IDEASTUDIO
THE MAKERSPACE @ WEST HOUSTON INSTITUTE

3D Printing Training Guide (FDM)

Introduction

3D printing is coming out of its status as an emerging manufacturing technique and becoming a more commonly accepted means of manufacturing. This is due to lowered costs and an easier overall process to end up with a finished product.

After reviewing this training guide, you should understand the concepts associated with FDM printing and be able go through all of the general steps it takes to 3D print a finished part.

Outcomes

After reviewing the instructions in this document, you will be able to:

1. Understand general concepts and terms associated with 3D printing
2. Create, acquire, and/or customize a 3D model
3. Prepare a 3D model for print using the appropriate software
4. Set up an FDM printer, including loading filament and starting a print
5. Post process a printed 3D model or part

Assessment

In order to be released to use the 3D printer, you will need to demonstrate competency and earn at least 40 points during the assessment at the end of the training. The following are the individualized criteria that you will be tested on.

CRITERIA	Needs Work (0 points)	Competent (5 points)	Exceptional (10 Points)
Identify the main components of an FDM printer			
Explain the FDM process			
Acquire and customize 3D model			
Slice a model with FDM settings			
Change out a roll of filament into an FDM machine			
Print a model with an FDM machine			
Identify what constitutes a print failure and what to do			
Post process an FDM printed model			

Overview

Fused Deposition Modeling (FDM) or Fused Filament Fabrication (FFF)

As defined in the terms section, Fused Deposition Modeling (FDM), also known as Fused Filament Fabrication (FFF), uses a string of solid material (filament), pushing it through a heated nozzle and melting it in the process. The printer continuously moves this nozzle around, laying down the melted material at a precise location, where it instantly cools down and solidifies. This builds up the model layer by layer.

A print fail, although self-evident in the term itself, can occur in numerous ways. A fail can be something as obvious as a portion of the part falling over mid-print and the remainder of the print is an unidentifiable stringy ball on the print bed. It can also be as subtle as poor adhesion between layers due to under-extrusion or the temperature is set too low. Also, keep in mind that a print that experienced a problem may not necessarily be a fail because it can still be used as it was originally intended. For example, a part may have inconsistencies in the surface finish because the print speed was too high, but still be usable because the aesthetics of the part are irrelevant because it's a purely mechanical part.

FDM 3D Printing Process

3D printing is a subset type of manufacturing known as *Additive Manufacturing*. As defined in the Terms section of this guide, Additive Manufacturing is the process of fabricating a part by adding material in layers. In this training, you will learn about specific points necessary to 3D print a physical object using an FDM printer.

1. The first step in 3D printing is to create/obtain a computer 3D model of an object in the computer. The creation process of a computer modeling is known as *Computer-Aided Design (CAD)*. Obtaining a file can also be as simple as merely downloading a file from an online repository. The typical file types for 3D modeling are *.stl* and *.obj*.
2. The next step is to convert the 3D file into a type that the 3D printer can understand. This step of conversion is called "slicing" and the file type is *.gcode*. Basically, "slicing" a file means to set the print parameters by which the object will be printed. Parameters include, but are not limited to, layer height, print speed, material temperature, and support. The G-Code file itself is merely a list of individual commands that set the parameters and move the tool head in an ordered path on the X, Y, and Z axes.
3. The final step in slicing is saving the G-Code file. This usually involves saving the G-Code file to a flash drive or an SD card. It depends what each printer accepts. Additionally, there are some printers that are network connected so a file can be sent to the printer online.

4. Before actually running the print, it is important to set up the 3D printer itself. This includes, but is not limited to, making sure:
    a. The print bed is cleared of debris and leveled
    b. The print material is loaded properly
    c. There is sufficient material to fully print the part
5. Once the printer is prepared, insert the flash drive or SD card and use the user interface to run the print.
6. Patiently wait for the object to print. 3D printing a model can take several hours to complete.
7. After the object has printed, remove it from the print bed and post process it as needed. Post processing includes, but is not limited to, removing support material, sanding, gluing and painting.

Instructions

Follow the instructions below to complete the 3D printer training for FDM. This tutorial will focus on process rather than creation, so you will be downloading a pre-designed file to use for printing.

Acquire the 3D Model

1. Go to: https://www.thingiverse.com/
2. Search for the test object using the terms “WHI Test Object” or “3106023”.
3. Download the file and unzip the contents.

Slice the Model

Open the File

1. Open your slicing software. We will use *Cura*, free software from Ultimaker.
2. Select your printer under *Settings*. Our studio has three types of FDM printers:
    a. Ultimaker 3
    b. Ultimaker 3 Extended
    c. Prusa i3 Mk2.
3. Go to *File* > *Open File(s)*. Navigate to the *WHI Test Object* folder and open the file, *WHI_Circle.stl* , from the *files* folder.

Print Setup

4. On the right-side menu, under Print Setup, select *Custom*. The Recommended settings typically offer a very abbreviated list of print settings to make the printing process easier for users. For this training, it is important to see a more complete set up options as you learn the process.

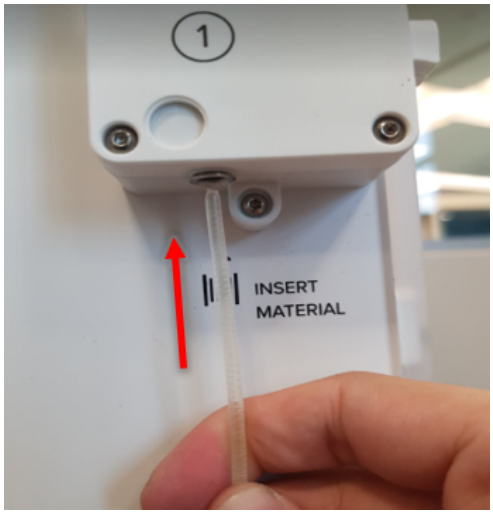
5. Select Material from the right-side menu:
    a. Under *Extruder 1*, in the *Material* section, open the dropdown menu and select *Manage Material*.
    b. Select any color PLA, click *Activate*, then click *Close*. The selected color here will not affect the actual color of the print. The point of this step is to familiarize you with materials management. Just be sure to load the same TYPE of material onto the printer that is select in this step.
6. Under *Quality*, adjust the layer height to 0.2 mm. As a rule of thumb, the larger the layer height, the less time it will take to complete a print. For our test print, the quicker print time will work. The trade-off is that the "resolution" quality will be lower. In other words, the layer lines will be more prominent.
7. Under *Shell*, adjust the wall count to 1. The rule of thumb here is the more walls, the stronger the part. The trade-off in this case is that more walls means a longer print and more filament used. A single wall is adequate in most cases, unless the part will be used as an actual mechanical part.
8. Under *Infill*, adjust the Infill Density to 10%. The higher interior density makes for a stronger part. The trade-off here is similar to Infill. A higher infill density makes for a longer print and it uses more filament.
9. Under *Speed*, adjust print Speed to 85 mm/s. The speed at which the tool head travels can impact quality of a print or even whether a print will even be successful. A fast print speed of, say, 120 mm/s might be OK if the print consists only of ling straight lines or large curve, but the print will likely fail on a small or intricate part.
10. Under *Support*, ensure that the General Support box is not checked. Supports are always recommended for 3D printed objects that have overhangs of a certain angle. The tradeoff for supports that not only will it increase the overall print time and use more filament, it will actually affect the surface quality of the part. In this case, the part is small enough where any steep overhangs will not be an issue due to the melted plastic's high viscosity and short distance needed to bridge.
11. Under *Build Plate Adhesion*:
    a) Select the *Brim* option. When deciding whether to print with bed adhesion, the determining factors are how much of the actual part touches the print bed and how much the bed adhesion material will affect the surface quality of the part.
    b) Click *Prepare*. This step calculates the tool path of the print nozzle.

Preview the File

Check Print Details. It's a good practice to change the view of the part from Solid View to Layer View. Unlike Solid View, Layer View shows exactly how the print will be executed at every layer. This may detect print issues. For example, sometimes a section of the 3D model might be thinner than the width of the print nozzle. When that is the case, Layer View will show where the software won't be able to print certain portions of the part.

Save the File to a Removable Drive

1. Insert your removable device. The Prusas use a standard size SD card and the Ultimaker 3 & Ultimaker 3 Extended use a flash drive.
2. Click *Save to Removable Device*. This step saves the gcode file to the device that will be read by the 3D printer.
3. Click *Eject*. Remove the removable device and take it to the 3D printer.

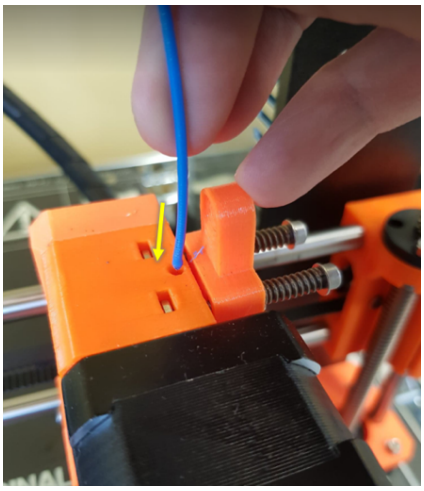


Printing with the Ultimaker 3/Ultimaker 3 Extended

1. Select/Inspect Printer
   a) Look around the front and back of the printer to make sure it is powered on and there are no obvious signs of disrepair or damage.
   b) Look to see that the glass print bed is secured in place by the two metal clips toward the front of the printer.
   c) Make sure there are no prints or other debris on the print bed.
   d) Insert the flash drive with the loaded G-Code file.
2. Change Filament (Optional)
   a) Use the scroll wheel to highlight the MATERIAL/ PRINTCORE. Press the center of the wheel to select the option.
   b) Use the scroll wheel to highlight the Material 1 option and click the wheel.
   c) Use the scroll wheel to highlight the CHANGE option and click the wheel.
   d) Wait for the print core to sufficiently heat and fully retract the filament.
   e) Remove the filament from the spool holder on the back of the printer and click CONFIRM.
   f) Select the correct filament that matches the type of filament you selected in the slicing software.
   g) You may want to snip the tip of the new filament to prevent jamming. Load the new filament on the spool holder and thread the end of the filament into the geared feeder at the back until you can see about two inches in the clear tube above the feeder.
   h) Click CONTINUE.
   i) Click SELECT MATERIAL.
   j) Scroll until PLA is highlighted and click the wheel.

k) Under the "PLA material is selected. Insert in feeder 1 and confirm" screen, highlight CONFIRM and click the wheel.

l) Wait until you see the filament from the newly loaded spool being extruded and click CONFIRM. Initially, some other color will come out. It's just leftover material.



m) At the MATERIAL1 menu, highlight and click return.

n) At the MATERIAL/PRINTCORE menu, highlight and click return.

3. Print
    1. Highlight PRINT and click.
    2. The Ultimaker sorts gcode files by date and so, if there are multiple gcode files on the flash drive, your file should already be highlighted. If so, click the wheel.

Printing with the Prusa i3 Mk2

1. Select/Inspect Printer
    a) Look at the printer to make sure it is powered on and there are no obvious signs of disrepair or damage.
    b) Make sure there are no prints or other debris on the print bed.
    c) Insert the SD card with the loaded gcode file. The port is to the left of the LCD screen.
2. Change Filament
    a) Use the knob to the right of the LCD. Press the knob (AKA Click). This will open the menu of options.
    b) Scroll down to Preheat and click.
    c) Choose the material and click.
    d) Wait. The next step requires that the print bed and nozzle be heated to temperature based on the selected material. Look at the LCD screen and notice both temperatures rise. Proceed to the next step when both temperatures have been achieved.

    e) Click the knob to open the menu.
    f) Navigate to Unload Filament option and click.
    g) The old filament will be pushed out. Remove the old spool and place the new spool on the holder above the printer. You may want to snip the tip of the new filament to prevent jamming.
    h) Insert the end of the filament into the extruder.

i) Pull the tab away from the main body and further insert the filament.
j) Navigate to the Load Filament option and click. The gear in the print nozzle assembly will advance the material to the heated nozzle and begin extruding material onto the print bed.
k) The printer will ask, "Is the color clear?". Answer "Yes" when only the single color of the proper material is extruded. Answer "No" to have the printer extrude some more filament until it is clear.

3. Print
   a) Use the knob to select Print from SD and click.
   b) Scroll to the file to be printed and click. The Prusa sorts gcode files by date and so, if there are multiple gcode files on the flash drive, your file should already be first on the list.

Post-Processing the Print

Post processing begins with removing the part from the print bed. If the part has had time to cool and the part is sturdy enough, removal may only be a simple matter of lightly pulling it off the bed. If it doesn't come off that easy, use a spatula or putty knife to gently remove the part from the bed.

Depending on final use of the part, post processing can be a simple or complicated process. As previously stated, post processing includes, but is not limited to, removing support material, sanding, gluing and painting.

For this print sample, you will only need to remove the brim and possibly any additional artifacts that appeared in the print. Follow these steps:

1. With your hand, remove the bulk of the brim material
2. With a knife, carve away any remaining pieces of the brim
3. Inspect the entire part and remove any stray pieces of material

3D Printing Resources

There are a number of online resources that can further your understanding and increase your skills in 3D printing.

Free Online 3D Print Ready Objects

Thingiverse - https://www.thingiverse.com/
Pinshape - https://pinshape.com/
Cults 3D - https://cults3d.com/en

Free Web-based Editing Software

TinkerCAD - https://www.tinkercad.com
Vectary - https://www.vectary.com

Free Download Software

Fusion 360 - https://www.autodesk.com/products/fusion-360

YouTube Channels for 3D Printing

Channel	Link
RCLifeOn	https://www.youtube.com/user/RcLifeOnSimon
Ivan Miranda	https://www.youtube.com/user/superazote
Josef Prusa	https://www.youtube.com/user/prusajr/videos
Maker's Muse	https://www.youtube.com/user/TheMakersMuse
Make Anything	https://www.youtube.com/channel/UCVc6AHfGw9b2zOE_ZGfmsnw
Thomas Sandladerer	https://www.youtube.com/user/ThomasSanladerer/videos
3D Printing Nerd	https://www.youtube.com/channel/UC_7aK9PpYTqt08ERh1MewlQ

3D Printing Terms

3D printing has a number of terms that is important to integrate into your own vocabulary. The following is a comprehensive list to get you started.

Name	Definition
3D Model	A 3D design typically produced on a CAD program.
3D Modelling	The act of using 3D CAD programs to produce a design.
3D Printer	An additive manufacturing machine that constructs a solid shape by building one layer at a time.
3D Printing	The act of using an additive manufacturing machine (3D printer) to produce a solid object one layer at a time (also known as additive manufacturing).
Additive Manufacturing	The process of fabricating a part by adding material in layers (also known as 3D printing).
Bridge	Occurs when the printer is required to print between 2 supports or anchor points. Because there is no support offered for the initial layer being printed (there is nothing to build upon) and it is required to "bridge" a gap.
Brim	A single flat layer printed around the base of a model to prevent warping. The width of the brim can typically be altered in a slicer program.
Build Plate	The area where a 3D print is printed upon.
Build Resolution	Typically refers to the layer height that a 3D print is printed at. Similar to the resolution on a television or computer monitor but in 3D the lower the build layer height the higher the part resolution.
Build Time	The total time it takes for a 3D printer to complete a 3D print.
CAD	Computer aided design - a method of design where a computer program is used to create 3D objects in the form of electronic files.
Filament	The general term given to the material used in FDM. Typically supplied in coils or rolls the filament is heated up and fed through the nozzle to deposit the material on the build plate.
File Type: G-Code	The common name for the most widely used numerical control (NC) programming language. It is used in computer-aided manufacturing to control automated machine tools (like CNC's and 3D printers).

File Type: OBJ File	A geometry definition file. CAD models are exported as OBJ files then imported into a slicer program. The slicer program then converts the file into G-code to be interpreted by the 3D printer. Similar to .STL files.
File Type: STL File	A geometry definition file that uses triangles to describe the surfaces of a 3D model... CAD models are exported as STL files then imported into a slicer program. The slicer program then converts the file into G-code to be interpreted by the 3D printer.
Hollow	A 3D print that is not solid and also does not contain any infill. Hollow models are much faster and cheaper to print but have very low strength.
Infill	A value usually represented in percentage that shows how much a solid model should be filled in with material when printed. 100% infill means the part is completely solid. Infill is used to make 3D printing cheaper and faster.
Layer Height	Sometimes called print resolution this is the height of each layer of a 3D print typically measured in microns.
Melting Point	The temperature a solid melts or turns into a liquid.
Micron	A measurement of distance regularly used to describe 3D printing layer height. 1000th of a millimeter. A human hair is approximately 17 microns thick.
Nozzle	The part of a 3D printer where the build material is extruded from.
Nozzle Diameter	The diameter of the material that is extruded out of the nozzle. This plays an important role in FDM where shells and walls should be a multiple of nozzle diameter.
Overhang	Overhangs occur when a newly printed layer of material is only partially supported by the layer below. Angled walls are considered overhangs and depending on the print technology and angle often require support to print successfully.
Polymer	A material whose molecular structure is composed of multiple repeating units. Natural polymeric materials include amber, wool, silk and natural rubber while synthetic polymers include resin, nylon, polystyrene and silicon.
Post Processing	Any act of improving the appearance or material properties of a 3d print after it has been printed. This covers a large range of processes in 3D printing that vary by technology (support removal, UV curing, heat treating, sanding, tumbling, polishing, painting etc).
Print Head	The part of a 3D printer where material is extruded/jetted from. Is an assembly of multiple components including the nozzle in the case of FDM.

Print Material: ABS	Acrylonitrile butadiene styrene (ABS) is a thermoplastic polymer commonly used in FDM 3D printing.
Print Process: FDM	Fused Deposition Modeling (FDM) uses a string of solid material (filament), pushing it through a heated nozzle and melting it in the process. The printer continuously moves this nozzle around, laying down the melted material at a precise location, where it instantly cools down and solidifies. This builds up the model layer by layer. The most common 3D printing technology.
Print Material: PLA	Polylactic acid (PLA) is a thermoplastic polymer commonly used in FDM 3D printing. It is derived from corn starch or sugar cane.
Print Speed	The speed the print head moves around the build plate typically measured in mm/s. 50mm/s is a common speed for desktop FDM printing.
Print Volume	The largest possible dimensions a 3D printer is able to print at. Varies significantly by technology.
Prototype	An early part or model of a design built before production to test form, function, aesthetics and interaction usually at a low cost. Prototypes are typically items to learn from to improve a design.
Raft	A thick grid with a roof that is added to the base of the part to limit the likelihood of warping occurring. Different to a skirt or brim.
Rapid Prototyping	The process of creating physical prototypes directly from digital data.
Shell	In FDM printing the shell refers to the walls of the print that are exposed to the outside of the model. FDM will print shells at the perimeter of the model and then fill the model with infill. Different to wall thickness.
Skirt	A line that is initially printed around the print (but not connected to the print) to clean the nozzle head.
Support	Support is the extra material that is printed during a 3D print allowing a design with complex geometry to be successfully printed. Support is required to successfully print overhangs and bridges and is removed and discarded in the post processing stage.
Surface Finish	In 3D printing this refers to the roughness of the surface of a 3D printed part. Generally qualitative.
Thermoplastic	A plastic material that becomes pliable or moldable above a specific temperature and solidifies upon cooling.

Wall Thickness	Generally associated with minimum wall thickness - the thinnest dimension a wall can be printed at such that it can support the model. Varies by technology. Different from shell thickness.
Warping	Due to the high heat involved in most 3D printing process differential cooling results in areas of a print cooling at different rates resulting in deformation.
X-Axis	The side to side (left and right) direction relative to the print bed
Y-Axis	The back to forth direction relative to the print bed.
Z-Axis	The up and down direction relative to the print bed.

Hubs.com (2018) https://www.3dhubs.com/knowledge-base/definitive-3d-printing-glossary