CreatorBOX

Unpacking the future.

ARENA2036 Rapid Prototyping Service Instruction Guide

Contents

Creating an Account 3
How do I use ARPS? 4
What Printer setup and Materials does ARPS use? 5
How do I use PrusaSlicer? 6
Set up the correct printer configuration 6
Slicer settings: 8
FAQ 9
Do I have to pay to use ARPS? 9
Where can I get help? 9
What should I do if I observe a malfunction or a defect? 9
I would like to print with my own filament. What do I have to do for this? 9
Where can I give feedback and report bugs? 9
When I click upload, I get an Error message. 9

Creating an Account

1. Be in the **ARENA2036** WIFI Network locally or using a VPN
    a. More VPN Info: https://www.tik.uni-stuttgart.de/support/anleitungen/it-basissupport-arena-2036/
2. Go to the ARPS2 Web-Portal: https://arps2.cb.arena2036.de/
3. On the top right, click “Login”
4. On the bottom of the “Sign in to your account” prompt, click “Register”
5. Fill out your information and click “Register”
6. Notify a CreatorBOX member and they will verify your account activation

Congrats! You now have an ARPS2 Account!

How do I use ARPS?

1. **Export** your desired part from your CAD Program as an ".STL" File
2. **Create your G-code:** For this, you typically have to import your .STL file into a slicing program
    a. We recommend using the free PrusaSlicer as we use their printers, so the correct settings are mostly already selected
        i. https://www.prusa3d.com/page/prusaslicer_424/
    b. Here you have to choose the following settings:
        i. Material: PLA or PETG
        ii. Printer: Prusa i3 MK3S (0.4mm nozzle size)
    c. For a more detailed description of the PrusaSlicer scroll down to the section labeled: "How do I use Prusa Slicer?"
3. **Upload your G-code:**
    a. Login to the ARPS2 Website
    b. Click on "New print job" on the left panel, you might have to clock on the three lines to show the menu panel.
    c. Click on "Choose a G-code file to print..." and upload the G-code you generated in step 2.
    d. Select the correct material you used in the slicing software.
    e. Choose the amount of copies of the print you would like printed.
        i. **NOTE**: It is much more time efficient if you copy and paste the amount of prints you would like in the slicer, so your one G-code already has the amount of copies you need
4. **Printing Process:** We will now automatically schedule your print for the next free printer. If you want you can view a live stream from the printers.
5. **Get your print:** Once your print is finished we will send a notification to the mail address you used for our account. You can now come to ARPS2 and pick up your part.
    a. Please clean the print off the plate and put the plate back into ARPS where you see this sign:

What Printer setup and Materials does ARPS use?

- Printer: **Prusa i3 MK3S (0.4mm nozzle size)**
- Material: **PLA** or **PETG**
- Temperature: Bed Limited to **80ºC**, Nozzle to **240ºC**

All printers are of the type “Prusa i3 MK3S” and equipped with a nozzle with 0.4mm diameter.

During the uploading process, the currently equipped filaments, usually PLA or PETG, can be selected as the printing material.

If you prefer a different nozzle diameter or want to print other materials, please contact us or use the contact button in the top right corner.

How do I use PrusaSlicer?

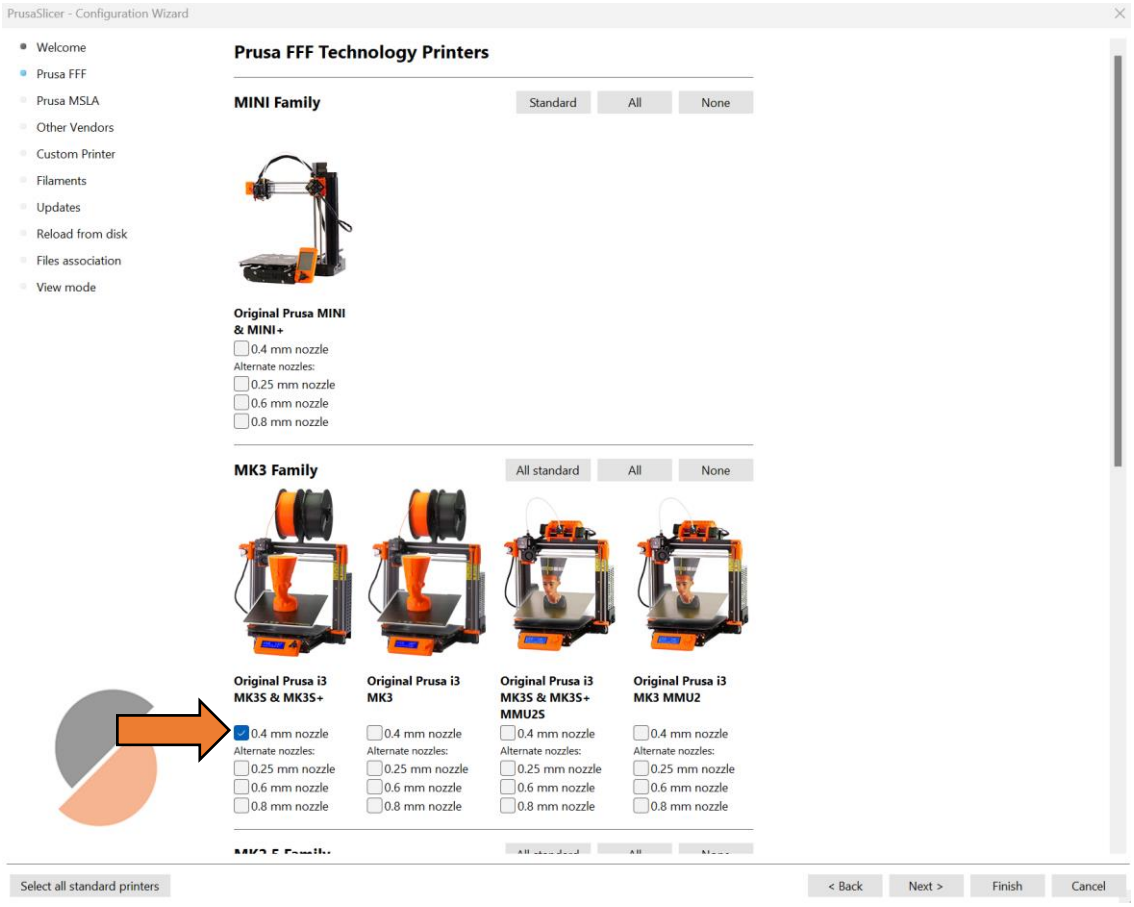
The PrusaSlicer is intuitive to use and works great with all the Prusa 3D printers in the **ARENA2036**. While it does offer near endless options to fine tune your 3D print, its default settings already yield great results which is why we recommend it.

If you have Installed Prusa Slicer and are now unsure how to proceed, this guide is for you.

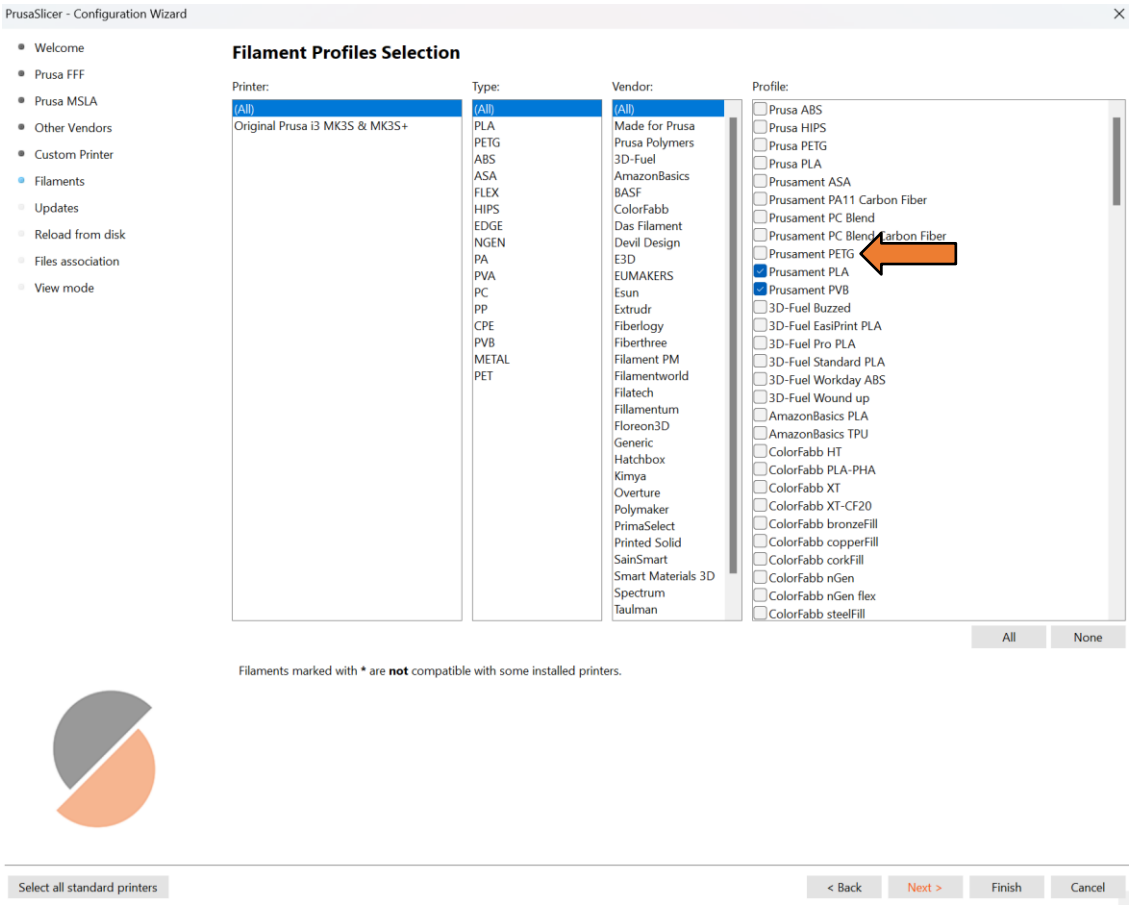
Set up the correct printer configuration

For the initial step you have to set up the correct printer and printing material:

1. Select the MK3 Family 0.4mm nozzle



2. Click “Next” until you reach the “Filaments” tab. Select the Profiles for “Prusament PLA”, “Prusament PETG”, “Generic PLA”, and “Generic PETG”:



3. Finish the setup by clicking “Finish”

Slicer settings:

1. **Print Settings:** Here you can select the layer height. The smaller you choose, the better your finished print will look but with smaller layer heights it will also take much longer. A good compromise is “0.2mm QUALITY”.
   You’ll see that some layer heights are listed multiple times. That’s because these are presets, which change much more than just the layer height. These presets work great for most situations.

2. **Filament:** You can choose which material you want based on the requirements for your print. Here are some options, that are available at the ARENA2036:
   a. **PLA** is a nice and forgiving material to print in
   b. **PETG** is more durable but often has a worse surface finish

3. **Printer:** In the ARENA2036 this is always the “Prusa i3 MK3S”

4. **Supports:** If your model has steep overhangs (typically larger than 45°) or free-floating bridges (longer than 5mm) than you likely need supports. This is added material that supports the part while printing, that can be removed afterwards.

5. **Infill:** To reduce the used material and the print time, the inside of a 3D print is filled with a honeycomb like structure. This percentage describes how much of the volume of your model is filled with material. Lower Values lead to a mechanically weaker part but also to much shorter print times.

6. **Brim:** If you want to print tall objects with a small base surface, it can be advisable to use a brim. This adds a brim around the base, so part adheres better to the build plate.

7. **Slice now:** Press this button to create the G-code

FAQ

Do I have to pay to use ARPS?

No, you don't have to pay to use ARPS, it is a service provided to you from us at the CreatorBOX and the **ARENA2036**, but we are always happy if you provide us with Filament, especially if you print a lot of parts with us. 😊

Where can I get help?

If you need help with setting up your account, accessing the ARPS2 Website, or creating G-Code, write us an Email at creatorbox@arena2036.de or come by and ask us in person, we are located on the **ARENA2036** factory floor next to the red container.

What should I do if I observe a malfunction or a defect?

If you see that the printer is malfunctioning, please go to the ARPS2 portal and press the "cancel" button under the printer with your print on it. Also, write us an email giving us a heads-up so we can fix the printer.

I would like to print with my own filament. What do I have to do for this?

Write us an email at creatorbox@arena2036.de or come by to our red container and ask one of us, if we have capacity, we can try to accommodate your request

Where can I give feedback and report bugs?

We are always looking to improve our service, so if you have any feedback or want to report bugs write us an email at creatorbox@arena2036.de

When I click upload, I get an Error message.

There might be problems with your G-code, one of the most common one is that the bed temperature is too high, we have limited the maximum temperature to 80°C to prevent the parts from sticking too much when removing the finish prints.

Also, if you are using a different slicer, you might be trying to print outside of the printers bounds.