

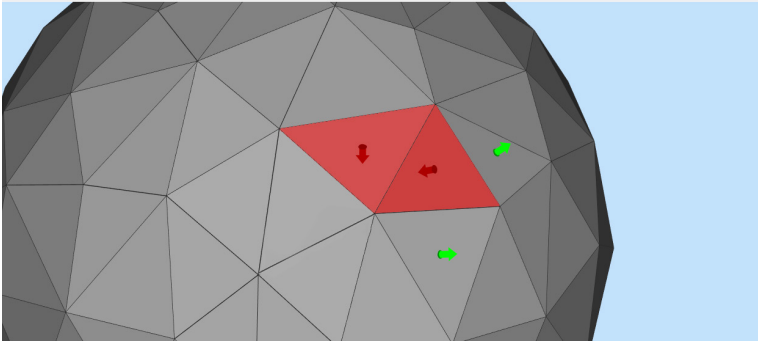


10 Basic 3D Model Repair Functions Every Data Prepper Should Know

If you have some experience with designing and printing 3D models, you know how important it is to repair your 3D file after conversion from CAD to STL. The design may look perfect, but when you start printing it, the build can fail. An error can easily slip the eye, which is why STL editing software can help you avoid printability issues and ensure a successful print. The STL repair and editing tools mentioned below help create a watertight, printable 3D model.

1. Flip inverted normals

There are two sides to any 3D model: the outside, which is the side you'll be able to see once it's been printed, and the inside, which you would only be able to see if you cut a hole through the side of your design. The triangles making up the body of your design also have an inner side and an outer side, and the outer side is called the "normal".

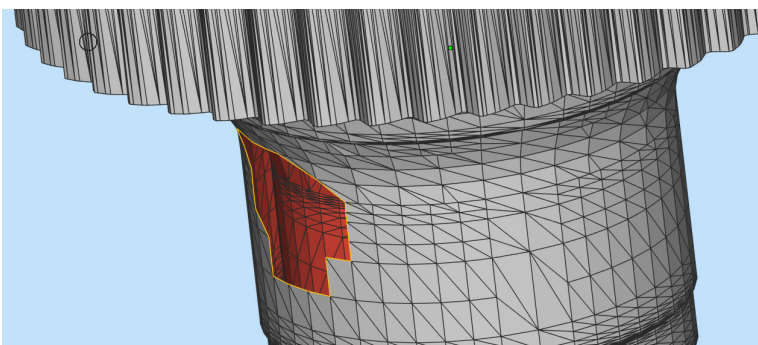


If a normal inadvertently gets flipped, so that it faces inward, the printer will get confused and think that the entire inside of your model needs to be filled in since the inside can now also be considered as the outside of the model. Essentially, it won't know where to stop printing and which parts it needs to leave hollow.

2. Fill holes

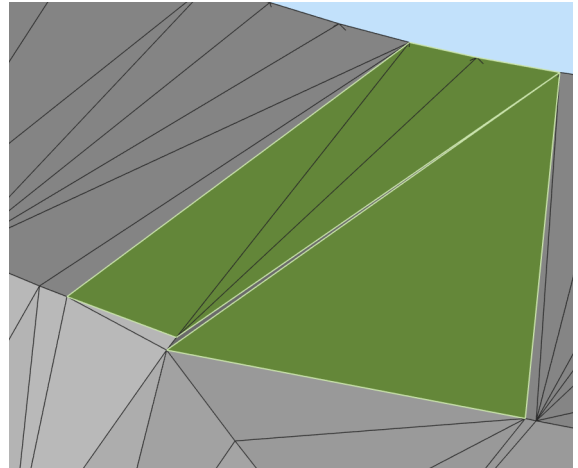
Sometimes triangles are missing, leaving a gap in your design. This is also confusing for the printer, as it won't have enough information to print the part properly. It tends to deal with these gaps in a similar way to the way it deals with inverted normal: your printer won't know where the design begins or ends and it will end up printing only the contour (and neglect the inside of each slice) or it will continue to print when it should stop.

It's important to note that it's completely possible to print a design with intentional holes; these holes just need to be properly defined with correctly placed triangles.



3. Avoid overlapping triangles

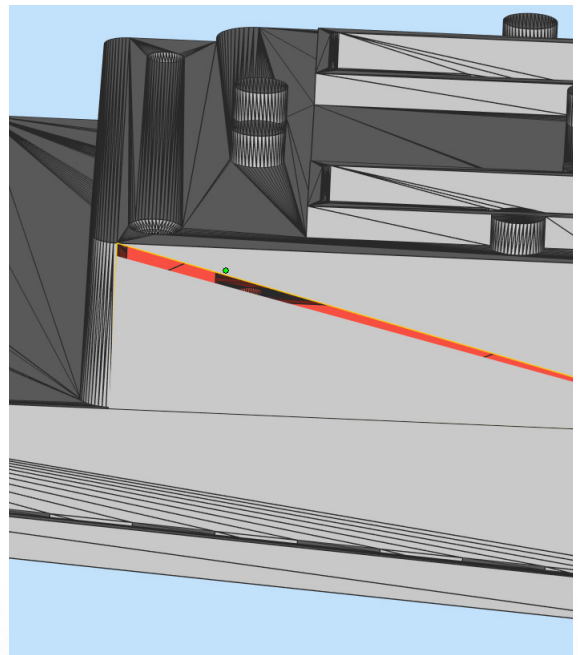
When designing a more complex model, you will probably need to combine two shapes or subtract a part of your model (i.e. applying Boolean operations). As a result, the internal geometry of your model is no longer optimal, and you'll need to clean up the redundant triangles that overlap each other. It isn't a fatal flaw – but it will cost the printer more time and material to print your model. It's also extremely tricky to spot on a finished model, as there could be layers upon layers of hidden redundant triangles worked into your design without you noticing



4. Stitch bad edges

When two or more triangles aren't connected to each other, we refer to it as "bad edges". In theory, bad edges can also simply be considered as gaps in the design, as the triangles that surround the gap can't connect to an edge anymore. When your design contains bad edges, it means that you don't have a watertight design. You need to stitch these bad edges to get a manifold, printable model.

Another variant of this phenomenon is called "near bad edges" and occurs when two triangles are close to one another, but with edges that don't completely touch. This isn't always visible on your screen, so it's always best to be careful and let your STL editor indicate the errors. When you have a whole series of bad edges, it's referred to as a "bad contour".



5. Remove noise shells

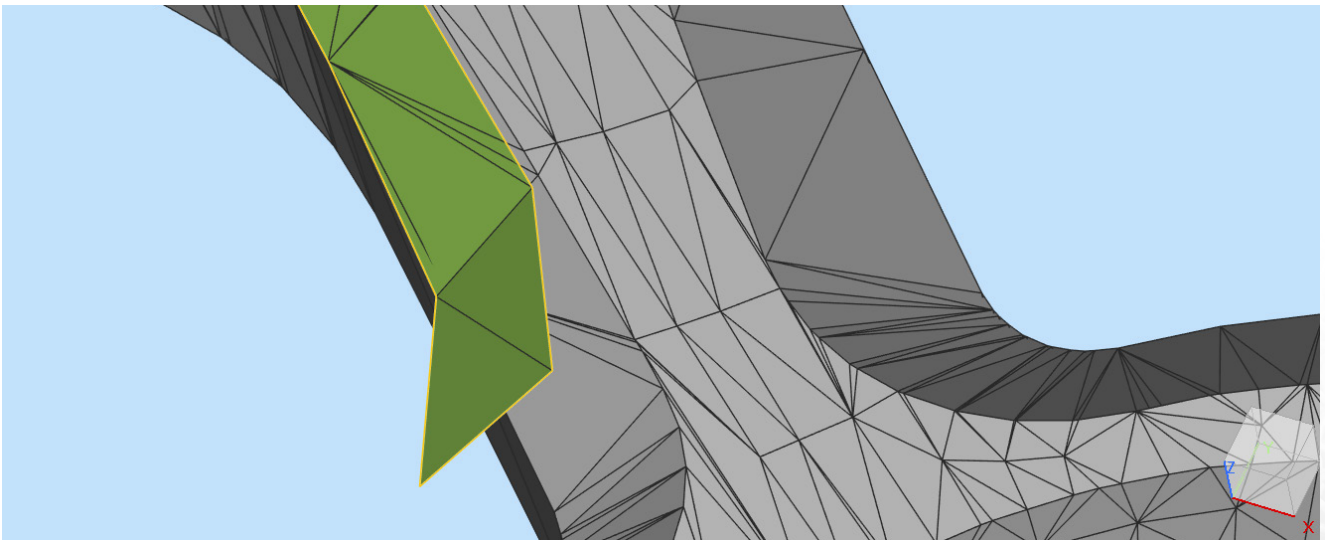
Not the kind you find on a beach, in 3D Printing a shell is basically a group of connected triangles which forms an autonomous 3D object in your file. These shells can overlap one another or be separated by space. A frequent problem with shells is when they have inverted triangles, or when the triangles on intersecting shells touch each other but one side is face the wrong direction.

Another phenomenon can occur, where a shell is so small that it becomes unnecessary. These types of shells are called “noise shells” or “orphaned shells” as they barely have any volume. Think of it as a wrinkle in an otherwise smooth piece of fabric that needs to be ironed out. The easiest way to remove noise shells is by flipping inverted triangles, and finally the remaining useful shells in your design will need to be unified into a single, solid volume. It is not necessarily a bad thing if your design has multiple shells – but the print time will increase significantly.

To make this all a little more clear visually, imagine drawing the outline of a shape on a piece of paper. If you trace the same outline a second time, it will become thicker, much like the way a 3D printed object gets thicker and stronger the more shells it is printed with. When printing a hollow design (a fairly standard practice as the model will be lighter and more cost-effective), the most common tactic is to print an outer shell (called an “outline”) and an inner shell (called an “insert”). The inner layer is composed of inverted normals in order to signal to the computer that it is designing a hollow model, and besides, a design with a single shell won't be strong enough and is likely to be printed badly. Too many shells are also bad for the object, and the maximum recommendation is five.

The ShrinkWrap tool

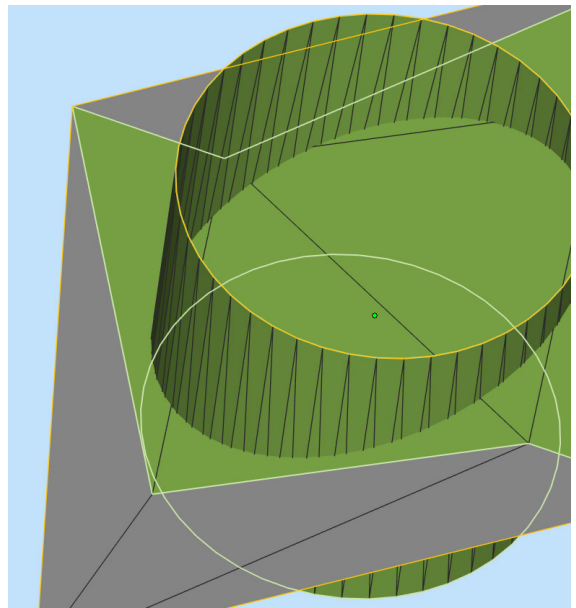
For files with very bad quality, the automatic repair of flipped triangles, bad edges, holes and rough surfaces may not be sufficient. In that case, you can apply one of Materialise's impressive fixing tools, called ShrinkWrap. This complex algorithm will place a thin layer around your model that shrinks afterwards and repairs difficult errors, while keeping details intact.



6. Trim or unify intersecting and overlapping triangles

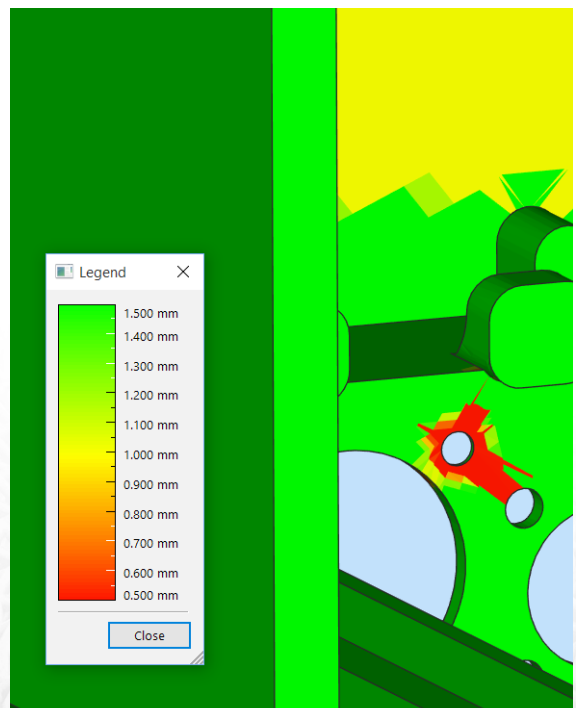
Another common problem that novices in 3D Printing may come across is triangles cutting each other. When triangles intersect, you will have to cut off the sharp edge and trim the triangles to obtain a unified design. If you forget to do this, the 3D printer will get confused again about the inside and outside of your model and will not know what part of the bounding box to fill up, resulting in a failed print.

Triangles can also overlap. When this happens an edge of a triangle is shared by more than two faces. In this case the printer will have a hard time calculating the production path. It's not clear to the machine what it needs to do, so it can slice your model in a way that the laser or nozzle simply continues to print your model with two identical edges on top of one another. Materialise Magics software removes the duplicates that are part of the overlapping triangles, thereby getting rid of the calculation problem and ensuring that you have a unified solid model.



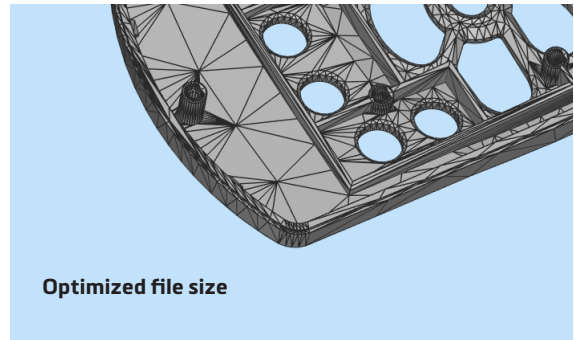
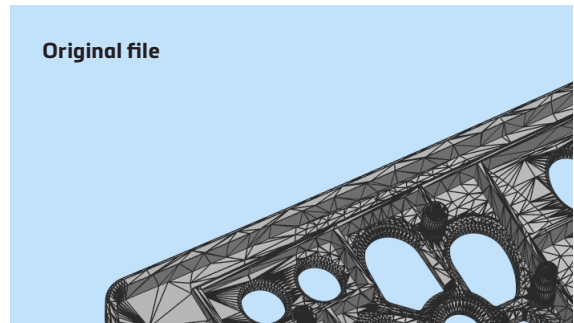
7. Check wall thickness

Fun fact: 1 in 5 orders received in 3D printing bureaus have to be cancelled due to incorrect wall thickness. It's exceedingly common, and it's a very important part of your design that you need to get right. When using 3D modeling software, it is possible to design a surface without implementing wall thickness. However, the 3D printer will need information about how thick you intend the wall of your object to be. It's a bit of a balancing act: the proper wall thickness means that you need surfaces which are thick enough to be 3D printed successfully and contribute to a strong, printed part on the one hand, and on the other hand thin enough to save as much material as possible. With Materialise Magics, you can analyze wall thickness and fortify the walls where needed.



8. Minimize file size

Every triangle your design is made up of uses memory in your computer. Therefore, STL files with more triangles are heavier to process and slice. In many cases, the 3D printer won't even be able to print a file over a certain amount of polygons, making triangle reduction – also known as polygon reduction or mesh decimation – an essential component.

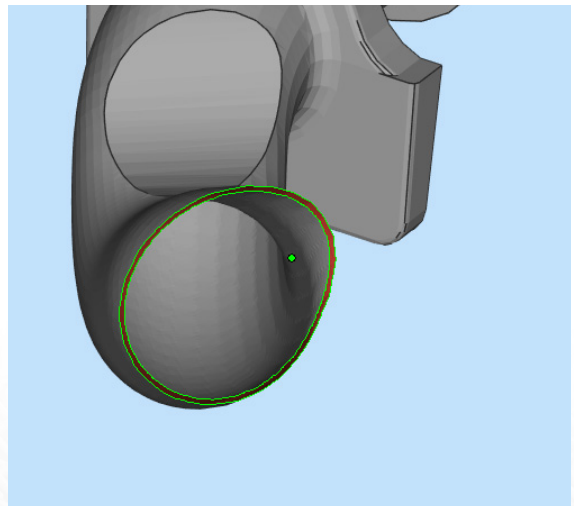


9. Rescale your 3D model

When can rescaling a 3D model be useful? First of all, since STL files don't contain distance unit information, you can use a rescale tool to rescale your model to the right unit and size. Secondly, to overcome some wall thickness issues, you can shrink or enlarge your model to the appropriate size. Thirdly, when ordering your 3D Print, size influences the price you will pay. And finally, parts are often rescaled for compensating shrinkage when printing.

10. Hollow parts

3D Printing can be a costly business, so to waste less material while printing, you can hollow out your entire model. With Materialise Magics, you can hollow models while maintaining the freedom of deciding the optimal wall thickness for printing. This ensures that the printed version will be as economical and environmentally friendly as possible, while still retaining its optimal structural integrity and great looks. In fact, all of the actions described above can be performed with Materialise Magics software.



About Materialise

Materialise incorporates nearly 30 years of 3D printing experience into a range of software solutions and 3D printing services, which together form the backbone of the 3D printing industry. Materialise's open and flexible solutions enable players in a wide variety of industries, including healthcare, automotive, aerospace, art and design, and consumer goods, to build innovative 3D printing applications that aim to make the world a better and healthier place. Headquartered in Belgium, with branches worldwide, Materialise combines one of the largest groups of software developers in the industry with one of the largest 3D printing facilities in the world.

For additional information, please visit:

www.materialise.com