Knowing Your 3D Printers

3D printers are available that print objects in materials ranging from plastic to titanium. The medical research world has even created machines that can build structures from human tissue. All that separates these technologies is cost. Affordable desktop 3D printers are limited to printing in plastics. So that 3D printer you bought for $500 can't print you a new kidney. (Not yet, anyway, but don't hold your breath.)

The two main ways 3D printers are compared to each other are in terms of *cost per cm^3* (that is, cost per cubic centimeter of printed object) and *minimum layer height*. Cost per cm^3 is the cost of one cubic centimeter of printed volume; it's typically the number used when comparing the cost of one 3D printing technology to that of another. Minimum layer height is the thickness of one layer in the object. The smaller the height between layers, the smoother the surface of the finished object will look and feel.

Desktop 3D printers

Desktop 3D printers are still an emerging technology. When starting out with your first desktop 3D printer, don't expect it to work perfectly the first time out of the box. The machines haven't reached that level of polish yet. It helps to see a 3D printer as a tool you're learning to use, rather than as an appliance you just plug in. Before you dive into printing your own creations, spend some time going through the printer's training materials. Print a few test objects and get a feel for using the machine.

Desktop 3D printers tend to be based on tried-and-true technologies that have been used in industrial machines for decades. The two technologies that are available now are fused deposition modeling and stereolithography. The following sections give you an overview.

Fused deposition modeling (FDM)

FDM printers represent the mostly widely available type of desktop 3D printer. They build objects using solid plastic fed through a fancy robotic glue gun-style nozzle. FDM printers are inexpensive to buy, with desktop models ranging from $500 to $2,500. FDM-printed parts are done the moment the printer is finished; no secondary process is needed to finish or strengthen the object. The parts that FDM printers make can be as strong as parts from injection molding, and cost around $0.04 per cm^3. On the downside, FDM 3D printers have limited maximum resolution and tend to have a lot of moving parts, which can impact their reliability.

FDM printers consume plastic filament as they build objects. Filament is drawn into the printer, heated, and then fused to form the object. Filament comes coiled on spools that look a bit like brightly colored weed-whacker wire. Spools are available inexpensively through vendors online and are starting to become available in large office supply stores.

Examples of FDM 3D printers are MakerBot Replicator (`www.makerbot.com`), Ultimaker (`www.ultimaker.com`), 3D Systems Cube (`http://cubify.com`), Affinia H-Series (`www.afinia.com`), and Solidoodle (`www.solidoodle.com`).

Stereolithography

SLA 3D printers represent one of the oldest 3D printing technologies. They use a laser to build solid parts in a vat of liquid resin, selectively hardening the resin layer by layer. Desktop SLA machines cost between $2,500 and $8,000, but you'll hear ongoing speculation about a printer emerging at a price below $1,000. Stereolithography offers superior printing resolution, which gives finished parts a glassy smooth surface. The machines are typically small and nearly silent, with a few moving parts. Prints from an SLA machine typically cost around $0.15 per cm^3 and require some cleanup after printing. Also, due to the resin's toxicity, printed objects must be washed in isopropyl alcohol before handling.

Stereolighography's consumable resin is an amazing bit of chemical engineering. It's a liquid at room temperature until it's exposed to intense ultraviolet light, which hardens the resin into a solid plastic. Resin comes as a bottle of liquid; usually you have to buy it from the maker of your 3D printer to ensure compatibility. Resin has a set shelf life and needs to be stored with some care; it can begin to harden if left in sunlight.

Two examples of SLA machines are Formlabs Form 1 (`http://formlabs.com`) and B9 Creator (`http://b9creator.com`).

As this book goes to press, DLP (digital light processing) printers are emerging on the desktop 3D printer market, with price points from $3,000 to $5,000 but likely to fall in the near future. The technology is similar to SLA, but uses less material.

Do-it-yourself and kit printers

Both FDM and SLA printers are available as kits and open-source DIY plans. In general, these 3D printers are good options for people who are interested in exploring printer technology and modifying their printers. Kits are cheaper than buying a printer, but often less reliable. The saying is, "If you build it yourself, you fix it yourself."

Two examples of DIY and kit machines are RepRap (`http://reprap.org`) and Printrbot (`http://printrbot.com`).

Professional 3D printers

Professional 3D printers are typically housed in a dedicated department and operated by full-time employees. They are expensive, with prices in the hundreds of thousands of dollars, but offer capabilities beyond what is available in desktop models. As desktop 3D printing grows, and patents continue to expire, expect to see these technologies becoming cheaper and move into the desktop realm. Here's the current lineup:

- **Selective laser sintering (SLS):** *Selective laser sintering* uses a laser to melt and fuse a finely powdered plastic. SLS machines can reproduce fine detail without adding any supporting structure to the model. (By and large, lasers that can manage these feats are still relatively expensive.)
- **Inkjet, powder-based:** Similar to SLS machines, these 3D printers use a powder that is

selectively hardened by liquid glue sprayed from an inkjet printhead. The ink can be colored and mixed; this is one of the few technologies that can offer full-color 3D printing.

- **Inkjet, resin-based:** These machines are an expansion of stereolithography technology. They use an inkjet printhead to spray fine layers of a UV-sensitive resin, which is then hardened by a powerful UV light. Advertising materials for these printers talk about layer heights measured in atoms.
- **Paper-based:** These printers build objects out of copy paper by gluing the sheets together then cutting away the excess. The parts are immensely strong, and can be treated like wood after printing.

3D printing services

3D printing services offer access to professional-level machines on a pay-per-print basis. Cost is based on volume of material printed, and you receive your printed object by mail in a few working days. Each company offers detailed instructions on minimum part sizes, wall thicknesses, and feature size. Pay attention to these rules. The services know what their machines can do, and you don't want to be stuck waiting a week for a part that didn't come out right.

Two examples of 3D printing services are Shapeways (`www.shapeways.com`) and Ponoko (`www.ponoko.com`).

CAN I 3D PRINT IN METAL?

Yes, 3D printers that print in metal exist. But unless you live in a hollowed-out volcano and race fighter jets in your free time, you probably can't afford to use them. Based on selective laser sintering (SLS) technology, these printers use an electron laser to fuse powdered metals in a hard vacuum. The parts they make are fantastically strong and complex, surpassing anything possible with traditional manufacturing technology. Current uses for SLS in manufacturing include custom medical implants, military jet engines, and spacecraft components. So if cost is no object, you can 3D print in metal.

If you need your 3D printed part made in metal, use a 3D printing service bureau or take a part printed with an FDM or SLA printer to a foundry. At the foundry, a process called a *burnout* makes a cast-metal version of the part.

Using Your 3D Printer

Your new 3D printer's manufacturer will have documentation to help you learn how to use the printer and the software, so we don't go into detail on specific machines. The following sections touch on a few general concepts of 3D printing that are often not explained well. For a closer look at 3D printers, check out *3D Printing For Dummies* by Kalani Kirk Hausman and Richard Horne.

Print early, print often

Desktop 3D printing is cheap. Really, unbelievably, remarkably cheap. Cost for running a typical desktop 3D printer is about 60 cents an hour. Once you've started the printer, it doesn't need any more input from you, which frees you to continue working on the design. With such an amazing tool that's so cheap to use, don't be afraid to print constantly. Print your SketchUp model over and over as you improve it; doing so develops it.

3D printers are built to print; they don't like to stand idle. Print more often! You'll become an old hand at using your machine, and get to see your design as it evolves in real time.

TIP Test prints are great for catching errors and mistakes. Test prints are also a great way to document the evolution of a design. It's a good idea to save a version of your SketchUp model that corresponds to each file you 3D print. If the print has an unexpected error, you can look back at that file and understand what went wrong, without having to dig too deeply into the version you have continued to work on. Keep some general points in mind:

- Test prints that use your 3D printer's Low Quality setting will print faster, but with a rougher surface.
- Save test prints as the project goes on. They're a great way to show progress, and clients love to see a physical expression of the work being done.
- Many 3D printing plastics are recyclable or biodegradable. When you're done with your models made of this stuff, toss them in the recycling bin or on the compost heap.

Inside your model

3D printed parts are unique in the world of fabrication. After you've created the outside of your part, you also get to decide what happens on its inside. Usually you let the 3D printer automatically handle the inside of the part, filling it with automatically generated structure. It's also possible to model a part's interior structure to change how it behaves. For example, you might want to hollow out the center of a part to make it lighter or add space for internal components. Some possible variations include

- **Internal voids:** On SLS and stereolithography machines, a common cost-saving strategy is to build parts as a thin shell with an empty interior. The resulting parts have compromised structural strength, but are printed using as little material as possible. Be sure to include drain holes so the un-hardened material you saved can escape from the part.

- **Part density:** FDM printers automatically generate a structure to fill the interior of their prints. The density of the structure is controlled by a setting called fill, which is stated as a percentage. Parts with 100% fill will be solid plastic all the way through, and are as strong as injection-molded parts. Typically, FDM printers default to building parts at 10% fill, meaning that 10 percent of the interior is filled with plastic. Higher fill percentage means more plastic used — and longer print times. Generally speaking, anything over 40 percent is a waste of time and plastic.
- **Outer wall thickness:** Another FDM printer setting, this deals with the amount of material put into the walls of the object before starting the Infill. Look for a setting called Shells, which is the number of layers of plastic the printer will use to make the outside of the model. If you're having trouble with parts crushing, increase the number of Shells; otherwise leave this setting alone.
- **Flexibility:** Flexible materials are available on most 3D printers, usually by using a special flexible plastic filament. You can also make structures flexible by making them very thin with 0 percent infill.

Going beyond Basic 3D Printing

The more you use your 3D printer, the more you may find it affecting your SketchUp designs. Don't be surprised if your designs become a bit more ambitious and mechanically complex. This section of the chapter outlines some possible directions and factors to watch out for.

Designing parts that connect

So you cut your model into parts. Now you need a way to get it to all stay together after it's printed. In this section, you discover different strategies and features, mechanical and otherwise, that you can build into your model for attaching its parts together.

Tolerance and clearance

Before we get into a discussion about mechanical connections, you need to understand two more of those pesky realities that crop up when you move out of SketchUp's idealized environment: tolerance and clearance.

- *Tolerance* is the difference between the measurement of your part in SketchUp and the measurement of the part produced by your 3D printer. If you draw a 10mm cube in SketchUp and 3D print it, none of the measurements of the printed cube will be exactly 10mm. The differences are small — just a few tenths of a millimeter more or less — but they can cause problems if your 3D printed parts have to connect to parts that already exist in the real world. The 3D printer's manufacturer provides a number for the machine's tolerance as a plus or minus value, usually something like pm0.05mm. This is the *maximum* variation for that machine, and you should be aware of it as you work.
- *Clearance* is the extra space you need for parts to slide past one another. If you try to install a 10mm peg in a 10mm hole, you're in for a bit of a surprise when the two won't go together. The peg and hole can fit perfectly in SketchUp, but that's not what's going to happen in the real world where you have to contend with friction. The surface of the peg is so much like the surface of the hole that the friction between them will keep the peg from ever going in if the fit is too exact. You need to add a small amount of space called *clearance* so the two parts can slide past each other. How much clearance you use depends on how the part needs to move. A spinning shaft, for example, needs more clearance than a simple snap fitting.

You'll come to an inherent understanding of clearance and tolerance as you do more designing in SketchUp for 3D printing. As you use your 3D printer, you'll be able to find values that work well with your equipment.

TIP As a starting point, add 0.2mm of clearance to all holes and 0.5mm of clearance to any points of rotation.

Glue

Glue, shown in Figure 9-19, is the universal way to stick something to something else. Unfortunately, it's also the weakest and most unreliable method. Glue joints in plastics have very little strength and will tend to break under stress, in response to temperature change, or if you look at them funny. If your part is meant to be anything more than a visual model, use one of the other attachment systems.

- The plastics used in 3D printers — ABS, PLA, PVA, nylon, and PETT — all require special glues to bond. These glues are available, but must be ordered from an online retailer.
- Biodegradable starch plastics such as PLA have a crystalline structure that doesn't work well with liquid superglue. The glue tends to stay liquid and migrate unexpectedly across the surface of the part.
- When in doubt, use epoxy. Its messy stuff, but will stick to pretty much anything.
- There are no glues that will stick to the flexible materials that are available for 3D printers. To attach parts made of that stuff, you'll need to look into a process called thermal welding, which is way outside the scope of this book.

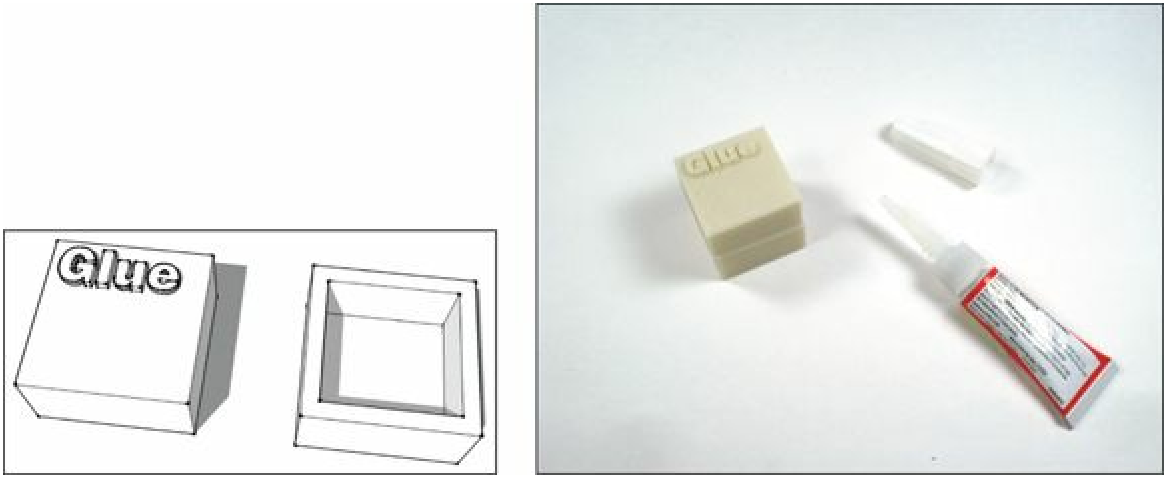

FIGURE 9-19: Works cosmetically, but don't expect much durability unless you use epoxy.

Snap fittings

Snap fittings are an awesome way to take advantage of your 3D printer. Properly designed, they'll let you pop your model together as soon as the parts come off the printer. Snap fittings can also be removable, so you can change out various parts of your model as the design evolves.

In SketchUp, you create snap fittings like those shown in Figure 9-20 by following two general steps:

1. Create the tongue with the Line and Push/Pull tools.
2. Create a matching capture point on the opposing part.

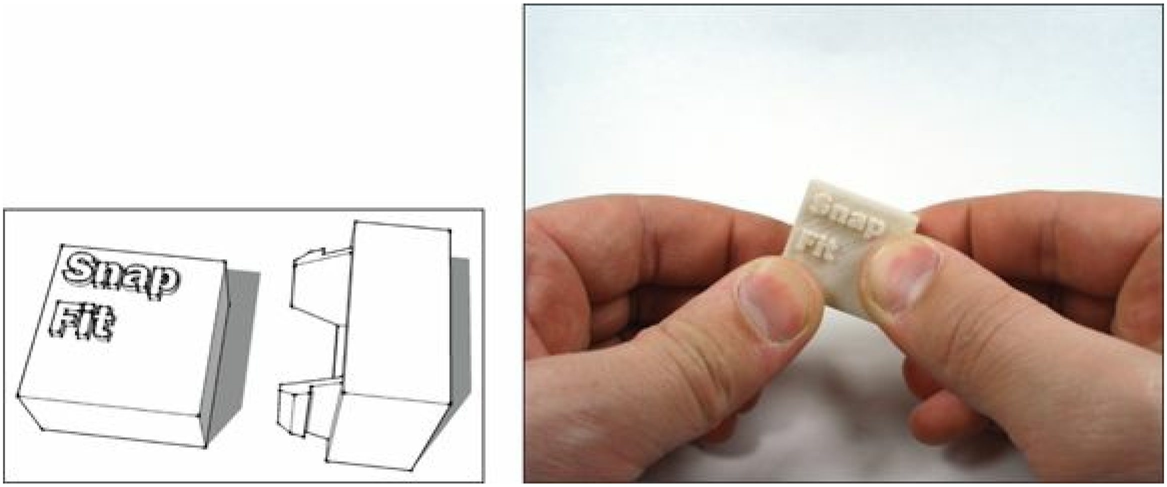

FIGURE 9-20: A snap fit joint is great for reusable connections.

REMEMBER As you create your snap fittings, keep these points in mind:

- Be sure to leave enough space for the tongue to bend backward as it slides into place.
- Include a clearance of between 0.2mm and 0.5mm, depending on how tight you need the joint to be.
- Always position snap fittings so they print horizontal to the 3D printer's build platform. Snap fittings printed in the Z direction, perpendicular to the platform, tend to break off.
- If you want your snap fitting to be re-openable, make sure that you provide a way for the tongue to disengage from the capture. This can either be a ramp in the geometry that forces the tongue backward as you pull on the joint, or an access point that lets you release the tongue manually.
- Don't make the tongue too thick. It has to bend for the joint to go together.

Press fit

A *press fit* is when two parts are designed with very little clearance between them. When the parts are forcibly pressed together, friction keeps the joint together with no additional hardware or glue.

To make a press fit like the one shown in Figure 9-21, use the Offset tool to create an outer lip on one side of the connection and an inner lip on the other side. Include a clearance of 0.2mm or less between parts, so you can still assemble them. Keep a few points in mind about press fits:

- For a press fit to work well, it needs to be a tight connection. You might need a small hammer, large clamp, or your whole body weight to press the parts together.
- A press fit is usually a one-way connection. After you put it together, don't expect to ever get it back apart.
- Press fits don't scale up well. Always design them at the size at which they'll be printed.

- The tight clearance of a press fit can make it difficult to get started. Running a hobby knife or deburring tool around the edge of the hole will widen that area a bit and help you get the parts together.

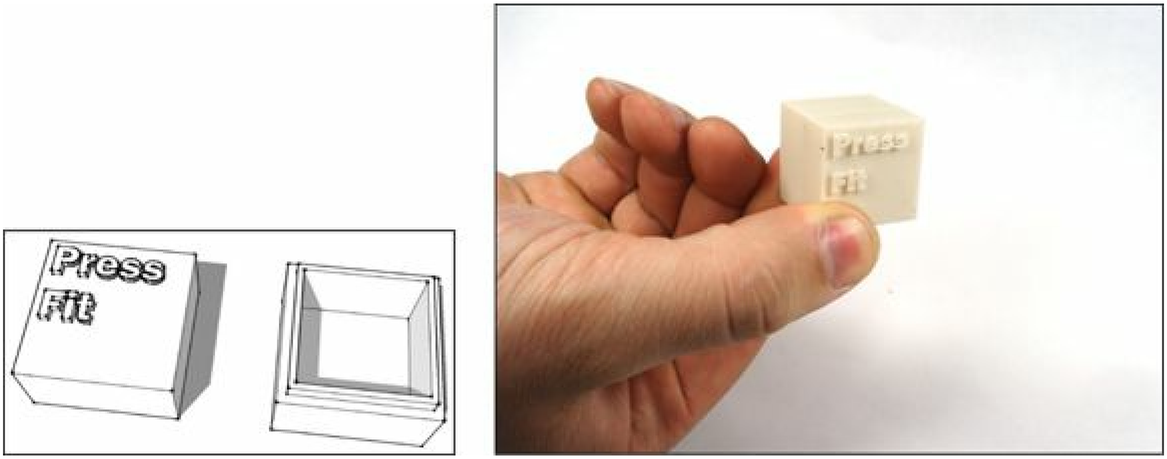

FIGURE 9-21: A press fit is an easy connection to draw in SketchUp.

Bolts, screws, and hardware

The strongest connection you can make between 3D printed parts is one held together with metal hardware. These connections are great for things that are more than just prototypes — for example, parts for robots or mechanisms.

When integrating hardware into your design, include the hardware in your SketchUp model. Take measurements of the parts you're planning to use and model them in SketchUp. After you make these components, save them to the component library so that you can use them again with one click. Keep these points in mind:

- Remember to include enough clearance in your holes for the hardware to be installed.
- If you're using bolts or screws and don't want to include a matching nut, a trick is to slightly undersize the holes. The threads of the bolt will cut into the excess plastic and hold it firmly in place, as shown in Figure 9-22.
- Online hardware suppliers like McMaster Carr and Amazon Supply stock every fastener known to the human race. If your local hardware store doesn't have what you're looking for, these online stores will have it.

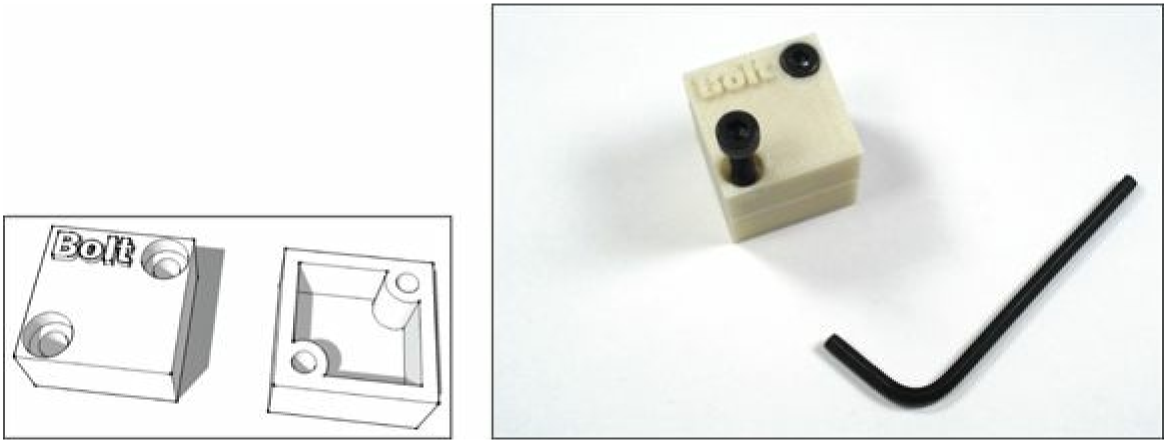

FIGURE 9-22: Nothing says strength like an exposed bolt.

Testing your model’s moving parts

Whatever connection system you are using, always test it first. Before you start printing a giant project, build a small version of the attachment. It only takes a few minutes to design and 3D print, and will give you a chance to make sure the joint works properly. Nothing is worse than modeling a system of joints and then discovering they don't fit together properly.

Designing Things That Move

3D printing something you've designed in SketchUp is cool. 3D printing something that moves is even cooler. This section focuses on a few features you can include to make your creations more than just interestingly shaped hunks of immobile plastic.

Captive joints

A *captive joint* is a moveable connection that comes out of your 3D printer already assembled and working. Captive joints tend to be mechanically simple hinges, ball joints, and chain links. Their simplicity is their power. A 3D printer can quickly build objects with hundreds of captive joints that would take days to construct by hand. Poseable action figures, clothing, and the chainmail in Figure 9-23 are examples of simple captive joints assembled into complex structures.

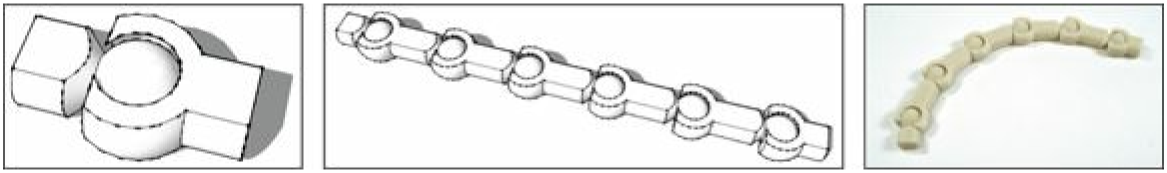

FIGURE 9-23: Captive joints bring the power of multiplication to life.

Creating a captive joint requires trial and error. You'll have to experiment to get the right combination of clearances and shapes. Keep these principles in mind while you work on captive joints:

- Use components when building structures with captive joints; they let you automatically modify all the joints at once as you work.
- How you design captive joints depends on the specific 3D printing technology you're using. SLS-based 3D printers can build captive joints that are a few millimeters across. FDM printers can make fantastically strong joints, but the printed objects need to be much bigger.
- A structure is only as strong as its weakest part. Don't make a joint so fine that it falls apart in your hands.
- Test print parts of your structure as you work. Remember, 3D printing is cheap, and with captive joints, you're pushing the limits of the technology. Test objects will help you check your work and keep your sanity.

Pins

Pins are small, round snap fittings that are pressed into place to make a connection. They can be a versatile replacement for hardware. In Figure 9-24, a single pin acts as a point of rotation, and two or more will hold parts firmly together.

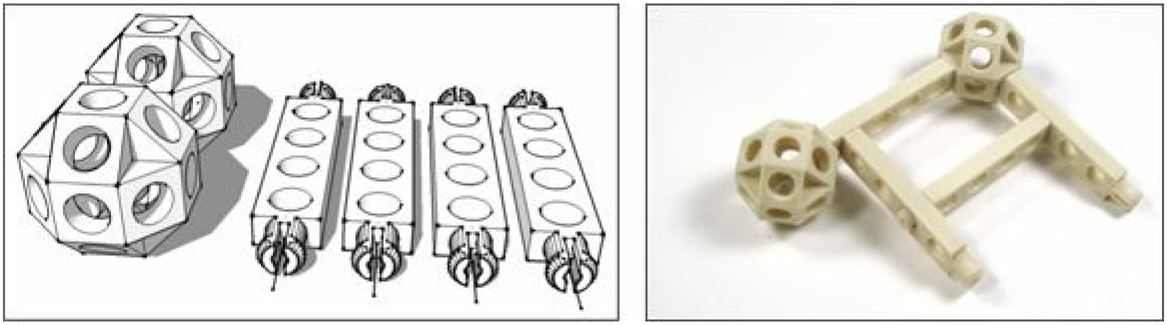

FIGURE 9-24: A system of pin joints used to make a simple toy.

Pins are like bolts, except you get to make them in exactly the size and shape you need. You can also print more when you run out.

- When designing your pins, give them one flat side. The flat side gives you a way to build the pins without using support material — while keeping the tongues horizontal to the build platform. This process also keeps finished pins from rolling off your desk.
- Make your pins into components so you can easily modify all of them at once if you need to.
- When designing a project with pin joints, make an effort to standardize around a small number of pin sizes. Standardization helps keep things tidy and simplifies assembly.

Gears

Gears are great for creating complex motions or transferring movement through a mechanism. That is probably the most understated description of the deepest rabbit hole of the industrial age. Gears are in every mechanical device you can imagine. They have existed in some form or other since before recorded history, and are the basis for an incalculable number of clever devices that make everyday life possible.

Making gears is fun and inspires lots of folks to start experimenting with 3D printing. Figure 9-25 shows a simple gear system that can become so much more. To help you get started, here a few basic principles for making gears:

- Gears need to have clearance between their surfaces to work properly. Gears that are too perfect a fit will bind up. Include a clearance at both the point of rotation and between the teeth of the gear and the teeth of its mate.
- A SketchUp extension called Involute Gears automates the process of making gears. The extension's creator hasn't made it available in the SketchUp Extension Warehouse, so you'll need to locate it by searching *sketchup gear plugin* through Google.
- Creating optimized gears from scratch is a technical art form that has fallen out of practice. If you want to learn more about designing gears, I recommend getting a copy of *Machinery's Handbook* (Industrial Press). After almost a century in print, this book is the gold standard for anything gear-related.

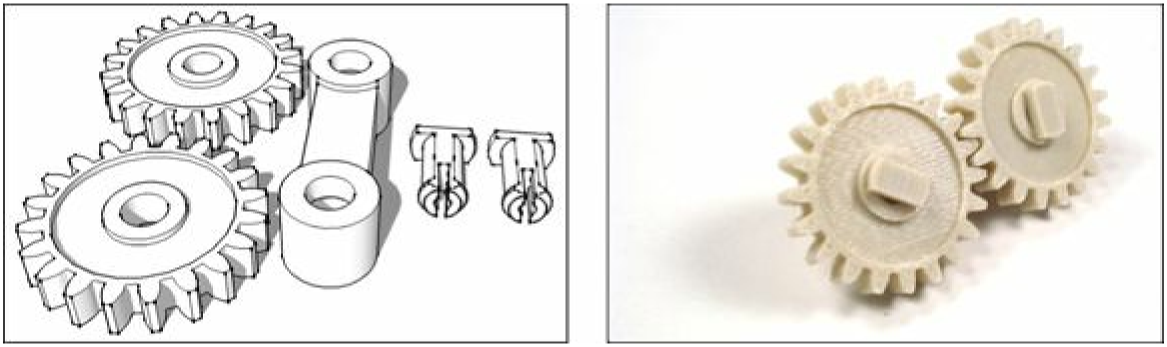

FIGURE 9-25: Welcome to the New Industrial Revolution. Time to gear up.