ADDITIVE MANUFACTURING CERTIFICATION INFORMATION

Additive Manufacturing Introduction

Lesson 1

Additive manufacturing is a rapidly expanding technology that's opening up new possibilities for how parts are made and what types of items can be printed. There are a few primary printing methods leading the industry, namely fused deposition modeling and stereolithography. I'll get into more detail in the next video about the differences between them. Each of these processes can produce different qualities of parts, and they come at a variety of price points. This makes it important to know what type of machine you'd be interested in purchasing and what the capabilities of that machine will be.

Beyond these two machine types, other companies produce additive manufacturing technology that can print metal, biomaterials, and even food. Additive manufacturing is changing the mindset of how parts are created. While most manufacturing processes have focused on a subtractive process, which removes material from an existing block, an additive process only uses material to make up the part geometry. This means that significantly less material is wasted, the cost of manufacturing goes down, and the part is generated immediately in its final shape.

There are a couple of additive manufacturing methods leading the industry. The most common type is called fused deposition modeling or FDM. In this process, a strand of plastic material is pushed through a heated nozzle that moves in a programmed pattern, placing the heated plastic strand in the shape of a single layer of the part. These layers stack on top of one another until the entire part is printed. Another common method is called stereolithography or SLA. In this process, a pool of liquid resin is heated to a specific temperature, and a laser moves throughout the liquid, hardening the resin at the point of contact. This laser moves in the pattern of a single layer of the part, and then the machine will adjust and move on to the next layer.

There are several other additive manufacturing processes out there. Many of these processes use a powder-based system that polymerizes the powder and forms a solid. While this process can make much higher quality parts along with the ability to print metal, it's a much more complex process and requires additional cleanup and post-processing. While these processes are revolutionizing the way parts are manufactured, the cost of buying a machine and printing parts varies between them. FDM printers are typically the lowest cost machines both in initial purchase and material. The hardware that makes up FDM printing machines isn't nearly as expensive as other machines that use a projected laser. In addition, the spools of plastic strands used by FDM printers are widely available and created at a low cost. So if you're wanting to get started with your own printer, this can be the cheapest option to get going.

SLA printers are more expensive than FDM printers. However, they often come in a smaller footprint and are available in desktop sizes. This means that they aren't nearly as expensive as the other powder-based printers, which use a more complex process with high-end components and are much larger dimensionally. However, SLA printers are making a strong push in the 3D printing market and are coming to a more competitive price point with other FDM printers.

As mentioned earlier, 3D printing technology is continually expanding and is able to print materials other than plastic. Metals such as steel and aluminum are in high demand, but other materials can have a tremendous impact. Medical industries can now print biomaterials such as skin cells, organ cells, and even dissolving implants. This could revolutionize healthcare and the procedures available to those in need. Even peripheral industries like the restaurant and culinary industry are benefiting from the ability to print food from a machine. While the possibilities of 3D printing aren't slowing down anytime soon, it's important to know how the technology works so that your prints are successful, high-quality products. In the following lessons, you'll learn more about the technical details of the machine and the best practices for printing your parts.

Machine Types

Lesson 2

3D printing is one of the fastest growing industries in manufacturing technology. Because of this growth, there's a variety of printing methods and machine types available. To begin, I'd like to walk you through some of the most common printing methods used today.

The first method, known as fused deposition modeling or FDM, takes a long coiled strand of plastic and pushes it through a heated nozzle. This process is similar to using a hot glue gun, where the material is pushed through the hot tip and applied to a surface. In the same way, an FDM printer pushes the material through the heated nozzle while the nozzle head moves about in a pattern in the shape of the part. This means that the nozzle moves along two axes in a single plane. This is done for a single layer at a time. Then the build plate that the part's printed on moves downward so the next layer can be printed on top of the last one. The build plate moving vertically is the third axis of motion.

The next method, known as stereolithography or SLA, is very different from the last method. In stereolithography, a laser is projected into a pool of liquid resin and hardens the resin at specific spots, which forms the shape of the part layer by layer. The laser is controlled by mirrors that move underneath the pool of resin. This means that only the build plate moves upward and downward in a single axis.

These two methods are the most common for smaller benchtop models. However, there are more advanced forms of 3D printing. The process known as selective laser sintering or SLS is similar to stereolithography but instead directs a laser in 3D space and hardens a powder at the contact point. A laser is still used in this process to produce a solid part. However, it interacts with a powder instead of a liquid resin. A subgroup of SLS, known as selective laser melting or SLM, uses a laser to melt and fuse metallic powders together. This process is used to create metal parts in particular where SLS can't achieve this result.

In the next video, I'll discuss how these different printing methods are incorporated into today's 3D printing machines. You'll learn why some of the printers have an open or closed printing area, as well as why some machines are larger than others.

Because the different printing methods vary quite a bit, the 3D printing machines also vary in size and structure. An FDM printer, such as this Ultimaker 3 model, has an open build area that's accessible during the printing process. This helps the heat dissipate during the print by increasing airflow. It also gives the build plate a much larger distance to move downward, which means you can print taller parts. Since FDM is one of the most common printing methods, many companies package their printers in benchtop size models that don't take up a ton of room and can be moved without too much difficulty. However, there are much larger versions of FDM printers available, which can print much larger parts and can print thicker spools of material at a faster rate.

One advantage of FDM printers is that multiple spools of material can be printed in the same part. This allows for one type of material to create the main structure of the part, while a second water-soluble material can be printed for the supports and then later dissolved away once the part's finished printing. Or if you want to use multiple colors in a single part, two different color spools can be used to intermix them in the printing process.

An SLA printer, such as this Formlabs 2, has an enclosed area that keeps debris and other foreign objects from entering because the resin needs to be consistent in order for the laser to harden the liquid. Floating debris or other objects can't be allowed to get mixed in; this could disrupt the printing process or even damage the machine. The contained space is temperature-regulated, so having a closed build area is not an issue for SLA printers. This printer type is also incredibly common in the market and is packaged as a small benchtop model that can be moved around if needed. However, there are also much larger SLA printers that can produce incredibly high-quality parts that are fairly large scale.

The other printing methods, such as SLS and SLM, are typically housed in much larger printers because the processes are more complex and require more parts. One advantage of using these larger, more sophisticated machines is that you can print some of the highest quality parts, eliminating the need for other manufacturing methods. They also can handle materials other than plastic, such as an SLM printer creating metal parts. These machines come at a much higher price point and are typically used by companies that offer 3D printing services.

With these different printers available on the market today, your designs can be made real in your own home in just a matter of hours. Imagine how this technology will continue to evolve and make some of the most complex products available at your fingertips.

Materials
Lesson 3

One of the most important aspects of 3D printing is the material used in the process. Not only does the material have to work directly with the technology, but it also has to have the right material qualities that make the printed part durable and functional. Fused deposition modeling printers, or FDM printers, use large spools of plastic that draw a continuous strand into the heated extruder nozzle. These plastic spools are typically comprised of either PLA or ABS; however, other plastic types are also available. Stereolithography printers, or SLA printers, use a pool of liquid resin that's hardened using a laser. The printer regulates the temperature of the resin to make sure it's consistent. Depending on the printer being used, a number of different resins can be used to produce different material qualities in the part, such as making it flexible or having a specific color.

In this lesson, I'll walk through how the different materials function within the printers and what options are available depending on the printer. To begin, I'll discuss which types of materials are used for fused deposition modeling printers, or FDM printers. To quickly review, FDM printers push a strand of plastic through a heated nozzle. The temperature of the nozzle allows for the plastic to become soft and malleable and can be printed in the shape of the part. The material must also fuse to the plastic printed on any lower layers. This means there are a couple of critical aspects to which materials can be used and how they can best be applied.

FDM printers most commonly use a plastic known as PLA, which stands for polylactic acid. This is the material that will be referenced for FDM printers throughout the remainder of the additive manufacturing lessons. This plastic has a few important properties: it can soften at a temperature that's easy to regulate in a 3D printer, it's a bioplastic which makes it biodegradable, and its fumes are less toxic than other plastic types, making it important to consider if the 3D printer doesn't have optimal ventilation. Many FDM printers also use ABS, which stands for acrylonitrile butadiene styrene. This plastic also melts easily and can print very evenly. The main advantage of ABS is that it's much more durable and produces higher quality finished parts. This is because ABS doesn't soften as easily as PLA due to its higher glass transition temperature material property.

There are many other plastic types that can be used in FDM printers. This Ultimaker printer primarily uses PLA and PVA for dual material extrusion. However, this model can also print polypropylene, nylon, ABS, CPE, PC, and TPU95A. Dremel also makes an FDM printer that utilizes some of the most common plastic types, namely PLA, ABS, and nylon. These materials are typically coiled around a large spool that attaches to one of the sides of the printer. This allows for the strand of plastic to feed up and over the top of the machine into the moving extrusion nozzle.

Now, I'll switch over to discussing which types of materials are used by stereolithography 3D printers, or SLA printers. These printers utilize the process of photopolymerization. In this process, a pool of liquid resin material has an ultraviolet laser that moves and projects the light into the areas that make up the solid model. X and Y scanning mirrors control the position of the laser. The material at this location in the resin pool then hardens and bonds with any other material that's been hardened beforehand. The laser will harden all the material in a single layer, and then the build plate will move in the third axis to work on the next layer. Because the laser shines upward through the resin pool, the part is printed upside down and rises upward out of the resin.

The resin used in the 3D printer is designed to have specific characteristics once the laser hardens the liquid into a solid. For instance, Formlabs offers a variety of resin types that produce different solid qualities. There's a standard resin that comes out semi-translucent, a white resin, a tough resin that has added durability, a resin that's castable which can burn out cleanly without ash or residue while still capturing fine detail and smooth surfaces, a flexible resin used to produce parts that bend and compress, and a dental resin that produces high-quality dental devices in-house such as surgical guides, orthodontic models, and retainers. These resins are typically loaded into a cartridge that slides into the machine and can be replaced as the resin runs out.

With all these resin types available for SLA printers, the limits of creating new parts are ever-expanding. You can be confident that your parts will be functional and serve their purpose right away.

Model Preparation

Lesson 4

When preparing the model to be printed, there are several factors to take into account before sending the file to the printer. Part orientation is one of the most critical aspects of achieving a successful print, which means minimizing overhangs on the model and making sure support material is in place. SLA parts may need to have a drain hole included so that liquid resin isn't trapped inside the part. The amount of infill or solid material also greatly affects the part as well as the desired wall thickness if the part isn't entirely solid. Adjusting scaling will also ensure that parts are sized correctly and can fit together. Lastly, the smoothness and finish quality are determined by a couple of factors such as material thickness and adding any post-printing procedures to polish off the final design.

With both FDM and SLA printers, orienting the part correctly is critical to achieving good results. With a part printed on an FDM printer, minimizing the number of overhangs will ensure that the layers will correctly adhere to one another and the geometry will be built correctly. Typically, anything more than about 55 to 60 degrees of overhang will require support material to keep the layer from collapsing. This would cause a layer to disconnect and potentially a print failure. Here I'm using Ultimaker's Cura software to prepare the model for printing. To orient the part, I want to make sure the overhangs are minimized and that the footprint is as large as possible, especially if it's a flat surface. This means I need to rotate the part so that this face is parallel to the build plate, then click on the lay flat button to make sure it's laying directly on top of the build plate and use the move tools to position it near the center. In this orientation, notice that the overhangs have been minimized, and the largest surface is flat against the build plate to make the largest footprint.

With SLA, it's also important to consider how supports will be added but not as critical as with FDM. There's also one important feature of printing SLA parts: if you're printing a part that is hollow, there should be either a natural drain hole so the part doesn't trap resin inside or you should model one in such a way that resin won't be trapped in a cavity in the part. Here I'm using the Formlabs PreForm software. Orienting a part in the software can be a very simple process. If I click on this icon, one-click print, the software will automatically orient the part to the optimal angle and position, followed by generating supports where necessary and calculating the layout. Notice that the part isn't flat against the build plate. If needed, these three steps can be manually adjusted to your preference. I can click on any of the icons, such as the orientation options, and make adjustments as needed. This is the same for the supports, such as adjusting the density or adjusting the layout by dragging this icon to position it on a different area of the build plate.

Next, I'd like to talk about the concept of infill, which refers to how much of the part volume is considered to be solid. The more solid the part is, the more rigid it will be, but it will also consume more material and time. Often, the infill isn't solid material but is created using internal structures that act like a solid. The slicer software can automatically incorporate this, removing the need for you to incorporate this into the 3D model. For parts that need to function without breaking, a solid infill is ideal. If a part is non-mechanical or doesn't have to sustain external loads or pressures, you can reduce the infill down to the 10 to 20 percent range to save time and material.

Another key aspect to consider is determining the part's wall thickness. The wall thickness affects multiple aspects of the printed part: strength and durability, amount of material needed, how long it'll take to print, and how refined the finished part will look. However, keep in mind that this only applies to parts that have less than one hundred percent infill. Most slicer programs will also let you define the number of layers a part will have before it begins to create the infill. In FDM printers, wall thickness is typically set as a multiple of the size of the extruder nozzle opening.

One of the key steps in creating a correctly sized part is adjusting the part's scale. Sometimes a printed part will shrink slightly as it cures, particularly for SLA printed parts. To account for this, the slicer program can be used to scale up the model by a few percent prior to printing. You don't have to account for this in the modeling software. In addition, if you're printing parts that fit together, such as a bolt going through a mounting hole in a part, you'll need to make sure that the part going inside isn't too large. So scaling the parts accordingly will help make sure the parts fit properly together. Scaling down one of the two parts will most likely take care of it.

With all these considerations in mind, it's important to adjust settings to achieve the finish quality you're looking for. Because 3D printed parts are created in layers, the layer height has a big effect on how smooth and refined the finished part appears. Thinner heights will produce a smoother part but will drastically add time to the total print time. If a very fine finish on a part is necessary, post-finishing of the part should be considered as a solution since it will keep the print time and material consumption low. This can include sanding, painting, and polishing.

File Export

Lesson 5

Export settings greatly determine how successful a printed part will turn out. This includes making sure that the 3D geometry is structurally sound as well as aesthetically pleasing. Some of the important export settings include saving the STL file from its CAD software with a certain amount of roughness or smoothness, orienting the part correctly in the slicer software, adjusting the layer thickness, and including supports in the print. In this lesson, I'll cover many ways you can refine your print, ensuring that your part will come out fully functional and looking just how you intended it to be.

In order for a 3D printed part to come out successfully, there are several settings to consider in both the CAD environment and the slicer software. With the file ready to be exported from SolidWorks, you'll want to go to File > Save As, and then choose STL from the available file types. This is the most common file format used to export 3D printable files. When I click save, SolidWorks breaks up the part into tiny triangular segments. The larger the number of triangles, the more refined the printable file. This results in a smoother part but increases both the file size and print time. If you want to change the number of triangles used to make up the 3D geometry, the options can be adjusted from the same Save As menu. I'll open it again, make sure STL is selected, and click on Options. There are a few ways to make adjustments. You can select coarse or fine, and when I check the box for Preview, the number of triangles is shown. If you select custom, you can adjust the deviation and the angle, which will increase or decrease how many triangles are created in the STL file. I'll click OK and then save, then yes, and the STL file is generated.

Other 3D printing file types that are supported are 3MF files and AMF files. Additionally, some slicer programs include the ability to directly import SolidWorks and other native CAD files. This eliminates the need to export them to a neutral format like an STL file. Here I'm using the Formlabs PreForm slicer software. Orienting the part and preparing it for printing is simple; just click on the One-Click Print icon, and all the work is done for you. However, you can always make adjustments as needed. For instance, if I knew I needed to rotate the part by 30 degrees around the X-axis, I can click on the Orientation menu, then type in 30 in the Orient X field. This adjusts the part orientation, and the supports disappear. To add the supports back in, I can click on the Supports menu and make any adjustments as needed. I'll leave the defaults and click Auto Generate Selected. From here, I'll make sure it's positioned properly on the build plate by clicking on the Layout menu and moving it into position. Now the part is in place and ready to be printed.

Now I'll briefly switch over to Ultimaker's Cura software, which uses an FDM printing process. With the STL file imported, I'll make sure that the part is oriented in such a way that it'll produce the optimal print. I want to minimize the number of overhangs and make the footprint as large as possible. This means that if I rotate the part so that this face is the base, the footprint is as large as possible and the overhangs are minimal. If a part like this one is intended to be structural and withstand stress, the printed layers should be perpendicular to the direction of applied stress. Since the direction of the layers on an FDM printed part is the weakest, the strongest part will be created if the applied load is perpendicular. In this software, once the model is oriented correctly and any additional supports are included, the printer type can be selected here along with custom settings such as the thickness of each layer, the amount of infill, or if you'd like to include build plate adhesion. Once your settings are in place, you can click the Slice button to prepare the model for printing, and you'll see Cura displays an approximate print time. Finally, you can save the file and begin the print.

In the Formlabs PreForm software, the part can be sent directly to the printer by clicking on this button and selecting the available printer connected to your machine. Or if you need to save the file to a thumb drive or other storage device, you can go to Save As and save the part as a PreForm job file.

Machine Preparation

Lesson 6

In order to successfully print a part, you'll need to ensure that the machine is prepared properly. One of the most important areas to prep is the build plate. Since the first few layers of a print are the most important, the build plate should be prepped so that the surface allows the material to adhere to it. You'll also want to make sure that there aren't any objects obstructing the build area.

Loading the material is also an extremely important process. FDM printers need to have the spools attached and oriented correctly, along with the strand feeding evenly into the extrusion nozzle. SLA printers need to have a sufficient amount of resin remaining in the cartridge and ensure there aren't objects floating in the resin tank prior to printing. Once these are prepped, the access doors can be covered, guards put in place, and the build area checked for clearance. From there, you can begin printing.

The build plate is one of the most important areas to prepare on a 3D printer. There are a couple of different methods depending on the type of machine you're using. When using an FDM printer, the build plate is most often made of glass, which is the most common practice with current generation machines. Due to the smooth nature of glass, the addition of some sort of adhesion material will aid in making sure that a print will stick to the build plate. Some suggestions for materials to use on the build plate include blue painters tape, hairspray, or the use of a glue stick. Because the adhesion of the first layer of an FDM print is so critical to the successful completion of a print, it's highly suggested that one of these methods be used.

In the case of using an SLA printer, it's best to make sure that the build plate is clean and free of debris and that the resin tank is clear of any foreign objects that may disrupt the path of the laser.

Loading the print material is another important aspect of preparing the machine. There must be enough print material present, and it needs to be loaded properly. Loading material into an FDM printer will vary by manufacturer, but the most common steps will include mounting a spool holder and then attaching the spool. Be sure to orient the spool the correct way, which means that the strand of material feeds smoothly into the guide without any kinks. If the spool is oriented correctly, it will rotate easily on the spool holder and allow for an even feed into the extrusion nozzle. Additionally, be sure to handle the spools of material in a way that the strand doesn't overlap and get tangled around itself before attaching it to the machine. This could bind the strand and keep it from feeding into the printer.

In the case of this Dremel machine, you'll need to remove the access cover and then manually feed the material into the guide tube. At that point, you can run the material load function on the printer to conclude the process. With this Ultimaker 3 printer, the spool holder needs to be mounted and plugged in, and the spool end holder should be used to keep the spool on the machine. The Ultimaker spool holder includes an NFC reader which can transmit the material type, color, and diameter directly into the machine and into the slicer software. Once mounted, you feed the material into the feeder here and then complete the process on the machine interface.

In the case of most FDM printers, once the machine has successfully started extruding material through the nozzle, it's best to let a little bit of material flow out to ensure that there aren't any clogs. You can also watch to make sure any previous material has flowed out in the case of a color change.

With an SLA printer like this Formlabs Form 2 printer, you can always add additional resin once the installed cartridge runs out. An SLA printer will first need to heat up the resin in the build tank prior to starting the print. This is an automated process that will take place once you begin a print. In the case of most modern printers, they will allow you to pause a print should it get close to running out of material, then the machine can be refilled with additional material, whether it uses a spool or a resin cartridge.

Once the build plate is prepped and the material is loaded properly, you're ready to begin printing. You want to make sure that all the access doors are closed. Not every machine incorporates safety switches. When beginning to print, it's also a good idea to make sure the printer is placed in a well-ventilated area, especially with SLA printers since the heated resin can produce a noticeable smell.

Printing the part

Lesson 7

When it's time to print your part, you want to make sure that the printing process will go smoothly. This means that you want to be sure you can send the file to the machine correctly, whether it's using a USB cable, thumb drive, or even over the internet. Then check that the first few layers are printing properly and continue to check on the part from time to time during the printing process. Gauging the progress at each of these stages will help ensure that your part prints as intended or that you can cancel the print at the right time if there's an issue with the printer.

3D printing software provides a variety of ways for the print file to be sent to the machine. One of the most direct ways to send the print file is to connect a USB cable from the computer to the printer. This will allow the file to print right away. Another way for the file to be sent to the printer is by exporting the file from the slicer software and then loading it onto a USB thumb drive. As long as the printer has an available USB port, the thumb drive can be inserted and the file can be printed. Similarly, some machines also have an available slot for an SD card. Just like the USB drive, simply export the print file to the SD card and then insert it into the printer. Additionally, some printers can receive files over a Wi-Fi network or a wired Ethernet connection. This makes it even simpler to print if both machines are using the same internet signal.

Some machines have more advanced monitoring systems. Some printers have web monitoring where you can watch the progress of the print in real time from your computer no matter where the printer is located. So you could be at the office watching a new file print at home. Some printers also have the ability to send a text message or email to the user when a print is finished or if an error occurs and the machine needs attention.

Now that I've discussed how to send a file to the printer and monitor it, I want to talk about the actual printing process. The very beginning of a print is the most important stage. This is where the first layers are formed, which is the foundation for the rest of the part. It's recommended that a user watches the first set of layers, maybe three to five, to make sure that the print is running smoothly. If an issue occurs at this stage, the part most likely won't print properly.

In an FDM printer like the Ultimaker and Dremel models, the first layers must adhere to the build plate for the part to turn out correctly. For every print you do, watching the early stages will help ensure you get the results you're looking for. With the first few layers printing successfully, you can gain some confidence in the printing process. However, it's still advised that you check on the part every so often to make sure it's still on track.

While the downside of not checking on a print can have minimal consequences should a part fail, it may save you time and material if you're able to stop a failed print before it spends hours printing incorrectly. Sitting near the machine during the print can also be helpful, as it may make strange sounds that indicate if a part isn't printing properly. This will allow you to easily check on it and cancel the print if needed. In general, 3D printers are fairly hands-off and can require little to no supervision when set up and started correctly.

Post printing

Lesson 8

Once a part is finished printing, it's time to take it off the build plate. You may want to remove the build plate from the machine to make it easier to pry off the part. If you do need to pry it off, be very careful not to injure yourself if you're using a tool with any kind of sharp edge. Once the part is removed, you can replace the build plate back in the machine, making sure it's set properly and is level once again. If you're done printing parts for the day or any extended period, be sure to shut down the machine and remove any printing material that may clog the machine or cause problems down the line.

It's a great feeling once a successful 3D print is finished, and you may be eager to grab it and quickly remove it from the build plate. However, there are some things to consider before taking it off. Because of the use of adhesion materials on FDM printers, like blue painters tape, hairspray, or glue stick, some prints may be tough to remove. Using a glue stick may be the trickiest of the three. You want to be sure that removing the part doesn't damage it in the process. Here are some suggestions on removing the part properly off the build plate.

Extreme care should be used when removing the part from the build plate. In many cases, you may need to pry a part off using a tool. A putty knife is a great tool to get under the part and pry upward. This doesn't have a particularly sharp edge. Whether using a semi-sharp tool like this one or potentially something with a sharper edge, you should always pry away from your body to avoid injury. In addition, parts will always be easier to remove as soon as possible after print completion as opposed to trying to remove them after they've sat around for a while.

You can also be strategic during the design phase by integrating a feature on the base of the part that will allow you to easily remove it. A pry slot located around the outer edge will greatly aid in this process, but this may not always be possible given the part's design and geometry. Once the part has been removed, it's also the best time to make sure that the build plate is free of any leftover material and ready to be used again.

In the case of the Formlabs SLA printer we're using, the manufacturer has supplied a fixture in which to mount and hold the build platform for easier part removal. Their slicer software also produces a continual pry slot around the raft section of the part, which you can use their pry tool to remove the part more easily. In many cases, it may be best to remove the build plate from the 3D printer to get better access and leverage when removing the part. Each machine is slightly different in how the build plate is removed and replaced. Once you know how to remove it, you typically follow the same steps in reverse to replace it in the machine.

After you pry the part off the build plate, make sure all the surfaces are clean and there aren't any pieces still stuck to it or any debris left over. This will help ensure that the plate will nest back into the machine securely. If you do need to replace the build plate, it's best to take a bit of time to make sure the plate has been securely placed and to be sure everything looks visually correct. This will help make sure it's ready for the next print. If the plate looks crooked or doesn't seem secure in its position, it's best to remove it and try again. Some printers also have a leveling procedure you can follow after the build plate has been replaced, which will guarantee that the build plate is flat before printing your next part.

If you're done printing parts for the day, you want to make sure the machine is either in rest mode or shut down. If it's not going to be used again right away, it's best to fully shut down the machine and power it off. Most FDM printers will do just fine leaving cold material in the extrusion nozzle, but it's recommended to purge any material from it using the load or purge function of the machine if it's going to sit for an extended period. For SLA printers, you can simply empty any unused resin back into the cartridge of the machine if it will not be used for some time. The resin is UV sensitive, so be sure it's not exposed to direct UV light prior to adding it back into the cartridge. This could cause misprints and other issues in the machine in the future.

Part finishing

Lesson 9

Adding the finishing touches to a part may seem like a short step, but there are many considerations to be made. First, removing the support material should be done with care and may require tools to get rid of all the leftover material. There are also washing and curing machines for SLA printed parts made from resin. Second, you want to make sure you check that the part will function as intended. This can mean fitting the part into the larger assembly or fitting it into other parts to ensure everything lines up. From here, you can continue the finishing process. You'll want to sand the part by hand or by using a micro file to make it smooth. If the part was printed in sections, you'll also want to make sure those will assemble properly and then glue them in place. Finally, you can paint the model however you'd like, putting it to use and on display in its final state.

Once a printed part is removed from the build plate, it's time to remove the attached support material. Some material can be removed by hand; in other cases, removing supports can be made easier by using a small set of cutters or needle-nose pliers. Because the process can be tedious, it's recommended to optimize the amount of support material in the slicer software to minimize how much it will generate. When using an FDM printer such as the Ultimaker or Dremel models shown here, the orientation of the part in the slicer software can aid in reducing the amount of supports or the need for them entirely. In the case of printing an open-ended box, if you printed it with the open end on one side, you'd need to add supports. However, if you printed it standing up with the open end at the top, you wouldn't need any additional supports. With the introduction of PVA material and being able to use it in a dual extruder machine such as the Ultimaker 3, you can print water-soluble support material that only requires a tank of water to remove the support material as shown. While it may take longer to complete, it also allows for internal support structures that you wouldn't be able to remove otherwise. Raft material can also be difficult to remove, but using a raft can be helpful in printing large parts on an FDM printer.

When printing a part using an SLA printer such as this Formlabs model, the part is typically printed at an angle intentionally, and support material is added by default. Oftentimes, these supports can be removed by hand; however, using small cutting tools may also be needed. Because these parts are made from liquid resin, the parts may still have excess resin coating their exterior. This resin can be removed by washing it by hand with isopropyl alcohol. Excess UV resin used in the Form 2 machine is easily removed by using their specially designed machine called the Form Wash. It's optimized for removing all the remaining resin simply by agitating the solution while the parts are submerged and lifting them out to air dry when complete. There's also a lid that keeps the alcohol fumes contained inside the machine. Parts average about 15 minutes of soak time to complete. Resin parts also need some time to cure under UV light. This can be done by placing them in contact with sunlight for a period of time. Formlabs also created the Form Cure machine that rotates the part on a turntable while shining UV light directly onto the part. It also includes a heating system that preheats to get the parts ready for optimum post-curing. Both processes help solidify the part to the correct material properties after it's been cleaned.

Before investing too much time in the post-processing of a part, it's best to visually inspect the part to make sure that all features are there in case it needs to be printed again. Because the part will tend to dimensionally change for a short period of time after it's printed, it's best to ignore any sort of dimensional issues until after it's cured and you can verify if it will work to meet your needs. Once the parts are clean and most of the support material has been removed, the use of some fine sandpaper or fine micro files will aid in removing leftover support attachment points. Layer lines can also be smoothed out by sanding. This will help achieve a smooth finished part, which gets much closer to the final look and feel of the part.

Once the printed part is cured and sanded down to the intended smoothness, you can begin checking that the part will function as intended. Because printed parts' dimensions will change, especially right after being printed, it's best to wait a few hours to allow it to stabilize. In addition, the features that fit together need to be checked to ensure they will work. For instance, printing a one-millimeter hole and a one-millimeter rod will not allow them to fit together after they're printed. It's best to allow for some clearance, which can easily be added by slightly scaling one of the parts in the slicer software. Because material shrinkage is never uniform in all directions, it may take a few iterations of printed parts to get the fit and scale just right. You can closely estimate the shrinkage of a material by printing any one of the many dimensional test parts found online, but be aware that each machine may produce different results, and results can also vary between different batches of material. In any case, always be prepared to print a part more than once to get the final fit and function you desire.

If you printed a part that needed to be broken up into multiple sections since it wouldn't fit in the machine's build volume, you want to be sure that they fit together before using any adhesive. This is also a good time to consider part orientation when printing on an FDM printer because layers can easily separate. Printing a part where the layers are perpendicular to any sort of stress will be ideal in making sure it won't break apart between weak layers. Once they're ready to be glued together, you can apply super glue and fit them together. This type of adhesive works well on many 3D printed parts and gets the job done quickly. If you're unsure if a certain type of glue will work, find some scrap parts and test glue them to make sure it works. With all the sections assembled and the parts sanded down to your desired smoothness, the parts can be painted with any standard spray paints or brushed on with model paints. As expected, the cleaner the part is from any sort of dust or cleaning solutions, the better the paint will adhere and dry. Now your model is ready to be put to use or set on display for others to admire.

Software Options

Lesson 10

With the rapid advancement of 3D printing technology, part manufacturing is being changed in ways that make all new products possible. CAD software now integrates 3D printing tools directly in the user interface and leverages functionality such as generative design to create optimized geometry that fits the exact needs of the part. Online communities and file-sharing platforms are also making models available to anyone, which can serve as a part to print directly or use as a base model and make adaptations. These tools and platforms are also being applied to materials and applications other than 3D printed plastic parts. Metal parts are now able to be printed, which simplifies the manufacturing process and keeps material from being wasted. In addition, other non-mechanical materials can be printed as well, such as implantable biomaterials and particular food types. The world of 3D printing is going to continue to expand quicker than we may realize, bringing a lot of excitement to what the future holds.

Most CAD titles now include a number of 3D printing tools integrated directly into the user interface. SolidWorks has partnered with machining and software vendors to allow you to set up and create sliced parts directly inside the software. This allows you to quickly iterate designs without the need to deal with time-consuming and hard-to-track exported files. You can also see where a part needs supports added based on the orientation. In addition, you'll be able to see if the part will fit into your machine's build space. Tools such as 3D Expert for SolidWorks from 3D Systems can also automate processes such as combining an assembly file into one single printable part. Slicer software isn't the only technology being implemented directly into CAD. Generative design tools such as Design Guidance within X Design are helping designers create more complex but stronger parts that may not be able to be machined but can easily be printed. This allows designers to potentially use less material while gaining structural strength. And because 3D printing can orient the part and print it in various shapes and sizes, the designer doesn't have to be overly concerned with manufacturability. The resulting shapes tend to also have a more organic look to them as opposed to the traditional mechanical look, which can always be refined if needed.

One great way to begin working on a design is to see if there are existing 3D models that you can adapt and work from. There are some great online resources that make models available for download and direct printing or bringing into a modeling software and making modifications. SolidWorks hosts the 3D Content Central where users in the 3D experience community can upload their designs and share them with one another. These can be downloaded in a variety of formats, such as the SolidWorks part and assembly files or even an STL generated directly from the website. This is a great place to find models you're looking for since the native CAD file type is one of the available download options. Other websites that host community-shared models only make STLs or graphics files available, which are much more difficult to make edits. The website Thingiverse is also a great resource with tons of models available for download and printing. Depending on the member that uploaded the file, they may have also posted the native CAD files for download. It simply depends on which files that user chose to upload. In the next video, I'll discuss some of the ways in which 3D printing is advancing and what other materials and applications are being leveraged.

As 3D printing continues to be adopted by companies and individuals, new technologies are becoming mainstream. 3D printing isn't restricted to just plastics but has made its way into the world of printing metal. Metal printers from companies like Desktop Metal and Markforged are allowing companies to print final design parts at the lowest costs possible. With these machines, different types of metal materials can be printed, but all will require some sort of post-printing curing to fully harden the part. The ease of use of the machines, along with the easy-to-use software programs, is allowing for this technology to gain momentum at a very fast pace.

Another emerging technology is the SLS printing process. This stands for selective laser sintering. It falls between the costs and complexity of SLA and metal printing. It's similar to the SLA printers in that it uses a laser to harden material; however, it's solidifying a powder instead of a liquid resin. One of the largest benefits of SLS printing is the ability to print multiple parts that are interconnected by support material that can be easily broken away. Formlabs also makes an SLS printer that provides a low-cost entry point combined with a user-friendly interface. This makes it easy to get up and running making professional-grade parts.

Regarding emerging applications of 3D printing, the medical field, cooking, and space travel are all benefiting from this advancing technology. In the medical field, there are many emerging materials in development that are allowing for printed parts to be placed inside a human body without the risk of causing infections or the body rejecting the part. Experimental materials such as skin cells and body organs are allowing for further exploration into printing human body parts. Regarding cooking and food preparation, 3D printing allows for food to be stored in sealed, hygienic containers and then printed out when ready to be consumed. This can be extremely helpful when needing to transport food to impoverished countries without worrying about how the food will last as it's transported. Finally, in the use of space travel, the use of 3D printers is allowing tools and parts to be readily manufactured without the need for waiting for costly resupply missions, which can also take months to occur. This allows the astronauts flexibility in what they need, and the preparation time and materials can be reduced prior to conducting a successful launch. With 3D printing technology taking rapid steps forward, the future holds great potential for manufacturing and how efficiently new products can be created.