CHAPTER 11

Troubleshooting

As you have seen in the book to this point, 3D printing is still a bit of an art. To go back to our cooking metaphor from Chapters 5 and 6, you might have a recipe that sounds great and still wind up with a cake that comes out of the pan in ten pieces. The preceding chapters have laid out what can go wrong for many common issues. This chapter covers some more systemic problems that might result from an unfortunate confluence of several smaller problems, as well as problems due to the environment around the printer.

The beginning of Chapter 9 noted that 3D printing requires a little bit of programming knowledge and a little bit of shop class. In Chapters 9 and 10 we cautioned those of you who came into this from the programming side to be a little wary of your limitations. That caution applies in this chapter, too.

However, this time around the reverse is also true: if you came into this as a strong mechanical shop-class wizard, be aware that debugging a system with a lot of software involved is a little different from figuring out what is wrong with something purely mechanical (or even electronic). Be very systematic about keeping track of what you have changed and what was different after you changed it. Otherwise, you may go in circles.

This chapter covers a few common issues that often are caused by a mix of hardware, software, and environmental factors. First, we look at the typical signs of trouble and then cover how to go about analyzing and fixing the problem—or preventing it in the first place. Many of the issues in this chapter are strongly affected by your printer's design, so take any suggestion here with a grain of salt and read your manufacturer's instructions first before taking any advice in this chapter.

Clicking or Grinding Noises

One of the first signs of trouble with a 3D printer is an unfamiliar noise. You will know you are an experienced 3D-printer user when certain noises make you immediately execute a high-speed dive across the room toward your printer. Because prints can take many hours, listening to your printer is often the best way to monitor it (with an occasional glance, of course).

A filament-based printer has to drive the filament into the extruder somehow. Clicking noises usually means that the filament is not going into the nozzle smoothly (or at all). Here are some possible causes:

- The filament might be hung up on the spool or catching on something (see the section "Handling Filament" in Chapter 7).
- The nozzle might be clogged (see the section "Clogged Nozzle" later in this chapter).
- The filament might not have been seated correctly when it was inserted. (This applies at the beginning of a print; if the ticking starts mid-print, this is unlikely.)
- The filament might be too wide to go into the nozzle (check it with calipers as discussed in Chapter 7's section "Filament General Issues").
- Something might be hitting one of the fans on your printer. Check that the fans are clear.

Sudden grinding noises usually mean that the print came loose and is jammed under the nozzle. This warrants a sprint across the room to turn off the printer, because the jammed part can exert some unexpected forces on the platform and its supports. Get your printer into a safe condition by stopping the print job. Then manually walk the extruder back out using the techniques in the section “Backing Out of a Bad Situation” in Chapter 6. You may have to re-adjust the build platform (see “Printer Internal Alignment Issues” later in this chapter).

■ **Caution** In addition to being sensitive to abnormal noises, be aware of what a “normal” print run smells like. A burning smell is never a good thing around electronics. Be able to distinguish problems from, say, the smell of ABS printing.

Environmental Issues

3D printers (and 3D-printing filament) are sensitive to drafts, temperature, humidity, and dust. Good printer design can minimize some of the impact of these factors, but you also should be thoughtful about where you put the printer. Some people think of their 3D printer as an item to have in a computer room, whereas others think of it as a machine tool to be kept in a shop. If you keep it in an office environment, you will need to figure out a good way to be certain there is enough ventilation. If you are in a shop, you may have some of the issues noted in the following sections.

Drafts

The 3D-printing process, particularly ABS prints and others built on heated platforms, is very sensitive to drafts. When you arrange your ventilation, do not have a big fan blowing *on* the printer, which may defeat its temperature management. Have the fan pulling air away from the printer—for example, have the printer on a table and a fan on the floor blowing air away. If your ABS prints are pulling away from the platform, check for drafts.

■ **Note** Having a fan blowing directly on the print can improve PLA prints. However, if you are printing it on a heated platform and/or do not have much insulation on your hot end, your printer may need more power to maintain those temperatures.

Some printers optimized for PLA have a small fan to cool the PLA more quickly to improve print quality, but that is different because PLA cooling fans are generally shrouded to keep them from blowing on the nozzle. If the nozzle is well insulated or otherwise shielded, a large fan off to the side can be just as good.

Ambient Temperature

If your printer is in a space that undergoes significant temperature swings, the temperature on the heated bed and/or extruders should still maintain the commanded temperature. However, we have heard anecdotally that very cold rooms and very warm ones have resulted in print-quality issues. If your printer is a true RepRap and has 3D-printed parts, avoid temperatures warm enough to soften the material that was used in the parts. Do not keep a spool of filament (particularly PLA) in direct summer sunlight in a hot parked car.

Humidity

Humidity can be more of an issue than ambient temperature. Most filaments are strongly hygroscopic—they absorb water from the ambient air around them. If damp filament is heated, the moisture can bubble and make the filament extrude unevenly, which can cause discoloration and poor inter-layer adhesion. If you live in a very humid climate, keep your filament tightly sealed in a plastic bag with its desiccant when you are not using it. Avoid leaving your printer outside overnight or allowing water to condense on it.

Dust

Is your 3D printer a machine tool or a computer? It is a little of both. If you are going to have your printer in a shop environment, you need to keep the printer as clean as possible. In particular, you want to keep the *filament* as clean as possible so that dirt is not carried down into the nozzle where it will clog it.

If you are working in a dusty environment, you can take a piece of dust-free cloth (a paper towel will do) and use a binder clip to secure the cloth over the filament, as shown in Figure 11-1. The clip should not be gripping the filament; the filament should slide freely.

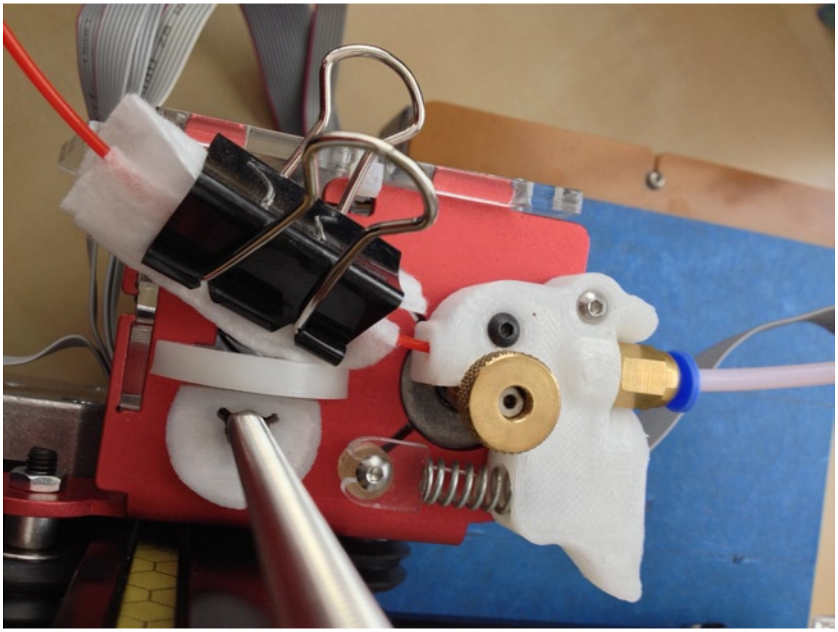

Figure 11-1. *A dust-catcher on the filament*

Be sure the clip and cloth do not interfere with the extruder mechanisms or the movement of filament, or get too close to the hot parts of the printer. The cloth will wipe off the filament as the filament is dragged through by the extruder mechanism. If the cloth gets dirty, you will know you needed it!

The filament will drag the dust-catcher next to the extruder drive. (This printer's drive gear is the round brass object near the center of Figure 11-1.) A dust-free cloth is a particularly good idea if your printer shares space with ceramic, wood, or glass shops, all of which generate a lot of fine dust. Do what you can to keep the printer clean, particularly when it is not in use. If your nozzle does clog, see the section "Clogged Nozzle" on unclogging your extruder later in this chapter.

Printer Internal Alignment Issues

A critical characteristic of 3D printing is having the printer's parts align correctly *to each other*. This requires that the printer be both *square* and *tram*. *Squaring* refers to ensuring that the axes are at right angles to one another, whereas *tram* and *tramming* are used specifically to refer to adjustment of the *build platform* to ensure that it is parallel to the *x*/*y* motion of the machine.

A 3D printer that is square but not tram will have the axes at right angles, but the nozzle will not maintain the same distance from the platform as the *x* and *y* axes move. A machine that is tram but not square will usually print fine, but the angles will be wrong so that rectangles will become parallelograms and circles will become ovals. There are several ways in which the printer can get out of tram. For a Cartesian printer to be in tram (see Chapter 2's section on types of printers), the *x,y,z* axes are all square to each other and then the platform is in a plane parallel to the *x* and *y* axes. For other geometries (like Delta machines), this alignment is more complex.

If you suspect that your machine axes are out of alignment because prints that should be rectangles are skewed or the width of an extruded line is not consistent from one end of the platform to the other, first check that the axes are square relative to each other (use a carpenter's square or, in a pinch, any other stiff, accurate rectangular object). If they are not, use whatever adjustments your printer manufacturer provides. Once you have done that, you should check the alignment of the build platform relative to the axes, hereafter referred to as *tramming the platform*.

■ **Caution** Some manufacturers refer to "leveling" a platform. However, do *not* use a bubble level to align your platform. If you have your printer on a table that is not level, but your printer is otherwise square, changing any printer adjustment based on a bubble level would most likely not fix the problem. We use the word *tram* to draw this distinction about internally consistent alignment of the printer axes relative to the platform versus level relative to gravity. Note that the axes are imaginary lines in space; normally, the frame of the printer will be lined up with the axes, but in principle software can compensate if they are not. Be sure you understand what your printer does and does not compensate for before attempting to fix problems.

If the build platform gets out of tram, either you can wind up printing in air in places where the platform is lower than expected or mashing the first few layers on parts of the platform that are too close to the nozzle. Prints that do not stick in one corner and are a bit mashed (with wider-than-normal extrusion lines) on another part of the platform are symptoms of this problem.

A typical way to fix this for most printers is to home the axes manually (see the section "Manually Controlling Your Printer" in Chapter 6). Then take a piece of paper (a sticky note works well) and move your extruder in *x* and *y* over each of the adjustment screws for your platform. For most printers, the piece of paper should be able to get between the platform and the nozzle, but there should be a bit of resistance. There is a device called a *feeler gauge* that does the same thing, but do not use one—a feeler gauge is made of a metal that is harder than your nozzle and might damage it.

As a quick test, you can print a small item with a big skirt (a skirt that is near the edges of the platform) to check platform tram. If the skirt does not lay down evenly, then you need to try again.

Some machines have "auto-leveling" features. For the most part, these cannot allow for the axes being out of alignment with each other, so if your auto-leveling is failing, take a good look at the overall frame geometry and check it with a square if you can.

Prints Not Sticking to the Build Platform

Getting a print to stick to the build platform can be challenging. Chapter 7 covered techniques for each type of filament material, and Chapter 8 walked you through some examples of the "right way" to analyze how a print job should be arranged to stick to the platform. However, often failures can be more instructive than success; this section describes in detail an example that does everything the *wrong way* so that you can see the result.

Figure 11-2 is an object that looks like a squared-off tornado created in OpenSCAD. It is designed to fail quickly in mid-print. This design has several problems: it has a tiny contact area with the build platform and slants off at an angle. It needs a raft, or at least a brim, and some support. However, it was sliced without adding any of those things, to create a worst case.

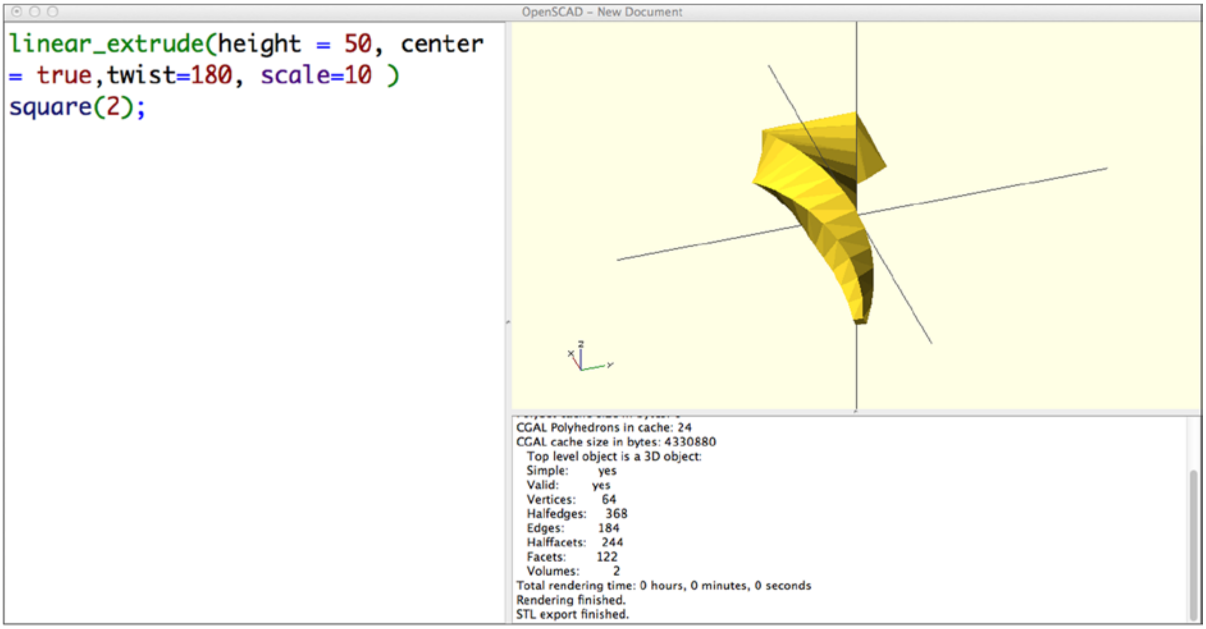

Figure 11-2. *The "designed to fail" print in OpenSCAD*

Figure 11-3 is the result, stopped part way through the print. The "tornado" managed to adhere to the platform for quite a while before it toppled over or was knocked over by the extruder. Then the extruder was printing in air, which created all the stringy stuff. The print was terminated prematurely.

Figure 11-3. *The "designed to fail" print after it was shut off part way through*

If your prints turn out like Figure 11-3, here are some factors to consider:

- Does your print have enough contact with the build platform? (See the "Brims" and "Rafts" sections in Chapter 5.)
- Is your print overhanging enough that either you should build it upside-down (that might have worked here) or build it with support? (See the section "Supporting and Orienting a Model" in Chapter 5.)
- Were you using the right build platform for the material? (Here, we were: this was PLA on blue tape, but the bad geometry overcame the correct settings.) See the appropriate Chapter 7 section for the material you are using.
- Was the platform aligned correctly (as described earlier in this chapter in the section about machine alignment)? If the build platform and extruder are too far apart, the first layer may print in air, and therefore all the subsequent ones have nothing to print on. (This print was created on a correctly trammed machine, but again extreme enough errors in model placement and slicing settings will cause a failure like this. If it had been a tramming issue, the print would not have partially survived as this one did.)
- Were the temperature of the nozzle and the build platform (if it is heated) set correctly? (See Chapter 7's discussions of these requirements for each material.)

If all of these factors check out, consider that the print may not be printable on your machine as-is. It may need support, a raft, or something else to get it to stick.

Clogged Nozzle Solutions

One of the more common problems with a 3D printer is that the printer stops extruding plastic because the extruder nozzle is clogged. The nozzle hole is small and it can fairly easily be blocked with debris that was embedded in the filament, dust, or plastic that got too hot and scorched or burned.

■ **Note** The nozzle-unclogging processes that follow are adapted from an online Deezmaker troubleshooting procedure written by Rich Cameron. The original, with any updates since the writing of this book, can be found at `www.bukobot.com/nozzle-cleaning`. (The photos in Figures 11-4 through 11-7 were taken by Cameron.) Although the instructions there were written for Deezmaker printers, they apply generally for any printer with a nozzle rated to the temperatures suggested here. If your nozzle is not rated to 240 degrees C, you should not use this procedure.

Cold Pull

In the past, the only way to get a clog out of a nozzle was to take the entire hot end (nozzle, barrel, and heater block) apart or take it off and put it in a solvent. Because that is not very convenient, and taking the hot end apart can damage it, the *cold-pull* technique was developed. A cold pull starts with inserting filament in the nozzle just as if you were going to print with the filament.

Instead of using the usual extrusion temperature, though, which would melt the filament, a cold pull uses a lower temperature. The temperature is warm enough to allow the plastic to stretch enough to pull away from the sides of the barrel so that it doesn't seize up entirely, but cold enough so that the filament stays in one piece. Usually, any debris in the nozzle will then come out with the filament. We estimate the best temperature to use for this *cold-pull temperature* in the following description.

The cold-pull technique works best with printers that have polished-smooth stainless steel nozzle barrels. It also works for nozzles that have a polytetrafluoroethylene (PTFE) internal coating. The cold-pull technique has been successfully done with ABS (cold-pull temperature of about 160–180 C). PLA is much more difficult to work with, but a cold-pull temperature of 80–100 C will sometimes work. Nylon 618 filament (pull temperature of 140 C) is far easier and more reliable to use for this purpose due to its strength, flexibility, and low friction. The cold-pull technique works like this:

- To begin, remove as much of the plastic that you've been using as possible. To do this, you can attempt a cold pull with ABS or PLA with the temperatures listed previously.
- Next, heat your nozzle to 240 C so that it can thoroughly melt the nylon and push the nylon filament in. Attempt to extrude the nylon slowly. Most clogs (especially those caused by accumulated dust) will not actually block the nozzle entirely, but will be pushed into the nozzle and stop it when the nozzle pressure increases and then float up out of the way when left to sit.
- If you do not have a hard clog (usually a solid foreign particle lodged in the nozzle), a slow, pausing extrusion should allow you to purge the old printing material.
- Once nylon starts coming out of the tip, you can begin cooling your nozzle to the pull temperature. See Chapter 6 for information on manually controlling your printer to see how to extrude some filament by hand.
- Finally, pull the filament out of your printer manually, if your printer mechanism allows it, or by retracting it using a host program.

■ **Note** The temperatures specified in the preceding list are maximum temperatures—temperatures above which the plastic is unlikely to come out solid. For best results, you should always pull the plastic at the lowest possible temperature, and it may help to cool the nozzle well below this temperature and then continually attempt to pull it as the nozzle heats up again.

Figure 11-4 shows some examples of cold pulls. If you pull the nylon out and the surface is rough, dark, discolored, or has black spots around the sides, this indicates that there is residue of overheated or carbonized, burned plastic in the nozzle. If you see this, you should clip off the end and repeat the process until the nylon comes out smooth, clean, and mostly white. Figure 11-5 shows similar examples from with a nozzle that was clogged with overheated plastic, rather than burned plastic.

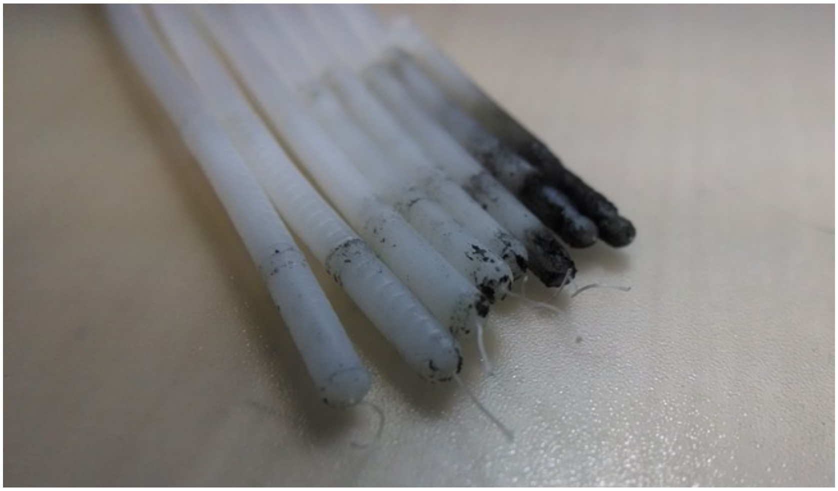

Figure 11-4. *Burned residue on cold-pulled filament*

Figure 11-5. *Nylon with residue of overheated plastic*

■ **Tip** You might want to do cold pulls from time to time as a preventative measure if your print quality seems to be degrading.

Wire Brush Bristle

Usually a cold pull will clean a nozzle. However, if not, there is probably a hard bit of debris in your nozzle. One way to poke out a hard clog is to take a bristle from a wire brush and push it into the nozzle. The bristle typically does not clear the nozzle. It only dislodges the clog so that hot plastic can flow past it and then harden around it so that it can be pulled out. Occasionally, this will also break up the clog enough to allow it to be pushed out of the nozzle in smaller pieces.

Figures 11-6 and 11-7 show this process. Pull a bristle out of the brush with a pair of pliers and poke it into the nozzle. Then try a cold pull to see if you can pull the debris out. Note that the nozzle has to be hot for this maneuver to work, because otherwise the nozzle will be clogged with solid plastic. Be careful not to get your fingers near the hot nozzle.

Figure 11-6. *Extracting a single bristle from a wire brush*

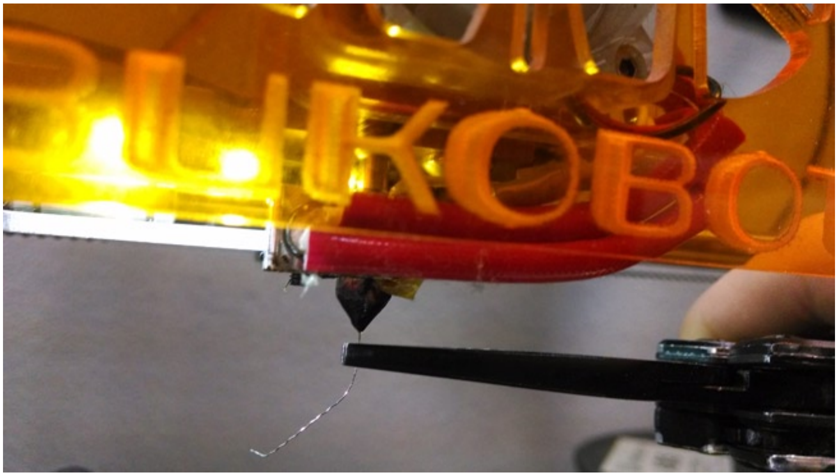

Figure 11-7. *Using the wire brush bristle to clear out the nozzle*

Extruder Drive Gear Teeth Clogged

If you have been trying to extrude filament into a clogged nozzle, the drive gear may have been digging into the filament and clogging up the gears with ground-up filament. Check this too if you had to clear the nozzle. Otherwise, filament may not feed well, and you might think that the nozzle is still clogged. Pick and/or blow it off carefully and be sure you are not blowing plastic dust someplace where it will cause other problems.

Eliminate Stringing

Sometimes a print will come off the machine with strings everywhere. Often this means that the retraction settings (under Printer settings in the slicing programs—see Chapter 5) are not correct. When a printer is extruding and has open spaces in a layer, printers pull the filament back so that stringing does not occur. For a Bowden extruder (a tube between the drive gear and the nozzle), 2 mm or so is enough, although with long Bowden tubes you may need to pull back as much as 10 mm. For direct-drive extruders (where the gear is right by the nozzle), generally 2 mm or less is more appropriate. If you have problems with stringing, you may want to increase the retraction length slightly. Figure 11-8 shows what it looks like if retraction is turned off completely versus being set correctly for a particular printer.

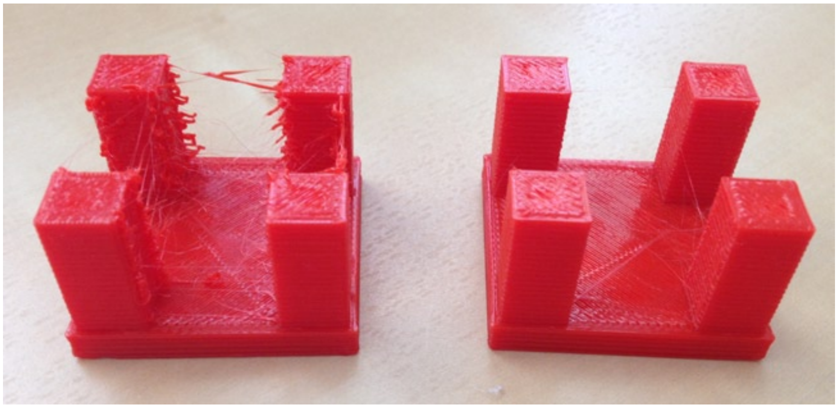

Figure 11-8. *A miniature table printed (upside-down) with retraction disabled (left) and set correctly (right)*

Software Upgrades

If you are using an open-source printer, be cautious about upgrading to a new version of any of the software packages as soon as they are released. Sometimes it is best to wait a bit and read the user reviews. Many of these programs have a terrific volunteer base (or a small paid staff), but their ability to test on all platforms usually is not exhaustive. If you are in the middle of a critical set of prints, wait to upgrade until having problems with the new software would not be disastrous.

■ **Tip** If you start changing slicing settings, be sure to write down what you changed and just change one setting at a time to see the result. Otherwise, reconstructing what does and does not work can be very hard.

Summary

This chapter covered some common symptoms of trouble during the 3D-printing process and ways to resolve, and in some cases prevent, these problems. This chapter reinforced some of the suggested best procedures described in Chapters 5 through 8 and left you with a bit more understanding of the 3D printer as an interacting hardware and software system.