Warnings (don't hurt yourself!):

Power supply: As discussed in camp, do not remove the inside cover on the power supply – that's where the 120V wiring is, and it's an electrocution hazard. (Just like you shouldn't open up the power supply of any other equipment!) If you ever need to connect additional 12V loads to the power supply, you can remove the cover underneath the socket to access the +12V and ground terminals. If you're not totally comfortable doing that, get someone to help.

Hotend and bed: The hotend and bed are capable of reaching temperatures of at least 275 °C and 120 °C, respectively, which are burn hazards. You can't tell that either one is hot just by looking at it, so always use caution – check the current temperature on the display or, if the printer is off, hold your hand near the part to check for heat before touching it.

Cautions (don't hurt the printer!):

Heat-Hotend: Don't turn off the printer immediately after finishing a print; let the hotend cool down to at least 120 °C first. If you turn off the printer while it's still hot, the cooling fan on the hotend will stop and heat can leak from the heater block up into the parts of the hotend that are supposed to stay cool.

Heat-Bed: The Buildtak surface on your print bed says it can withstand a maximum temperature of 120 °C, but the heater on the bed is capable of getting hotter than that – so don't try to push the bed to its limit. In general, try not to go hotter than 100 °C, even if printing high-temperature materials.

Electrical: Always turn off power to the printer before plugging or unplugging anything electrical. The stepper motor drivers are especially sensitive to this – unplugging a motor while it is powered is a good way to burn out its driver. Make sure you plug in connectors the right way around; some connectors, like for the endstops, can be plugged in either way around on the RAMPS board and getting it backwards can damage the electronics.

Mechanical: If you notice any belts slipping or if the X-gantry or bed assembly seem to be getting crooked, STOP! And figure out what is going on. This is most important on the Z axis – if one of the leadscrews stops while the other one keeps turning, the motor can exert enough force to break or bend part of that assembly.

More information:

We have uploaded the .stl files for the printer parts and firmware configuration files to https://github.com/HypatiaStudioLLC/SummerPrinter ; more information will be added there as quickly as I am able.

If you have questions about the printer or problems with it, please create an issue in the issue tracker there to discuss it. That way, others will also be able to join the conversation and learn from it.

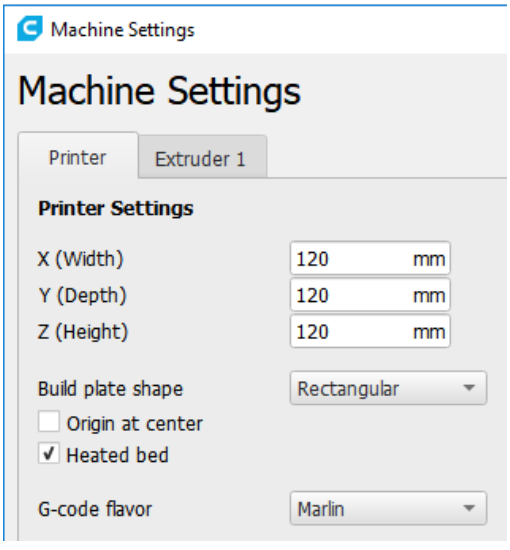
Operating the printer:

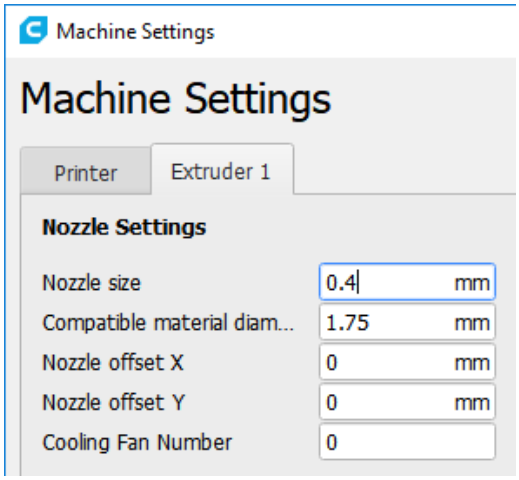
Filament: You need filament to print with. If you're new to printing, start with regular PLA from a good brand – we mostly use Hatchbox and it is almost always trouble-free. Esun also works well, in

my experience. Don't go to more difficult materials (PETG, ABS, etc.) or filled materials (wood-fill, metal-fill, etc.) until you have some printing experience.
It is possible to get a bad spool of filament from any brand, though it becomes more likely with especially cheap filament. Bad PLA will jam or underextrude repeatedly – if you just can't get good prints, it might be the filament's fault, not the printer; try a new spool.
For ABS, we have had good experiences with eSun's ABS+ filament. (That's what most of the printed parts in your printer were printed with.) It is best to print ABS in a well-ventilated location. Cheap ABS tends to have a stronger smell, but the fumes aren't healthy to breathe in any case.
We don't have enough experience printing PETG or other materials to recommend specific brands.

Printing pipeline: The general 3D printing pipeline is to create a design with 3D modeling software or a 3D scanner; export it as an .stl file (triangle mesh); use a slicer to convert the mesh to instructions for the printer (G-code); and send the G-code to the printer to print. Different software is used for each step.
3D modeling: Consider Tinkercad, Fusion360, Meshmixer, OpenSCAD, DesignSpark Mechanical, or BeetleBlocks. Other software options (that we have less experience with / do not like as much) include Vectary, FreeCAD, Blender, and Sketchup.
3D models: The most well-known site for downloading .stl files is thingiverse.com .
Slicing: We mostly use Cura 3.6; Repetier also works.
Printing: You can either connect your computer directly to the printer with a USB cable and have the slicer feed lines of G-code one-at-a-time during the print, or save the G-code file to an SD Card and insert it into the card slot in the display of your printer. The latter option is usually more reliable; you won't have a print fail if your computer goes to sleep or installs updates in the middle of a print. The printer can be picky about cards; you might find that some work in your computer but not the printer.

Slicer settings: Slicers have dozens, if not hundreds, of settings that influence how they translate meshes into printer instructions. Here are some basic settings to start from; you will have to experiment to get the best performance from your printer!
When initially setting up Cura you can set basic parameters of your printer:





Dimensions: The print volume is rectangular with 160 mm size in x, y, and z. (Note that you will have to avoid the screw holes in the print bed if you want to use the full width of the bed.)
The bed is heated.
Filament diameter is 1.75 mm. (Watch out! Cura defaults to 2.85 mm.)

Temperatures: Nozzle and bed temperatures depend on the material and brand; start with the brand's recommended temperatures – usually PLA is around 200 °C for the nozzle and 50 °C for the bed.
Layer height: Start with 0.2mm layers; we have printed jewelry successfully at 0.05mm layer height. Use layer heights that are multiples of 0.01mm.
Print speed: We normally use 80mm/s with 50mm/s outer perimeter speed and 200mm/s travel; Cura's defaults are a bit slower.
Cooling: For PLA, enable cooling with the fan speed at 75% or more. For ABS, use 35% fan speed.
Retraction: Enable retraction, with a retraction distance of 2mm. Retraction speed can be increased to 50mm/s or so. Cura defaults to 6.5mm retraction distance, which is way too high.
Bed adhesion: If you have your bed leveling and first layer spacing dialed in well, you should usually not need any bed adhesion. If you are printing a model with a small base or that mostly has support material touching the bed, you might wish to enable a small brim to help the support material stay put.

About the printer:

Hardware: This printer is based on a frame made of 2020 aluminum extrusions, which is pretty standard for printer kits and, for a printer this size, makes for a very rigid frame. In general, any accessories that are made to be mounted to 2020 or 'v-slot' extrusion will fit on this printer frame. The motion system is designed to fit entirely inside the frame so that you can place panels on the sides and bottom to partially enclose the printer.

The motion system is Makerbot-style, in which the hotend moves on a gantry in the X direction (side-to-side), and the whole gantry, including X-axis motor, moves in the Y direction (front-to-back). The bed itself moves in the Z direction (up-and-down). The speed of this style of printer can be limited by the need to sling the weight of the X-axis gantry around, but in this design we took care to keep the mass of that assembly low, so high print speeds and accelerations are possible.

> [Aside: Low-cost printers more commonly use a Prusa-style motion system, in which the bed moves in the Y direction, but it is difficult to print at high speed and quality with such an arrangement. (Genuine Prusas do very well, but they are not low-cost!) Other common motion systems include CoreXY (fast, but pretty complex) and Delta (fast, but prone to subtly distorting prints if the geometry is not exactly right).]

The bed is supported by leadscrews on both sides of the printer, which prevents vibration from movement of the printhead from translating into shaking of the bed.

The hotend (where the plastic gets melted with the nozzle) is a clone of the all-metal E3D V6 hotend made by Trianglelab. The E3D documentation at https://e3d-online.dozuki.com/c/V6 applies to this hotend as well. Any nozzles that fit an E3D V6 hotend should fit this hotend.

The extruder is a clone Mk 8 extruder.

The print bed is 160mm x 160mm and has an integrated 12V heater. The printing surface is genuine Buildtak, which provides very good adhesion for PLA prints. You can find troubleshooting and maintenance information for Buildtak at https://www.buildtak.com/support/#FAQ .

The stepper motors are 1.8° per step (200 steps per revolution); the Y, Z, and extruder steppers are rated for 1.5A max and the X stepper is rated for 1.0A max. Don't run the steppers at more than about 2/3 of their rated current; they will get hot enough to soften the 3D printed parts they are attached to.

Electrical: The power supply is a Meanwell 12V power supply rated at 29A; the printer should draw around 17A max, so there is room to grow or add accessories. The control electronics consist of an Arduino Mega board with a RAMPS V1.6+ board on top of it and A4988 stepper motor drivers. If you should ever wish to use these electronics in a larger printer with a more demanding heatbed, you can attach the bar-shaped heatsink to the power transistors on the RAMPS board to make sure they can handle the higher current without overheating; the heatsink is not needed for this printer. (See https://reprap.org/wiki/RAMPS_1.6 .) The labels on the screw terminals on the RAMPS board are rather mysterious; D10 is for the hotend heater, D9 is for the print cooling fan, D8 is for the bed heater, and the terminals below them are for power input. The display is a RepRapDiscount Full Graphic Smart Controller. The Arduino is running Marlin firmware version 1.1.9.

Maintenance:

Electrical: Periodically check the large wires that connect to the screw terminals through the bottom of the electronics case – they can work loose over time. Just give gentle tug on each wire and, if it budges, tighten down the screw for that terminal.

Mechanical: We greased the leadscrews for the Z axis with synthetic grease (Super Lube with PTFE), they may require additional grease over time if they start making noise. Use the same type of grease; don't use oil.

The bearings, both linear and rotational, are lubricated at the factory and should not need additional lubrication.

The hotend cooling fan will eventually go bad; the bearings on these small fans just wear out over time. If it doesn't spin freely or starts to make noise, it is time to replace the fan. For replacements, look for an E3D V6 hotend fan. (If the replacement fan comes with long-enough wires, you might wish to wire it directly to the 12V terminals on the power supply rather than the control electronics.) The part-cooling blower (on the back of the printhead) should last a long time.

Heatbed: The Buildtak surface should last a long time for PLA. The surface may get dirty or oily over time; you can clean it with isopropyl alcohol (rubbing alcohol). Don't use acetone or other harsh cleaning agents. If cleaning the Buildtak doesn't restore its stickiness, you can also try lightly sanding the surface with medium-grit sandpaper. At ABS temperatures (100 °C or more), the surface will have limited life, mostly due to the adhesive under it wearing out. To increase its life with ABS, run the bed at the lowest temperature that gives adequate adhesion, which will probably be around 90-100 °C. There's a good recent video by Maker's Muse on Youtube that discusses how to clean print beds here: https://www.youtube.com/watch?v=ShFaJ027pFs .

Expect to have to adjust the bed's height periodically, using the screw on the Z endstop, especially if you move the printer or change materials. You shouldn't have to go through the full leveling procedure again. If you find that prints don't stick well on one end of the bed, that may indicate that you have to raise that end of the bed slightly. Look at the amount of 'squish' the first layer of prints has and see if it is different in different areas.

Troubleshooting:

General print troubleshooting: There is an overall troubleshooting guide for printing problems on the reprap website here: https://www.reprap.org/wiki/Print_Troubleshooting_Pictorial_Guide . Another good guide can be found by MatterHackers here: https://www.matterhackers.com/articles/3d-printer-troubleshooting-guide . Finally, Simplify3D has one as well, although some of it is specific to their slicing software, here: https://www.simplify3d.com/support/print-quality-troubleshooting/ . It is good to peruse these, just to get an idea of what can go wrong in a print and what to watch out for.

Here are some troubleshooting tips specific to this printer.

Consistent under-extrusion: If the printer seems to operate correctly but the print is uniformly under-extruded throughout the entire print, first check your printer settings in Cura – make sure that the printer configuration has the filament diameter set to 1.75 mm instead of 2.85. Find the printer settings in Cura preferences in the printers category, then click on the 'machine settings' button while your printer is selected. If/whenever you upgrade Cura you should check this parameter again; Cura likes to reset it to 2.85mm.

Occasional jamming/under-extrusion: If the extruder jams periodically – you hear or see the extruder motor skipping – there are a few possible causes.

- Make sure the hotend cooling fan is running; if the upper portion of the hotend gets warm it can lead to jams (heat creep).
- Make sure the retraction distance in Cura is not excessive – around 2mm is usually best, but Cura defaults to 6.5mm. Excessive retraction pulls hot plastic up into the cool part of the hotend, where it can freeze and jam.
- Old PLA filament, or low-quality PLA filament, is prone to jamming. PLA picks up moisture from the atmosphere over time, and as it gets more problematic. In our dry Colorado climate you can expect PLA to last at least six months after it comes out of its packaging; if you want a roll to last a year or more you should store it in a sealed bag or box with dessicant.
- If the extruder motor is cold or cool to the touch even after running a print, you can adjust the motor's current higher, which will allow it to push the filament harder. It is difficult to describe how to adjust the motor current in writing – look for tutorials on adjusting A4988 driver current online. For these boards, the motor current will be 1.25 times the voltage measured between ground and the potentiometer on the driver board. Make small adjustments and check the motor temperature after a print before increasing current further; it is fine if the motor gets warm to the touch but it should not get uncomfortably hot.

Layer shifts: This printer is somewhat more susceptible to layer shifts in the X direction if the printhead catches on something due to the smaller-than-usual X axis motor. If possible, orient your prints such that any steep overhangs run in the Y direction. You can also enable Z hop in Cura for difficult cases; only 0.5mm or so of hop should be needed.

Occasional printing slowdowns/pauses: If you print files with many small triangles, you might find the printer occasionally slowing down or pausing while printing. This is caused by the controller being unable to process lines of G-code fast enough. (Each triangle in a layer translates to one line segment and one line of G-code.) Possible cures include:

- Increase the maximum resolution setting in Cura (default is 0.01mm, try going to 0.03mm); that will force Cura to combine very short moves and reduce the number of lines of G-code.
- Slowing down the printing speed helps the controller keep up.
- Use a program such as Meshmixer or Meshlab to reduce the number of triangles in the .stl file.
- Upgrade the printer's controller to one with more processing power. You can add a Raspberry Pi to the existing controller and run Klipper firmware; or replace the Arduino board with a Re-

ARM board (while keeping the RAMPS board); or replace the controller electronics altogether with a Smoothieboard, MKS-Sbase, SKR 1.3 motherboard, Duet, etc.

Upgrading/going further:

Now that you have your own 3D printer, you will probably find that you want to do more than you can with this printer. The following are some suggestions for various improvements/applications:

I want to print high-temperature materials (ABS, Nylon, etc.): Your printer can print ABS+ without any modification. You will get better results from adding side and bottom panels to the printer to eliminate drafts. If you plan to print lots of ABS, consider replacing the Buildtak bed surface with PEI, which will stand up to ABS temperatures for longer. Higher-temperature materials than ABS may require a full enclosure, which may in turn require an upgraded hotend cooling fan to prevent jams.

I want to print filled PLA (glow-in-the-dark, metal-fill, wood-fill, carbon fiber-fill, etc.): These materials are abrasive and will wear away a standard brass hotend nozzle over time. If you plan to print a lot of these materials you should have a spare nozzle on-hand, and you may want to consider getting a steel or other hardened nozzle.

I want to print large things faster: The standard Cura settings come nowhere near maxing out the speed of this printer. You should be able to reach 100mm/s print speeds or more, but expect to have to adjust your hotend temperature upwards somewhat and adjust other settings as well – this takes experimentation. You might also want to consider getting a nozzle with a larger opening, such as 0.8mm. Larger nozzles allow you to extrude plastic more quickly, but will reduce the detail in your finished prints. See E3D's V6 documentation to see how to replace nozzles.

I want to print small, detailed models (miniatures, etc.): You might want to consider installing a smaller nozzle, such as 0.25 or 0.3mm, to allow the printer to print finer details. Printing thinner layers also leads to smoother, more detailed prints. Choose layer heights that are multiples of 0.01mm, as that is the distance the bed moves from a single full step of the Z motor. You might also consider switching to a geared extruder or a 0.9 degree stepper motor for the extruder, as either change will allow the extruder to push small amounts of plastic more precisely. For geared extruders, I personally prefer the Voron Mobius extruder, but that is a fairly complex build.

I want to print larger models than fit on this printer: You can use Meshmixer or Blender to cut models into parts, print them separately, and glue the parts together. Or, you need a larger printer! You can use this printer to print parts for a second printer – consider a Voron, RailCore, or Hypercube Evolution (NOT the original Hypercube); all are considered to be good. Or you may wish to purchase a new printer. I don't have much advice there, except to avoid Anet A8s (they are fire hazards).