

Table of Contents

G-code preparation for MMU1	3
Step 1 - How to generate G-code and print on Original Prusa i3 MMU1	4
Step 2 - Download the latest drivers	5
Step 3 - Add the first part of the object (.stl)	6
Step 4 - Loading the remaining parts of the object (.stl)	7
Step 5 - Assign extruders	7
Step 6 - Position the Wipe tower	8
Step 7 - Print settings and Filament	9
Step 8 - Slice, export and print!	10

G-code preparation for MMU1

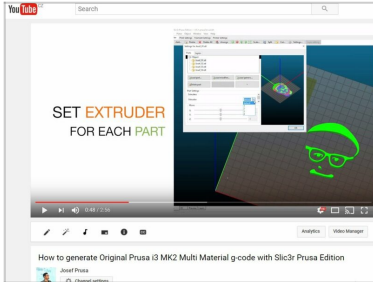


help.prusa3d.com/g105784

Scan the QR code to
display the latest
version of this
chapter.



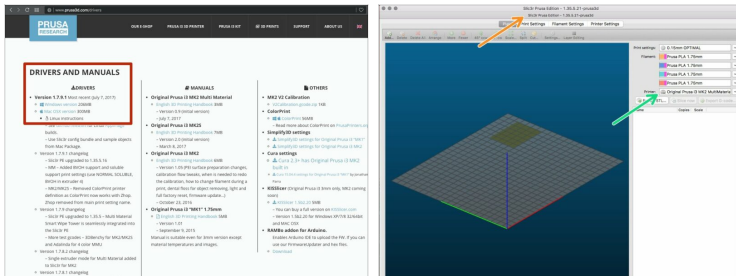
STEP 1 How to generate G-code and print on Original Prusa i3 MMU1



! This is a legacy guide, For the current software version and **MMU2S** units, please move onto G-code preparation for **MMU2S**

- **To generate Multi Material G-code, you can:**
- Watch a YouTube video <http://www.prusa3d.com/mmuvideo>
- Or follow this guide. The approach is the same.

STEP 2 Download the latest drivers



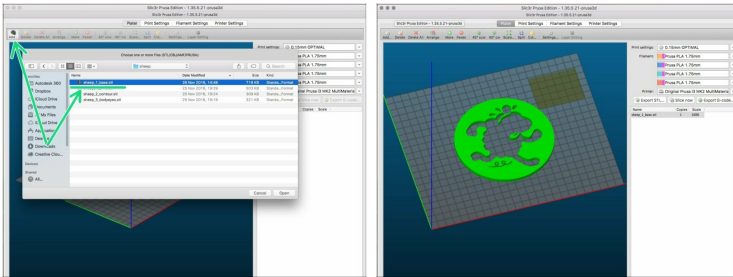
🛡️ **We recommend to have installed always the latest release of Prusa drivers, you can download them here:** <http://www.prusa3d.com/drivers>

🟡 After you finish the installation, run **Slic3r Prusa Edition**.

🟢 Ensure you have selected the proper settings for Multi Material print. You need to select **Original Prusa i3 MK2 Multi Material** printer.

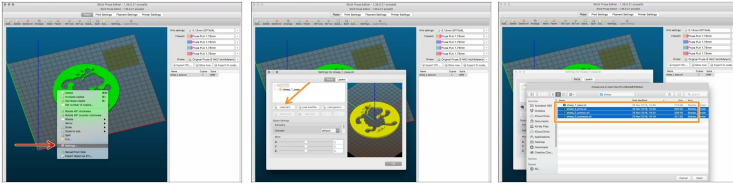
📄 This guide is based on drivers - version 1.7.9.1 (Slic3r PE 1.35.5.21).

STEP 3 Add the first part of the object (.stl)



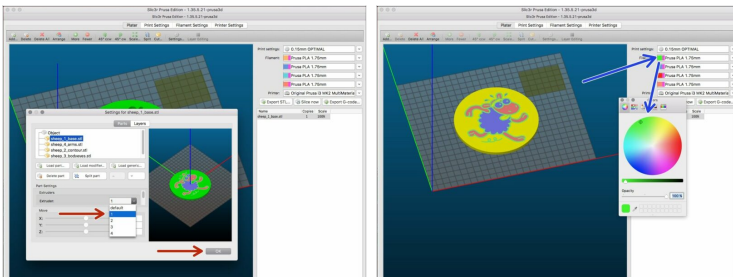
- Prepare your model separated in multiple .stl files, if you don't have it yet, please visit our guide: [Creating MMU models: Export model from Fusion 360](#)
- Load the first .stl file part by clicking on the button **Add**, you can also Drag&Drop the file into Slic3r PE.
- ⚠ **The remaining parts (.stl files) must be added in a different way!!! Please see the next step.**
- ℹ Sample model used in this guide 3D modelled by Roman Tyr.

STEP 4 Loading the remaining parts of the object (.stl)



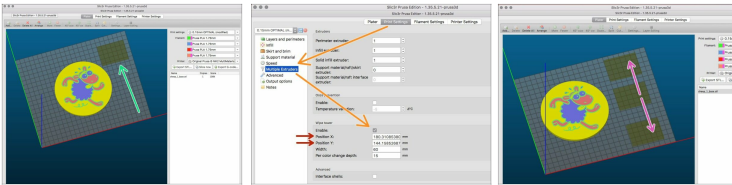
- Right-click on the object and select **Settings** from the context menu.
- Pop-up dialogue for adding additional parts of the model will appear.
- Use the button **Load** to add all parts of the object.

STEP 5 Assign extruders



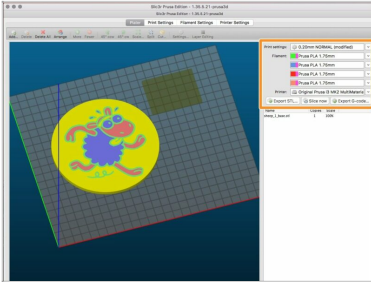
- Assign an extruder for each part of the object. Click **OK** when all parts are assigned.
- You can assign a colour for each extruder to preview the final look. Double-click on the left part of each Filament field a colour palette will appear.
- Note the colour preview will be available in the tab **Preview** only after you slice the model by **Slice now**.

STEP 6 Position the Wipe tower



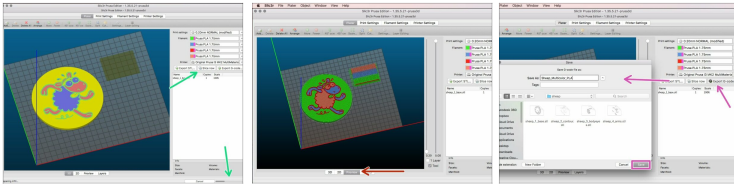
- ◆ Ensure the **Wipe tower** is enabled.
- ◆ If you can't see the Wipe tower in the 3D view, open **Print settings** and click on **Multiple Extruders** and select **Enable**.
- ◆ You can adjust tower's position by:
 - ◆ Changing values of Position X and Position Y.
 - ◆ Using left-click and dragging the tower across the heatbed.
- ⚠ **Ensure the Wipe tower ISN'T COLLIDING with the printed object!!!**
- ◆ **Experienced users** can also experiment with the size of the tower by changing **Width** and **Per colour change depth**.

STEP 7 Print settings and Filament



- Before you proceed to slicing, please select the **Print settings** (layer height) and your **Filaments** (material).
- For the latest list of supported materials and print quality please always read our Multi Material Handbook
- i** Slic3r PE offers many more settings to be changed. You can leave the recommended values or adjust them. However, this is for **experienced users**.

STEP 8 Slice, export and print!



- **We are almost done, now just:**
- Press **Slice now** and wait until slicing is complete.
- Check your print including Wipe tower in the **Preview** tab.
- When you feel satisfied with the Preview, click on **Export G-code** and name your new model.
- Your file is ready! Happy printing!
