



Cura for Type A Machines

Quick Start Guide

Table of Contents

- [About Cura for Type A Machines](#)
- [Downloading Cura for Type A Machines](#)
- [Installing Cura for Type A Machines](#)
 - [Mac](#)
 - [Windows](#)
 - [Linux](#)
- [About the Configuration Wizard](#)
- [About QuickPrint Mode](#)
 - [Overview](#)
 - [Load a 3D Model](#)
 - [General Slicing Settings](#)
 - [Slicing and Saving Your Model](#)
- [Extended Controls](#)
 - [Model manipulation](#)
 - [View modes](#)
- [Expert Mode](#)
 - [The Basic Tab](#)
 - [Advanced Tab](#)
 - [Plugins](#)
 - [Start/End GCode](#)
- [Troubleshooting](#)
 - [Install](#)
 - [Slicing Problems](#)
- [Contacting Support](#)



About Cura for Type A Machines

Cura for Type A Machines is a desktop slicing software package optimized to support the 2014 Series 1 3D Printer. It contains the settings and components necessary to get the highest print quality from your Series 1.

Slicing software is used to prepare 3D models for printing. Models saved in the industry standard STL file format are "sliced" into thin layers that the printer builds, one at a time, to create a 3D object.

Cura for Type A Machines is a branch of an open source project. It is based on [Cura 14.07 by Daid](#).

Downloading Cura for Type A Machines

Visit www.typeamachines.com/gettingstarted to download the latest version for your operating system. We currently offer easy-to-use installer packages for:

[Mac OS X 10.9](#)

[Windows 7, 8](#)

[Ubuntu Linux](#)



Installing Cura for Type A Machines

Mac

Locate the installer image in your Downloads folder. Click to open it. Drag the Cura for Type A icon into your Applications folder to install it. Now open your Applications folder, open the Cura for Type A folder and click on the Cura icon to open the program.

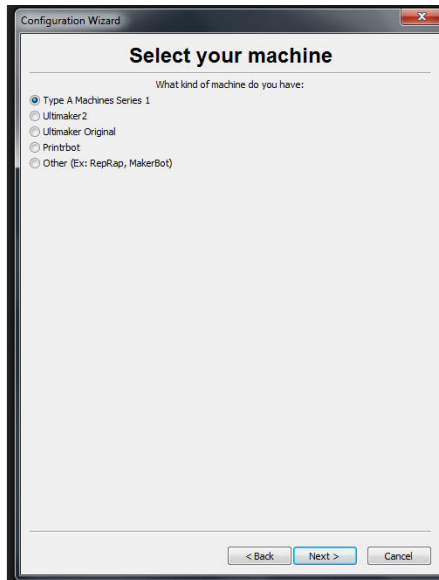
Windows

Open the installation wizard by double clicking the Cura for Type A Machines installer you have downloaded. This wizard will prompt you through setting up your installation. After your installation is complete, click the Finish button to launch Cura for Type A Machines.

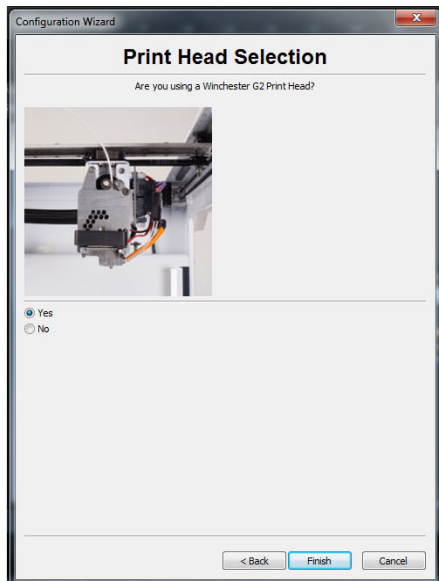
Linux

Download the DEB package for the architecture of your machine (amd_64, or i386) and click on it to open the Ubuntu Software Center. The Software Center window should open and load the Cura for Type A Machines package. Click Install on the right hand side. After the program is installed, close the window and use the Unity launcher to launch Cura for Type A Machines.

About the Configuration Wizard



The first time you open Cura for Type A Machines, the Configuration Wizard will appear. The Type A Machines Series 1 should be selected automatically.



Next, select the print head your 3D printer uses with the photo as a guide.

If your printer's print head looks like the photo, choose Yes. If not, choose No. Then click the Finish button. Cura for Type A Machines will now open in QuickPrint mode.

Note: *If you make a mistake or want to set up an additional 3D printer, go to Expert>Run First Run Wizard to change the settings.*

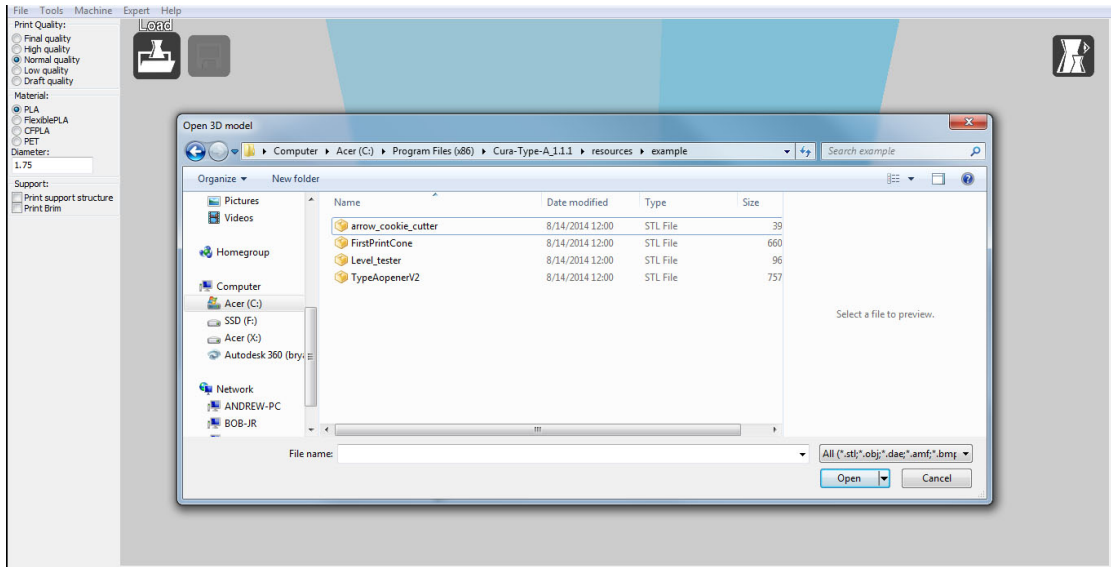
About QuickPrint Mode

Overview

The first time you open Cura for Type A Machines, the program will be in QuickPrint mode. This mode limits the number of printing options to make it easier to quickly create 3D prints. QuickPrint mode uses print resolution and material settings optimized to produce high-quality results for general printing needs.

Load a 3D Model

To prepare a 3D model for printing, you first load it into Cura for Type A Machines by clicking the Load button near the upper left corner of the window. Cura for Type A Machines works with STL, the 3D industry standard file format, as well as OBJ, DAE, and AMF files. The first time you open Cura for Type A Machines, a test 3D model called FirstPrintCone will be loaded automatically for use as your test print.



Tip: You can find a cookie cutter and other sample STL files to print in the `...cura\resources\example` folder in the location where you installed Cura for Type A Machines.

Note: Not all models are created equally, although we have made every effort to design a software and hardware stack to allow our users to create anything, there are some limitations to the process. In general, best results are achieved when the 3D Model to be printed is a CLOSED MESH, meaning it has a clearly defined interior and exterior, and does not contain naked edges. If you're having problems with a mesh import, see our troubleshooting guide.

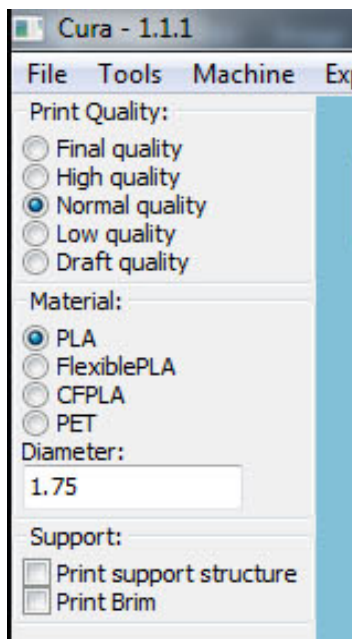
General Slicing Settings

After loading a model, inspect it to make sure it is the correct size and orientation. It should sit as flat on the bed as possible.

Print Quality/Resolution

In the upper left corner of the program window, you will find the Print Quality, Material, Diameter, and Support options. Here you can specify the quality level (or resolution) for your model. There is always a tradeoff between quality and the time required for a 3D print. Higher resolutions result in smoother surfaces, which take longer to print.

We have optimized the settings in QuickPrint mode to balance speed and quality. When you select a new quality setting for a model, the program will automatically update the time estimate for printing. That allows you to experiment with different quality settings and select the one that best fits your project and time frame.



| | |
|--------|----------------------|
| Final | 50 microns (0.05mm) |
| High | 100 microns (0.10mm) |
| Normal | 150 microns (0.15mm) |
| Low | 200 microns (0.20mm) |
| Draft | 250 microns (0.25mm) |

Material

After you've selected your resolution settings, check the type of material you have loaded in the printer.

Note: *It is very important to confirm the type of material you are printing with . Failure to do so can result in very poor print quality and damage to the printer.*

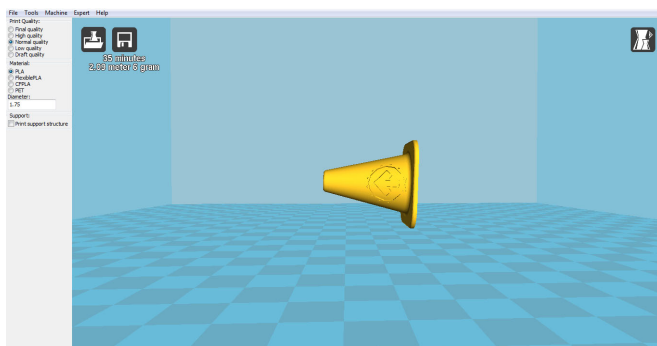
The final general setting is *Support* and *Brim*. Because the Series 1 3D Printer builds material up layer by layer, the model may need additional material to support the build in process. This additional material can be automatically generated and added to the model file.

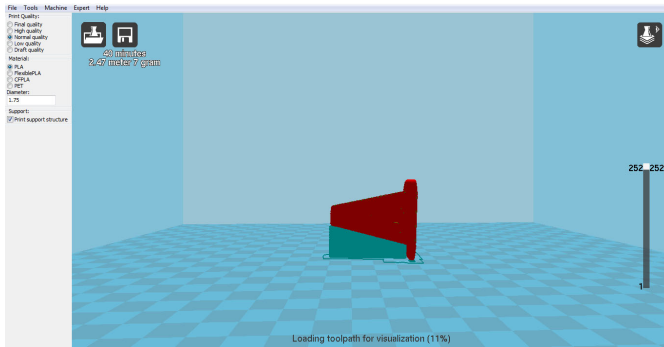
Generally support material will be necessary when any part of your model contains 'overhangs', or unsupported sections of the mesh with an angle over 45 degrees. It's a good idea when printing a model for the first time to turn on support.

Brim turns on a ring of extra material around the base of your model, this additional material sticks the print to build platform. If you are printing large flat features, small features, or if you are experiencing problems getting your prints to stick turning on Brim may help.

For most models additional support material is easy to remove by hand after the print has been removed from the printer. Depending on your model, you may not need to use support material. To preview the support material that is generated, click on the view icon in the upper right corner then the 'Layers' mode.

The cone below is shown in an orientation that would require support. Support is shown in the bottom image. For most models you will have better results printing with Support turned ON.

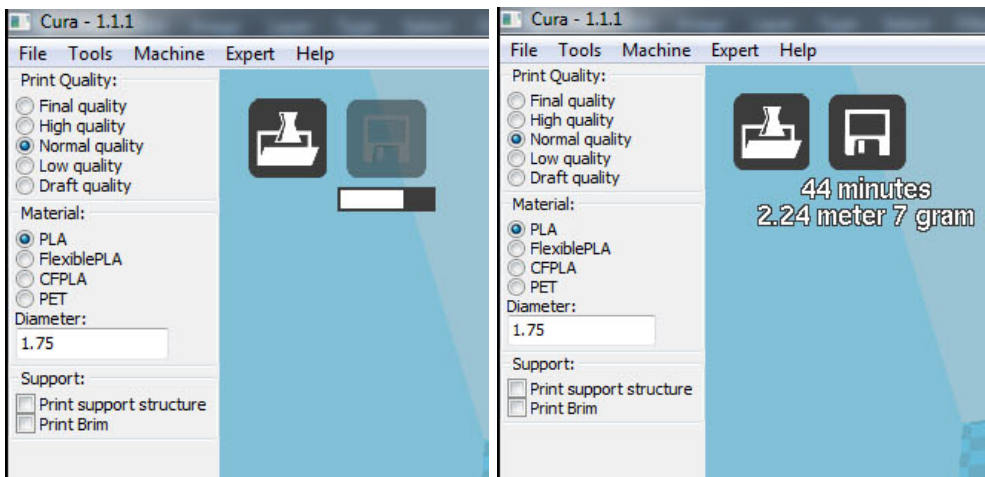




Tip: Support material generation is dependent on the orientation of your model. You can often decrease the amount of support used or even eliminate it completely by re-orienting your model geometry.

Slicing and Saving Your Model

After every change, you will notice a progress bar in the upper left corner. This is the slice progress. When it's done your model is ready to be sent to the printer. For large and complicated models, a slice may take a few minutes to complete.



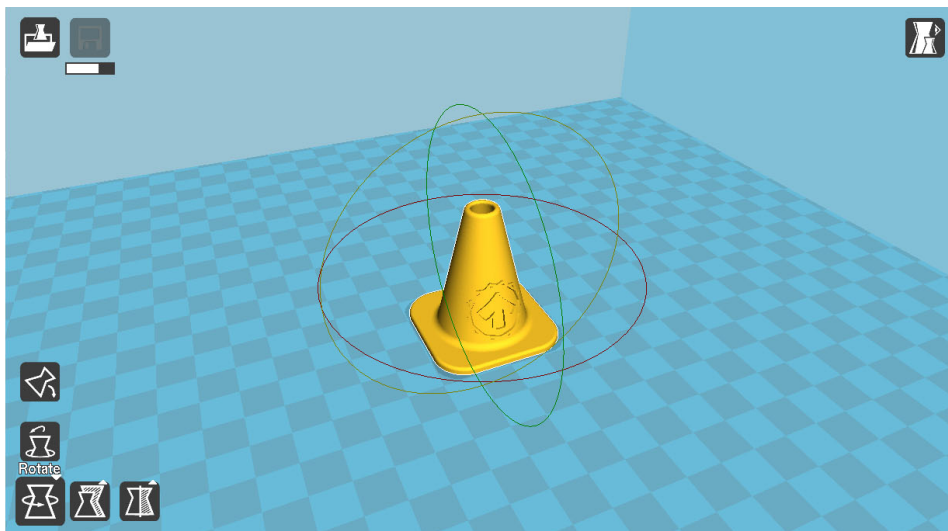
When a slice is completed, it will tell you the estimated print time, as well as the amount of material the print will consume. If you are happy with the selected settings, click the 'Save' icon to save the GCode, which will run on the printer. This saved file in .GCODE format is what you will upload to the Series 1 Web Interface.

NOW YOUR MODEL IS READY TO 3D PRINT

Extended Controls

Model manipulation

Cura for Type A Machines comes with a set of basic tools for 3D Model repair and manipulation built into the interface. To activate the tools, click on a model that has been loaded into the main screen. You will notice a bar of icons pop up in the lower left of the screen. The first button allows you to rotate your model in 3 dimensions, as well as automatically lay it down as flat as possible on the build platform.



The second button allows you to scale your model. This is useful for conversion from inches to mm. If your model comes into the viewer very small, there is a good chance your modeling program is using inches. To correct the size, open the scale tab and scale by 25.4 to convert your model from in to mm.

You can also automatically scale your part to the maximum size of the build volume here.

View modes

In the upper right of the scene view is the View icon. Click to enter the Views menu, and from here you can select from the Normal, Overhang, Transparent, X Ray and Layers mode. Overhang mode can be useful for detecting areas of the model where support will be necessary.



Expert Mode

If you're already an expert and know what you want from your slice, go ahead and switch over to expert mode. Go to expert>expert mode. The interface will reload and expose many more of the settings controlling the printer. You will need to enter the settings you want to use.

Note: Expert mode allows you to directly modify parameters of the print, direct editing can cause failed prints. If you are unsure of your settings and just want a print see "QuickPrint".

The Basic Tab

Quality

Quality settings control the resolution of the print as well as the thickness of the side walls of the print.

| File Tools Machine Expert Help | |
|--------------------------------|-------------------------------------|
| Basic | Advanced Plugins Start/End-GCode |
| Quality | |
| Layer height (mm) | <input type="text" value="0.15"/> |
| Shell thickness (mm) | <input type="text" value="0.8"/> |
| Enable retraction | <input checked="" type="checkbox"/> |

Fill

Fill controls the interior components of the print. The bottom/top thickness controls how many layers are laid down on horizontal surfaces of the print. It's a good idea to keep this number at a multiple of the layer height. We've found that multiplying the layer height by 4 works well. For example, if your layer height is 0.20mm, then your top thickness should be 0.80.

Fill density is a very important part of controlling print quality, strength, and speed. With too low of an infill, the print will not come out well; too high and the print will take a disproportionately long amount of time. We've found that a good window is 12-25% infill for most prints.

Fill

Bottom/Top thickness (mm)

Fill Density (%)

Speed and Temperature

Speed controls how fast the printer will move while printing. The 2014 Series 1 has been tested to operate between 90-150mm/s. Some materials may require slower print speeds, and quality can be impacted by printing at high speeds.

Print temperature is material dependant. Check the recommended printing temperature of your material. If you are having problems printing, you may want to try at a higher printing temperature. Go up in increments of 10 degrees to determine proper print temperature settings.

Speed and Temperature

Print speed (mm/s)

Printing temperature (C)

Support

As discussed earlier, support is necessary for prints with overhangs over 45 degrees. The support drop down changes the amount of support material.

Platform adhesion affects how strongly the print is stuck to the bed. If your print has a large flat footprint, you may not need to print with any platform adhesion. For most prints, a Brim type platform adhesion will help the print stick to the platform.

Support

Support type

Platform adhesion type

Filament

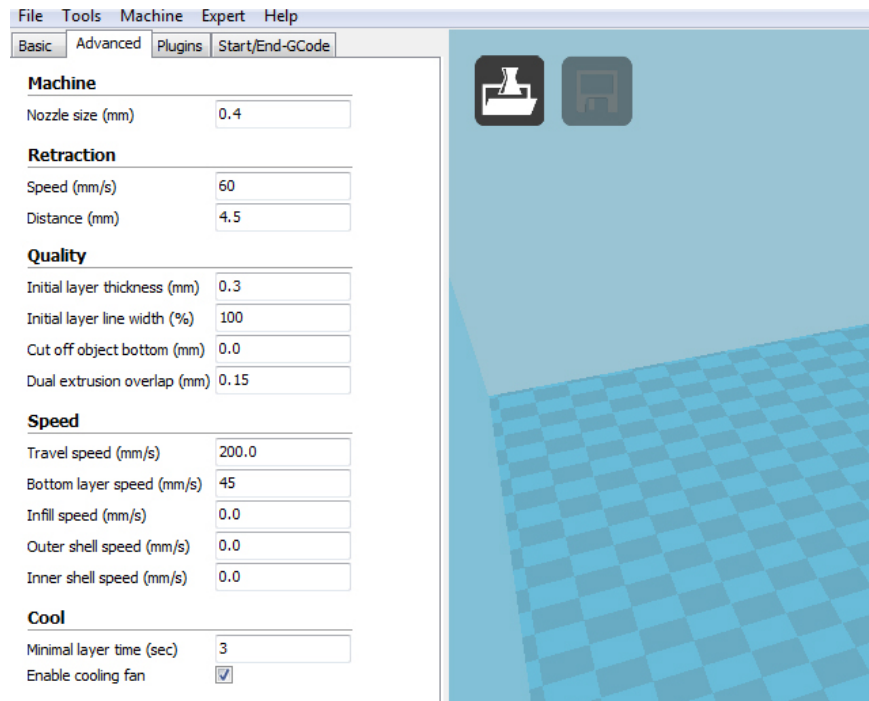
Set the parameters of the filament that you are using. Use a caliper to precisely measure your filament diameter. If you are having problems with under extrusion, adjust the flow percentage.

Filament

| | |
|---------------|-------|
| Diameter (mm) | 1.75 |
| Flow (%) | 100.0 |

Advanced Tab

The Advanced tab controls the advanced options for the slicer. These are preconfigured and changing them can cause major problems in your slice.



Nozzle Size is the actual size of the nozzle on your machine. Only change this if you put another size nozzle in.

Retraction controls the *speed* and *distance* at which the extruder pulls back at the end of a path. Getting the correct setting here insures prints are clean.

Quality controls fine resolution changes to the print. *Initial layer thickness* sets the thickness of the first layer independently of resolution. This can help with a not completely aligned build platform. *Cut off object bottom* offsets a printed object below the print bed allowing messy parts to be removed pre-slice.

Speed settings control the motion of the machine. *Travel speed* is how fast the printer moves when not extruding. A setting of 150-250 works well. Moving too fast can cause the belts to skip. A slower *bottom layer speed* can help bed adhesion.

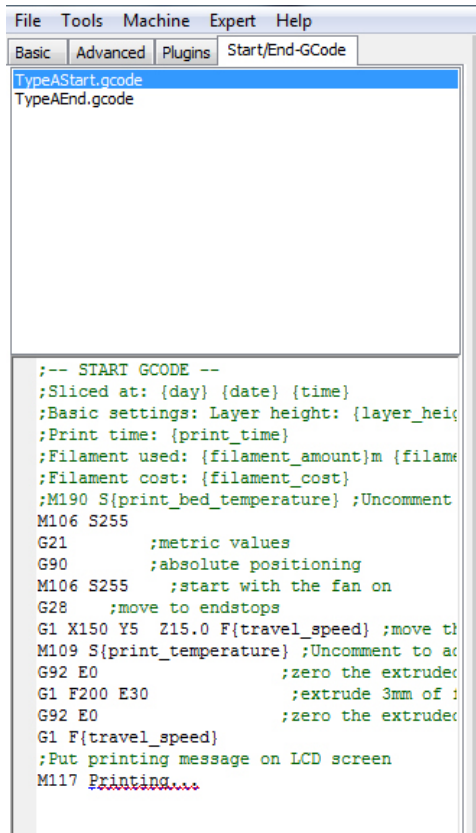
Cool controls the cooling features of the printer. For high print quality, the plastic must cool as soon as it leaves the extruder. For small features, the printer will slow down to allow for cooling. The *cooling fan* can also be controlled from here.

Plugins

Plugins are user created content that allow direct post-processing of GCode in the Cura for Type A Machines environment.

Start/End GCode

Start/End GCode controls the prefix appended to a sliced GCode file. This file is necessary to prepare the printer for beginning the print. If your Machine Setup was completed successfully then your GCode prefix by default will work with the Series 1 2014. Refer to the [RepRap Wiki](#) for a key to GCode Values.



```

File  Tools  Machine  Expert  Help
Basic  Advanced  Plugins  Start/End-GCode
TypeAStart.gcode
TypeAEnd.gcode

;-- START GCODE --
;Sliced at: {day} {date} {time}
;Basic settings: Layer height: {layer_height}
;Print time: {print_time}
;Filament used: {filament_amount}m {filament_color}
;Filament cost: {filament_cost}
;M190 S{print_bed_temperature} ;Uncomment to
M106 S255
G21          ;metric values
G90          ;absolute positioning
M106 S255    ;start with the fan on
G28          ;move to endstops
G1 X150 Y5 Z15.0 F{travel_speed} ;move to
M109 S{print_temperature} ;Uncomment to activate
G92 E0          ;zero the extruder
G1 F200 E30     ;extrude 3mm of filament
G92 E0          ;zero the extruder
G1 F{travel_speed}
;Put printing message on LCD screen
M117 Printing...

```

Troubleshooting

Install

If you are having trouble installing the software, ensure that you are downloading the correct version for your operating system.

Ensure that you are running a compatible version of an operating system. Our Mac release is compatible with OS X 10.9 and later. You can check what version of OS X you have by going to system preferences.

Our Windows release will run on Windows 7, 8, and later. In Windows go to Start>Control Panel>System and Security>System.

If you cannot find the installer after you have downloaded it, look in your browser's downloads folder.

Slicing Problems

In general, if you suspect a slicing problem, you will need to confirm the problem is software related, not hardware. To do this, print a known good GCode file such as the ones that shipped

on the printer. If that prints successfully, then open the quick print interface, load the problem model and slice. Print that file using the QuickPrint settings. If your model is still not printing correctly, proceed to troubleshooting.

Note: Many slicing problems can cause hardware problem symptoms. Before contacting support or proceeding to hardware troubleshooting make sure to print a known good GCode to confirm if the problem is isolated to slicing settings.

| Symptom | Cause | Fix |
|--|--|---|
| parts of print drooping | overhanging areas not supported | enable support and reprint |
| warping or print peeling off bed during printing | too little print on build surface | enable Brim or Raft and reprint |
| first layer not sticking | lack of adhesion | enable Brim or Raft, confirm correct temperature, speed |
| inconsistencies or gaps in the sides of parts | poor extrusion | confirm temperature, slow down speed and reprint |
| holes on top of parts | too little infill or horizontal layers | turn of infill % and horizontal layer thickness and reprint |
| incorrectly sized parts or features | too much/little extrusion | confirm the correct temperature for your material, slow down speed and reprint |
| bubbling in filament or no extrusion | incorrect temperature setting | confirm the filament type and select corresponding button |
| shifts in print | running too fast | slow down the print by selecting a higher resolution or changing the travel speed |

Contacting Support

If problems persist after printing known good GCode files, or if known good GCode files do not print correctly, please contact support at support.typeamachines.com.