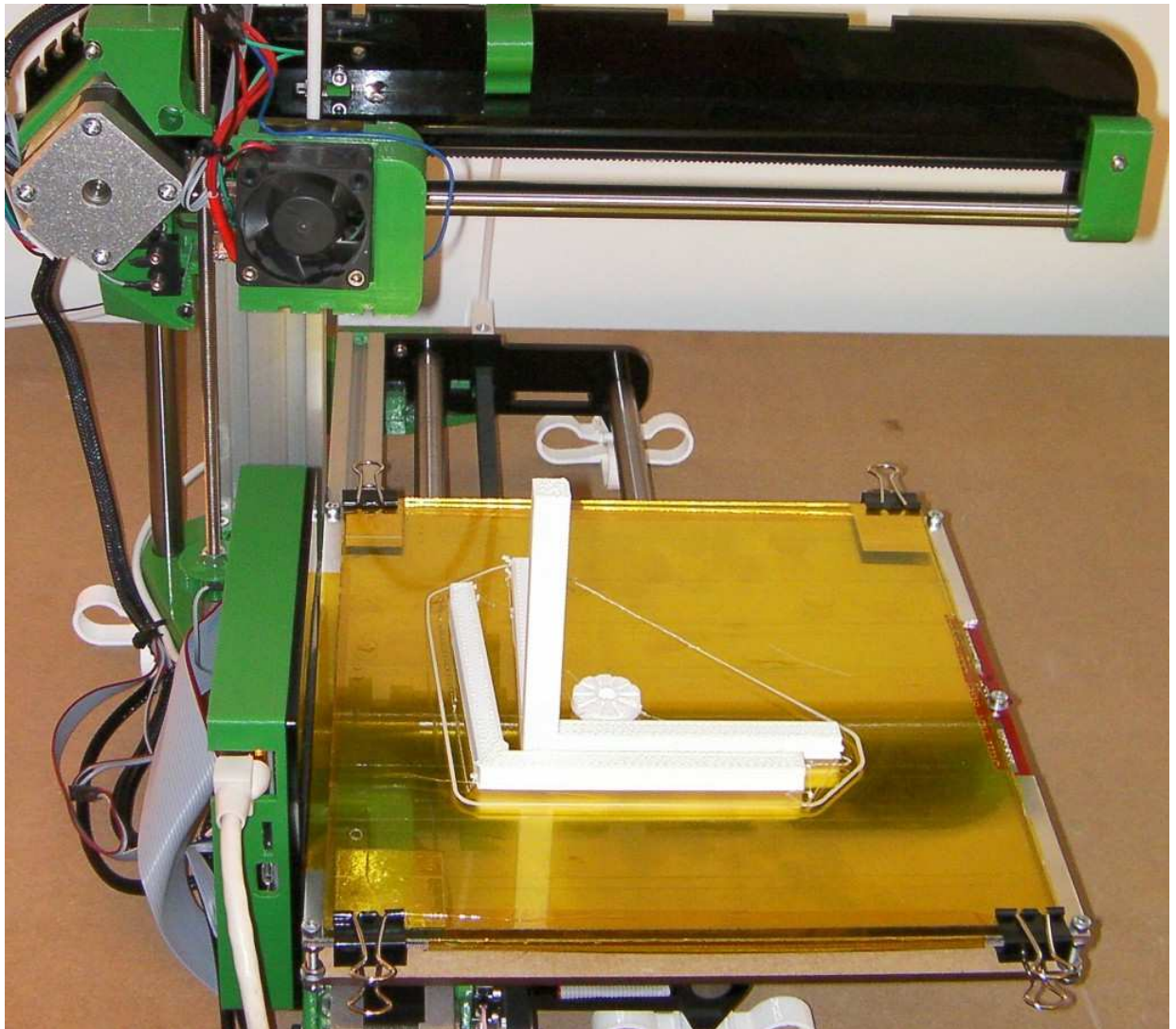


Printing

Contents [[hide](#)]

- [1 Printing the Orthogonal Axis Compensation Test Pieces](#)
- [2 Converting CAD files for printing](#)
 - [2.1 File formats](#)
 - [2.2 Slicing software](#)
 - [2.3 Slic3r profiles](#)
- [3 Printing from the SD card](#)
- [4 Print an Ormerod](#)

Printing the Orthogonal Axis Compensation Test Pieces



The first thing to print is the orthogonal axis compensation test pieces, as in the picture above. The G Codes for this are in a file called ORMAXIS.G on the SD card.

(The 3D models for the test pieces are in the **Data** folder of the RepRap Firmware download, incidentally: **gauge.stl**, **thumbwheel.stl**, **testpiece.stl**, and **calibration.scad**.)

For help on the Ormerod user interface, [see here](#).

Plug in the USB and power, power on your Ormerod, wait a short time for it to connect to the network, open the browser, and type the IP address of the Ormerod in the address bar.

Log in, and go to the **Control** page.

If the nozzle is low, lift it with the up-Z buttons to about 10mm above the bed.

Now home the X and Y axes.

Normally, we would use the Z probe to home the Z axis as well, but as this is the first print we will home Z by hand. Move to the first point you used for setting the Z probe (in the [Axis Compensation](#) Instructions).

Lower the nozzle using the small-increment Z move buttons until it is just clear of the bed. You want it as near touching as possible, but so you can just see light in the gap.

Send a G92 Z0 command to set this as Z=0.

Now run your bed plane compensation. If you recorded it in a **setbed.g** file, simply go to the **Print** page and run that.

Go back to the Control page and check all the numbers for position and temperature look as you would expect.

Now run the ORMAXIS.G file. It should print the objects above. Before you take them off the bed, use a felt-tipped pen to label the axes of the three-legged piece. In the picture above, X runs left-right, Y runs front-back, and Z runs down-up.

You can now go back to the [Axis compensation instructions](#) and set the orthogonal axis compensation.

The G Code files for printing a [snowman Christmas tree decoration](#) and the traditional [RepRap coathook](#) are also on the SD card. You can print those in just the same way as above.

Converting CAD files for printing

File formats

Most CAD software can output files that can be used for 3D printing. There's a useful list of free CAD systems [here on the RepRap website](#). Professional software like SolidWorks and Autocad, as well as free options such as RS DesignSpark Mechanical, Sketchup, Blender, Sketchup and OpenSCAD (as well as many others) can export stereolithographic (STL) files. They use the file extension '.stl'.

An [STL file](#) is a list of triangles completely covering every surface of a 3D object. This is not a very robust way of representing a solid, but it has become a universal standard.

Slicing software

Like all 3D printers, Ormerod is actually controlled by G Codes – low-level instructions that say things like *go-to-this-point*, or *print-filament-in-a-line-from-this-point-to-that*. We need a program to convert STL files into G Codes. A very good program to do that is called Slic3r. It is open-source and free, and it is [available here](#). There is also documentation on Slic3r available [here](#), so that is not reproduced here.

Slic3r profiles

Slic3r needs to be set up for each type of 3D printer for which it translates STL files into G Codes. We have performed that set-up for you for Ormerod. To get it you need to download the main Ormerod repository (if you haven't already), which is stored [here on Github](#). As with downloading the RepRap Firmware, click on the **Download ZIP** button with the little cloud and arrow. In the download is a folder called **Slic3r-settings** that contains the files Slic3r needs to convert STLS to G Codes for Ormerod.

To use our profiles with Slic3r, the contents of the 'Slic3r-settings' folder that is in our github repository here [[github.com](#)] needs to replace the contents of the default Slic3r profiles folder. The location of this folder depends on which operating system you are using. You need to run Slic3r once, so it creates the 'Slic3r' folder that holds the profiles.

On **Windows 7/8**, the profiles are stored in folders under
c:\Users\{username}\AppDataRoaming\Slic3r

AppData is a hidden folder; you will need to change the View settings to see System folders, so you can find it.

On **Windows XP**, the profiles may be stored in the same folder as above, or in
C:\Documents and Settings\{username}\Application Data\Slic3r

On **Ubuntu**, the profiles are stored in an invisible .Slic3r folder, in
/home/{User}/.Slic3r

On **Mac OS X**, the profiles are stored in /Users/{username}/Library/Application Support/Slic3r

This is a hidden folder, but you can get to it by going to the main menu option ‘Go’ then ‘Go to Folder...’ and typing ‘~/Library/Application Support/’

After replacing these files, when you open Slic3r the next time, there should be three ‘Ormerod-0.5’ (for Print settings, Filament and Printer) options at the bottom of the Slic3r application window.

Printing from the SD card

You may have noticed that, so far, all printing is done from the SD card. This is the preferred way of printing. It is possible to print directly from Pronterface (ie load gcode files into Pronterface and press print), but the serial communication is slow. The printer will pause a lot during printing, and the printed part will not be of good quality.

Always copy gcode to the ‘gcodes’ folder of your SD card to print. This is where the firmware looks for files. There are currently two ways to transfer files to the printer:

1. Turn off the printer, remove the SD card from the Duet, insert it into the USB adapter, then into your computer. Copy the gcode file from your PC to the SD card, then eject the USB adapter. Wait for it to finish writing, then remove the SD card. Reinsert it into the Duet. Restart the printer. Your gcode file should be accessible from the Pronterface ‘SD’ menu.
2. In Pronterface, click the ‘SD’ button, and click ‘SD upload’. You can then upload the gcode file to the SD card while it is still in the Duet. This is quite slow, and is suitable for small files only (1MB will take a couple of minutes).

Future firmware updates will improve the speed of serial communications, and give more options for uploading files to the SD card.

Print an Ormerod

Ormerod is a RepRap – a *Replicating Rapid Prototyper*. So, to experiment, why not print out some of Ormerod’s own parts? These are available in the folder **stl/individual parts** in the [Ormerod download](#). When Slic3r has generated a G Code file for you from an STL file, you can put it in the **gcodes** folder of Ormerod’s SD card and then print it. Either put the SD card in your computer to copy the file (remember not to take it out of Ormerod unless all power including the USB is off), or you can upload G Code files using Pronterface. The Duet controller in Ormerod expects file names to be in [8.3 format](#) at the moment. This restriction will be removed on a future release.

And, now you have started, why not print out a full set of Ormerod parts for a friend? Hardware-only kits for Ormerod will soon be available, giving all the things you will need other than the printed parts to build another complete Ormerod.

Happy 3D printing!