ISSN 2229-5518

Fused Deposition Modeling (FDM) Mechanism

Fawaz Alabdullah

Abstract— Fused Deposition Modeling (FDM) is technology used in 3D printing. This research explains the foundation, mechanism and the economy efficiency of Fused Deposition Modeling (FDM).

1 INTRODUCTION

In 3D printing area, people say "If you can draw it, you can make it". Additive manufacturing (AM) allows complex parts to be built without the need for tooling, dies, or molds, using little human intervention. Fused Deposition Modeling (FDM) technology is an AM process that builds 3D shapes by taking filaments of thermoplastic polymer materials and driving them into a heated liquefier to be extruded through a small diameter nozzle onto a build platform. Technologies capable of processing metals include, for example, electron beam melting, direct metal laser sintering, laser engineered net shaping and selective laser melting. Utilizing FDM technology to extrude metals poses advantages and disadvantages when compared to methods that currently build using metal alloys. An advantage of using FDM is the lack of expensive lasers equipped in sintering processes or an electron beam as is present in the electron beam melting process. Less expensive materials and systems are available that use FDM technology compared to sintering and melting technologies. A difference is also the ability to build using both thermoplastics and metals within the same build which is not possible with other direct metal systems.

2 HOW DOES IT WORK?

Fused deposition modeling, which is often referred as "FDM", is a type of fabrication commonly used within engineering design. Throughout the development and manufacturing production cycle, FDM systems are invaluable every step of the way including conceptual prototyping, design verification and direct digital manufacturing.FDM is ideally suited for the designers who demand part stability and strength. Unlike other additive processes, FDM 3D printed models are created with actual thermoplastics. The result is a prototype that can endure exposure to chemicals, mechanical stress, and a variety of climate extremes. FDM has paved the way for functional use testing, and DDM manufacturing. Extruded prints are supported with soluble enabling complex cavities and geometries. This also makes the process perfect for jigs and fixtures.FDM is a process using molten plastics or wax extruded by a nozzle that traces the parts cross sectional geometry layer by layer. FDM creates tough parts that are ideal for functional usage. FDM works on an "additive" principle by laying down material in layers. A plastic filament or metal wire is unwound from a coil and supplies material to an extrusion nozzle which turns the flow on and off. The nozzle is heated to melt the material and can be moved in both horizontal and vertical directions by a numerically controlled mechanism which is directly controlled by a computer-aided design software package. The model or part is produced by extruding small beads of thermoplastic material to form layers as the material hardens immediately after extrusion from the nozzleplant.

3 MATERIALS CAN BE USED

Many different materials can be used for 3D printing, such as ABS plastic, PLA, polyamide(nylon), glass filled polyamide, stereolithography materials (epoxy resins), silver, titanium, steel, wax, photopolymers and polycarbonate.

4 MECHANICS OF FDM

In order to create a complex physical object from a digital set of instructions, many mechanical systems must work together to get the job done correctly. In addition to these mechanical systems, software used to control the nozzle temperature, motor speeds & direction, and methods in which the printer lays out the material are equally important to create a highly accurate model. This section will describe these various systems and how they contribute to the overall operation of the 3D printer.

The nozzle in a 3D printer has one of the most important jobs of all the mechanical systems. It is the last mechanical device that is used to build up a 3D object and it's design and functionality is extremely important when it comes to the accuracy and build quality of the printer. The biggest contributor to the performance of the nozzle is its orifice size. Typically, the nozzle size used on many 3D printers is 0.4mm. This size is small enough to produce high quality parts while maintaining reasonable build times. Printers such as the Makerbot Replicator use this size nozzle. Depending on the overall goal of the part being printed however, these nozzles can be changed to larger diameters in order to increase the speed of the print job. While doing so will decrease the horizontal accuracy, parts that will be used as rough drafts or that will be post processed with fillers or paints will still perform as intended. It is important to never set the layer height higher than the the nozzle size. This will dramatically decrease the bond strength between the layers and overall build quality. For example, if a 3D printer is using a 0.6mm nozzle, then the maximum layer height should not exceed 0.5mm.

While the nozzle is used to direct molten plastics in a precise manner, it's other job is to convert the solid coil of plastic material into the molten state by utilizing a heating element within the extruder assembly. This heating element can be a vitreous enamel resistor, a nichrome wire, or a cartridge heater. In addition to the heating element, there is usually a thermistor (temperature sensor) integrated into the extruder assembly to control the required temperature for the specific material being used. For example, one of the most common materials used in FDM is PLA (polylactic acid) which has a melting temperature of around 160 degrees Celsius. In contrast, another very popular material used is nylon. This material requires extrusion between 240 and 270 degrees Celsius. It is very important to use the correct extrusion temperature in order to minimize the risk of the nozzle jamming and also maximize the bond between bead layers. The design of the extruder is very important to not only the printing accuracy, but also to the overall performance and maintenance of the printer. While the bottom end of the extruder must be able to heat the material to a desired temperature within a few degrees, the upper end must remain as cool as possible in order to avoid jamming. This is due to the feed mechanism located above the extruder, which requires the filament material to be in a completely solid state in order to function properly. One way to decrease heat transfer from the heating element to the feed mechanism and in turn decreasing the chance of jamming, is to use fans to cool the top end of the extruder. Depending on the type of model being printed, and the type of material being used, a heated bed may be important to maintain the structure's shape while it cools. Since plastics shrink as they cool, a quick temperature drop could cause the corners of a part to curl up off of the printer bed. To minimize this risk, some printers incorporate an electronically heated bed that keeps the temperature steady. This allows the model to cool at a more even rate and improve its overall dimensional accuracy.

There are many factors that contribute to the build quality of a 3D printed part. As mentioned previously, the extruder assembly which includes the extruder, heating element, & nozzle contribute greatly to the overall build quality. In this section, additional factors that contribute to build quality will be discussed..

5 Filament

When a conventional FDM printer is broken down, it is surprising to see how basic the technology really is. In its most basic form, the FDM printer is just a hot glue gun that is controlled with XYZ motors. In order to create accurate models however, these components must be able to work together under the assumption that the filament flow rate is constantly known. While the printer can easily estimate the flow rate based on the extruder speed and known nozzle diameter, it must also rely on the quality of the filament being used. In the figures shown, it is easy to see the difference in cross-sectional accuracy of the two different brands of filament. Since the software bases its calculations under the assumption that the bead cross-section is constant, using a low quality brand filament can cause undesired surface finishes. If we take a look at the cross-section further, we can better understand the importance of using high quality filament. The cross-section on the left shows a perfectly round filament (blue area) and the amount of tolerance allowed for it to pass QA (green area). Because the melted filament is bounded by the bottom bead layer and the bottom surface of the nozzle, the rest of the material flows to the sides. This causes any excess or deficiency in the filament cross-section to greatly affect the width of the bead produced. According to ProtoParadigm (a supplier of high quality filaments), it can be said that the error amount in the width of the bead is about two times that of the error in the filament cross-section.

Depending on the software being used to print an object, the user can utilize a wide range of different tools to modify the method in which the model is printed. Many basic printing machines ship with their own software. An example of this is the MakerBot type printers.This printer ship with a basic software package that is designed to be as simple as possible. This means that the user does not have as much control over how the model is actually printed. Some users prefer to use their software of choice in order to open up these additional options. One such software is called Slic3r, an open-source software that provides users with additional features. Some of these additional features are:

5.1 Layer Height

This is the vertical height change from one layer to the next. While a smaller layer height yields a higher resolution part, the build time is much longer. On the other hand, large layer heights take much less time to produce but also decrease the surface quality. As mentioned before, the layer height must not exceed the nozzle diameter. This will lead to little or no bond between the layers.

5.2 Perimeters

This is the number of times the printer will draw the outer surface of a layer before proceeding on to the infill. Usually, there are 2 layers printed before the infill is done. The user may select to add more layers in order to increase the strength of the outer surface. This will however, increase the build time.

5.3 Solid Layers

This option allows you to specify the number of bottom and top layers. Usually it is advised to use 3 layers for the top and/or bottom surfaces. A setting of zero for the top layer will produce an outer shell type object much like a vase or pot.

5.4 Fill Density

This percentage number determines the amount of plastic infill there will be in an object. For example, a setting of 0.35 will fill 35% of the inner structures volume plastic material. Usually a setting of around 0.30 is used.

5.5 Generate Support Material

It is common for more advanced software to allow the user to modify how the support material is utilized. Usually, this task is handled by the software automatically. There are certain cases where this may be beneficial. For example, if a user finds that a printed object is warped after it is completed, then adding additional supports to the object during the print process can help prevent this. Another example would mostly be used as a last resort. If the user finds that the bed of the printer is not level, then the Raft option could be used. This function builds a level layer of support material on the bed in order to create a level surface for the object to be printed on.

5.6 Speed

The maximum speed of the machine is governed by the firmware installed on the motor controllers. However, it can be beneficial to adjust the speed in order to decrease the build time (fast), or increase the build quality (slow). The speed settings can be split in three categories; the perimeter, infill, and travel speed. The perimeter speed is the speed at which the print head moves while printing the perimeter of the model. The infill speed is the speed at which the print head moves during the infill operation. And lastly, the travel speed is the speed at which the print head moves from one location to another while not printing. For example, if there are multiple parts being printed at once, then the travel speed will be the speed of the print head when it is traveling to the other part to be printed. Typical speeds for the perimeter and infill are 50mm/sec and 70mm/sec, respectively.

5.7 Brim Width

The Brim Option is used to help large object being printed stick to the printer bed. Sometimes, large object will want to curl up at the corners due to warping. This option will print a horizontal ring around the first layer perimeter in order to increase the amount of material on the bed. This added bond strength will help prevent the part from warping.

5.8 Sequential Printing

While most 3D printing software can handle more than one object being printed at a time, sequential printing is a little different. The problem with printing multiple parts at once is that if one of the parts fails to print properly, then all of the other parts must be discarded as well. Sequential printing still prints multiple parts on the bed, however the parts are printed one at a time. This allows the printer to successfully complete part A before moving onto part B. Since the extruder has the potential of crashing into the already printed parts, the user must specify a clearance distance in order to avoid any interference. While this process does take longer, it will save the user from having to discard a whole bed of parts in the case of an error.

6. Economy of FDM

By understanding the book Fabricated the new world of 3D printer, Lipso and Kurman shows that how the FDM technology why cheaper than the traditional manufacturing process, that due to the factors in cost Keyes as the following:

- **Manufacturing complexity free**. In the traditional manufacturing the more objectives complicated the more cost to make it.
- **Verity is free.** 3D printer has flexibility to makes different objectives.
- **No assembly required**, by using FDM that makes assemble process not exists, because all the parts of product printed as one part and depend on the 3D model.
- **Zero lead time**. the FDM can print in demand when objective needed.
- **Unlimited design spaces**. The traditional manufacturing technology and human artisans are limited comparing with FDM technology.
- **Zero skill manufacturing**. There are no human skills required in FDM technology.

7 Conclusuon

Summing up, Fused Depostion Modeling (FDM) is technology used in 3D printing. A research has been done to explain the foundation and maechainsm of Fused Depostion Modling (FDM).beside the kyes that detrermind the FDM economy efficiency.

References

[1] A Wiley Brand, 3D Printing for Dummies. John Wiley & Sons, Inc. 2014, Jimmy Wohl, MakerBot Blog.
[2] Fused Deposition Modeling of Metals (n.d.): n. pag. Web.
[3] ""Introduction to 3D Printing." Instructables.com. N.p., n.d. Web. 02 Mar. 2015.
[4] Junker, Gregory. Pro OGRE 3D Programming. Berkeley, CA: Apress, 2006. Print.
[5] "Prusa Mendel (iteration 2)." - RepRapWiki. N.p., n.d. Web. 19 Mar. 2015.
[6] Kurman, Melba, and Hod Lipson. Fabricated: The New World of 3D Printing. N.p.: Wiley, 2013. Print. 3 May 2015.