

Implementing 3D Printer to Produce Parts in Medical Applications

Roger P. Johnson III¹, Charles Taylor², G. H. Massiha³

^{1,3}Department of Industrial Technology, University of Louisiana at Lafayette, USA

²Department of Mechanical Engineering, University of Louisiana at Lafayette, USA

Article Info

Article history:

Received Mar 15, 2017

Revised May 16, 2017

Accepted Jun 30, 2017

Keyword:

3D printing

Additive manufacturing

Agile manufacturing

Fused deposition modeling

PLA filament

ABSTRACT

The purpose of this project is to show the ability of a 3D printer to produce temporary parts, molds, and jigs for industrial applications. In the industry, it is common for any replacement parts to be milled by a CNC machine or a large inventory of replacement parts to be kept on standby. This represents an underutilization of company capital. This is because there should either be a CNC machine delegated to remake the part, while the other machine is down, or have capital dumped into parts that will not be utilized until a part breaks. A 3D printer can create a temporary part that can take the place of the broken one until another, more permanent; one can be produced or ordered. Although, if this was the only thing it would be used for it also would be underutilized and not bringing out its full potential. It can also make jigs in a fraction of the time, and cost in materials, it would take for a CNC machine to mill it from metal. This increases flexibility of the 3D printer and does not leave it underutilized, thus not being capital that is underutilized.

Copyright © 2017 Institute of Advanced Engineering and Science.

All rights reserved.

Corresponding Author:

G. H. Massiha,

Department of Industrial Technology,

University of Louisiana at Lafayette,

Lafayette, LA 70504, USA.

Email: massiha@louisiana.edu

1. INTRODUCTION

3D printing has seen much growth in the past decade. There are many different methods and materials that can be used, depending on what is wanted with the product. There has been much research into different application for different fields. The aim of this paper is to consider the application of a 3D printer for use in the industrial sector, specifically for use in the creation of temporary parts, jigs, and molds. The application of the research done in this paper will be in terms of a pump impeller used in the Heartmate II, seen in Figure 1.



Figure 1. Heartmate II pump impeller

The 3D printer that is being used for this Project is the Makerbot Replicator 5th generation, Figure 2. It has a lot of automated settings and calibrations that make it easy on a person who is just starting out in 3D

printing. The Replicator operates under the process of Fused Deposition Modeling (FDM) [1, 9]. It uses a Standard Tessellation Language (STL) file, to slice a model into thin horizontal levels that it can then convert into a toolpath that the Replicator can use to print the part. The tool path used by the Makerbot Replicator is generated by placing the parts STL file in the proprietary program made by Makerbot, Makerbot Desktop. It is a basic program that can take STL files and slice them up into layers. The thickness of each layer is determined by the programs set layer thickness. Makerbot Desktop then takes these layers and uses them to determine the toolpath. Because this is a free software it is not optimized to take the best path there is.

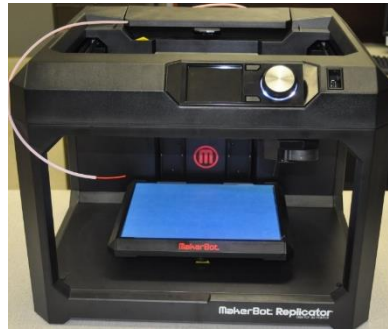


Figure 2. Makerbot replicator 5th generation used in the project

FDM is the only 3D printing method currently available that can produce parts that can both be a working prototype and an end-use product [9]. The parts can be used as such since the part that is printed with production-grade thermoplastics, giving the part excellent mechanical, thermal and chemical properties. It produces this effect by building up the part layer by layer, both the sliced layers and the outer (shell) wall. As the extruder nozzle is laying down the filament strands it is also heating up the filament strands above and to the sides of the one currently being extruded. This causes the strands to fuse to the one being laid down and allows for greater strength. Another aspect about FDM that makes it perfect for this project is that it has a high precision, around 1mm, that allows for great resolution on small parts [8].

One of the good qualities of FDM is its capability to print with different materials. The most widely used print material is Polylactic acid (PLA) [7,9,10], this is a non-IUPAC name. PLA is a biodegradable material that is more environmentally friendly than most other plastic materials. PLA has a harder structure than its predecessor, Acrylonitrile Butadiene Styrene (ABS), and its material melts at 180 C - 220 C, this is a lower temperature than ABS. PLA can glass transition between 60 C and 65 C [7]. PLA also does not require a heated build plate, unlike ABS, to be able to stick to the build plate. One of the downsides to using PLA is that it is biodegradable, thus it cannot be exposed to moisture too often or it will start to disintegrate making it a poor choice for outdoor use. Due to PLA having great attributes it will be the main material used for the project.

2. PRODUCTION OF IMPELLER THROUGH USE OF 3D PRINTER

2.1. Initial Assumptions

For the initial print, seen in Figure 3, a baseline of what the print would look like using the printers default high quality settings needed to be established. It is assumed that the result will be close to what is required but since the part is small, making it harder to clean off all the supports and stringing, it will need many minute adjustments to the settings. Stringing, occurs when small strings of plastic are left behind on a 3D printed model, is the main concern since this is such a small part. There is also a concern about the extruder head staying too long in one place, near the end of the print, causing a small amount of filament to 'blob' up on the end of the print.



Figure 3. Initial print of the outflow stator, rotor impeller, and inflow stator

2.2. Initial Print

2.2.1. Settings

The default settings for the printer are good for moderate to large sized objects. They are not good for small objects, which is what is needed. To learn what modifications are needed, knowledge of what the settings do and what changes need to be made are necessary. The initial print helped to determine what changes were needed. This includes extruder temperature, travel speed, minimum layer duration, first layer, infill, inset, outlines, infill percentage, infill pattern, layer height, shell number, raft margin, support density, retraction distance, retraction speed, and restart speed.

Extruder temperature, set at 215 C, is the temperature that the filament is extruded out at. It is pivotal that this is set at the right temperature or it could cause severe harm to the printer and to the part being printed. If the temperature is set too high the printer could start over extruding plastic or cause major issues with stringing/oozing. On the other hand, if the temperature is set too low it could start under extruding plastic, cause holes/gaps in the top layers, or cause the layers to separate and split. It can also cause the part not to stick to the print bed. Although this is mainly caused by an A/C vent over this printer or the active cooling being set too high [14].

Travel speed, set at 150 mm/s, is the speed at which the extruder travels when not set to extrude materials. It is set high so that the 'string' of filament that is still oozing from the extruder nozzle, due to the high temperature, can be broken easily and not deform the part due to placing filament in the wrong place. The speed cannot be too high due to the 3D printer running on an open-loop system. If the speed is too high, it could cause the extruder assembly to be jarred out of place when the assembly comes to a sudden stop to start extruding material. This will cause layer shifting or misalignment [14].

Minimum layer duration, set at 5 seconds, affects the minimum time the printer will spend on a layer. If the printer would spend less than the set time on that layer it will change the print speeds to be at or above the minimum time. This setting is a way to roughly adjust print speeds to ensure the printer does not rush the print on any layer. One good use of this setting is to use it to regulate grinding filament. This is caused when the part is being printed too fast and needs to be slowed down overall but you do not know which settings to change [14].

First layer, set at 30 mm/s, is crucial to the precision of the part, depending on the area of the side the part is placed on. If the area of the first layer is large this setting needs to be set low. This is done to reduce the chances of the filament bonding at an improper angle and causing problems with the base of the part. It can also cause the print not to stick to the bed if the first layer speed is too high [14]. If it has a small area it can be set higher.

Infill speed, set at 90mm/s, is the internal supporting structure within a part. This structure allows the part to be printed quickly without having to make great sacrifices in its properties. This setting needs to be high to print the part in a reasonable amount of time, since the infill constitutes the majority of the time spent printing. Inset speed, set at 90 mm/s, is the inner layers of the shell. This is faster but for a quality looking part it will need to be slowed down. If the inner layers are not placed properly the inner structure of the part can be compromised.

Outline speed, set at 20 mm/s, is the outer most shell layer. This must be slow so the outer part conforms to all set specifications. If it is set too high the outer face of the part might not look like it should, causing problems when it needs to be set in place. The issue of improperly aligned extruded filaments on the external shell layer can cause major issues in terms of aligning. If the external shell layer is distorted it can cause problems with the part fitting where it needs to.

Infill density, set at 40 %, is the percentage of the internal structure of the part is filled with filament. Setting the infill at 40% is normal for a part that needs to work as a product, any higher will end up risking the part warping. The part could also have trouble with their being holes or gaps in the top layer if the infill is set too low [14].

Infill pattern, set at diamond (fast), is the pattern which the infill structure is printed as. Depending on the size and complexity of the part, the infill pattern might have to be changed to a simpler one if the STL file is too large. If the infill is too weak, too much spacing between infill layers, or if it is necessary to keep the infill percentage the same, the infill pattern can be changed to suit the needs. There are also the effects that the different infill patterns might have on the properties of the model.

Layer height, set at 1 mm, is the height each vertical layer of the shell layer. This controls the quality of the print, with 1 mm being the highest precision. The larger the number is the lower quality print you will have, but the lower the quality the faster your part will print. Shell number, set at 2, is the number of shell layers that the printer will use on the part. Most of the time you will keep the quantity of shells at 2. If you need a more solid part, you may increase the amount of shells. Keep in mind that anything above 5 will drastically increase print time and cause some problems that need to be addressed by changing other settings to get a good quality part.

Raft margin, set at 4 mm, is the maximum distance from the parts outside edge the printer will print a raft from the part. If there are troubles with warping or the raft being hard to remove, this setting will need to be increased. It is noted that while having a larger raft is a good idea it must always be kept in mind that rafts require a lot of filament to be made. If sustainability factors need to be considered, this is the first setting that can be changed to bring about a measurable change.

Support density, set at .05, controls the support density the printer will extrude supports at. The value can be anywhere from .01, being minimal supports, to 1, being a solid support structure. The density of supports needs to be carefully considered. It needs to be set high enough so the part will not have any overhang that will cause issues with the print quality due to sagging filament strands. It will also need to be low enough so that filament is not wasted on too much support structures or be too hard to remove after the print is completed.

Retraction distance, set at 1 mm, is the distance the filament is retracted by the extruder assembly before it switches to a travel move. This makes it so no excess material is extruded on the part while the extruder assembly makes the travel move. Since the Extruder nozzle is heated and in contact with the part to fuse the material it is extruding to the material previously extruded, it is essential the during a travel move, when the extruder nozzle has retracted from the part, that no material is extruded to get placed anywhere on the part causing issues with the properties of the part. Increasing the retraction distance can help to alleviate the issue.

Retraction speed, set at 35 mm/s, is the speed at which the filament is retracted. A low speed can result in some filament being dropped onto the part while the extruder assembly is waiting for the filament to retract. A high speed can cause gear slippage and other various problems to the extruder assembly. Restart speed, set at 30 mm/s, is the speed at which the filament is lowered back into place to begin extruding filament, this is after the extruder assembly has traveled to the point at which it needed to move to. If the restart speed is set too low, it can cause the previous layer to be overexposed to heat which will cause a breakdown in the properties of the part at that specific point. If it is set too high, it will cause gear slippage and other various problems in the extruder assembly.

2.2.2.Results

The part came out at a good level of precision to what is needed for the part. Everything came out in the dimensions specified on the model and in the position specified. The main issues came down to: excessive stringing, on what side of the part to place it on the plate, how it should be aligned on the plate. The first issue of excessive stringing, as seen in Figure 4, is a bit tricky to deal with. This is due to the fins being so close together. Stringing is when small strings of plastic are left behind on a part. It can be alleviated, or solved depending on the part specifications, by changing the retraction distance, retraction speed, temperature, and travel speeds. This issue poses the greatest concentration of effort since many different settings can affect the amount and diameter of the stringing. Stringing can cause a lot of issues with the look of the final product. It takes time and a lot of effort to clear up the stringing.



Figure 4. Outflow stator with stringing issues

The second issue of what side of the part to place on the plate is somewhat easier to deal with in most cases, but for the case of the impeller it is not so easy. If you place it on its bottom flat portion, as sense would dictate, the printer would run into issues of ability to print the part and issues with the fins separating from the main shaft of the impeller.

The third issue of how it should be aligned on the plate is something that needs to be paid strict attention to. The operator of the program needs to be aware of where the part is in the three-dimensional space on the Makerbot desktop software, Makerbot's proprietary software that controls the 3d printer and slices up the STL model loaded into the program. It has also been noted that using the auto leveling function within the Makerware Desktop software is the best way to get the part as low to the plate as possible, so that

little in the way of support structures are used, and, in terms of the impeller used in this project, at the right rotation to place it on the plate in the optimal position.

2.2.3. Changes Needed

To address the first issue, several variables must be slowly changed to get the best results. First, the temperature should be lowered. This will make it so the plastic is more viscous and less likely to string. Second, the retraction distance will need to be increased in small increments. Since the temperature will already be lower, the less viscous plastic will be less likely to ooze a great amount onto the part during a travel move but be hot enough to be able to fuse to the part when it is extruded. Third, the retraction speed will need to be increased in manageable increments. This will cause the filament to be retracted faster from the extruder tip causing less filament to be allowed out to cause stringing. Fourth, the travel speed can be adjusted to find a speed that is fast enough for the string to be broken but slow enough not to jar the extruder assembly out of place due to the high speed.

To address the second issue, it is mainly a point of knowing the ambiguities of the make and model of 3D printer that is being used. For this printer, it has problems when the outflow stator is placed vertically on the build plate. It starts to have trouble with printing the main shaft and printing the fins attached to the main shaft. To solve this issue, it needs to be placed horizontally on the build plate. Although this requires more support structures to be printed underneath the part it will print the fins properly attached to the main shaft. To address the third issue, it is only a matter of the operator knowing the proper procedure for prepping a model in the Makerware Desktop software. So long as the operator makes sure that the part is leveled and centered by the auto-leveling and auto-centering functions, then there will be no problems with printing the part.

3. PRINTING SMALL PARTS

These settings were acquired from the Makerbot blog site, see table 1 under suggested. It stated that these settings are good for printing small parts [15]. It is apparent that most of the settings will be useful for the project since it mainly deals with the printing of a small part. It assists in the printing of small parts by slowing down the print process at critical points to allow a higher accuracy in the print. Although, there is an issue with the temperature. Since most small parts are usually thin and don't have close in edges, it usually works well, but in the case of this project, with fins very close together, it increases the stringing issue.

The reason this works well for small thin parts is that it allows more filament to be extruded which will fill in the spaces that are left behind on the part due to the thin nature of the part. After some testing it was found that 3 modifications were needed to these settings, as shown in Table 1 under implemented. The scale of the part was increased to make it easier to manipulate and remove supports and stringers with minimal damage to the part. The temperature was reduced to ensure the properties of the part since higher temperatures can negatively affect the part. The infill speed was reduced to ensure an accurate print.

Table 1. Printer Settings for Printing Small Parts

setting	Part Scale	Extruder Temperature	First Layer	Infill Speed	Outline Speed
Suggested	100%	220 C	10 mm/s	45 mm/s	15 mm/s
Implemented	125%	199 C	10 mm/s	40 mm/s	15 m/s

4. OUTFLOW STATOR

4.1. Using Outflow Stator to Find the Best Settings

Since the initial print went well it was necessary to find the part that would give the most trouble and use it as the base to find the optimal settings. To this end, it was determined by visual comparison. That the outflow stator would have the most issues during printing. It is imperative that whenever a model has multiple parts, joined together through an assembly, that the part with the most issues in printing is used as a base to rapidly ascertain the optimal print settings. This allows for great potential in terms of mass production, since the weak link in the model is optimized the rest of the models parts will be optimized too. A fringe benefit of optimizing the model in this way allows for faster print time, thus achieving overall optimization of the model faster and with less materials used.

4.2. Thought Process

The aspect of the outflow stator that makes it the best part to find the optimal settings is that the fins are so close together that it causes a lot of stringing. Several settings must be modified to allow for better printing. This includes the following: extruder temperature, travel speed, minimum layer duration, first layer, infill, inset, outlines, infill, infill pattern, layer height, shell, raft margin, support density, retraction distance, and retraction speed. Each one can drastically change the quality of the part. It should be taken slowly with each setting being individually adjusted to find the optimal mix of settings to easily and accurately print the part.

4.3. Settings

Extruder temperature is set at 199 C to help control the stringing issue. It is better to have the extruder set to a low temperature so that the filament does not come through as easily. This will make it so the filament is more likely to be placed only where it is needed, thus reducing the number of thick stringers and globs of filament on any thin stringers or the part itself.

Travel speed is set at 100 mm/s to help break off the thick stringers from the smart extruder+. Thick stringers are large amount of filament that are approximately the same thickness of the filament being used to make the part. Travel speed is not only used to move quickly to the next build point, it is also used to break off the filament and, at its worst, leave behind only thin stringers. The higher the travel speed is, the greater chance there is that it will overshoot its build point but there is also less of a chance for it to leave behind a stinger. It was determined that putting the travel speed at a slightly lower speed would be better for the overall quality of the part.

Minimum layer duration is set at 0 seconds so that the settings dictated by the user have full control over the printing of the part. Minimum layer duration will slow the print speed in a greater amount if the layer can be printed faster than the set time. Since the print speeds have been sufficiently slowed to ensure the quality of the part. If the print speed is slowed down too much, the quality of the print will decline due to overheating of the layers thus warping the part.

First Layer is set at 10 mm/s to ensure the accurate starting of the part. This layer needs to be slowly laid out to that warping and inaccuracies are reduced on the bottom of the part. It is also important that the supports be started accurately. If they start to warp, then inaccuracies can come up in the part due to complications caused by improperly printed supports.

Infill speed is set at 90mm/s to speed up the overall printing. Since the infill deals with the internal structure it is necessary, to ensure the rapid production of parts, to keep this setting high. The whole point of this project is to rapidly produce parts to the specifications given. If this is slowed down too much it would cause the total print time to increase substantially.

Inset speed is set at 45 mm/s to speed up the overall printing yet maintain the proper shape of the shell of the part. Inset controls the printing of the shell layers, specifically all the layers except the outermost layer. While it is important that the insets are extruded properly, it does not dictate the outer dimensions of the part. Therefore, it is feasible for the inset to be printed somewhat faster than the outlines.

Outline speed is set at 15 mm/s to ensure the proper shape of the outer shell of the part thus giving it the proper dimensions. This is one of the more important settings due to the fact that it sets the standard for what the dimensions of the part is. Usually this can be set higher for a large part, or one with simple geometry. Since this project focuses on a small part, the speed is set lower so the accuracy of the dimensions is correct. This setting is also correct for a part that has complex geometries that need to be properly printed.

Infill density is set at 50% to give the part the proper amount of strength since it needs to be used as a working part. Infill provides the internal structures and support for the external shell structure. Fifty to forty-five percent is normal for a part that will be used as a working part, as the part this project is working with is. If the part will only be a working prototype the infill can be set around fifteen to twenty-five percent infill. This will provide a faster print time while still providing a decent amount of strength to allow it to work properly but not under the circumstances it is normally designed for.

Infill pattern is set at diamond (fast) since this is a basic setting. The diamond (fast) setting was used since it was the default setting and could be quickly printed out. This setting works well with most parts since a diamond structure provides a decent amount of strength to the external and internal structures. If they are overly large a different setting might be considered to reduce the file size. The infill pattern used can provide different structural aspects. This project did not consider the different aspects provided by the other infill patterns.

Layer height is set at .1 mm to allow the part to be printed at a high quality and precision. This amount of precision is necessary since the part is so small and has details that need this level of accuracy. If the part does not need this kind of accuracy, the layer height can be set to .2 mm, for standard quality, or 3mm, for low quality. The only differences in the quality is the feel of the part. The high quality has a smooth

surface and is almost like a normally produced plastic part. On the other end of the spectrum, the low-quality part has a relatively rough surface. The tradeoff of these settings is the print time. Although high quality is smooth it takes longer to print, approximately three times longer than low.

Shell is set at 2 to ensure the shell cannot be easily pierced. Shell layers provide the external structure of the part. Depending on the parts use, it may be necessary to increase the number of shell layers. This project only uses two shell layers since the part is so small and does not have major load requirements. For a working part, four shell layers are recommended for normal use. This project also investigates the use of a large number of shell layers, later in the paper, for the soft jaw vice.

Raft margin is set at 8 mm to decrease the chance that the part will be affected by warping due to the nature of the room in which the Makerbot Replicator, used in the project, is housed. The main purpose of the raft is to provide a surface that can easily adhere to the build plate and the part while it is being printed. It is also designed to be easy to remove from the build plate and part, when the settings are properly set otherwise the raft cannot be easily removed from the part.

Support density is set at .05 to allow enough support to keep the fins in their proper shape, but not too much that it would cause difficulty in the removal of said supports. The range that the support density can be set is from 01, a small amount of supporting structure, to 1, solid support structure printed on the part. The solid support structure would only be used in conjunction with HIPS (include proper name), since it is a dissolvable filament. This project does not use HIPS since it requires a heated build plate, the Makerbot Replicator does not have the capability to support the heated build plate in factory condition or be modified to use it.

Retraction distance is set at 5 mm to reduce or eliminate the stringing issue on the part. This setting had to be slowly increased to find the best distance to reduce the stringing issue. Normally, this setting can be left on one mm if the part does not have any elements that are close together; in the case of this project the fins were close together. If a large number of parts are being printed, then the operator must be sure to space the parts far enough apart from each other to ensure no stringing occurs. If there is still issues with stringing, a retraction distance of two mm should suffice. A lot of care must be taken to ensure the smart extruder+ does not clog due to the fact that a five-mm retraction distance is very rough on the extruder.

Retraction speed is set at 50 mm/s to reduce the amount of time spent retracting the filament thus reducing the chance that there will be blobs of thick stringing on the part when the smart extruder+ makes a travel move. This setting is left at the default since this is the maximum it can be set at. There is no reason, for the purpose of this project, to change the setting since a fast retraction is needed.

4.4. Results

One of the big obstacles printing the outflow stator was the constant jamming of the smart extruder+ between prints. This was due to the high retraction length needed to reduce the stringing issue, as shown in Figure 5 for examples of stringing. Settings that reduce stringing as shown in Table 2. The best way to make sure that the jam does not happen is to unload and reload the filament. Before reloading the end of the filament needs to be clipped. More than likely there will be a small blob of filament at the end of a string of filament, this is the part that needs to be clipped off. If the filament is not removed correctly there is a high chance of it breaking off in the extruder and being unable to melt it and extrude more filament. The way it is taken out of the Makerbot Replicator is first activating the unload filament function.

Once it indicates that it is ready for removal, depress the tab on the side of the smart extruder+ then pull the filament out. If this is done properly the possible filament jam will come out. If the filament string is not thick enough it will fail to be removed and the smart extruder+ will need to be taken apart and the jam manually removed [16]. If the jam is the kind that has a long filament string, then you can easily pull it out by the filament string. If it cannot be pulled out, then it might be a thermal blockage which can be removed by removing the extruder tip while it is hot and forcing the filament through the smart extruder+ which will force out the thermal blockage. If both solutions fail, then the extruder has a blockage that cannot be removed through means that will not keep the warranty on the smart extruder+ valid. If the warranty is expired, then it is possible to do some other forms of removing the blockage. These methods were not investigated since the smart extruder+ was still under warranty.

During printing, it was noted that the printer was missing some areas causing some spaces to be left in the main shaft of the outflow stator between the shaft and the fin. The solution that was proposed was to place a fillet between the main shaft and the fin. Unfortunately, since the impeller design was contributed to this project by an outside source, it was unable to be modified. Whenever the attempt was made, the fillet would mess up and would fillet the edge of the fin without taking the main shaft into account. The conclusion was that this happened because of the way the model was designed. The fins are placed inside of the main shaft instead of being extruded from the main shaft. This makes the fins act like an independent body when you are trying to edit them.



Figure 5. Outflow stator with stringing issues resolved

Table 2. Settings that Reduce Stringing

setting	Extruder Temperature	Retraction Distance	Retraction Speed	Minimum Layer Duration
Default	215 C	1 mm	35 mm/s	5 secs
Suggested	220 C	2 mm	35 mm/s	5 secs
Maximum	199 C	6 mm	50 mm/s	5 secs
Implemented	199 C	4 mm	50 mm/s	0 sec

During the process of finding the proper settings for printing it was found that if the retraction distance is set over 6 mm. The part starts to thin out in the middle, as seen in Figure 6, increasing as the retraction distance is increased. During the project the part sometimes came out looking like swiss cheese when the retraction distance was set at 6 mm or greater. It was found that it is best to keep the retraction distance at 2 mm for normal parts. If printing a small part, it is best to keep the retraction distance at 4 mm or less. Settings that improve the quality of small parts as shown in Table 3.



Figure 6. Retraction distance set at 7mm

Table 3. Settings that Improve the Quality of Small Parts

Part	Time Required	Materials Used	Cost of Materials- 900g reel	Cost of Materials - 18lbs (8164.66g) of pellets
Jig 100% Infill	6 hr 58 min	151.96g	\$8.44	\$1.57
Jig 50% Infill	4 hr 56 min	101.79g	\$5.66	\$1.05
Outflow Stator	18 min	1.56g	\$0.09	\$0.02
Full Impeller	59 min	5.19g	\$0.29	\$0.05

5. FULL PART

5.1. Merits of Printing

Printing the full impeller is more difficult than it seems. This is because the close arrangement of fins causing a large amount of stringing between all the parts. Although this stringing is not as bad in the sense that it does not have small little blobs of filament within the stringers. These blobs sometimes end up being extruded at the start or end of the stringer causing issues to arise with the part. These blobs are difficult to remove without altering or breaking the impeller. Since the stringing issue was reduced, all that remains of the stringing are relatively thin strings that contain no blobs.

5.2. Issues with Printing

The main issue with printing the full part was a problem with the STL file causing the fins to be treated as flat surface instead of as a body with volume. This issue arose from a problem with the assembly. It was not this project's objective to find out why it caused a problem only to find a way around it. This was solved by saving the assembly as a part. Doing so allowed the fins to be recognized as bodies with volume instead of flat surfaces. Thus, the full part could be properly printed, as shown in Figure 7.



Figure 7. Full part with stringing and supports removed

The amount of stringing increases in the full part compared to only printing the outflow stator. Although there is increased stringing it is not the serious kind that cause complications in the part. It is the very thin type that is a nuisance but not a major problem. It also contains few blobs on the stringers that might be extruded directly on the part causing as much trouble as a thick piece of stringing.

6. SOFT GRIP VICE PRODUCTION

6.1. Merits of Printing Jigs

Jigs are used with mills heavily in the industrial sector. Even with this heavy use it takes time and resources to create specialized jigs to work the raw stock in the way it is needed for certain projects. This is where using a 3D printer comes in handy. It can easily and rapidly make jigs to the exact specifications needed for the jig. Although a jig made by a 3D printer is not as durable as a metal jig, it is close to what is needed.

This project focused on the soft jaw type. A soft jaw jig is a type of jig that utilizes sacrificial material on the jaws of the jig that can be cut into whatever shape is needed. In this project the soft jaw jig is printed with a 8-inch diameter hole. It is then milled into a one inch hole. This is done to improve the accuracy of the size and location of the hole, taking out some of the inaccuracy in the part. This makes it easier to place the jig into the CAM program with the part that needs to be milled.

6.2. Process

There were a few issues in executing the printing and milling of the soft jaw vice. There were issues with the method of printing to get enough sacrificial material printed, an issue with the design of the vice, an issue with some of the settings, an issue with preparing the vice to be milled, and an issue with the GCode used to mill out the part. For the milling the project used the Pocket NC mill, as seen in Figure 8.

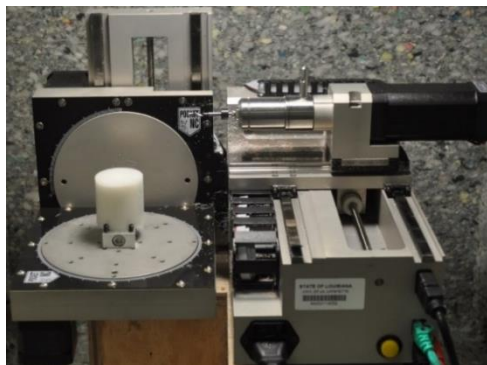


Figure 8 Pocket NC mill used in the project

The design of the vice had to go through a few revisions before it could move on to the milling phase. The main issue with it is initially it was designed to use a single screw, designed and printed with the 3D printer, to clamp down on the 1-inch stock. There were also two guide posts that ensured that the two parts of the vice lined up, these were later removed along with the screw since it is better to use metal hardware in place of the printed hardware since it is better suited to take the forces acting upon it. The vice also had to be shortened since there was no more printed hardware. Since there was a need for space to place holes for screws to be put in the overall vice portion was widened and four holes were placed in this area. The diameter of the hole was set to .8 inches with there being a quarter inch taken out from between the vice to allow for tightening. The base of the vice had to be increased in size to allow the screw holes to reach the outer holes which are much deeper than the holes currently used. Finally, the right angles on the outside edges of the base and the ones on the transition section, from the base to the vice, had fillets placed on them to make printing easier.

The issue of enough sacrificial material was mainly one of should it be accomplished through large amounts of shell layers or through 100 percent infill. When a large amount of shell layers was used, 20 layers which was calculated by the amount of material needed to be left behind after the milling and the amount of material needed to be removed by the milling, there ended up being gaps in the vice after printing. The base screw holes were also able to be easily peeled off. It was later discovered that the temperature was too low and did not allow enough material to be extruded, causing gaps in the part, or to properly bond to other extruded filaments, causing the layers to be easily peeled off the vice. In the end, the part was printed with 100 percent infill. The downside to this is that there is a lot of warping on the bottom of the part.

There was some issue with the settings due to the temperature of the room and the fact that the Makerbot Replicator does not use a heated build plate. Since the build plate is not heated, the placement of the Makerbot Replicator in a room is crucial. It cannot be placed under or near any vents or external fans. It must also be in a room that is equipped with a thermostat so that the temperature can be regulated. If the room is too cold, the filament will cure too fast causing the base of the part will start to come off the build plate and warp, sometimes it even comes all the way off the build plate resulting in a failed print and a waste of filament. If the room is too hot, it can cause the filament to cure slightly slower. This can be a problem with a small part but not with a large part that has a lot of area to be printed.

6.3. Results

The first attempt at milling ended up as a failure. Due to some issues with the GCode the mill cut out a notch in the middle of the vice. This could cause an issue with clamping down on the aluminum stock. It is unknown why the GCode made the notch in the vice wall. There was also an issue of the drill bit starting to drill approximately a half inch above the top of the vice. The issue was resolved by using a file that was known to work properly. For some reason, there was a drawn point in the file that made the drill bit start drilling at the proper point above the top of the vice. A proper explanation as to why this was occurring was never found out nor was it the objective of this paper to find out why. The second attempt, as seen in Figure 9, was a success. It only had a slight problem with milling stage, but nothing that would impede its use in the project. The issue with the milling is that there is discrepancy with the code that causes it to mill a smaller hole diameter approximately 3 inches down from the top of the jig.

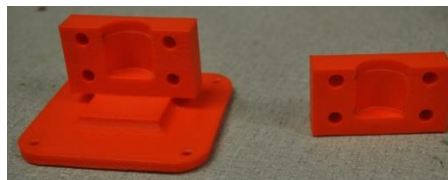


Figure 9. Complete soft jaw vice

7. COST AND TIMER MANAGEMENT

Part of whether something can go from proof of concept to being implemented is what the cost of the material is and how much time is required to do it. Currently the industry utilizes mills heavily to create the necessary parts and jigs. Using mills, it takes a long time and a lot of resources to make a part or a jig. It also requires a person to be present a good portion of the time thus tying up personnel on that mill for a large portion of the workday. On the other hand, a 3D printer can accomplish a task with less expensive materials and only a minimum of personnel time expended keeping an eye on the printer.

Earlier in the project it was pointed out that it would decrease the cycle time and amount of resources expended to find optimal settings if just the outflow stator was printed. As can be seen in Table 4, the outflow stator can be printed in 1/3 of the time as the full impeller and with 1/5 of the required material. For the jig, it depends on if a soft jaw jig is the desired outcome or not. During the project, it was discovered that if the infill is reduced to 50 percent, down from 100 percent infill, the print speed is decreased by 1/3 of the original along with reducing the amount of material used by 1/3. The amount of time that a part needs to have after printing for finishing also needs to be taken into account. The finishing is the removal of anything on the part that is not needed, or wanted, to make the part function properly. This includes rafts, support structures, and stinging. It can take anywhere from 5 to 30 minutes depending on the size and complexity of the part in question.

When calculating the cost of the project the following formula was used to find the cost of materials used if a 900g reel of pre-extruded filament from Makerbot was used to print the parts $Y=(X\div 900)\times 50$. In the equation, Y is the cost of the materials and X is the amount of materials used in grams. If there are capabilities to extrude the filament in house buying the filament as pellets in bulk is the better option. The website, Open Source 3D printing, has an option of buying the filament pellets in bulk at \$84.28 per 18 pound bags [17]. The equation to find the cost of materials is $Y=(X\div 8164.66)*84.28$ with Y being the cost of materials and X being the material amount used.

Table 4. Cost and Time Required for the Parts Made in the Project

Part	Time Required	Materials Used	Cost of Materials- 900g reel	Cost of Materials - 18lbs (8164.66g) of pellets
Jig 100% Infill	6 hr 58 min	151.96g	\$8.44	\$1.57
Jig 50% Infill	4 hr 56 min	101.79g	\$5.66	\$1.05
Outflow Stator	18 min	1.56g	\$0.09	\$0.02
Full Impeller	59 min	5.19g	\$0.29	\$0.05

8. CONCLUSION

The problem that this project faced was the flexibility of a 3D printer in a medical application that also has viability in the industrial sector. It is generally known that a 3D printer can make rapid prototypes of part to show where modifications might be needed. This project has shown that it can be taken a step farther. Through the use of various methods, the possibility of reducing the inventory down to the bare essentials is a viable option. A good 3D printer can cover this reduced inventory by printing the parts in a fraction of the time it takes for a CNC machine to mill out the part. It can be used to substitute replacement parts temporarily until the new part can be ordered or milled saving the company money by reducing the amount of parts lying about doing nothing to make the company money. If this was the only thing the 3D printer could be utilized for it would also be wasting the money of the company that owned it. It was found that a 3D printer could make jigs for use in mills with a reasonable precision. This adds to the flexibility of the 3D printer thus reducing the amount of unused capital by the parts being unused or unutilized machinery.

9. FUTURE RESEARCH

During the course of this paper, the quality of the part was unable to be improved to the point of where the only thing that needed to be removed would be the supports. The use of different slicer programs would need to be investigated. The program that was used, Makerware Desktop, is not efficient in the tool paths it sets for the extruder assembly to take. This is part of what caused the stringing issues this project was having. Different programs will need to be investigated and implemented to improve the quality of the part. Another area that can be investigated is the ability of the parts to hold up under the conditions the parts would normally be put under. This will test the viability of the concept of temporary parts and make it more than just a proof of concept.

ACKNOWLEDGEMENTS

The authors would like to thank Dr. Charles Taylor and Clint Bergeron for their valuable assistance.

REFERENCES

- [1] The MakerBot Replicator+ | MakerBot [Internet]. Makerbot.com. 2017 [cited 24 April 2017]. Available from: https://www.makerbot.com/replicator/?ab=toughpla&utm_expid=65420830-9.IUwf5r7YS8GRRM_QNCgAMw.1&utm_referrer=https%3A%2F%2Fwww.makerbot.com%2F%3Fab%3Dtoughpla.
- [2] Pocket NC [Internet]. Pocket NC Company. 2017 [cited 24 April 2017]. Available from: <http://www.pocketnc.com/pocket-nc/>.
- [3] Martorelli M, Gerbino S, Giudice M, Ausiello P. A comparison between customized clear and removable orthodontic appliances manufactured using RP and CNC techniques. *Dental Materials*. 2013;29(2):e1-e10.
- [4] Kun K. Reconstruction and Development of a 3D Printer Using FDM Technology. *Procedia Engineering*. 2016;149:203-211.
- [5] Herrmann K, Gärtner C, Güllmar D, Krämer M, Reichenbach J. 3D printing of MRI compatible components: Why every MRI research group should have a low-budget 3D printer. *Medical Engineering & Physics*. 2014;36(10):1373-1380.
- [6] Satyanarayana B, Prakash K. Component Replication Using 3D Printing Technology. *Procedia Materials Science*. 2015;10:263-269.
- [7] 3D Printer Filament Types Overview | 3D Printing from scratch [Internet]. 3D Printing from scratch. 2017 [cited 24 April 2017]. Available from: <http://3dprintingfromscratch.com/common/3d-printer-filament-types-overview/>
- [8] Direct and Binder 3D Printing techniques | 3D Printing from scratch [Internet]. 3D Printing from scratch. 2017 [cited 24 April 2017]. Available from: <http://3dprintingfromscratch.com/common/direct-and-binder-3d-printing-techniques/>
- [9] Types of 3D printers or 3D printing technologies overview | 3D Printing from scratch [Internet]. 3D Printing from scratch. 2017 [cited 24 April 2017]. Available from: <http://3dprintingfromscratch.com/common/types-of-3d-printers-or-3d-printing-technologies-overview/>
- [10] Top 10 Materials for 3D Printing [Internet]. Asme.org. 2017 [cited 24 April 2017]. Available from: <https://www.asme.org/engineering-topics/articles/manufacturing-processing/top-10-materials-3d-printing/>
- [11] Dehue R, Strikwerda P, Strikwerda P, Strikwerda P, Dehue R. What is 3D printing? How does 3D printing work? Learn How to 3D Print [Internet]. 3D Printing. 2017 [cited 24 April 2017]. Available from: <http://3dprinting.com/what-is-3d-printing/>
- [12] Dieter G, Schmidt L. *Engineering design*. 1st ed. New York: McGraw-Hill; 2013.
- [13] Ross L. *Robotics*. 1st ed. Tinley Park, IL: Goodheart-Willcox Company; 2011.
- [14] Print Quality Troubleshooting Guide | Simplify3D Software [Internet]. Simplify3d.com. 2017 [cited 24 April 2017]. Available from: <https://www.simplify3d.com/support/print-quality-troubleshooting/>
- [15] 7 Tips For Better 3D Printing [Internet]. MakerBot. 2017 [cited 24 April 2017]. Available from: <https://www.makerbot.com/media-center/2014/12/05/makerbot-7-tips-better-3d-printing>
- [16] Sink A. Fixing a Jammed/Clogged Makerbot Replicator Smart Extruder [Internet]. Sink Hacks. 2017 [cited 24 April 2017]. Available from: <http://sinkhacks.com/fixing-a-jammedclogged-makerbot-smart-extruder-teardown/>
- [17] PLA 4043D Week Long Box - Domestic Free Shipping (Within US) - Open Source 3D Printing [Internet]. Open Source 3D Printing. 2017 [cited 24 April 2017]. Available from: <http://store.os3dp.com/bundles/38-pla-4043d-pellets.html>