Download:

1. Download and install Prusa Slicer: https://www.prusa3d.com/prusaslicer
    1a. When downloading make sure to choose the option for "Drivers and Apps Package Full Installer" for your OS. (On windows when asked what files to install make sure you check the box that says drivers).

2. The first time you launch Prusa Slicer you will see the Configuration Assistant. Click Next to get started.
    - In the assistant uncheck the box for Original Prusa Mini .4 mm nozzle.
    - Check the box for Original Prusa i3 MK3S & MK3S+ MMU2S.

3. It is suggested that you use the "Expert" view.

Setup:

1. Under the "Filament" section from the main screen you will see 5 drop down boxes which correspond to the up to 5 filaments the MMU2S can print at once! To select a material other than Prusament PLA click on the Filament dropbox and select either one of the predefined options or select "Add/Remove Filaments" for even more options. You can navigate to the Printer Settings tab and change the color of each of the filaments to better reflect what is currently on the machine. To do this go to Printer Settings -> Extruder 1/2/3/4/5 -> Preview -> Extruder Color -> Change to what is on the machine.

2. Cick on the "Plater" tab to load the model. All the other printing settings can be left on default (this is for only Prusament PLA, for other materials please consult a technical supervisor about best printing settings).
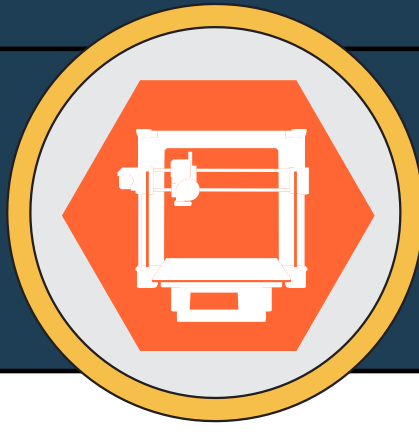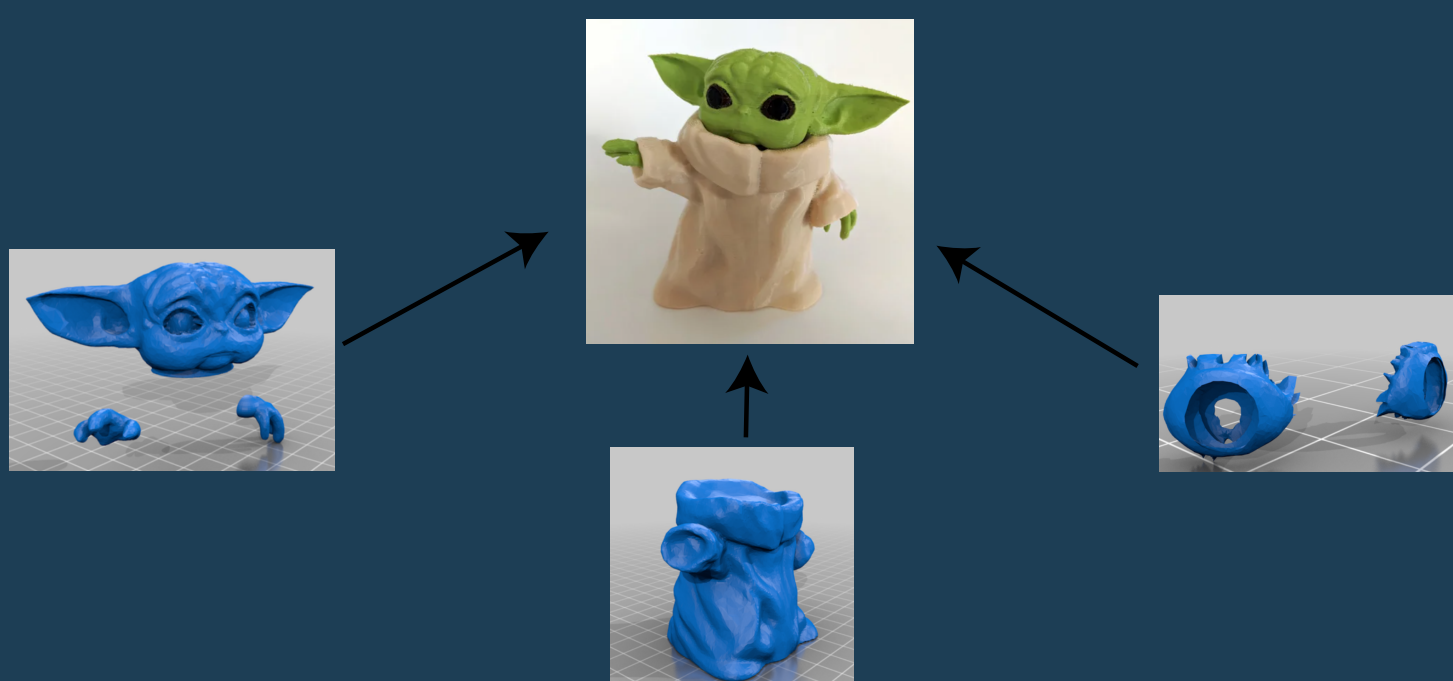
Slicing

1. All MMU prints are specifically designed to be MMU prints. Which means they will come in multiple parts and fit together as one piece in the end (See below for example). Each part can be assigned an extruder and in theory use a different material/color hence multi material print.

2. File -> Import -> STL (either add them one at a time or shift click to import a group of STL files, you will see a dialog box saying "Multiple objects were loaded for a multi-material printer" click yes on this box).

3. Your print should come in multiple parts that can each be assigned to one of the extruders/colors. If your STL parts have warning triangle boxes just click on them to conduct an auto repair (Prusa slicer has detected weird geometry but usually it's not a big deal).

4. You should see the Name of your STL, Extruder, and Editing in the right hand side of the Prusa slicer software. To change the extruder/color for one of your STL parts, click on the part, click on the part name from the list and right click the extruder dropbox and assign it one of the five colors available. Repeat for each of your parts.

Slicing Continued:

1. Once you have selected all the extruders/colors to be correctly assigned on your file right click on the gear in the "Editing" column and click on "Wipe option", this should bring up a box at the bottom that says "Wipe into this object's infill" and you want to click on the checkbox to allow for this setting. Repeat for each of your parts.

2. Add support (if needed) by clicking on the "Supports" dropbox and selecting the appropriate option.

3. Once done click Slice now. You have two views to choose from "3D View" and "Preview View". The 3D View is what you have been working with thus far and will give you a 3D representation of your print. The Preview View allows you to step through your print layer by layer to see how it will print (these views are located in the bottom left corner).
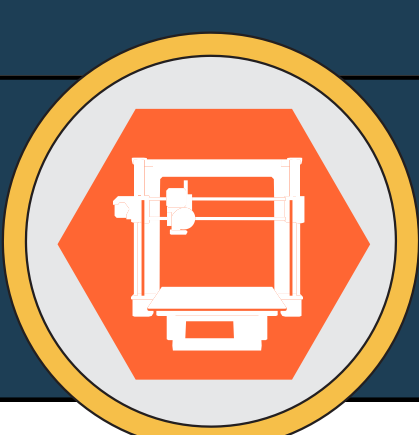
Printing Process:

1. Talk to a technical supervisor or program specialist about loading filament before you start your first print!

2. Grab the SD card from the Prusa, put into computer (we have adapters if you don't have sd card reader on your laptop) and export the .gcode onto the SD card from the Prusa Slicer.

3. Print. Using the scroll wheel, go to "Print From SD" and select your exported .gcode (press scroll wheel in to select).