	<ul style="list-style-type: none"> + Similar to ABS or PP materials + Partially flexible + Suitable for mechanical parts - Low resistance to high temperatures
Heat-resistant resin	<ul style="list-style-type: none"> + Highly temperature-resistant + Used for injection forms - Expensive
Bio-compatible resins	<ul style="list-style-type: none"> + Non-toxic + Suitable for dental implants manufacturing + Abrasion-resistant - Expensive
Flexible resin	<ul style="list-style-type: none"> + Similar to rubber (70A hardness) - Lower resolution of printed parts

10 Software/driver package

We provide owners of Original Prusa printers with a bundle of **useful programs and tools** called **Drivers & Apps package**, which can be downloaded from www.prusa3d.com/drivers

This package contains (among other things) the latest stable version of PrusaSlicer and also a bundle of sample models, which can be used to test the print quality of your 3D printer.

You can also obtain alpha and beta builds of PrusaSlicer from our official Github, located at <http://www.github.com/prusa3d>

11 Controlling the printer via a web browser

You can check the **current status** of the Original Prusa SL1 3D printer during printing through a **web browser** - even on mobile phones.

1. Connect the printer to your network as described in chapter [4.4 Network connection](#)
2. Check the printer's **IP address and API key**. See chapter [7 Connecting Slicer to SL1 over a network](#) for information on obtaining the IP address and API key.

3. Open a **web browser** of your choice, enter the printer's IP address
4. Enter "**maker**" as the username and the **API key** as the password

12 Optional Accessories

There's a couple of things you might want to consider buying to make your 3D printing experience better, safer and more convenient.

12.1 Original Prusa Curing and Washing Machine - CW1

Objects printed from liquid resin are usually a bit soft and sticky. To clean them and reach optimal surface properties, the **objects need to be washed, dried and cured**. This process involves **washing the object in isopropyl alcohol, drying using hot air and curing using UV light** - see chapter [Drying and Curing Printed Objects](#).

To make this process **much faster, simpler and more convenient**, we have designed a 3-in-1 device called [Curing and Washing Machine](#) (CW1 for short). This device is sold as an **optional accessory** and it reduces an otherwise rather complicated and potentially messy process **into a few minutes**. To learn more about this device, visit our e-shop or check the Curing and Washing Machine chapter.



Curing and Washing Machine (CW1)

12.2 FEP Films - Original and Aftermarket

The Original Prusa SL1 comes with a **set of FEP films**, which act as a **transparent bottom of the resin tank**. Since this part is a **consumable** (it will degrade over time by continuous use), you will have to replace it from time to time. Compared to certain other 3D printer brands, you **don't have to replace the whole tank**, you can remove just the film.

You can order a new FEP film from our e-shop. The process of replacing the FEP film is described in chapter [Replacing the FEP film](#) at the end of this handbook.

Several things to keep in mind regarding FEP films:

- **The FEP film we're using is not proprietary** - you are free to buy FEP films from online or retail stores.
- **The FEP film is a consumable** - it means the quality of the surface will slowly degrade through continuous use. However, we wanted the part to be easy to replace and inexpensive. You can buy a replacement FEP film on our e-shop.
- **You should be extremely careful** when handling the tank. Keep it away from sharp objects that could potentially damage the bottom of the tank. FEP films that were pierced, scratched or damaged by improper use are not subject to warranty terms.

You can, of course, **use a FEP film from your own source**. However, keep in mind that the **thickness of the film affects the quality of the print** and that the FEP films from our shop have the correct size and holes for screws. Thicker films disperse light more, so the printed object may lose some details. Very thin FEP films are, however, prone to damage.

⚠ WARNING: Keep sharp and/or metal objects away from the FEP film! Do not use isopropyl alcohol to clean the FEP film!

Never use the bundled metal spatula to remove stuck objects from the FEP film. Do not use fingernails, screwdrivers or other sharp objects to remove stuck objects from the FEP film! Do not clean the FEP film with isopropyl alcohol. Use a combination of warm water and a dishwashing product.



Resin tank with the transparent FEP film bottom

12.3 Additional accessories

Optional accessories you should consider buying:

- **Paper towels** - for cleaning up the printer, drying printed models...
- **Isopropyl alcohol** - for cleaning the prints
- **Plastic tablecloth** - to contain resin spillage and to make cleaning up easier
- **Pipette** - for removing larger resin drops/stains
- **Respirator / protective glasses**

13 Regular Printer Maintenance

Although the Original Prusa SL1 has a number of smart sensors and has been carefully designed to be easy to use, it's still a machine and **requires regular maintenance and care**. There are several key areas that require attention if you want to keep your printer in good shape for a long time.

13.1 Keeping the printer clean

The key to a trouble-free experience is first and foremost **keeping the printer clean**. Resins are liquids and they are easily spilled. Every time you use the printer, make sure you are **not leaving drops or stains of liquid** anywhere. Liquid resin will solidify **even in regular daylight**, making it much more difficult to remove compared to its liquid form.

When removing the printing platform / printed objects, use the black plastic tray to shield the printer from drops of resin. Always keep a bundle of **soft paper towels** near the printer, so you can wipe any spilled resin/resin drops **immediately**. To clean drops of resin more effectively, use a piece of cloth / paper towel dipped in isopropyl alcohol. For more delicate surfaces (such as the display), use a mix of water and dishwashing product.



To ensure trouble-free printing, keep the printer clear

A good practice is to **place a plastic table-cloth** under the printer for easier cleaning. However, be careful - the table-cloth **must not block the fan** on the underside of the printer.

⚠ Do not use acetone or ethanol to clean the lid!

The lid is made of acrylic glass and it has a special surface finish to block UV light from reaching the printing chamber. You can clean it with standard products for cleaning glass and/or isopropyl alcohol. Don't use acetone or technical alcohol - it would damage the acrylic glass.

13.2 FEP film / Tank maintenance

Empty the resin tank **after each print**. The tank has one corner which has a different shape than the other three. Use it in combination with a funnel to pour the resin out of the tank into a bottle. Don't mix used resin with unused liquid. Instead, use a new bottle.

Inspect the resin tank bottom before and after each print. Look for any scratches, dents or holes. If the FEP film is damaged, replace it immediately, otherwise, you're risking a resin spillage.

If the FEP film has a grey-ish tint, you can restore it to its original look by washing it with warm water with a bit of dishwashing soap (or even regular soap) before you start printing again. Don't print with a dirty or damaged FEP film, as it will negatively impact the print quality. It may even lead to damage to your printer. **NEVER use isopropyl alcohol to clean the FEP film!**



Cleaning the FEP film - before and after

Check the FEP film / Tank bottom replacement chapter to learn how to replace the FEP film.

13.3 Printing platform maintenance

The platform should be **cleaned thoroughly after every print** - usually, there will be some leftover resin on the platform. Once a print finishes, **place the black tray** on the printer and use paper towels or a spatula to wipe the top of the platform clean, then proceed to remove the platform and the printed object.



Once the object is removed, **use isopropyl alcohol and paper towels** to clean the printing platform thoroughly. Leaving liquid resin on the platform for extended periods of time can lead to solidifying of the material.

After removing an object, use a metal spatula to scrape any remnants of the previous print and use paper towels to wipe the underside of the printing platform clean - make sure that the printing platform is **nice and shiny** before you start the next print. Any leftovers on the printing platform (leftover supports, pads) could potentially damage the bottom of the resin tank or even the display. Surface scratches on the print platform **won't affect the quality** of the print.

After you clean the printing platform with isopropyl alcohol, dry it thoroughly!

13.4 Exposure screen maintenance

Use a **damp cloth or LCD cleaning spray** to keep the glass / LCD display clean. Don't use any rough materials (sanding paper) or low-quality paper towels, otherwise, you might scratch the surface. Make sure that the **print display is clean** before you start a print and that there are no smudges, stains, scratches or cracks.

⚠ Do not use sharp objects, such as metal spatulas, knives, flat screwdrivers and similar tools to remove dirt or resin from the protective glass. Instead, use IPA or dishwashing products in combination with soft paper towels to avoid scratching the display.

13.5 Acrylic lid

The lid has **three major functions**: it protects the printed model from potential damage, helps to contain resin fumes inside the box and **blocks a significant amount of UV light** so it doesn't reach the printing chamber. Especially the last part is important - if you leave the resin in the tank and keep the lid open, it can **start to solidify** quickly depending on the environment.

⚠ The lid won't stop 100% of UV light - do not place the 3D printer next to a window, otherwise the sunlight might cure the resin in the tank.

This also means that you **shouldn't disable the cover check**. It may be tempting to watch the print process, but for the sake of better results and safer operation, **keep the lid closed during the print**. You can, of course, open it for a quick check and then close it again.

Don't run the printer with the lower case panels removed or with any parts missing. The case helps to contain the UV light.

You can wash the lid with isopropyl alcohol, **do not wash it with ethanol! Do not use rough materials to clean the lid (e.g. sanding paper, rough sponges)**.

13.6 UV light and potential health risks

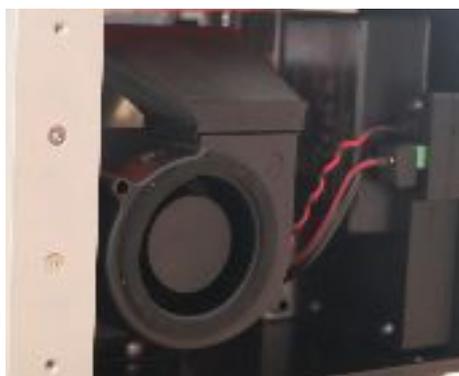
Eyes are very sensitive to UV light and UV radiation, because the cornea absorbs high amounts of UV light. Extended exposure to strong UV light may lead to corneal damage, macular degeneration or cataracts - these conditions can ultimately lead to blindness.

This is why the Original Prusa SL1 is designed to contain the UV light inside, which means that when used according to the instructions in the manual, there are no health risks related to UV radiation during regular use.

⚠ DO NOT LOOK DIRECTLY INTO THE UV LIGHT AND DO NOT PRINT WITH THE BODY COVER REMOVED! If you want to check the UV led panel, look at it through your phone camera app or take a photo, so you won't look directly into the UV light.

13.7 Fan cleaning / Removing dust

It's a good practice to **clean the fans from time to time**, especially if you have pets at home - the recommended time period will vary depending on the environment, in which the SL1 is used. **In general, it's good to inspect the printer every 6-8 weeks**. There are three fans placed inside the printer. You can get to the side fan and bottom fan by removing the main body cover.



First, **turn off the printer** and disconnect it from the power source. Locate eight screws on the sides of the printer (four on each side) and remove them with an Allen key. Then carefully slide the body cover out. Be careful not to

damage any cables which are **attached to the power button**. This will give you better access to two fans. Use a can of compressed air to clean the fans carefully.

The third fan is located on the backside of the printer. Locate two screws holding a small door closed and remove them. Open the door and use a can of compressed air to clean the fan.

Never blow compressed air on running fans!

Dust can also accumulate on the underside of the display and on the reflector. You can clean these parts **carefully** with compressed air. First, unscrew the eight screws holding the display in place, then **slowly and carefully** remove the display - be careful not to damage the display's wires. Then, use compressed air from the distance of about 30 cm to clean out the dust particles gently. **Do not use sharp or rough objects!**

13.8 Carbon filter replacement

Check the **carbon filter** from time to time to ensure it is in good shape. The carbon filter is located **on the back side** of the printer under a small door. Use an Allen key to remove two screws holding the door shut. Open the door and remove the filter. Check if it is in good shape, and consider replacing it if it isn't.

13.9 Tower maintenance

The tower (Z-axis consisting of the trapezoidal rod and the cantilever) requires **little to no regular maintenance**. In case the printer starts making unwanted noises when lifting or raising the platform, consider **applying a lubricant on the trapezoidal rod** - any good super lube will do the trick.

13.10 Tilt maintenance

The key to a long-lasting and smoothly running tilt bed is simple: **keep it as clean as possible**. Use the included plastic tray every time you manipulate with the resin or the printing platform with finished print. The tray will stop the resin from spilling over the tilt bed.

14 Hardware troubleshooting

Please note that detailed hardware troubleshooting articles will be available only online through manual.prusa3d.com or help.prusa3d.com.

14.1 FEP film / Tank bottom replacement

Please note: Every time you replace the FEP film, tighten the screws properly, but don't use excessive force. Overtightening the screws may lead to issues with their removal during next FEP film replacement.



An extreme case of FEP film damage

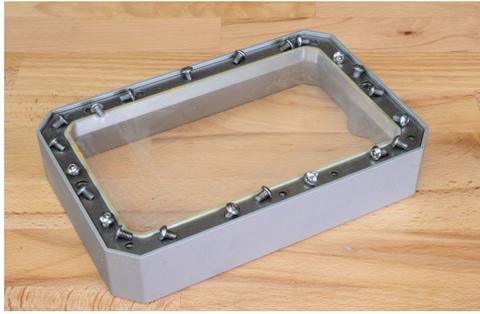
The FEP film is a consumable, which means it will degrade after a certain time. You can order replaceable FEP films from our e-shop.

How to replace the FEP film:

Before you start, clean the tank thoroughly and wipe it clean



Use a 2.5mm allen key to loosen the screws. DON'T use a cordless screwdriver



Remove all screws and gently press your fingers against the FEP film to release it



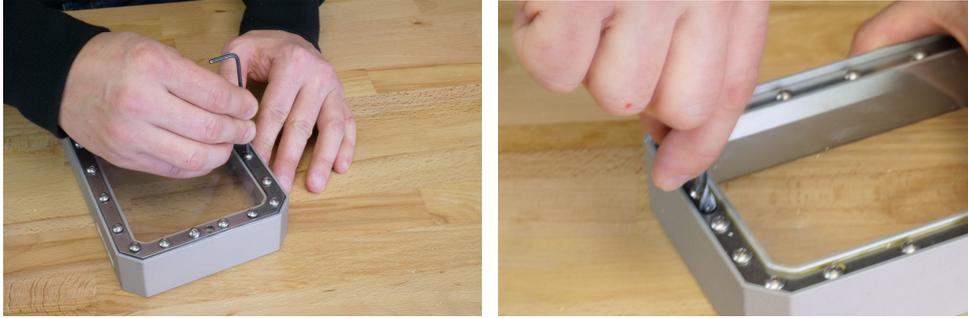
After you remove the FEP film, take a spatula and scrape all bits of leftover resin from the edges. The edges must be perfectly clean, otherwise, the tank will leak



Clean the tank and frame using isopropyl alcohol



Align the frame with the new FEP film, so the holes match. Start adding screws one by one. Place the assembly onto the tank body and align the screws with the holes. Don't create holes for resin tank screws yet!



Start tightening the screws in a criss-cross pattern to ensure that the film will be stretched evenly. Make sure that all screws are tightly screwed. Once everything is in place, take an object with a sharp tip (e.g. a 6mm drill bit) and puncture two holes for resin tank screws. Remember to re-calibrate the printer using the tank with the new FEP film.

⚠ If you replace the FEP film, re-calibrate the printer!

14.2 Spilled resin

Always use the **protective tray** and exercise caution when handling liquid resins. If you manage to spill resin inside the printer (e.g. through the tilt bed), **turn off the printer immediately** and disconnect the power cord.

Locate eight screws on the body cover (four on each side of the printer), remove them and take off the metal body cover. **Inspect the damage:** in case there's just a few drops of resin, it's very likely that they've been caught by the tilt bed assembly. Wipe them clean and make sure there are no leftover drops anywhere. Put the cover back and check whether everything works fine.

In case there's a **major spillage** (e.g. on the mainboard), keep the printer turned off and **contact our tech support**.

15 Print quality troubleshooting

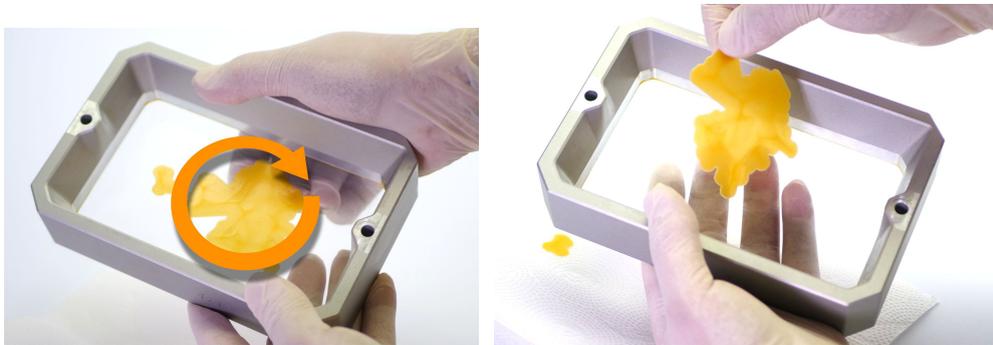
When you print on SLA printers, you may encounter several print quality issues. The following section will help you recognize them and solve them.

15.1 Removing an object stuck to the bottom of the tank

In case the first layer did not stick to the printing platform, it remained **stuck at the bottom of the tank**. No matter how big or how small the stuck layer is, there's a **quick safe way** that will help you **remove it from the FEP film** without damaging it:

1. Unscrew the resin tank screws and remove the tank
2. Take an empty bottle and insert the bundled funnel/filter in it
3. One of the tank corners has a different shape, use it to pour the resin from the tank into the bottle. Always pour used resin into a new bottle; don't mix it with unused liquid - especially if the print failed.
4. Wipe the tank with a soft paper cloth. The layer that got stuck should be now fully visible
5. Keep the tank in a horizontal position and use your fingertips (as shown in the picture below) to detach the solidified layer from the bottom. Use a reasonable amount of force and don't use your fingernails. If the layer is larger, start from the edges and use a circular motion to detach it evenly. You will usually hear the sound of the layer detaching from the FEP film - this indicates that it's working.

Once the stuck object is removed, clean the printer and the tank and try printing a sample object. If the print fails again, re-run the calibration process. The most likely cause of this issue is an incorrect Z-offset of the printing platform.



15.2 First layer doesn't stick (Non-Adherence)

If you notice during the printing process that **no object appeared** on the printing platform, it usually means that the first layer **didn't adhere properly**, or the object detached mid-print. To rule out slicing-related issues, try printing a sample (pre-sliced) object first.

There are a **number of reasons** why the first layer does not stick to the printing platform:

a) The printing platform is not secured properly

The printing platform is not properly secured. Make sure that you didn't forget to tighten the black knob at the top of the tower. A freely moving printing platform cannot reach an optimal position and the first layer won't stick.

Solution: Tighten the black knob at the top of the platform.

b) The bottom of the tank is cloudy/dirty

Before you pour the resin into the tank, make sure that the **bottom of the tank** (the FEP film) **is completely clear**. A cloudy, dirty or damaged FEP film will have a **negative impact** on the printing quality and it may even **block UV light** enough to stop the resin from solidifying properly.

Solution: Clean the FEP film using warm water and a bit of dishwashing product, and wipe it with soft paper towels. Low quality / rough paper towels could scratch the surface.



Dirty FEP film (left) vs clean FEP film (right)

c) Incorrect slicing settings

There's a couple of things that may go wrong during slicing - **incorrect exposure times, an incorrect object orientation or a lack of supports**.

Make sure that you are using the **recommended exposure times** for the resin of your choice. We have tested over a hundred of various resins and we're constantly testing new ones, so we can add tweaked resin profiles into PrusaSlicer.

Solution: **Reslice your model using default settings** and try printing the object again, or try **printing one of the sample objects** to rule out the possibility that slicing settings are the cause. The exposure times for initial layers should be much longer - e.g. 45 seconds to ensure a strong connection to the printing platform.

d) Poor object orientation / insufficient supports

Incorrect object orientation can be another source of issues with adhesion. After a layer is solidified, the printer has to detach it from the bottom of the tank - if there's a single support holding a large part of the object, it won't withstand the peel force, causing it to detach.

Solution: Please refer to the chapter [Slicing - Best practices](#) to learn how to position an object correctly.

e) Resin has been mixed with IPA

The resin has been mixed with IPA (e.g. while cleaning the printing platform) and its properties have changed, causing it to not solidify.

Solution: Clean the resin tank properly, use a new batch of resin and try again.

f) The printer is not calibrated correctly

It's possible that something changed since the last calibration of the printer - you have replaced some parts or moved the printer to a new location.

Solution: A re-calibration is required. Please refer to the [4.5 Calibration](#) chapter for step-by-step instructions.

g) A faulty print display or UV LED panel

If the first layer didn't adhere to the printing platform and it's not attached to the bottom of the tank either, it usually means that the first layer wasn't created at all. A possible cause is a faulty print display or a faulty or disconnected UV LED panel.

Solution: Please refer to the online troubleshooting section at <https://help.prusa3d.com> to learn more about diagnosing the function of these two parts.

15.3 Printed object has missing parts

In some cases, a print can finish even if some parts of the object weren't fully printed, but a missing part of an object will more often cause the whole print job to fail. Try printing a sample object to see whether it is also affected by this issue.



The warrior's right arm is missing completely and a part of the shield is damaged

There are **two main reasons** (and a third, less common one) why printed objects have missing parts and they both usually come down to incorrect slicing settings:

a) Lack of supports

Please check the [6.5 Slicing objects for the SL1](#) chapter to learn how to correctly position an object and generate supports for it. Keep in mind that parts of the object cannot start mid-air and that the printer has a threshold for overhangs.

Solution: Improve the supports in PrusaSlicer, re-slice the object and try printing it again. If the problem persists, try printing a sample object and see if the problem is still present. If it is, there might be a problem with the printing display or UV LED panel. See online troubleshooting at <https://help.prusa3d.com> to learn more.

b) Incorrect exposure times

If you set incorrect exposure times (especially too low times), the resin will not have enough time to solidify, which may result in missing details. Always make sure you're using recommended exposure times for your prints.

Solution: Make sure you are using exposure times recommended for the resin of your choice.

c) Faulty hardware

Another possibility (a less common one) is that your print display or UV LED panel is faulty, so either the mask is incorrectly displayed or only a part of the object is cured by UV light. Please refer to the Hardware troubleshooting section at <https://help.prusa3d.com> to learn how to diagnose the print display and UV LED panel.

15.4 Printed object is missing small details

The Original Prusa SL1 3D printer can produce objects with an incredible amount of details - tiny wrinkles on clothes of a figure, structure of a skin, thin railings on buildings... however, in case the printed object is missing incredibly tiny details (not entire parts of the model), it can be due to incorrect exposure times.

a) Insufficient exposure

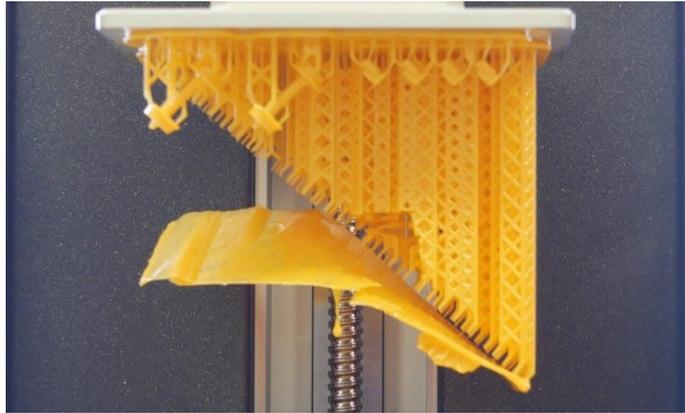
Too short exposure times will result in details being “melted” together. Increase the exposure times to recommended values to fix this issue.

b) Overexposed layers

Long exposure times will result in a “staircase” effect - the layers won’t blend into each other perfectly and some details may be lost due to this effect. Use recommended exposure time values to fix this issue.

15.5 Printed object detached from supports

If the printed object detaches from supports mid-print, it usually falls into the resin tank, while the supports are still attached to the printing platform. It’s usually caused by the combination of the printed object’s weight and insufficient supports. Another possibility is that you are printing a hollow object without drainage holes - this creates suction on the bottom of the tank and the object cannot detach properly.



Consider using Meshmixer to hollow out the object. Too heavy (solid) objects may be too much for default supports and as a result, they often detach from supports once they reach a certain weight threshold.

Solution: Increase the number of supports or the thickness of supports. Check the exposure times and change them to recommended values.

15.6 Broken supports

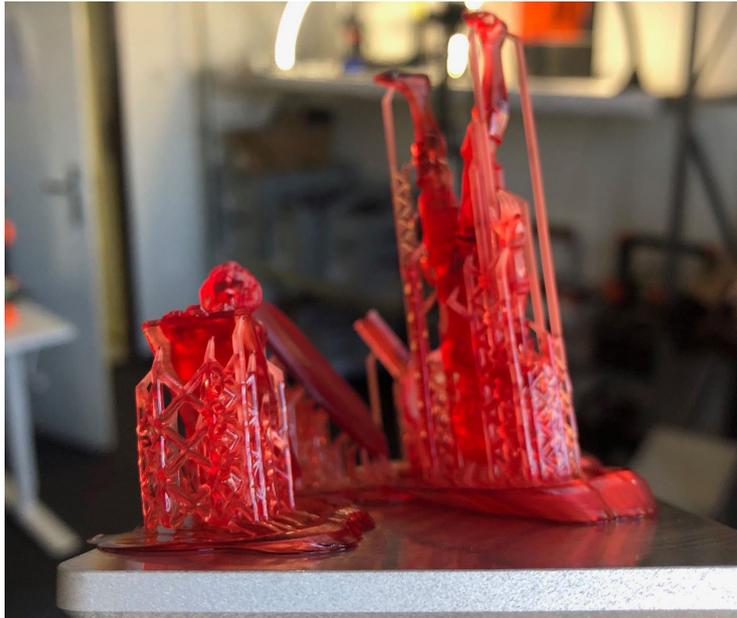
Broken supports (or missing parts of supports) are usually caused by incorrect settings in PrusaSlicer - the supports are either too thin or the exposure time is set to too low value. In some cases, this can be also caused by a faulty UV LED panel or dirty FEP film.



Solution: Try printing one of the sample objects to see whether the issue remains. If the sample object was printed without trouble, reset your PrusaSlicer settings. If the sample object has the same sort of trouble, inspect the FEP film in the resin tank - it may be dirty. In very rare scenarios, a faulty UV LED panel can be the cause.

15.7 Printed object is skewed / deformed

A skewed/deformed print is usually caused by loose screws - either the large black knob at the top of the cantilever, or the resin tank screws are not properly tightened.



Solution: Make sure the printer is properly calibrated, tighten the black knob at the top of the tower and tighten both resin tank screws evenly and with the same amount of force.

15.8 Print doesn't appear / resin does not solidify

If the print does not appear on the printing platform, nor at the bottom of the resin tank, there are a couple of possible causes: insufficient exposure times, faulty UV panel, faulty LCD screen, or there's too much isopropyl alcohol in the resin.

Solution: First try a new batch of resin. If you kept re-using one batch several times, it's possible the resin lost its original properties and won't solidify anymore. Check the UV panel and LCD screen, whether they work as intended - run the printer's self-test and calibration to make sure the hardware is not faulty.

15.9 Printed object has very visible layers

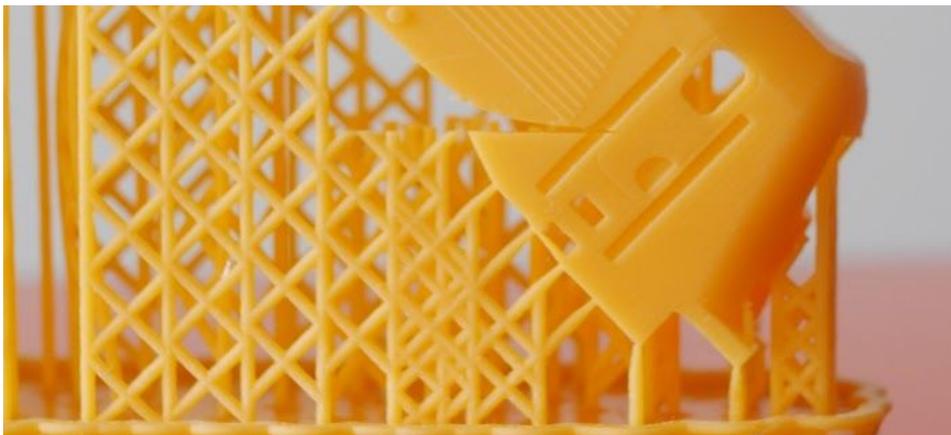
Even though SLA printers can achieve much better resolution than standard FFF/FDM 3D printers, layers on objects can be still visible if you look very closely. If you feel like the layers on your prints are too visible, there's a couple of things you can try.



Solution: Print a sample object with a good/tested resin (like the sample bottle of Prusa Orange) and see if the issue is still present. If it's not, then the cause is usually related to slicing settings. Try rotating the object to eliminate gradually rising / declining surfaces. Extreme exposure times can also produce visible layers. Mixed / low quality resins can also affect the print quality - use a fresh, high-quality resin and shake the bottle for at least 20 seconds before you pour the resin in the tank.

15.10 Delamination / Separated layers

If you encounter delamination, separated layers or missing layers, the cause is usually in a broken SL1 / STL file - it's a different case than a lack of supports. Another cause can be a faulty UV LED panel or dirt in the resin tank.



Solution: Try printing a sample object and see if the issue persists. If it does, clean the resin tank properly and try again. Continuous issues with separated / missing layers can be caused by a faulty UV LED panel - check whether it works correctly. Otherwise, the cause is most likely a broken object file. Either try re-slicing the model or download a new one.

15.11 Warped layers

Most resins have a tendency to expand by several micrometers after they are cured. These expansions and resulting forces can lead to layer warping and layer separation. However, these errors are usually limited to large solid layers.



Solution: This printing error is caused by the physical properties of resins. Consider hollowing problematic objects, if you encounter layer warping.

15.12 Large holes in printed objects

Large holes in printed objects are almost exclusively caused by dirt on the FEP foil. Tiny pieces of debris on the bottom of the tank will block the light from reaching the resin correctly. Dirt particles have a tendency to grow during the print, which increases the size of the hole.



Solution: Make sure the FEP foil on the bottom of the tank is perfectly clean. Carefully remove any dirt / solidified bits. If the FEP foil is cloudy, use hot water and dishwashing soap to clean it and restore it to the original look.

16 FAQ - Frequently Asked Questions

Q: UV LEDs don't light up

A: If you want to check whether the UV LED panel works correctly, NEVER LOOK DIRECTLY IN THE UV LIGHT! The safe way to check whether the panel works or not is to use your smartphone's camera - either look at the LED panel through your phone's screen or take a picture and inspect the photo.

Q: Printer cannot be calibrated - the prints don't stick to the printing platform

A: Try printing a sample object. If the print remains stuck to the bottom of the tank, re-run the calibration process. If it doesn't help, try replacing the FEP film. The FEP film must not be washed in IPA.

Q: The printer displays a message related to overheating

A: Make sure the printer is located in a room with ambient temperature between 18-32°C. Check if all fans are spinning and there are no objects that would block the air inlets/outlets.

Q: USB is not recognized or 'File is corrupted' error

A: Try to export / copy the SL1 file again. In case the error is still there, try a different USB drive.

Q: Wifi browser does not list my network

A: Please, make sure that your wireless network name does not contain non-standard characters. Try renaming your wifi network and see if the problem is solved.

Q: I can't connect to unprotected (public) wireless networks

A: Due to security reasons, the printer can be connected only to WPA2-protected networks.

Q: Wifi / LAN does not connect

A: There can be various reasons why the printer won't connect to your network, but since every network is different, it's difficult to pinpoint the exact cause.

- Can you see the list of other wifi networks in the browser?
- Are you sure you are entering the password correctly?
- Can other devices connect to your network?
- Do you have a DHCP service running on your network, so IP addresses are assigned automatically?

Q: Printer doesn't turn on

A: Is the power cord attached firmly? Is the switch on the back side of the printer switched to the "On" position? Try disconnecting the power cord and then reconnecting it after a few seconds.

Q: LCD touch screen does not respond to touch

A: The display uses a capacitive touch screen layer and it might not recognize a touch with wet fingers. Make sure your hands are dry and the display is clean.

Q: Resin vapours are very noticeable

A: Some resins have a more prominent smell than others. Always run the printer in a well-ventilated room. Consider switching to a different brand of resins that promises less odours when printing. However, it's almost impossible to eliminate resin vapours completely.

Q: The lid of the printer has been left open and the resin started to solidify in the tank

A: In most cases, it's possible to "save" the tank, sometimes even the FEP foil depending on how much has the resin solidified. Remove the tank from the printer, find a large volume container (e.g. a bucket) and start removing the resin with a metal spatula carefully. If you damage the FEP film, you can replace it according to the instructions in this handbook.

Print and share!

Do not forget to tag your prints with #prusas1 while sharing so we can find, pin and showcase them with our

Pinterest

<http://www.prusa3d.com/original-prusa-i3-prints/>

Happy Printing :)