3D Printing with SketchUp Models

IN THIS CHAPTER

- **Dividing your model into layers**
- **Using SketchUp files for 3D printing**
- **Preparing a SketchUp model for 3D printing**
- **Knowing your 3D printers**
- **Going beyond basics with joints and motion**

Seeing your masterpiece in SketchUp is cool. You can twist and turn and view it from every angle. But wouldn’t the world be so much cooler if you could hold your creation in your hands?

This chapter talks about using SketchUp with a 3D printer.

To follow along with this chapter, you need a working understanding of SketchUp and its tools. We assume you’ve heard about 3D printers and are curious to learn more. The focus is on desktop 3D printers, the most common class of 3D printers on earth. You get a look at 3D printing technology in general and touch briefly on a few types of professional machines.

Most of this chapter discusses guidelines, methods, and tools to help you modify your SketchUp models to be 3D printed. We also point out a few limitations of 3D printers, and how you can work within them.

Building Up a View of 3D Printing

It may sound like magic, but 3D printing is a process that uses the 3D information from your SketchUp model to build a physical version of that model in the real world. It's science fiction come true.

3D printing got its start as the hot new technology of the mid-1970s — and spent 35 years stuck as the high-priced plaything of prototyping engineers and the lucky people who build fighter jets. And that's where it stayed until the first DIY desktop 3D printers appeared in 2009. Overnight, the cost of a 3D printer fell from $500,000 to $500. What was once the coveted technology of the chosen few is now found on the desk of any well-equipped designer, engineer, or model maker.

Building a Model in Layers

There are many different 3D printing technologies and manufacturers, but they all use a process called *additive manufacturing:* An object is built from thin horizontal slices of material, with each new layer extruded slightly above the previous layer and then fused to it. Layer by layer, the printer builds up the object until it's finished.

Supporting layers from below

SketchUp's world is an amazing place where you can rapidly build a 3D model while ignoring little things like the laws of physics. An object being 3D printed, however, is subject to all the forces of the physical world, including that pesky omnipresent one, gravity. New layers can't be printed floating in open space; they need to have something below them.

REMEMBER The need to support each layer is the most important guideline to keep in mind while you're designing objects for 3D printing. You can use either of two strategies to support new layers as you're printing objects:

- **Use the 3D printer's support-material function,** which creates a secondary structural lattice around the part. The lattice holds up any layers that would otherwise be free-floating in space. Support material may be dissolvable after printing or have to be manually removed with fingers and tweezers, depending on your type of 3D printer. Either way, removal of support material can become the most labor-intensive part of 3D printing; it also increases the amount of materials used and the time it takes to print the object.
- **Design and print your parts in a way that limits the conditions that allow unsupported layers to exist.** Smooth transitions and sweeping curves not only look awesome; they can also be easily printed without resorting to the use of support material.

Designing to avoid support material

With a little forethought, you can avoid the use of support material entirely by adhering to a few basic guidelines:

- **Think about your parts orientation.** Orient the part so that it prints with the smallest number of overhangs. This may mean printing your object upside down or on its side. The capital T in Figure 9-1 could be printed standing upright, but laying it on its back allows it to print much faster — without support material.
- **Try rethinking your angle.** Most printers can print a slope between 45 and 60 degrees from vertical without using support material. Keep this limit in mind as you design. Chamfers and fillets are great for supporting features and smoothing out rough transitions. In Figure 9-2, a chamfer turns a part with an unsupported overhang into an easily printable part. As an added bonus, fillets

and chamfers make your part stronger by eliminating stress points.

- **Consider breaking your object into multiple prints.** The sphere in Figure 9-3 could be printed using support material, but it will print faster and with less cleanup if you split it in half and attach it back together afterwards. Later in the chapter, you learn about systems to make the parts lock together quickly.

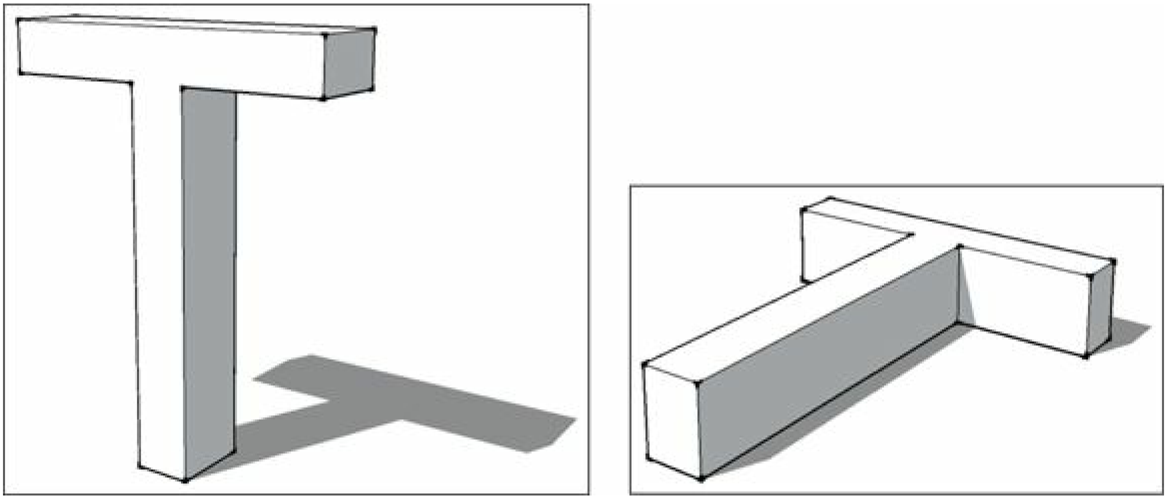


FIGURE 9-1: Re-orienting a part can make it print faster and leave less support material to clean up.

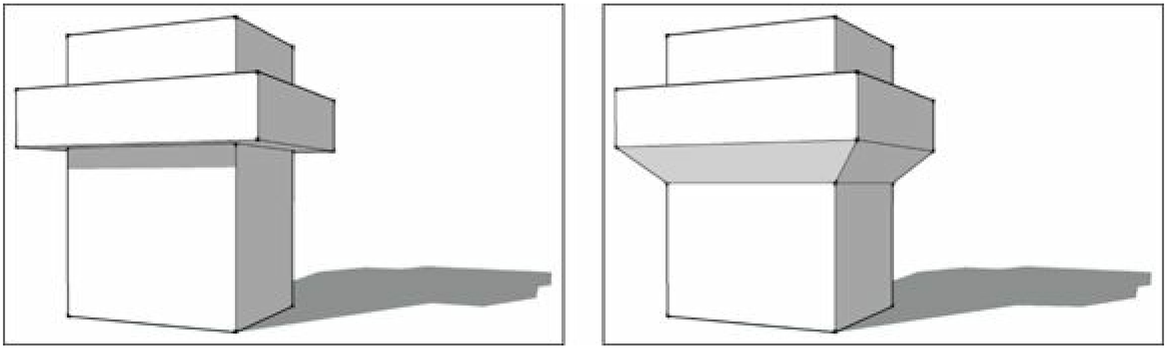


FIGURE 9-2: A chamfer can turn an unprintable 90-degree angle in to a printable 45-degree one.

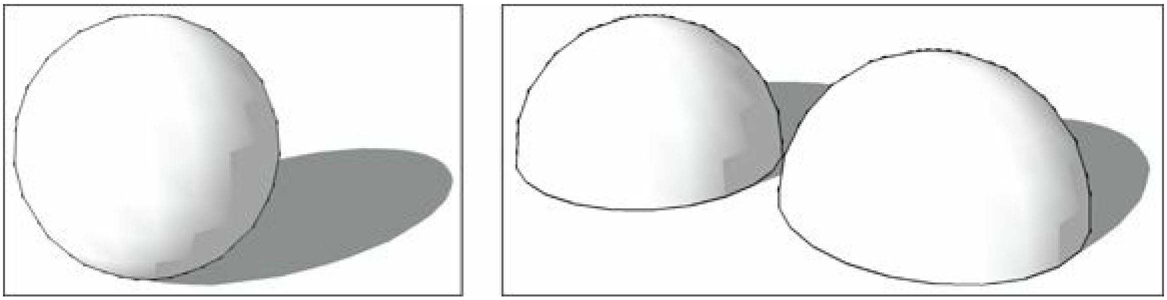


FIGURE 9-3: Often a subdivided part is faster and cleaner to print than a part printed all at once.

Bridging

Bridging is a feature on 3D printers that lets you print a structure across a gap without using support material. To bridge a gap, the structure being printed must be parallel to the build platform and have a secure point of attachment on either side, as with the top of the door frame shown in Figure 9-4. The printer will attach plastic to one side of the gap and stretch a line across to the other side. The process is repeated until the gap is filled. Future layers are printed on top as usual.

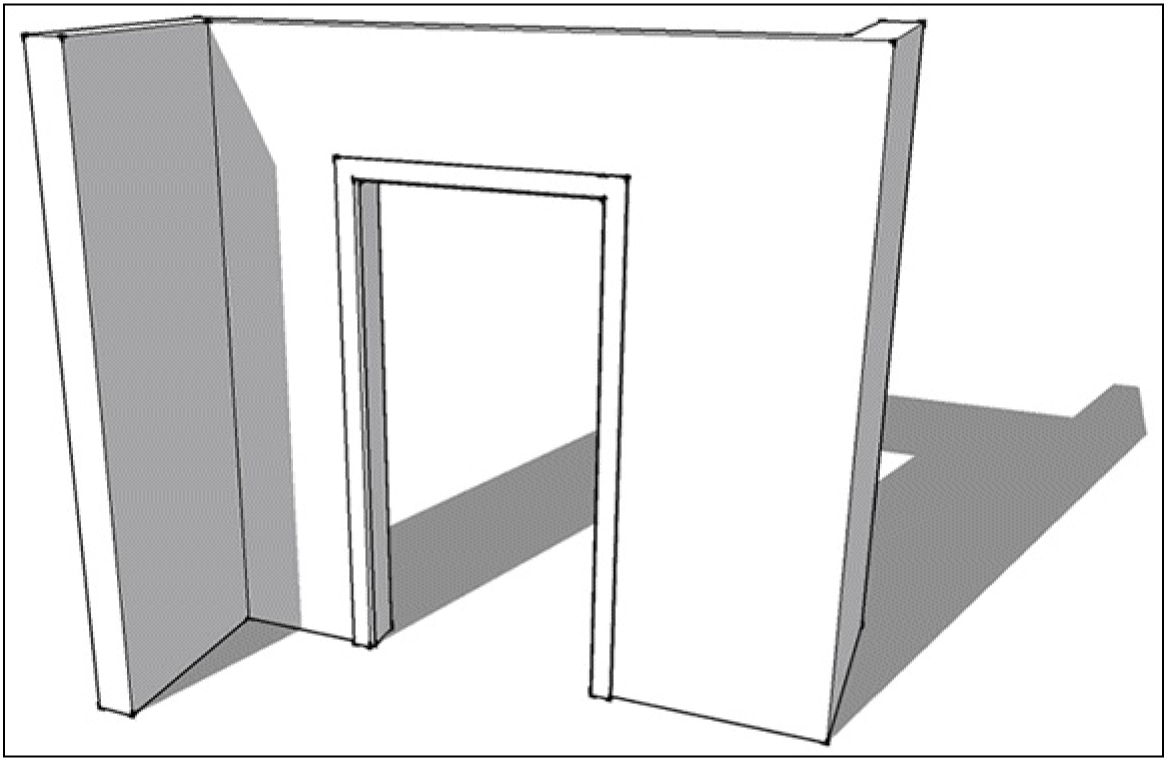
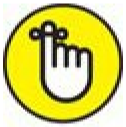


FIGURE 9-4: When is a door frame a bridge? When it's 3D printed.

Preparing a SketchUp Model for 3D Printing

The longer you work on a SketchUp model, the more it tends to fill with illogical intersections, free-floating cantilevers, and other quick shortcuts. When you're making an image or walkthrough, these are minor trade-offs that help get a big job quickly. Drawing something that looks right on the outside in SketchUp doesn't necessarily mean it can be 3D printed with one click of a button. A 3D printer can't interpret that mishmash of geometry to guess what you were really thinking.

In this section, you discover how to clean up that SketchUp madness and make it into a 3D printable object. The cleanup process uses tools discussed in earlier chapters, but the way you use these tools differs. Cleaning up a messy SketchUp model can look like a daunting task when you start. Remind yourself that it's just like eating an elephant: Divide the job into manageable chunks and work on them one at a time … even if the trunk can be a bit chewy.



REMEMBER Before you start cleaning up geometry to make your model 3D-printable, make sure you've saved a separate copy of your SketchUp file, just to be safe.

Peeking inside a model

Although you can use the Section Plane tool to create sectional views through your model, this awesome tool has another use: It can show you what's going on inside your model's geometry. 3D printers see a SketchUp model as a series of 2D horizontal slices. Figure 9-5 moves the Section Plane through the model horizontally to show what the 3D printer is going to see.

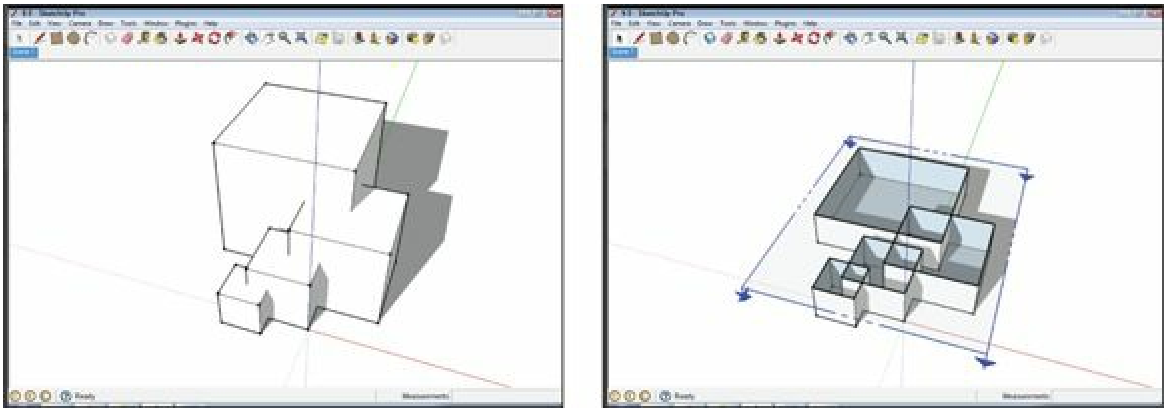


FIGURE 9-5: Using a section plane to look at all overlapping parts that make up your model.

To get a watertight solid object, all groups need to be combined into one, and all that extraneous geometry has to be eliminated from the model. Chapter 11 discusses the Section Plane tool in detail.

Knowing what makes a solid model

To 3D print your parts successfully, they need to be solid objects, or as close to solid as possible. No missing or extra faces, overlapping sections, or extraneous bits allowed. Your SketchUp model needs to describe all aspects of the outer surface of the shape. For example, to print an egg, a SketchUp model would need to describe 100 percent of the outside of the egg's shell. That's not as simple as it sounds.

A solid model that's 3D-printable meets the following criteria:

- **The model walls have a thickness.** Zero thickness = Zero printing. A wall that is a single face has a thickness of zero and will not print. You need to make sure all the walls have some thickness.
- **The model has only an outer surface.** In SketchUp, you can create a model quickly (in the abstract, anyway) by pushing parts through each other and building revisions on top of old geometry. To your 3D printer, however, such models are an illogical mess that would make M.C. Escher cry. Your 3D printer doesn't stand a chance. You'll need to clean everything up so that only the outer surface remains. Fortunately, SketchUp tools and extensions discussed later in this chapter can help.
- **Groups and components are merged.** Chapter 5 explains groups and components, and how to use them to keep your SketchUp model organized as you work. Groups and components are great for keeping parts from sticking together as you work, but to make your SketchUp model a solid shell that can be 3D printed, you need to merge everything together.

  TIP Before you start exploding things and sticking them together, spend a moment thinking about how you want your actual 3D printed model to work. For most projects, you need to merge all your groups and components into one printable object before exporting your model from SketchUp. For larger projects, think about assembling the blocks into sections that you can 3D print and attach together later.

The upshot is that 3D printers aren't very smart. If you make them guess what your part should look like, they will usually guess wrong. Figure 9-6 is a testament to the carnage.

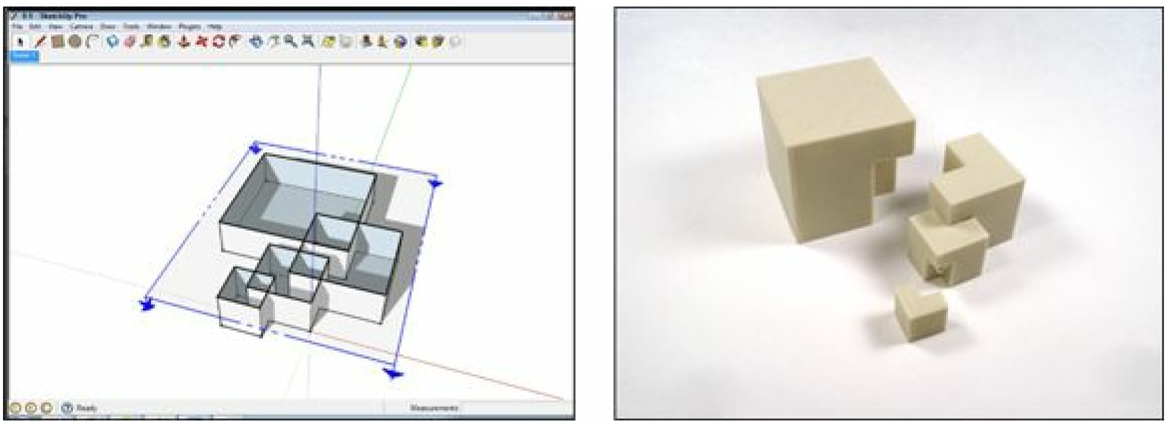


FIGURE 9-6: This poor print never stood a chance.

Using Solid Tools to combine groups

Chapter 6 explains how to use SketchUp's Pro's Solid Tools to perform Boolean operations. Solid Tools are great for unifying groups made up of simple solid parts. They can save significant time. Unfortunately, complex shapes cause the Solid Tools to act unexpectedly — and, after multiple iterations, to break down.

In Figure 9-7, Solid Tools had no problem intersecting several simple cubes together, but adding a complex shape to the object caused the tools to break down. The accompanying sidebar takes a closer look at why this happens.

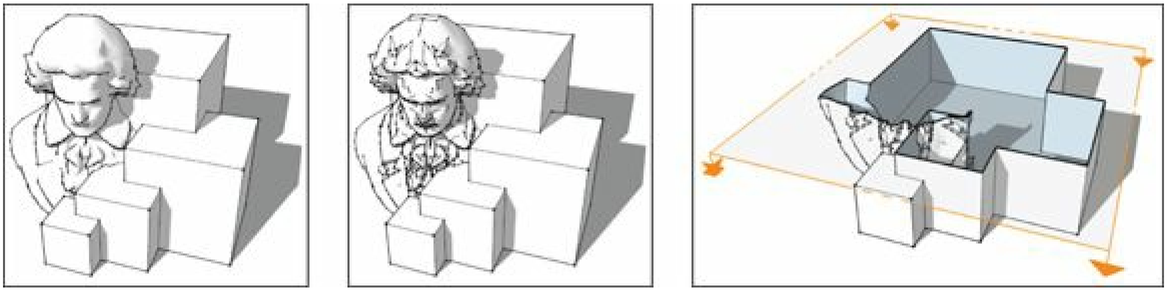


FIGURE 9-7: Booleans are great for simple things. But don't rely on them for complex cleanup.

WHAT’S WRONG WITH SOLID TOOLS?

You may have been looking at the Outer Shell button in the Solid Tools menu and thinking, “Wow, making this into one outer shell is going to be easy.” So you decided to save your work, select your whole model, and click Outer Shell. Oops. (Come back when you're done screaming.)

In the discussion of Solid Tools in Chapter 6, you find out that they are also called Boolean operations. Boolean operations started out as computer programming tools that help sort and manipulate data. In the early days of computer graphics, programmers used Boolean operations to work directly with the data that makes up 3D models. Boolean operations still exist in every piece of 3D modeling software today. If you take a class in 3D Modeling, they’ll be the first thing you’ll learn, and then you’ll be told to never use them ever again.

The problem is Boolean operations are the blunt instruments of 3D. They work within a narrowly defined set of parameters, and are notorious for the way they indifferently modify geometry. After a few uses, the damage caused by the tool itself starts to compound, and the model becomes too garbled for the Boolean to interpret again. Basically, after being confounded by the sum total of its own mistakes, it gives up in failure. Outer Shell is trying to run a whole series of Boolean operations at once, and if your model is made up of anything except simple shapes, those operations won’t succeed.

Unfortunately, cleaning up your model just isn’t going to be single-click simple.

CleanUp^3 and Solid Inspector^2

Two tools from the SketchUp Extension Warehouse are essential for 3D printing: CleanUp^3 and Solid Inspector^2, both created by Thomas Thomassen.

CleanUp^3 checks and simplifies the geometry of your SketchUp model. It combines multiple faces, eliminates extraneous data, and erases any lines that don’t make a face. Two of the most useful CleanUp^3 options are Erase Duplicate Faces and Repair Split Edges, which can be enabled in the CleanUp^3 menu. Duplicate faces and split edges are errors that inevitably appear in your model as you work in SketchUp, and they drive 3D printers nuts. Both errors are hard to recognize and repair manually.

CleanUp^3 is also great at simplifying STL files (3D-printable files) you import into SketchUp. In Figure 9-8, which shows the triangulated data you get from an imported STL file, the faces have been broken into hundreds of triangles. By removing that triangulation, CleanUp^3 makes files downloaded from 3D printing communities (such as Thingiverse at `www.thingiverse.com`) easier to edit in SketchUp.

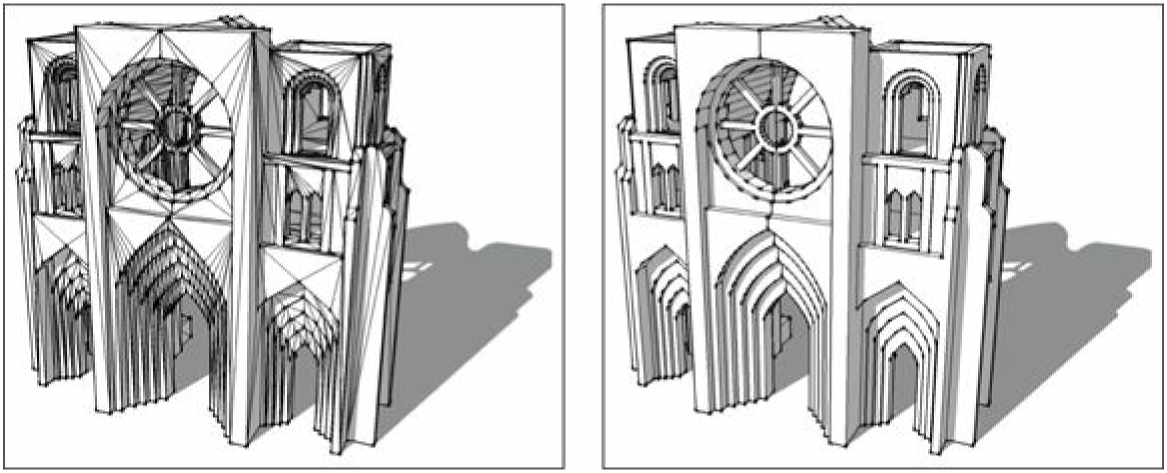


FIGURE 9-8: An imported STL file, before and after running CleanUp[3].

TIP Solid Inspector[2] finds and highlights problems that are preventing your model from being a solid shell, and we can’t overemphasize its usefulness. It highlights problem areas and helps you automatically move from one error to next, making repairs much faster (see Figure 9-9). It’s a tool that everyone using SketchUp for 3D printing should have.

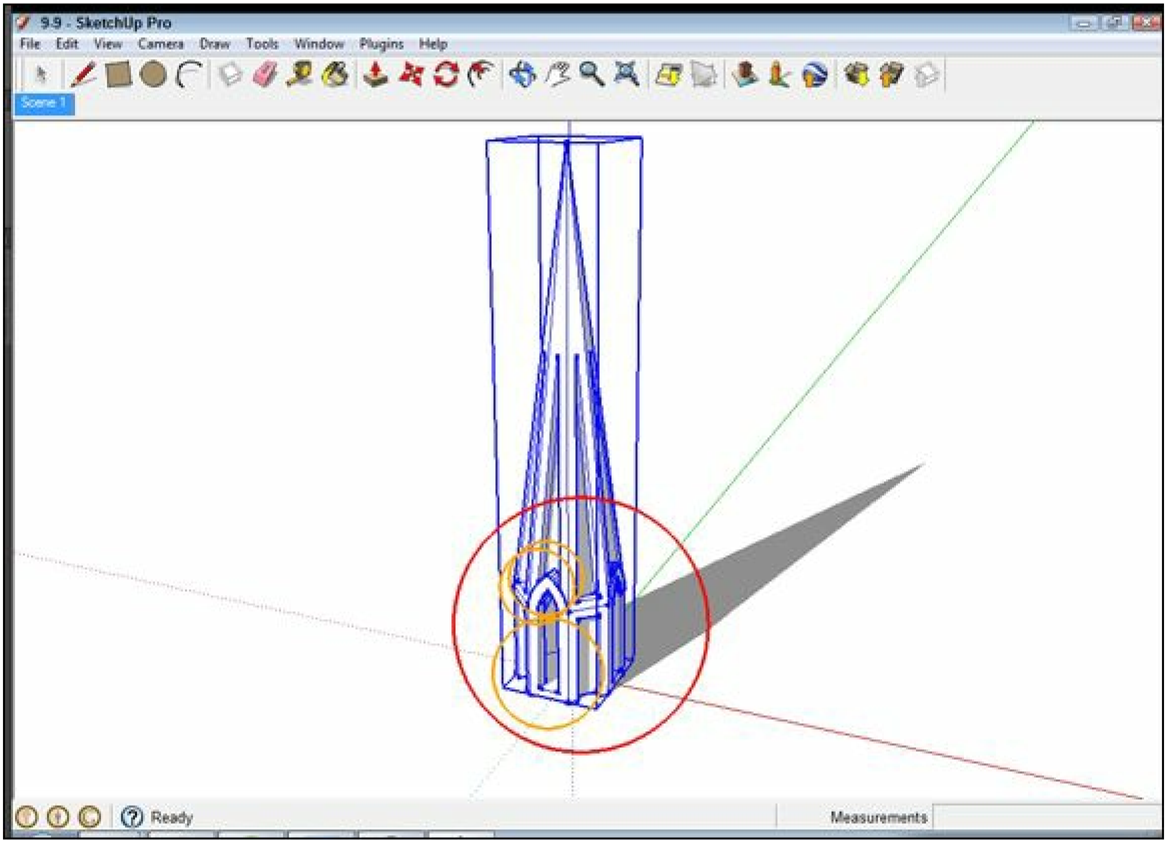


FIGURE 9-9: Solid Inspector^2 is great at finding problems that need a quick fix.

REMEMBER As you work on your model, get in the habit of running CleanUp^3 periodically. Run Solid Inspector^2 before you export from SketchUp to 3D print. It will catch any errors that would cause the print to fail.

Combining groups with Intersect Faces

An alternative way to assemble your groups into one object is to use the Intersect Faces command, introduced in Chapter 4. For 3D printing, Intersect Faces can eliminate overlapping geometry and leave behind a solid object. You can use Intersect Faces on very complex objects and groups that aren't already solid.

In a complex model, isolate and work on one small part of the SketchUp model at a time, making each section into one solid part. Repeat the process on those sections until you have one solid model. The following steps outline this process in more detail:

1. **Select the groups you want to combine and group them together.**

   Your original groups are now subgroups that exist together inside a new group.

2. **Choose Edit ⇒ Intersect Faces with Context.**

   Doing so opens your new group and selects all the subgroups, drawing lines at all the places that groups intersect. These new lines of intersection exist outside the subgroups and aren't stuck to anything yet, as shown in Figure 9-10.

3. **Select the lines of intersection and copy them by choosing Edit ⇒ Copy.**

4. **Open each of the subgroups and paste the lines of intersection into them, using Edit ⇒ Paste in Place.**

   Figure 9-10 shows the procedure.

5. **Open each subgroup and erase all the areas of overlap beyond the lines of intersection.**

   Work deliberately, moving back and forth between the subgroups to make sure you're erasing the right areas.

6. **Back in the main group, select all the subgroups, context-click the selection, and choose Explode on the menu that appears.**

   You end up with one outer shell of an object in its own group. Figure 9-10 shows this result.

7. **Run CleanUp[3] on the new part and use Solid Inspector[2] to check for any accidental holes.**

   TIP Individual faces and short line segments can get lost during this process. You may need to make minor touch-ups with the Line tool.

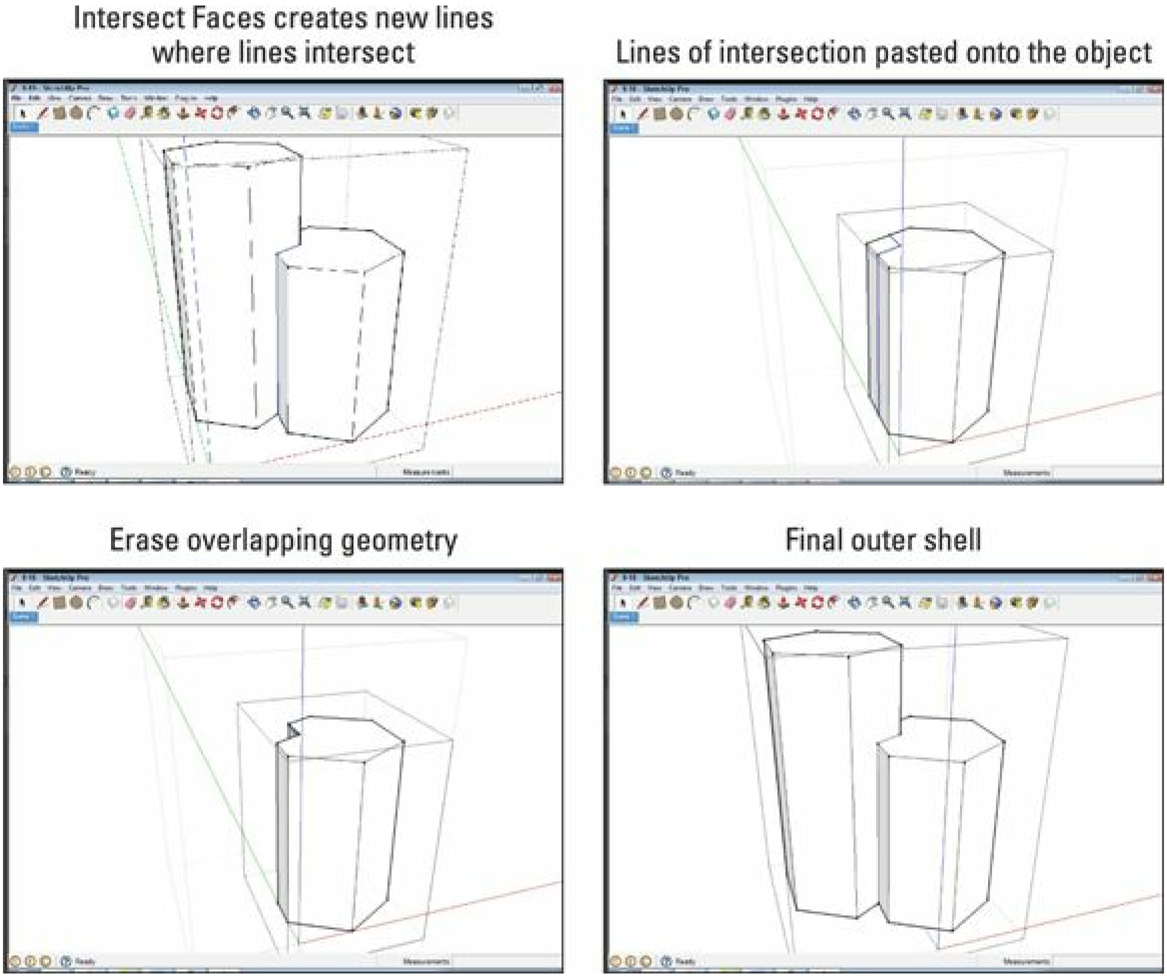


FIGURE 9-10: Erase overlapping geometry to create an outer shell.

Checking a model's normals

From a highly technical computer-science perspective, every face in your model has two sides: an inside and an outside. The two sides are set apart by a bit of data called the face’s *normal*.

SketchUp is smart enough that it doesn’t matter how the normals (also known as the front and back) are oriented; both sides are treated the same. 3D printers aren’t that clever; they need to have all the normals oriented so the outsides are pointing out and the insides are pointing in.

To check your model’s normals, you’ll need to look at it styled with SketchUp’s default texture. Choose View ⇒ Face Style ⇒ Monochrome to hide any colors or textures you’ve added to the model and see the default material. (Chapter 10 covers styles in detail.)

Any faces that are shaded the default blue have reversed normals. These faces will need to be corrected, or the 3D printer will see them as missing.

- **To correct a single face,** context-click the face and select Reverse Faces from the menu that appears.

- **To correct several misaligned faces,** context-click one of the faces that is white and select Orient Faces. Doing so flips all the faces in the model to match the view shown in Figure 9-11.

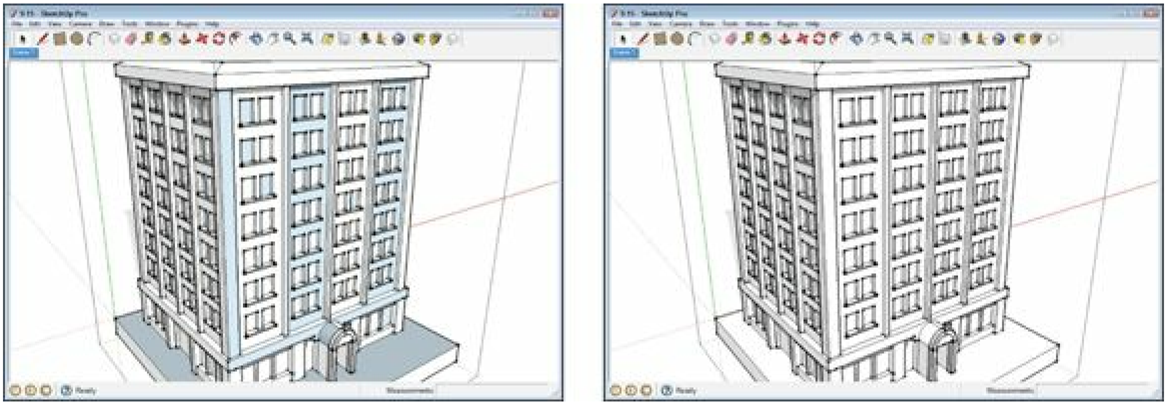


FIGURE 9-11: A correct normal is a happy normal.

TIP If the Orient Faces tool causes everything to flip randomly, it's a sign that you have extraneous faces inside your model. Use the Section Plane tool to look inside, find the face, and erase it.

Checking your model's size

Every 3D printer has a minimum and maximum size of object it can build, as shown in Figure 9-12. These sizes are usually set by the size of the tool printing the material and by the overall size of the printer. To build something bigger, you have to get creative. To build something smaller, you'll need a more expensive 3D printer.

FIGURE 9-12: Too big and too small. There is something there on the right; it's just very small.

Too small to print

In the SketchUp world, you can design a skyscraper small enough to fit on the head of a pin, but the

3D printer can’t print it. Every 3D printer has a minimum size for what it can build; anything smaller than that won’t be printed. You find these values listed as Minimum Feature Size and Minimum Wall Thickness. Feature Size and Wall Thickness can turn into stumbling blocks when you’re trying to 3D print a SketchUp model that was constructed at full size and then scaled down.

- Minimum Wall Thickness tells you how thin a freestanding piece of geometry can be and still be printed. That thickness is typically between 1.0mm and 0.5mm.
- Minimum Feature Size is the smallest size that a feature can be on the surface of the object that will be printed. It’s typically between 0.7mm and 0.2mm.

If you're modeling a small object for 3D printing, create the model in small units, such as millimeters, from the beginning. To set or change the default units in your SketchUp model template, choose Window ⇒ Preferences (SketchUp ⇒ Preferences on a Mac) and select a template that uses millimeters. You can also adjust the default units in the Model Info window by choosing Window ⇒ Model Info and selecting the Units option on the left.

Too big to print

The biggest object you can 3D print at one time is set by the printer’s *build volume*. If your model won't fit in the build volume, you have to either scale your object down, or print it in parts. The next section talks about how to split up a model so you can print it in parts.

REMEMBER Although the maximum size of any single part is limited by the size of your 3D printer, the size of what you can 3D print is limited only by your creativity and patience. A man in New Zealand is 3D-printing himself an Aston Martin, one 6-inch cube at a time.

Here are some handy hints for making best use of build volume:

- The build volume, or *envelope*, is given by manufacturers as measurements of X, Y, Z. X and Y are the width and depth of the build surface; Z is the maximum height.
- It’s helpful to create a representation of your 3D printer’s maximum build volume in your SketchUp model. Make a translucent block representing the maximum build volume and check to see whether your SketchUp model or its components will fit inside that volume.
- A 3D printer’s build platform is much longer diagonally than it is on any one side. To take advantage of the extra length, rotate long parts so they stay inside the build volume.
- Printing large objects comes with its own issues and complexities. Large objects are more prone to failure and breakdowns. Make sure you're comfortable with your 3D printer before you start printing your own sports car.

Breaking Your Model into Parts

As you do more 3D printing with SketchUp, you’ll run into the need to split your model into parts. Some ideas are just too big to fit into your 3D printer’s build volume. Other ideas want to be 3D printed in a rainbow of colored plastic. And sometimes a model just needs to be split to make it easy to print.

Where to cut

When subdividing a SketchUp model into printable parts, start by thinking about what you're going to do with the seams. If you're going to sand, paint, and finish the model, then have at it and cut wherever you want. But sanding and finishing is a huge amount of work, especially if you've never done it before. It's much easier to make your cuts and seams look like they're intentional parts of the model.

REMEMBER For a seam to look intentional, it has to work with the logic of the object. Every object has its own logic, an underlying order that informs how it's shaped and structured. For example, in Figure 9-13, the object is symmetrical left and right. A seam on the axis of symmetry is less objectionable than one that runs randomly in another direction. By following a line that's already conceptually present in the object, the seam reinforces what the eye already perceives.

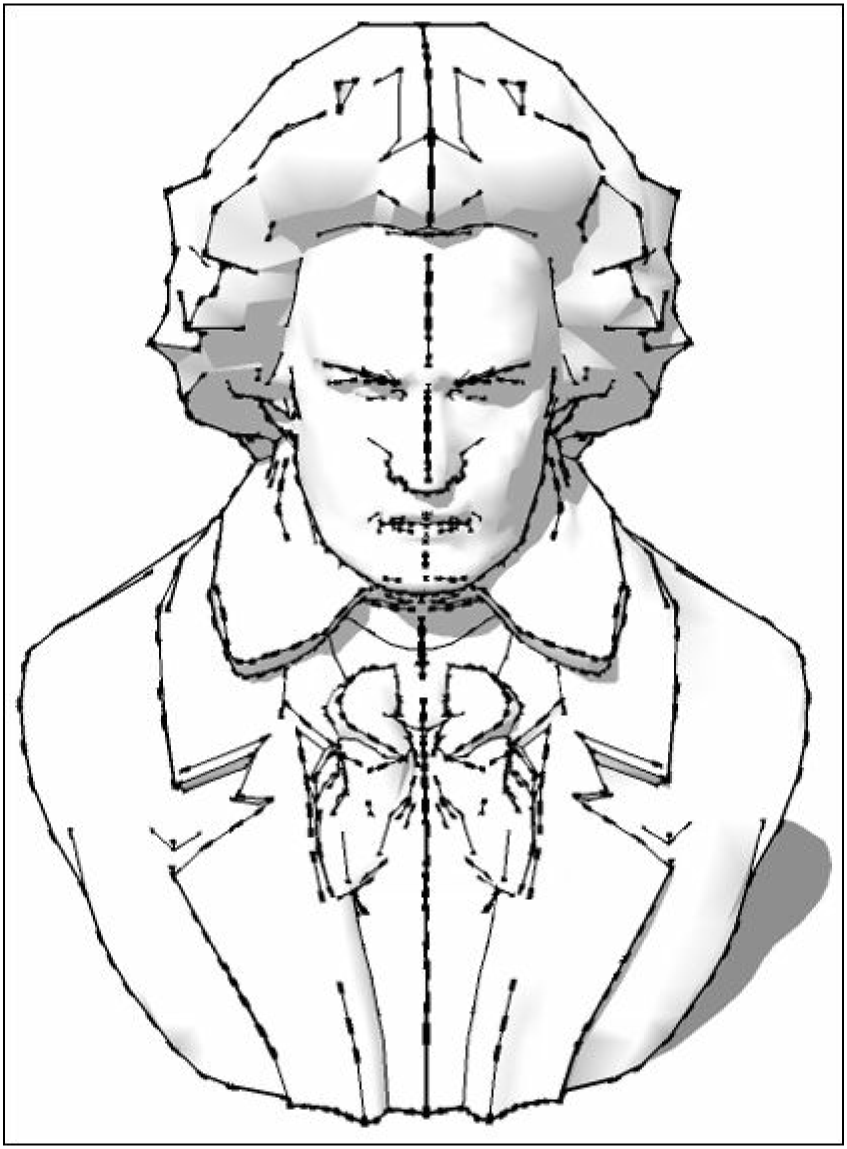


FIGURE 9-13: The line is far less objectionable when it's well thought out.

Another good place to hide a cut is at a change in elevation, curvature, or color, as shown in Figure 9-14. Placing a seam where the surface of a part is already interrupted or in transition will make it far less noticeable. This is the strategy most commonly employed for injection-molding parts. Pick up something around you that's made of plastic and find the seams where the parts come together. If you can imitate that type of seam, people will readily accept that your object is a "real" thing. It will feel correct among manufactured things in our injection-molded world, and you'll fool people into thinking your 3D printed part came from a factory somewhere.

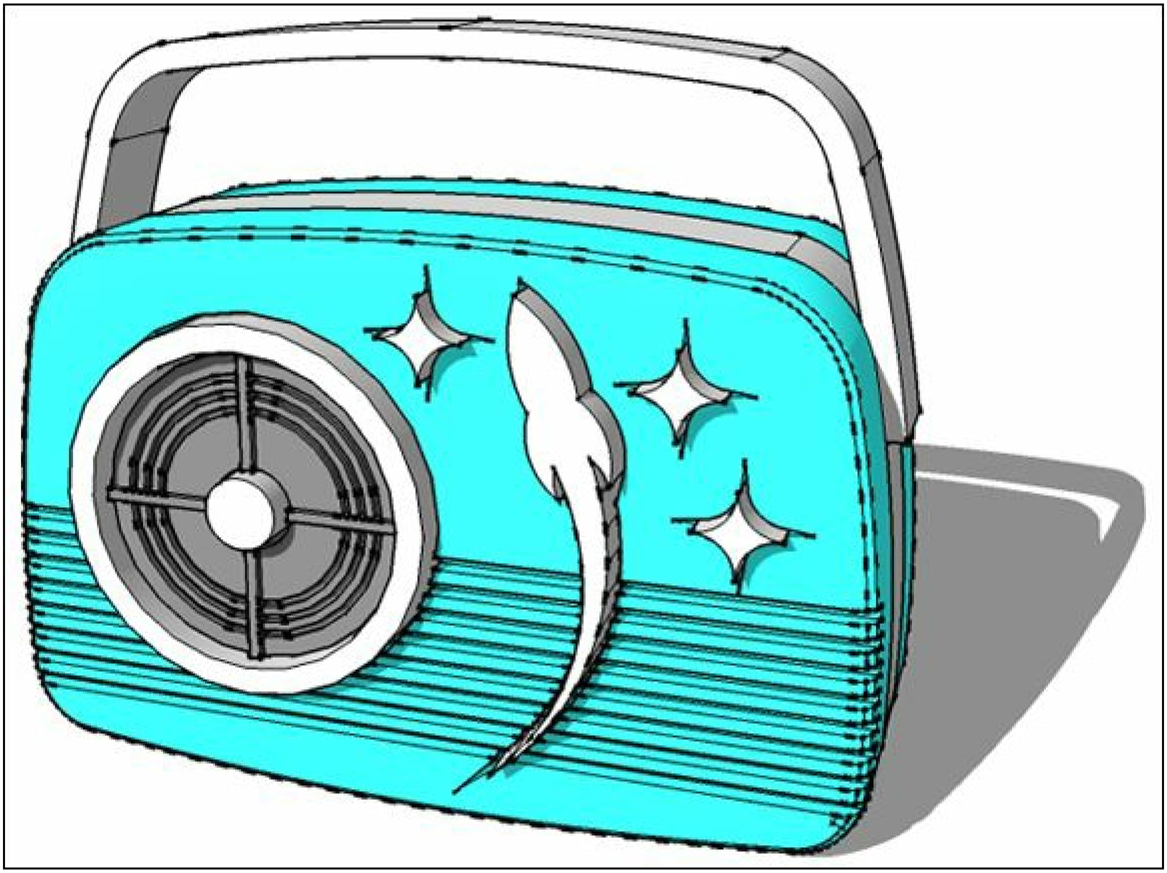


FIGURE 9-14: Do the seams look out of place?

For very large constructions, the only option may be to establish a grid and cut into build-volume-size blocks. Sanding, finishing, and painting can work well on large parts, but expect to spend substantial time doing it right. If you've spent the energy to build something that big, it's worth the extra time to make it amazing.

How to cut

Cutting a model into parts is very similar to using the Intersect Faces tool to combine groups (discussed earlier in the chapter). You use a piece of geometry as a cutter that will be intersected with the larger object and become the new edges of the cut.

If your model is fairly simple and you have SketchUp Pro, you can use the Solid Tools to short-circuit this process. Create a solid block as a cutting object and use the Solid Tools ⇒ Trim command. Remember to run CleanUp^3 and Solid Inspector^2 on the new parts when you're done. For more complex models, or users with SketchUp Make, use the Intersect Faces method:

1. **Select the group you want to cut and make a new group around it.**

   The original group becomes a subgroup.

2. **Working inside the new group, create geometry in the shape of the cut you want to make.**

Work on top of the subgroup, so you place the cut correctly, as shown in Figure 9-15.

3. **When you're done placing cuts, make the new geometry into its own group.**

4. **Select the Cutter object and choose Edit ⇒ Intersect Faces with Context (or choose Edit from the context-click menu).**

   Doing so draws a line at every place where groups touch. These new lines of intersection exist outside the subgroups and aren't stuck to anything yet, as shown in Figure 9-16.

5. **Inside each group, use Explode to stick the intersection lines, cutting object, and base object together.**

   The surface of the cutting object becomes the sides of the new part.

6. **Move back out to the master group and make as many copies of that group as the number of parts you're dividing it into (see Figure 9-17).**

7. **In each copy, open the group for editing and erase everything you don't need in that part. Do the same in the other parts.**

   Just make sure you don't erase what the object needs to do its job.

8. **Run CleanUp[3] on the new parts and use Solid Inspector[2] to check for any accidental holes.**

9. **Position the parts back together to make sure everything lines up as expected, as shown in Figure 9-18.**

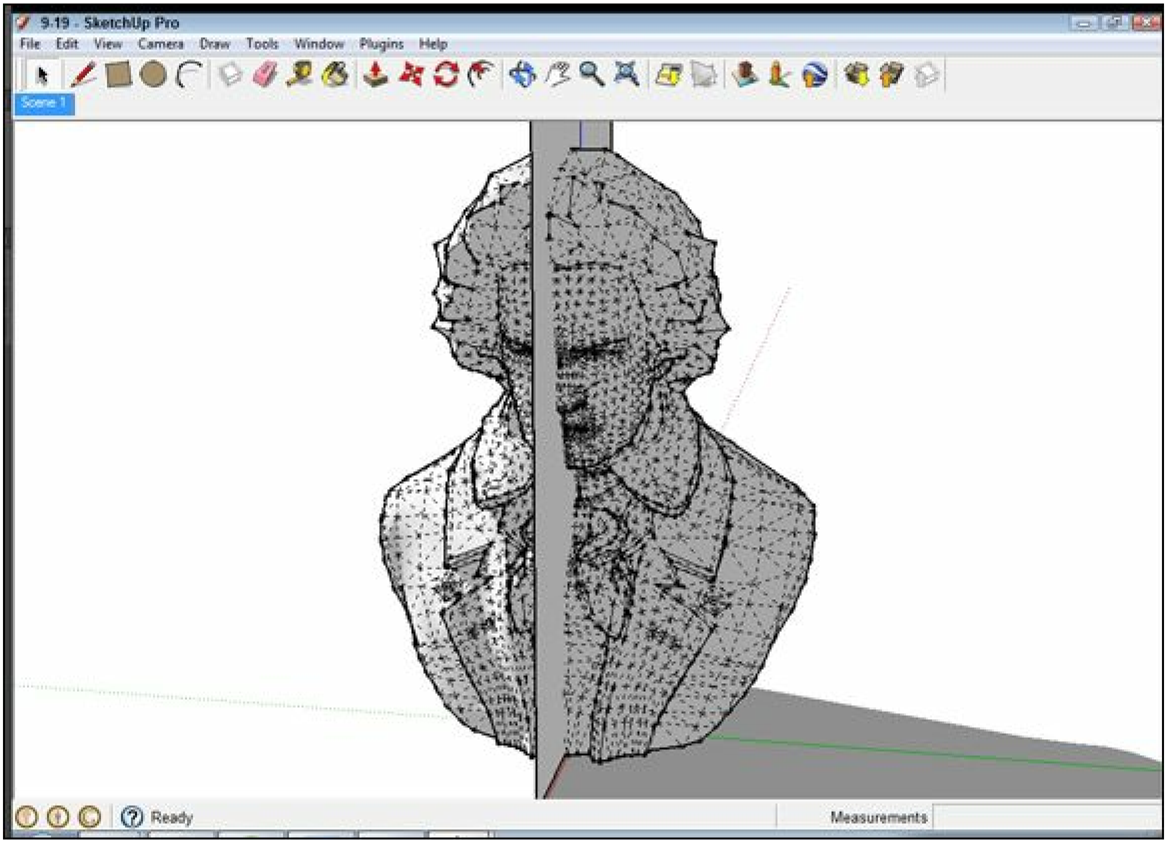


FIGURE 9-15: The cutter object can be a single plane; it’s going to become part of the solid object.

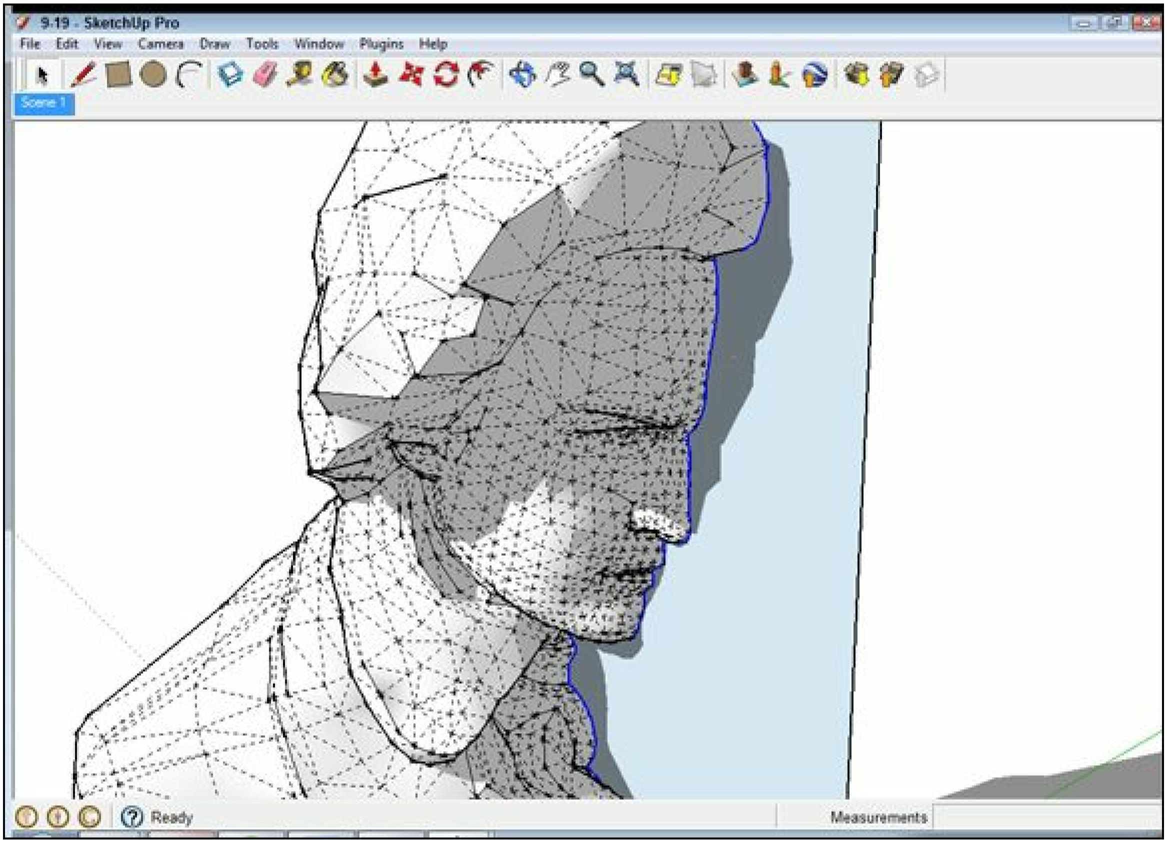


FIGURE 9-16: Lines of intersection created between the two groups.

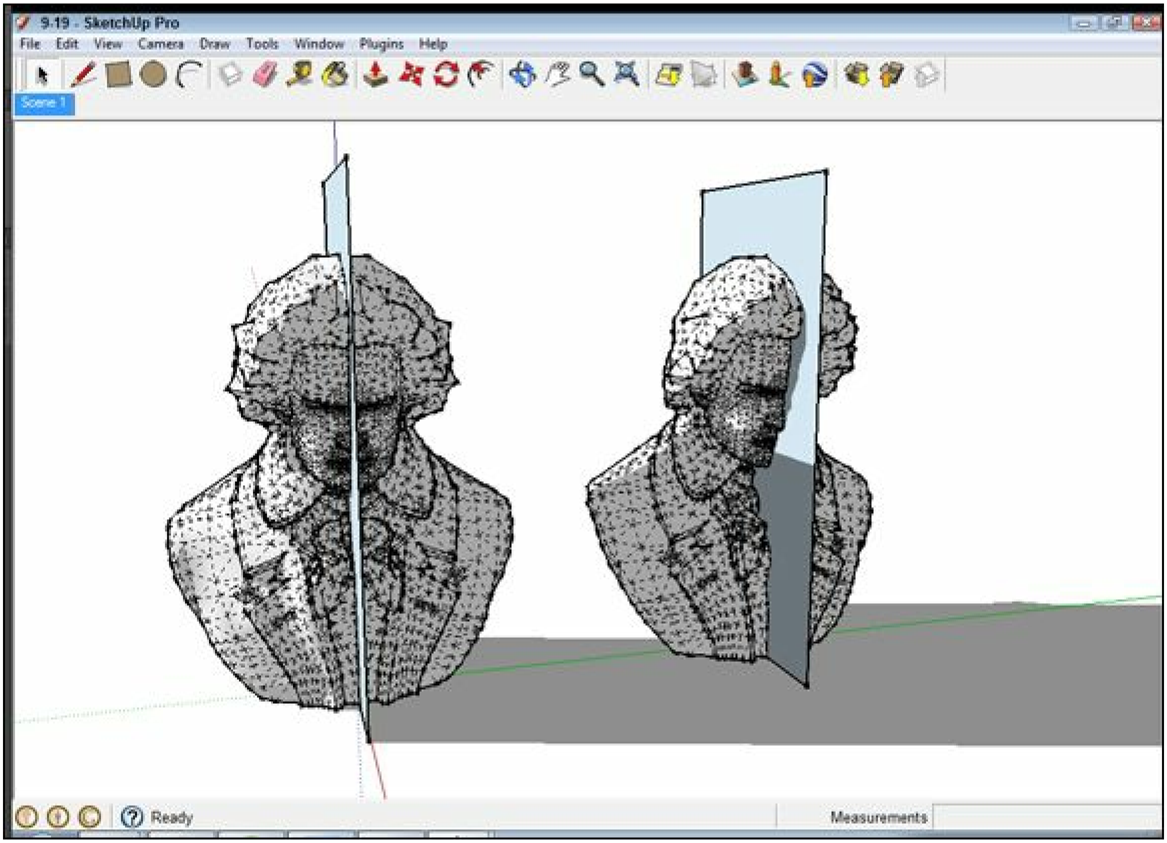


FIGURE 9-17: Make one copy of the group for each part of your model.

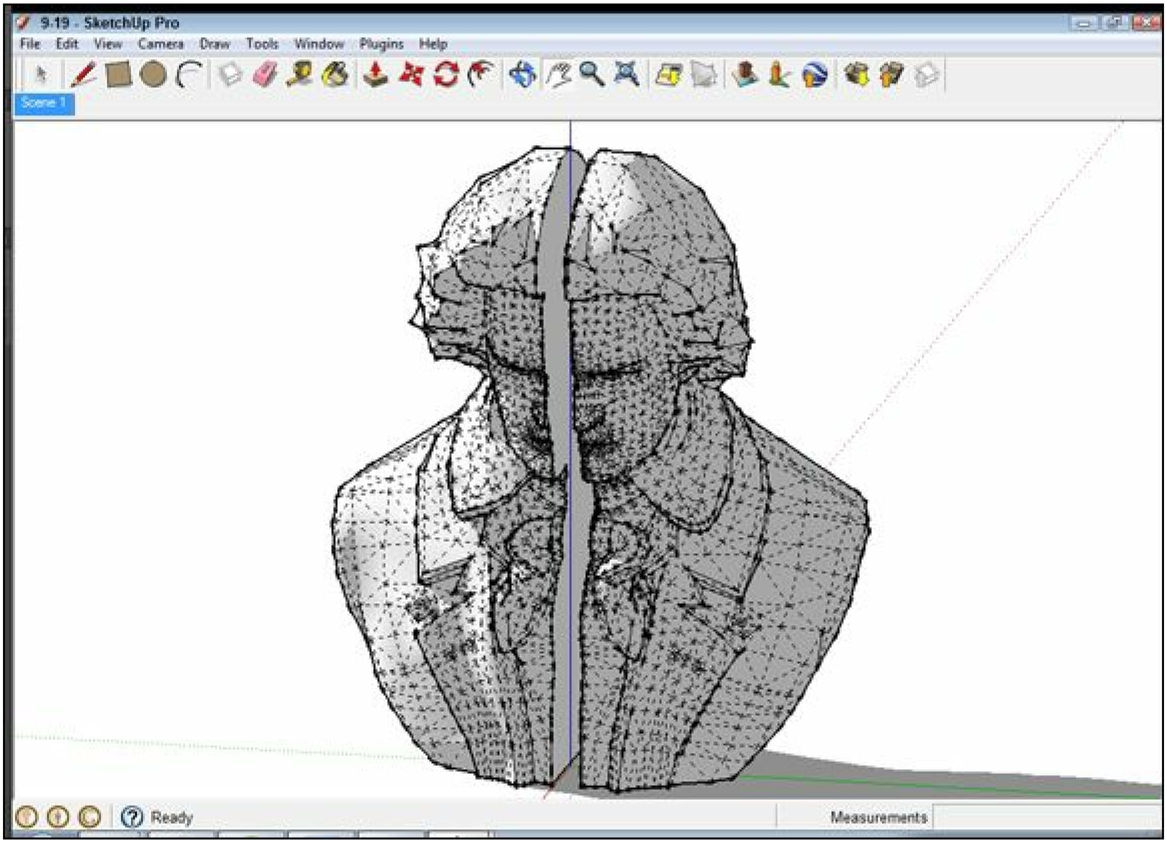


FIGURE 9-18: It still fits!

Exporting Your SketchUp File

After you’ve cleaned up your SketchUp model and you’re ready to print it, you have to get your 3D model out of SketchUp and into your 3D printer’s control software. Before your 3D printer can open your model, you have to export the model in either the STL or OBJ file format.

If you're using SketchUp Pro, you already have the option of exporting an OBJ file. To export an STL file from either version of SketchUp, you’ll need to install a free extension from the SketchUp Extension Warehouse called SketchUp STL. It’s a great tool created by the SketchUp team to make 3D printing easier.