File Name	Narration	Special notes
1.1	[Laura] Welcome! You are beginning your journey into three D printing. New printers, parts, and add-ons are being developed constantly. While this hobby can be engaging, it can also be incredibly rewarding, and it will rarely be boring or monotonous. This course is designed to help you understand the basics of an Ender 3 3D printer, the types of filaments available, and what some of the most important settings are within your slicer. To begin the course, click Start. If you need a tutorial on how to navigate this course, please click on Navigation.	
1.2	[Laura] There are a few different options to help you navigate this course. The two most important buttons are the next and the previous buttons. The next button will take you to the next slide, while the previous button will take you to the slide before. You may see the submit button on quiz questions or knowledge check slides. Click submit to send your answer and move to the next slide. If you need to adjust the volume, please use the volume button. The seek bar will allow you to move forward or backward on the audio, video, or animations on the slide. Use this if you need to reread or listen to the audio again. The play and pause button will allow you to pause any animation, video, or audio until you are ready to listen again. Simply click pause or play to resume. You may use the menu on the left to navigate to different slides in the course, for example, if you need to review information before taking a quiz. When you are ready, click the next button to begin the course.	
1.3	[Laura] In this course, you will be exploring 3D Printing. By the end, you will be able to identify the different parts of an Ender 3 printer. Order the steps needed to correctly level the build plate. Describe the different types of filaments used in 3D printing and Manipulate settings in your slicer to obtain the desired printed object.	
1.4	 [Telephone ringing] [Laura] Thank you for calling Creality Customer Service. My name is Laura. May I ask whom I am speaking with? [Cole] Hello, Laura. My name is Cole and I'm calling for assistance in getting my Ender 3 working properly. I've tried a few different things but I can't seem to have much success. Between prints not sticking to the bed, a ton of stringing, uneven layers during the first few layers I'm frustrated! [Laura] I'd be happy to help you work through some of the issues you are having. Many first time 3D printing hobbyists find themselves in the exact same place, so don't feel like you are alone. We've all started at the beginning, just like you are now. 	Cree - a -lity Pause at the Emphasis frustrated

	Have you gotten to know the different parts of your printer yet and how they work? This will be important because as we talk about how to adjust, level, and fine tune your printer, you'll need to know what parts I'm talking about. [Cole] I've got a few things down, but most of them I don't know what they do. I'd really appreciate it if you can help with this part. [Laura]	
	No problem! Why don't we get started? Click next to Meet the Ender 3	
1.5base	[Laura] Ok, Cole. Now you are going to learn the names and functions of different parts of an Ender 3. Use the slide bar to rotate the printer 360 degrees. As you rotate, click on the information buttons to learn more.	
	Within this small black box, you will find the motherboard. This is an integrated circuit where all of the different electrical components connect to. It is the brain of the printer.	
	[Cole] How can I find out what motherboard I have? I've seen posts on social media that talk about different versions.	
	[Laura] In order to find out which specific board you have, you have to remove the cover from the box and look at what is printed on the integrated circuit inside.	
	[Cole] That seems easy enough!	
1.5a	[Laura] The gantry is the horizontal bar that the extruder moves along. This is also called the x axis. It is important to make sure that this bar is level and tightened by adjusting the concentric nuts. If it isn't, you will likely have printing issues.	
	[Cole] I've noticed that the right side of my gantry is pretty loose. So I should tighten those concentric nuts?	
	[Laura] That is the first step. Sometimes you must loosen all three, get it level, then retighten them. We will learn about another option to keep the gantry level a bit later in the tour.	
1.5b	[Laura] The hot end assembly includes fans for cooling and the hot end. The hot end melts the filament and allows it to be pushed through the nozzle. You will also find the thermistor here. If you ever have issues with thermal runaway, you may have to replace your thermistor.	therm-ist-er

	 [Cole] So the hot end assembly includes the fans, the hot end, the nozzle, and the thermistor? I'll have to make some notes about that so I remember it later! [Laura] Yes. The nozzle is here too. You may want to have some spare nozzles on hand so you can easily swap them out if you get a clog. [Cole] That's great advice. Thanks! 	
1.5c	 [Laura] The filament spool holder comes with the printer. Depending on your model, it may sit at the top of the printer, or it might sit at the bottom. This holds the spool of filament so that it can freely spin as the printer uses filament. Many 3D printing enthusiasts will use other after market. [Cole] I think I've seen some of those after market options on social media before I purchased my printer. I'll stick with the factory version for now. 	
1.5d	[Laura] The z-stop is a small switch that will prevent the z-axis from lowering too far and damaging the print bed. The stepper motor which is also located here moves the z-axis screw to move the gantry up and down. [Cole] Damaging the print bed? Yikes. That sounds like it could be an expensive fix. I better make sure that's in the right position!	gan-tree
1.5e	 [Laura] The build plate is where the magic happens. Also commonly referred to as the bed, it can also be the source of much frustration if your build plate isn't leveled properly. The build plate moves forwards and backwards during printing. The filament needs to stick to the bed in order for prints to be successful. There are different settings that will allow you to change how your print will adhere to the bed. You will learn this later in the course. If you hit a snag with your 3D printing, always start by making sure you have a clean, level build plate. [Cole] This is definitely one of the skills that I'm struggling with. No matter how often I try to level the bed, the middle is always so wavy. [Laura] We will talk about other ways to fine tune your printer in a little bit if you know your bed is level. 	
1.5f	[Laura] This stepper motor is for the x-axis. It is what moves the belts to which the hot end is attached. It moves the hot end left and right to allow it to reach all areas of the print bed.	

	[Cole] Do I need to do anything to maintain this part?	
	[Laura] Not really. Just make sure that the belt moves freely and isn't too dusty.	
	[Laura] The extruder is also shown here. Depending on the model of your printer, you may have an all-metal extruder, or one that has some plastic parts to it. The extruder is responsible for advancing and retracting the filament during printing.	
	[Cole] My extruder is all-metal. I've noticed that sometimes little particles of filament end up around the gears. I'll have to make sure to keep that area clean.	
1.5g	[Laura] The y-stop and the stepper motor shown here is responsible for stopping the bed from moving too far forward or backward. The stepper motor is attached to the belts which move the print bed. Between the print bed moving forward and backward and the hot end moving left and right, all areas of the print bed can be reached by the hot end.	
	[Cole] Are these two parts housed together like the hot end assembly or are they separate?	
	[Laura] They are separate, but they are right next to each other. The stepper motor will be attached to the belt that moves the bed.	
1.5h	[Laura] The power supply is where the electricity enters the printer. The power supply has its own cooling fans and can run on 220 volt or 110 volt. Check the red switch on the back to ensure that your printer is running at the correct voltage.	
	[Cole] I've got my printer plugged into a normal US outlet. That's 110v, correct?	
	[Laura] Yes, 110 v is typical in American homes.	
1.5i	[Laura] Some hobbyists will choose to add different components to their printers to enhance their experience. Since this printer has a long gantry, the owner opted to add a dual z-axis. This allows the gantry to move up and down on both sides equally. If you are finding that you cannot get the gantry level, this might be an option for you to add later. Earlier when we were discussing the gantry, this is what I was talking about. The Ender 3 Max has a much longer gantry than the Ender 3 or Ender 3 Pro, so that's why this model has the dual z-axis.	
	[Cole]	

	Ok. That's helpful to know. I have the Max model, so this will be something to look into if I can't get the gantry level.	
1.5j	[Laura] The interface is where you will control the printer. Some models will have a touch screen, while others have a rotating button. The interface is the display for what the motherboard is communicating. You will use the interface to preheat the extruder and bed, calibrate your e-steps, and fine tune other aspects of your printer.	
	[Cole] Quick question.	
	[Laura] Sure, what can I help with?	
	[Cole] The way the menus show up on the interface may be different depending on what firmware the machine has installed, correct?	
	[Laura] Yes. Some people opt to change their firmware. This is very important if they choose to add an auto-leveler to their printer.	
	[Cole] Cool. Thanks. This makes it much easier to understand all the different parts.	
	[Laura] I'm glad that's been helpful. If you feel like you know all the names and functions for each part, click next to test your knowledge!	
1.6base	[Laura] There will be two parts to this knowledge check. On this slide you will be showing that you know the name of the different parts of the printer. Move the stars from the images to the star within the rectangle. When you are finished, click submit. You will have two chances to get them right.	
1.6a	[Laura] Amazing! Keep going, rock star!	
1.6b	[Laura] You didn't quite get them all – Go back and move the pink stars and see if you can get them correct.	
1.6c	[Laura] Unfortunately, you didn't get them all. In order of stars numbered, here are the correct answers:	
	 Hot End assembly Motherboard Build plate Interface 	

	 Power supply Gantry Extruder Y-stepper motor 	
1.7base	[Laura] Your job on this slide is to identify the function of each of these printer parts. Drag the function to the gray rounded rectangle. When you think you have them complete, select the Submit button. You will have two chances to get them right. Good luck!	
1.7a	[Laura] Wow! You got them all! Nicely done!	
1.7b	[Laura] You didn't quite get them correct. Try again, but this time look for the red ones which indicate they are incorrect.	
1.7c	 [Laura] Unfortunately, you didn't get them all correct. Since this is key information, I would recommend going back and reviewing the functions before moving on. If you choose to move on, you can come back to that slide to review at a later time. The correct answers are: Motherboard communicator -interface Integrated circuit for components – motherboard Heats filament - hot end assembly Advances/retracts filament -extruder Moves build plate forward/backward - Y stepper motor Provides electricity -power supply Horizontal bar, hods hot end -gantry Prints are produced here - build plate 	
1.8	 [Laura] Wow Cole! You did well on those knowledge checks. Are you ready to move on to the steps to level your bed? [Cole] Absolutely! I've tried so many times to level my bed and I just get so frustrated. I thought I had it, but then when I went to print, I just got a horrible mess because the filament didn't stick to the build plate. [Laura] That's understandable. As I've mentioned before, there is a handy add-on called an auto-leveler that will help adjust for any nonconformities in your build plate. When you're ready to go to the next slide, click next. 	
1.9base	[Laura]	

	Now that you know the names and functions of the parts of an Ender 3, the next step is to understand how to level a build plate.	
	Having a level build plate is one of the most important steps for successful 3D printing. Click on each section to read more about the 5 steps to level your bed. Some sections will show a video to display the steps as well. When you are finished, click on the same section to collapse the accordion.	
1.9a	[Laura] The first step to leveling your bed is to click Auto-home. This process will move the hot end and nozzle to the center of the bed. It will also lower it to the bed where it would normally start printing. This is the best place to start when leveling your bed.	
1.9c	[Laura] Next, you will lower the build plate so you don't run the risk of damaging it while moving the hot end assembly and nozzle. Turn each of the 4 screws under the bed counterclockwise. You may want to turn them until they no longer turn. However, if you are just fine tuning the leveling, a half turn or a quarter turn might be sufficient.	
1.9e	[Laura] The steps that follow will disable the stepper motors. If you recall, these motors move the X-axis, Y-axis, and Z-axis. By disabling them, you will be able to freely move the hot end assembly side to side and the print bed forward and backward.	
1.9g	[Laura] You will need a sticky note for the next few steps. Sticky notes are the perfect thickness to establish the correct spacing between the print bed and the nozzle. Move the nozzle to one corner. Turn the corresponding screw clockwise while moving the sticky note to feel for friction. You should feel a small amount of resistance between the nozzle and bed for the correct spacing. Move the nozzle to the next corner and repeat this process. Do all 4 corners and then check the middle of the bed. It can be a good idea to redo all 4 corners to ensure that there is still the correct amount of spacing.	
1.9i	[Laura] The final step of leveling your bed is to rehome the printer and repeat all of those steps. You will want to ensure that the spacing in different areas of the bed are uniform.	
1.9j	[Laura] Now that you've learned the 5 steps, I'm going to have you test your understanding with a knowledge check. When you're ready, click Next to continue.	
1.10base	[Laura] There are five steps to successfully level your build plate. Drag the steps to place them in the correct order.	
1.10a	[Laura]	Emphasis "woo

	Woo hoo! You got them all correct! Great job!	hoo!" extend vowels a bit - whoooo hoooo!
1.10b	[Laura] You didn't quite get them all. Try one more time and see if you can get them in the right order.	
1.10c	 [Laura] Unfortunately, you didn't get them correct. Remember the steps! 1. Auto-home 2. Lower the bed 3. Disable the steppers 4. Level the bed 5. Rehome and repeat 	
1.11base	[Laura] Sometimes a level build plate is all you need to successfully print. Unfortunately, many times, you will need to fine tune your printer. [Cole] I've found that out the hard way, unfortunately!	
	[Laura] First, it will be important to understand what issues are occurring and what is causing them. Below are 3 different printing problems that can lead to frustration with this hobby. Click on each picture to reveal some common problems that can be fixed with fine tuning your printer.	
1.11a	 [Laura] When your printer is under extruding, the easiest way to solve this problem is to calibrate your e-steps. The printer thinks it is extruding 100 mm, but in reality, your printer is extruding less. To fix this issue, you need to calibrate the e-steps. Step 1Measure 100 mm from the extruder and mark filament. Step 2After heating the hot end, extrude 100 mm using the printer settings. Step 3Measure the distance from the extruder to the mark. Step 4. Calculate correct e-steps by multiplying the current value by 100, then dividing by the measured distance. Step 5. Adjust your e-steps to reflect the difference. Step 6. Repeat Steps 1-4 until it successfully extrudes 100 mm. [Cole] That sounds easy enough. I've seen some of those gaps in my prints, so I bet my e-steps need to be adjusted. 	
1.11b	[Laura] When your print lifts from the bed, you may have a draft near your printer. The best way to fix it is to put it in an enclosure.	

	You can build an enclosure using a cardboard box, PVC frame with some insulating material, or you can purchase one made specifically for your printer.	
	If you choose to build an enclosure, make sure that the bed can move forward and backward freely without hitting the walls of the enclosure.	
	[Cole] I like the idea of building my own enclosure.	
	[Laura] Just remember the temperatures at which the printer works can lead to a fire hazard, so be thoughtful of which materials you choose.	
	[Cole] Oh, that's a great point. Thanks for the reminder!	
1.11c	[Laura] This image shows places where the nozzle is too close, too far, and just the right distance from the bed.	
	When the print appears blobby or clumpy, like you see here, the nozzle is too close. This occurs because the nozzle is moving through the previous layer and remelting that filament.	
	[Cole] That explains why I was having so many issues with my first layer!	
	[Laura] If you get strings of filament that aren't touching as you can see in this area, this indicates that your nozzle is too far from the bed. This occurs because the filament is cooling before adhering to the bed and it isn't being squished down.	
	Lastly, the area where the print appears smooth and there are no gaps is the perfect adjustment for your nozzle. You can see that here. If you have either of the above problems, here's how to solve it.	
	First, you will start with the main menu of your printer. Then click Tune. From there, you will scroll to Probe Z Offset. If your nozzle is too close, you want to increase the number. If the nozzle is too far away, you want to decrease the number. Keep in mind that if the numbers are negative, smaller numbers have increasing decimals. It is easiest to make these adjustments while the printer is running. You will be able to see immediately if you have adjusted it enough.	
	[Cole] I didn't even realize there was an option to move the z-axis while it was printing. I probably would have been happier with my results!	
1.11d	[Cole] Thank you so much for going through all those steps with me. I missed a few of those steps, so it is no wonder that I wasn't having success.	

	[Laura] I'm glad that the bed leveling section was helpful. Now we are going to discuss how to fine tune your printer. The way that the filament extrudes can show us a lot about what your printer is doing. We will go through a few different areas where you can fine tune your printer.	
	[Cole] Yeah. I've seen some recommendations for this, but I had no idea where to start. This is super helpful!	
1.12base	[Laura] Once you have become comfortable with adjusting the physical aspects of your printer, you may want to explore different types of filaments.	
	There are several different types of filament that 3D printing hobbyists can use. Each type of filament has a unique set of characteristics, so it is important to know what they are so you can choose the correct one for your model. Click on each type of filament to learn more about these characteristics.	
1.12a	[Laura] Short for Polylactic Acid, PLA is the best filament for beginners. It has a low melting temperature, and it can provide strength in prints. It comes in many different colors and finishes: matte, silky, and regular. One benefit is that this filament is biodegradable. However, if you want a model to withstand higher temperatures, such as the interior of a vehicle in the summer, this filament is not the one to use.	Poly - lactic - acid
	[Cole] Huh. I didn't know that PLA was biodegradable. I suppose that makes sense since they say that prints that are meant to last a long time shouldn't be printed in PLA.	
1.12b	[Laura] This filament is something you are very familiar with. Short for Polyethylene terephthalate glycol, Pet G is used in many plastic products, such as water bottles and take out cups. Pet G is food safe and has a higher durability. Since it melts between 230 and 245°C, it can withstand environments with higher temperatures.	Poly-eth - a -leen Tera - thaalate Gly-col
	[Cole] I've seen some videos where people take a 2 liter bottle and melt it into their own filament. Maybe someday	
1.12c	[Laura] Second in popularity, ABS is most frequently used for injection molding, but has many uses for 3D printing. Acrylonitrile butadiene styrene (ABS) has chemical resistance, toughness, and impact resistance, but is also rigid. The interesting thing about ABS is that you can use acetone vapor to help smooth the surface after printing.	Acrilo-nitrile Buta - dying Sty-reen
	[Cole]	

	The finishing aspect is one of the things that I am really looking forward to trying. The prints look amazing after that's done. So much better than sanding it a million times!	
1.12d	[Laura] New models that provide flexibility require the use of TPU. Thermoplastic polyurethane (TPU) is helpful for models that require flexibility but are also impact resistant and durable. In order to use TPU filament, many hobbyists recommend having a direct drive extruder upgrade on your printer.	Thermo-plastic poly-your-a-thane
	[Cole] I've heard great things about using a direct drive extruder. I'm not sure that is something that I will do, but I appreciate the introduction!	
1.12e	[Laura] What type of filament have you been using?	
	[Cole] I have only used PLA because I figured if I couldn't successfully print with PLA, why would I further my frustration with the other types?	
	[Laura] That makes sense. We are going to take a few minutes to go over the different types of filaments that the Ender 3 can use. This way when you are fully comfortable with PLA, you can expand to the other types of filaments that are available.	
	[Cole] I've read some information about ABS which sounds really intriguing. I know nothing about the other types that are available though.	
	[Laura] Great! Let's get started.	
1.13base	[Laura] Which filament would you choose for a project that requires these qualities? Drag each quality to the correct filament type. Each filament type will have more than one quality to place. You will have two chances to get these correct.	
1.13a	[Laura] Way to go! You really understand the different characteristics of filament types!	
1.13b	[Laura] You're on the right track, but you didn't quite get them all. Try again!	
1.13c	[Laura] Unfortunately, you didn't get them all correct. Remember that PLA is biodegradable and provides strength while PETG is food safe and can withstand harsh conditions. ABS and TPU are similar with durability, but ABS is chemical resistant while TPU is flexible.	
1.14 base	[Laura]	

	When adjusting the physical aspects of your printer just doesn't get you the results you want, it is time to start adjusting settings in your slicer.	
	Slicers are applications that will take a 3D model and turn it into g-code, which is the file format that your printer understands. Each of the settings below are found in all slicers.	
	[Cole] Those slicer settings are intimidating! I've only really played with layer height and supports. I'm ready to learn more!	
	[Laura] Great! Click on the left or right arrow to learn about these most common print settings. When you've visited all 5 sections, click next to continue to the next slide.	
1.14a	 [Laura] The settings for bed temperature will vary depending on the type of filament you are using. You will want to check the recommended settings on your spool of filament, but some general guidelines are as follows: PLA 50-70 degrees Celsius Pet G: 70 degrees Celsius TPU: < 60 degrees Celsius ABS: 110 degrees Celsius 	
	[Cole] This is great! I knew the other filaments needed different bed temps, but I didn't know what they were.	
1.14b	Depending on which type of filament you are using, the nozzle setting will vary. The best way to determine the perfect temperature is to print a temperature tower. This 3D model will change the hot end temperature at various heights so you can see the temperature at which it prints the best. Common temperature ranges by filament are: PLA: 190-210 degrees Celsius Pet G: 230-245 degrees Celsius TPU: 220-250 degrees Celsius ABS: 220-240 degrees Celsius	
	You can download a file for a temperature tower. This will change the temperature of the nozzle based on the height and will give you a good idea at which temperature your filament prints best.	
	[Cole] I've heard of temp towers before. I wasn't sure what their function was, so I'm glad you went over that.	
1.14c	[Laura] Bed adhesion may be one of the most frustrating problems to troubleshoot. If your print does not stick to the bed, you may find yourself wasting a lot of filament.	

	There are a few different settings you can choose from: None – your project has a large surface area on the bottom and may be able to stick on its own. Skirt – a thin layer of filament that surrounds your print but does not touch it. This helps to prime the extruder. Brim – this is similar to the skirt, except that it will touch the outer edges of your model. This helps improve the adhesion to the build plate and is especially helpful if your model has thin areas on the build plate. Raft – Provides a few layers of filament under the entire model and is discarded after the print. Printing with 3D filament will give you many different options for your project. [Cole] The raft feels like such a waste of filament, but I understand why it is useful. The skirt also seems like a weird feature since it doesn't actually touch the print, but	
1.14d	so many pictures that I see online only have the skirt. [Laura] Depending on the model that you are trying to print, you may need to have your slicer generate supports. Supports are temporary filament that is printed to ensure that overhangs can	
	print successfully. These are removed after the print is finished and are discarded. This feature is important for projects that have overhangs of more than 45°. When you have large overhangs, the printer will try to add filament to nothing – the filament will still extrude, but with no surface, it becomes a threaded mess. Failure to add supports to projects with these overhangs will ultimately lead to frustration and wasted filament.	
	[Cole] I've made that mistake a time or two. Such a mess and wasted filament. These supports makes me wish there was an easy way to recycle the waste filament produced.	
1.14e	[Laura]When you choose layer height, you need to keep in mind how much detail there is within the model that you are printing. If the model has many small, fine details, you will want to print at a lower layer height, such as 0.12 mm. If your print is more functional with few details, you may be able to get away with a 0.2mm or even 0.28mm layer height. Also keep in mind that when you choose a small layer height, the time it takes for your printer to complete the process will be significantly longer due to an increased number of layers.	
1.15 base	[Laura] We've covered the 5 most frequently used settings in the slicer. Now we are going to cover some of the more obscure settings. Keep in mind that there are a LOT more settings than just these 10 presented in the course. As you become more comfortable with your printer and adjusting print settings, you may find yourself more interested in those settings.	

	In the meantime, we will be covering retraction, speed, infill, initial layer speed, and the initial fan speed.	
	[Cole] I've used infill before, but I haven't messed with the speed or retraction settings yet, and those initial fan and layer speeds are just out of my league!	
	[Laura] Well, hopefully when we are done here, you'll feel more comfortable with those settings.	
	When you're ready, click on each of the boxes to learn more about each setting.	
	When you are finished, click next to move onto the knowledge check.	
1.15a	[Laura] The settings for speed may change based on a few situations.	
	The slower you print, the better the finish of your final product will be. If you are using a silk PLA, the shine will display best at slower speeds.	
	Printing speeds may also vary by filament. Since most 3D printing hobbyists will use PLA, the range is between 45 mm per second and 60 mm per second. Some individuals will print at higher speeds, especially if the finish of the model is not important.	
	[Cole] So you're saying I can make my printer go a lot faster if I don't care if the plastic is shiny? Let's go! This is a game changer. I'm always so impatient for the printing process to complete.	Emphasize Let's go - elongate the oooo a bit.
	[Laura] You'll want to be careful though because if it goes too fast, the layers may not adhere properly, so your model might be weaker.	
	[Cole] Good to know. Thanks for that information.	
1.15b	[Laura]	
	When 3D printing, retraction settings will allow you to determine how filament is pulled back as the nozzle moves between parts on a layer. There are a number of different settings that you can change to get the best possible print quality.	
	Retraction Speed – the speed at which the filament is retracted Retraction Minimum Travel – this depends on the distance between parts. It will allow the printer to reduce the number of retractions, especially if there are many small parts. Retraction Distance – the length of filament retracted	
	You can experiment with these settings to reduce stringing on your print.	

	 [Cole] So is there an easy way to get rid of the strings that are present on models without sanding it? [Laura] Many people will use a few short blasts from a heat gun to melt those tiny filament threads. What remains can often be smoothed down with fingers or they disappear completely. Just don't burn your fingers if the model got too hot. [Cole] Great tip! I never thought of a heat gun! 	
1.15c	 [Laura] To determine the infill that you need for your project, you must first ask yourself if you are seeking strength and durability, or something that is going to be purely decorative. If you need strength and durability, you will want a larger infill percentage. If you choose 100% infill, your model will be solid with no gaps inside. Most models that are more decorative can get away with infill as low as 15%. This reduces the amount of filament used during printing. [Cole] This makes sense – if I have a model that is just going to sit on a shelf, then I don't need much for infill, but if I am going to make something that is functional, like a hook or something that needs to support weight, I should go with a larger infill percentage. 	
1.15d	 [Laura] The initial layer is the very first layer of filament that is being placed on the build plate. If you are using a raft, this setting is not quite as important. When you slow down the speed of your initial layer, you provide ample time for the filament to go down without too many complications. The slower speed on this layer will help minimize vibrations from the printer and it allows the nozzle and extruder to reduce issues such as clumps or gaps. Many hobbyists will choose 20 mm per second as a speed for the first layer. [Cole] Maybe this is another place where I've been failing. I'm not sure what my initial layer speed is in my slicer. Do you know if 20mm/s is the default? [Laura] I believe it is, but you will want to double check your settings to make sure. If your prints aren't sticking, you could try slowing down your initial layer, adding a raft, or increasing your bed temp. [Cole] 	When you see mm or mm/s, don't use the abbreviations - use millimeters and millimeters per second.

	There's so much to know. I feel like my brain is going to explode with all of this new information.	
1.15e	[Laura] As your printer begins printing the first layers of your project, you may want to adjust the fan speed. How your first few layers of your project print will determine some of the most important parts to having a successful print.	
	Many hobbyists will turn off the fan for the initial layer. This allows the filament to cool slowly, providing a strong base layer. This can also help reduce your project from warping on the build plate.	
	[Cole] I've noticed that my printer is pretty loud right from the beginning of a print. I wonder if that's just the fan from the power supply, or if it is a combination of the fans from the hot end and the power supply. I will have to investigate this further!	
1.16 base	[Laura] Which slicer setting would you change to begin troubleshooting the following problems?	
1.16a	[Laura] Woo hoo! You got them all correct! Great job.	
1.16b	[Laura] Try one more time.	
1.16c	[Laura] You didn't choose the correct matching pairs. Here are the correct answers.	
	Bed Adhesion – Print doesn't stick to the bed Infill – Print is weak and breaks easily Retraction – Stringing between model parts Speed - Silky finish looks dull Initial fan speed – Print warps off the bed Supports – Strings occur by overhang	
1.17	[Laura] Now it is time for you to show what you have learned. The following quiz has five questions. You must earn an 80% or higher to pass. To earn an 80%, you must have four correct answers.	
	Click the begin quiz when you are ready to start.	
1.18	[Laura] What is the first step to level your build plate?	
1.19	[Laura] What is the best tool to use to determine if the spacing between your build plate and nozzle is just right?	
1.20	[Laura]	

	What are some of the qualities of different types of filaments? Match the filament type with the set of characteristics.	
1.21	[Laura] What setting would you manipulate in order to provide a brim around your project? Choose the best answer.	
1.22	[Laura] What are the functions of the different parts of an Ender 3 printer? Drag the function to match with the part.	
1.23a	[Laura] Amazing! Time to celebrate! You passed! You are well on your way to understanding the basics of 3D printing. The information in this course has prepared you to troubleshoot some of the most common problems during 3D printing.	Show excitement on "Amazing! Time to celebrate!"
1.23b	[Laura] Sorry, you didn't pass. Take some time to review your quiz results. When you are ready, click Retry Quiz to try again.	
1.24	[Laura]	
	That was a lot of information to digest! Hopefully you've found it helpful to learn these key pieces of information to help you be successful with your new hobby.	
	Now that you have reached the end of this course, you should be able to	
	 Identify the different components and the function of these parts on an Ender 3 printer. 	
	• Recognize the steps needed to correctly level the build plate and fine tune the printer.	
	• Describe the different types of filaments used in 3D printing.	
	 Manipulate settings in the slicer to obtain the desired printed object. [Cole] 	
	Definitely! I feel so much more prepared now. I wish there was a little cheat sheet that I could use to help me remember this after a while.	
	[Laura] You're in luck! In the resources section of this course you can find a PDF that you can print out if you'd like.	
1.25	[Laura] Congratulations!	Emphasize Congratulations!
	Through your hard work, you have learned some of the most important aspects of 3D printing.	
	The more that you work with your printer, the easier it will be to understand the issues that it presents you.	

Click complete to finish this course.	