CHAPTER 10

Large Prints and Post-Processing

For many applications, a part works fine as soon as it comes off the 3D printer, perhaps after picking off a bit of support. However, there are circumstances that require a bit of post-processing, either because the part had to be made in multiple pieces or because it needs to be smoother than the printer can achieve.

This chapter describes how to deal with a model that is either computationally big (an STL file over 50MB or so) or that, if printed, would be physically too big to fit in a printer's build volume in one pass. This chapter also covers how to lower the resolution of an STL file to make it more manageable. You saw how to divide a model in software back in Chapter 5's section on avoiding support by cutting a model into pieces. The purpose is different here, but the process is the same. In this chapter we will talk a bit about gluing together the parts after the print is done.

A printer will also leave behind layer lines, scars from support, or the odd bit of stringing across open areas. This chapter also talks about how to post-process a model by sanding, painting, dyeing, or chemically smoothing it.

3D printing is a bit of a crossover discipline; it's part computer science and part shop class. Any one person may not come into the field comfortable with both aspects. If you are entering this space from the software side, team up with a shop teacher, a sculptor, or someone else who routinely builds real physical objects. The techniques in this chapter are likely to be familiar to them.

Printing Computationally Complex Objects

Sometimes the 3D model of an object is very complex, even if it is specifying a physical print that fits comfortably into the build volume of a printer. For example, this issue arises in some scientific visualizations, as we will talk about in Chapter 13's discussion of 3D-printed models of complex molecules. It is tempting to want to keep all the detail, but the reality is that a consumer-level printer will probably not reproduce detail of a very complex model anyway.

If the model is going to be both physically big and very complicated, cutting it into pieces and slicing and printing those pieces separately can lower the STL file sizes of the pieces to plausible levels.

Tip An STL file more than a few tens of megabytes is probably too big both for most slicing implementations to handle and for most printers to reproduce. About 50MB is a practical limit—less if you run the slicing programs on an outdated computer. For a larger file, consider the two techniques discussed in this section.

If you find yourself with an STL file that is too big to work with, you can use the MeshLab program to re-mesh the model with fewer triangles. MeshLab is available at http://meshlab.sourceforge.net and is discussed in several places in Chapter 4. MeshLab is a powerful program but can be a little daunting to navigate because of its terminology. Because it is open source, it can change at any time, but as of this writing the following process will re-mesh an object:

- Open MeshLab and import a "mesh" (i.e. your STL file) under the File menu.
- Then click Filters ➤ Remeshing, simplification and construction ➤ Quadratic Edge Collapse Decimation. (You can see the issues here with somewhat non-intuitive menu items for the new user!)
- In the dialog box that pops up you can try cutting the number of surface triangles relative to what the default displays and experiment with the percentage reduction (from no reduction to 100%). A value of 0.5 will cut the number of triangles in half. You can see what you can get away with based on how the rendering in the program looks. If you have lost an unacceptable amount of detail, don't save the new mesh; import it again and start over.
- When you like the result, you can export a new STL file.

Alternatively, you can cut the file into pieces as described in Chapter 5 and then glue together the physical pieces after they are built. You will learn about that in the next section.

■ **Note** Cutting a file into pieces to reduce the file size (rather than to reduce the dimensional size) will reduce the amount of data that the slicer needs to process, but it can still result in a G-code file with segments so short that they will execute in significantly less time than it takes to transmit and process them. This will result in the firmware slowing down or even pausing between segments, and these unexpected speed changes will result in print artifacts. Printing from an SD card rather than over USB mitigates these issues, because reading G-code from the SD card is faster than transmitting it over the USB connection. (Thanks to Rich Cameron for this point.)

Printing Physically Big Objects

Consumer-level 3D printer build areas are for the most part relatively small, because it is a lot harder to manage temperature control and precision of the build platform and the rest of the machine's structure. As a result, most consumer printer build areas are under 200 mm or so in any dimension, and many are way under that. This means that if you want to build something bigger than that, you are going to have to cut the object into pieces in software, print the parts, and then somehow construct the final object out of those pieces.

If you want to print an object that will not fit within the build volume, you can cut it into pieces. For example if the object is long and skinny, you can cut it into a few objects that you can then arrange on the build platform and print all at the same time. Otherwise, you will have to print the object using multiple runs. In either case, you will need to glue or assemble the piece afterwards.

Objects That Are Too Long for the Build Platform

If you have a long, skinny object that will not fit within the longest build dimension of your printer, first check to see whether it will fit diagonally on the platform. You will also want to keep the orientation that will minimize support, so there may be some tradeoffs there. If you are considering printing diagonally in all three dimensions, it may be easier to cut the object in half and later glue two pieces that would both lay flat rather than pick off a lot of support.

If you do need to print a thin but long object (for example, a chopstick), you probably will be able to arrange the pieces on the platform next to each other and print them simultaneously. If, for instance, you were trying to print a tall, skinny tower, you could cut it into several pieces, lay out the pieces on the build platform in one or more groups, and then glue the pieces together.

Objects That Are Too Big in More Than One Dimension

If the object you are trying to print is too big in more than one dimension, you will need to split it along two axes—that is, into at least four pieces, with a lateral slice and a horizontal slice. If any parts require particular precision (if there will be critical joints, for instance), consider where your printer has the highest precision and arrange cuts so that those pieces will fall there. In most cases, the best resolution is at the center of the print bed.

■ **Tip** Even when the full part is too large in two or even three dimensions, it may be possible to divide it into just two or three pieces by judiciously cutting and then orienting the pieces. For example, a letter H that needs to be printed slightly larger than the build platform (in both *x* and *y*) can be cut into two halves, each of which may fit diagonally. A pyramid-shaped object that is too tall for the print volume and too wide only in one dimension may be cut into three pieces (the top and two halves of the base). An L-shaped object may be cut into four quadrants, one of which will be empty. Be careful if your printer does not have consistent resolution throughout its build area (most Cartesian printers do). (Tip courtesy of Rich Cameron.)

Chapter 5's section on cutting to avoid supports describes the process of cutting a part in two. To cut into multiple pieces, follow the same process but cut the piece in half and then cut the halves again (or into however many pieces make sense to make the part fit into your printer's build area). The general principles apply. Depending on which program you use to cut up the model, you might be able to slice along several axes at once, or you might have to explicitly make one cut at a time, save the pieces, and then cut them in turn.

■ **Caution** The techniques discussed in the rest of this chapter are more similar to a "shop" environment (or a chemistry lab) than a computer lab environment. If you are not used to this type of environment, work with someone experienced first to learn safety procedures. Use eye protection and gloves, be sure your space is well-ventilated, and follow the chemical manufacturer's instructions.

Gluing the Pieces Together

Once you have printed the pieces, use glue that works on plastics. Cyanoacrylate adhesives ("superglues") work pretty well on PLA and ABS. Nylon is difficult to glue with any adhesive appropriate for home use.

■ **Caution** Before using any glue, read the manufacturer's instructions and use glues in well-ventilated areas. You might want to try out a particular glue on a few scrap pieces first to be sure that it does not discolor your material. Glues may dissolve pieces a bit, which allows the "welding" process described later in this chapter.

Using an Acetone Slurry

If you are printing in ABS, there's another alternative for adhering pieces to one another (but see the Caution that follows). Acetone will melt ABS and so can be used to weld one piece of ABS to another. Some people put a little bit of acetone in the type of bottle used for nail polish (ones with a small brush) and add the skirt, support material, or other scrap from the print. The acetone will dissolve the scrap into a slurry that will weld the pieces together without melting the edges of the print too much. Or, if precision edges are not important, you can use a drop of acetone to bond the parts together. We will talk about using acetone to smooth a print later in this chapter.

■ **Caution** Acetone is flammable and volatile. Its vapor is invisible and heavier than air. The vapor can pool if you are in an unventilated area and cause a fire or explosion. Use it in a well-ventilated area without open flames or sparks. Follow the cautions on the manufacturer's label.

Acetone welding only works for acetone-soluble plastics, such as ABS, MABS, HIPS, and polycarbonate. Most other 3D printing filament materials (such as PLA and nylon) are not acetone-soluble. Some PLA formulations (depending on additives) may partially melt, whereas others have been known to warp and crack when exposed to acetone.

■ **Tip** When you paint acetone on a part, the acetone evaporates, and you will get a bit of condensation from the air. This can make the surface of the object a bit cloudy (you will see this in Figure 10-2). If you are just "welding," then clouding does not matter, but if you are smoothing (described later in this chapter as *chemical smoothing*), it is an issue. On the other hand, if the acetone is too warm, the surface will bubble. Keeping the acetone at about 50 degrees C (120 F) works well. You can put your acetone nail-polish bottle in a shallow bowl with an inch of warm water in it. If your printer has a heated build platform, you can set your heated platform to 50 degrees and put the (glass) bottle on it (see "Manually Controlling Your Printer" in Chapter 6). *Never* microwave acetone or ever place it anywhere near a stove; an explosion or fire can result.

Sanding, Chemical Smoothing, Painting, and Dyeing

Filament-based 3D printers will always produce fine layer lines in printed objects. You can think of them in one of two ways: as inherent in the medium (like brush marks on an oil painting) or as a problem that needs to be resolved. If your application falls into the latter category, this section gives you some ways to get rid of those lines and to color your printed part other than by choosing colored filament. We will talk about sanding and chemical smoothing to get rid of layer lines, and then about issues in painting and dyeing finished prints. Thanks go to the good folks at Deezmaker for the techniques in this section.

Sanding

A bit of sandpaper applied gently to PLA prints can significantly smooth the surface. Just use a piece of sandpaper and sand it by hand—any mechanical sanding is likely to melt the plastic. ABS can be sanded but tends to whiten a bit in the process, the same way it bruises when bent. Acetone will reverse this discoloration, so sanding can be used in conjunction with acetone smoothing to reduce layer lines while maintaining sharper features that would be rounded off using acetone alone to remove the layer lines. Nylon just comes out worse than you started if you try sanding it. Figure 10-1 shows a PLA piece being sanded by hand. A sanding block may also be useful, depending on the geometry of the part.

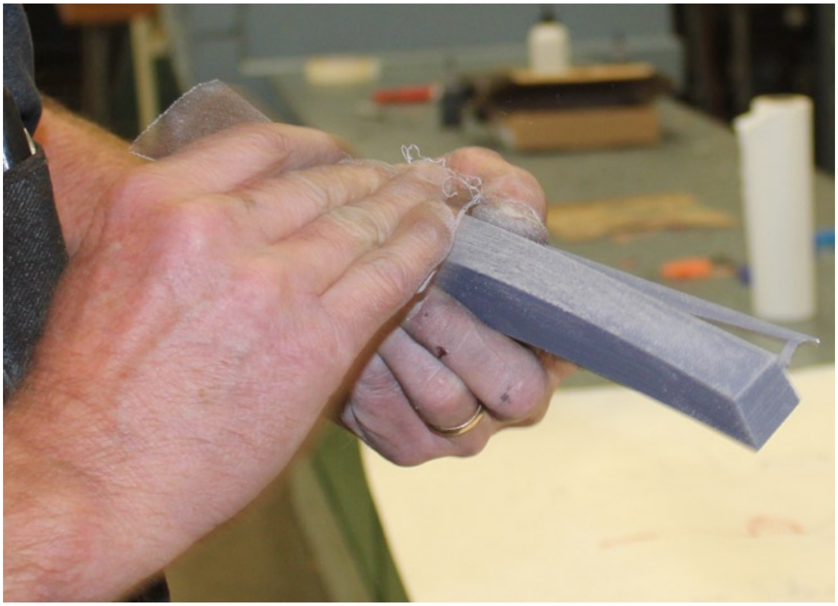

Figure 10-1. *Sanding a PLA piece. (This was part of the preparation for sand casting in Chapter 9)*

Smoothing and Bonding ABS with Acetone

In the earlier section "Using an Acetone Slurry," we talked about using acetone to weld ABS pieces together. Similarly, a little bit of acetone can be used to smooth an ABS piece using the same techniques (and cautions!) as those described previously. You may need to experiment a bit to see what works.

■ **Tip** A lot of smoothing techniques available online use acetone vapor and do-it-yourself devices to create and handle the vapor. It is this author's opinion that many of these are unwise and/or too hazardous for the intended general audience of this book; therefore, none of these is described in this chapter.

Do not use acetone on nylon or PLA; nylon will be unaffected by acetone. Unfortunately, chemicals that can smooth PLA are too toxic to be used in a home environment.

Figures 10-2 and 10-3 show an ABS piece (the Leopoly bear we met several times in Chapters 4, 5, and 8) that has been smoothed in places with acetone to remove scars from supports.

Figure 10-2. *The Leopoly bear from previous chapters, smoothed with acetone (front view)*

Figure 10-3. *The Leopoly bear from previous chapters, smoothed with acetone (rear view)*

As was the case in the Chapter 8 case study, where the bear exemplified a print with support, the bear was printed with his tail down. Because the support was at the back of the legs and under the tail, these were areas where we particularly would want to smooth the printed part. In this case, the acetone was a little cold, though, and as mentioned earlier in this section, the part got a little cloudy at the tail area.

Painting ABS and PLA

You can paint ABS and PLA parts with acrylic paint, like that available in a typical hobby store. If you need a multi-color print and you have a one-extruder printer, painting the object after the fact is a good workaround to produce the colors you need.

Dyeing Nylon

Although 3D-printed nylon may not look the same as it does in fabric form, it will take up dye in a similar way. Nylon filament typically is white, which makes it easier to dye. People have had good luck with household cloth dyes to dye a printed nylon object; check the label on the dye package to be sure that it will work on nylon.

Summary

In this chapter you learned how to work around some of the limitations inherent to the filament-based 3D-printing medium. We covered how to deal with prints that were too complex or physically too large, or that needed smoother surfaces than the printer alone could create. We looked at some suggestions of types of paint and glue to use, and noted some cautions about using chemical smoothing if you are new to the "shop" environment.