3D Industries – Software Installation

1   Preamble

This is a guide to installing the software that is used to convert CAD source files to the Gcode files used by the printer.

Other relevant documentation describes the operation of the printer itself.

IMPORTANT
The first time after installing Slic3r you may be prompted to go through a set up process. **DO NOT DO THIS**. Cancel the procedure. The config.in files provided by 3D Industries contains all the correct settings for the machines.

2   Overview of the process

CAD files showing the 3D image cannot be used directly by the 3D printer. These files are converted to Gcode files first. Gcode files have the extension **".gcode"**
3D Industries recommends and supports the Open Source **Slic3r version 1.2.9 software**.
3D Industries sandboxes and tests the latest Slic3r version and provides this with the printers when delivered. The versions supplied contain tailoring for the 3D Industries printers so that no printer related set up is required.

The software is supplied on a USB stick. It can be operated from the USB stick (slower) or copied to the laptop or PC

Step 1	The Software is loaded and a configuration file is selected.
	The CAD file (typically .stl) is loaded and position on the bed in a 3D or 2D Image
Step 2	Any changes to the software settings can be made here and saved if required.
Step 3	The CAD file is converted to Gcode and exported to a directory.
Step 4	**Preview** The generated gcode can then be viewed in 3D and each layer structure can be seen by using a slider.
Step 5	The pattern of the gcode generated sowing the perimeters infill and any support material can be shown under the "**LAYERS**" tab by using a slider.
Step 6	If the user is satisfied with the result then the Gcode file can be copied to the SDCard that is inserted into the side of the LCD controller.
Step 7	The printer can then be pre heated the bed prepared and the file printed.

Note, at step 4 if as a result of looking at the preview it is decided that the infill for example is too dense or the perimeters should be increased then Step 2, 3 and 4 can be repeated.

3   Software installation – Slic3R 1.2.9

3.1   Supplied software

The software supplied on the USB stick is in the following folder structure:
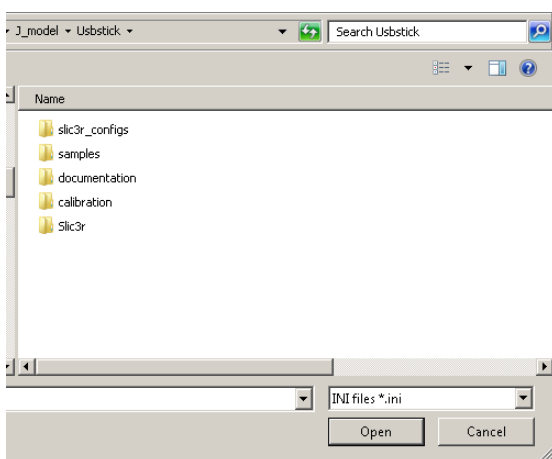
It is recommend that this is copied to the PC or Laptop

3.2   The Config.ini file

- The config.ini contains all the setting for the printer supplied.

3D Industries – Software Installation

- The **config.ini** file is a default set up for printing PLA. Do not modify this file instead copy it and rename it keeping the .ini extension.
- When performing the gcode conversion for CAD files different settings may be required. For example if printing in PLA ABS or flexi different setting will be required for the temperature and cooling and possibly the skirt or brim.
- Depending on the object, support material may or may not be required.
- Different percentages and types of infill may be required.

The settings can be made prior to performing the Gcode export, or settings may be made and these saved to a **config.ini** file with a different name. For example you could have config.ini files\:

- PLA.ini
- ABS.ini
- FLEXI.ini etc

To save the settings used for these different materials. You could also have additional variations on these for settings used frequently for example

- PLA-solid.ini
- PLA- Hollow.ini,
- PLA – 50%.ini indicating the infill required.

Tips:

- Save the original config.ini to config.ini.ori for example
- The config.ini has a text file ini but changes should not be made to it directly as some settings are dependant on others. A new config file should always be changed only by exporting the changed setting to a new config.ini name.

4 Slicer 1.2.9 Installation testing

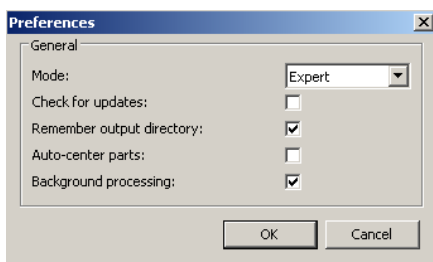Navigate to the directory (Slic3r-X86 or x64) and execute the file "**slic3r.exe".**

The home screen will be displayed:
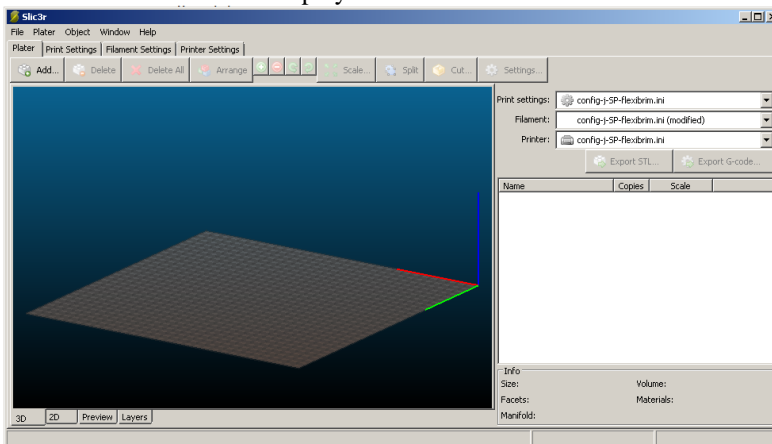
The first time Slicer is executed the user may be prompted to go through a set up routine. Cancel this because all the set up settings is in the supplied config.ini file.

Click on "**File** " – "**Preferences**".
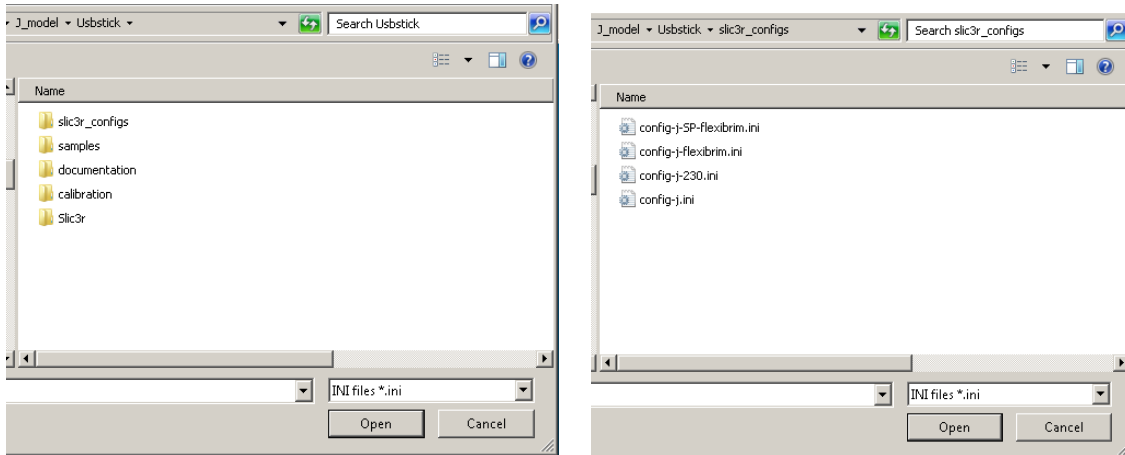A menu with check boxes is displayed:

Check the following and set as necessary as shown

3D Industries – Software Installation

If settings were changed - restart Slic3r.

Click on "**File**"– "**Load configuration**" . An explorer window will open.

Navigate to the directory containing the config .in file (Slic3r_configs) and select that then select the config.ini file

A CAD file can now be selected and sliced ready for printing. Refer to the relevant guide.