Metal FDM Testing Report (Using BASF Ultrafuse 316L)

Parker Hannifin

Andrew Bok, AMLDC Intern

Summer 2023

Table of Contents

Executive Summary

The production of custom metal parts is an important business matter for the AMLDC and many divisions of Parker. Producing complex parts is typically dependent on immense investment costs associated with acquiring tools suitable for the job, which can be one of the challenges of divisions adopting metal 3D printing technology.

BASF Ultrafuse filament is a material for FDM 3D printers which enables the production of stainless-steel parts at a fraction of the cost of other metal 3D printing technologies. The basis of its low cost is the use of traditional FDM 3D printers and outsourcing of the debinding and sintering post processes which use expensive equipment to transform a part from a mixed polymer and metal “green state” to a finished metal state. The outsourced processing is already an established system organized by MatterHackers, the vendor of this filament, meaning users just have to buy a processing ticket and ship their parts out. Post processing runs only happen twice per month, so parts are typically returned in approximately two to three weeks, which is a primary drawback of this process.

This report goes into the details of preliminary experimentation with this filament, covering my experience of doing ten test prints, post processing the parts, and analyzing the final sintered results. It also briefly compares the investment cost of performing metal FDM printing to other methods of metal 3D printing. The goals of this project were to learn how to successfully print the material and learn about the limitations that it had for producing metal parts. Prints included basic tests to identify good settings, functional parts, and some “torture tests”, which ultimately allowed the project to be a success with those goals. The report also has a collection of external appendices, which include many pictures taken during the printing process and of final products, 3D models and printer files used for printing, external resources like data sheets, and more.

BASF Ultrafuse filament was found to produce quality metal parts so long as the inherent constraints of the material are kept in mind during part design. Some challenges presented by the material include the tendency of the part to warp away from the print surface during print (which requires cleanup work on parts), fragility of small features in the green and brown states, and poor overhang quality after sintering (which can be remedied by supports that are removed via machining as a final production step). Developmental work with this filament should be continued and should now be directed towards real use cases within Parker. These production grade parts can likely be copies of 3D prints running on the AMLDC’s metal PBF machines, which would allow for easy comparison between methods. Good results in these tests would open the door for divisions to adopt metal FDM 3D printing as a low-cost, in-house solution to produce high complexity metal parts. If engineers at Parker look to adopt this tool, one of the available resources to maximize the likelihood of success is a design consultation service specific to working with this material, again offered by MatterHackers. It costs $500 normally, or $200 when added on to the purchase of a spool of BASF Ultrafuse filament.

One-Page Guide for Metal FDM 3D Printing with BASF Ultrafuse

Part Design:

- Parts should try to have a uniform thickness. A low height to width ratio is also advised.
- The minimum wall thickness of parts should be at least 2mm.
- Features with a cross sectional area smaller than ~20mm^2 (as modeled) are very likely to break during shipping or handling. Avoid small features where possible.
- Avoid sharp corners by chamfering or rounding where possible. Sharp corners are more likely to suffer from manufacturing defects like warping or delaminating.
- Support vertical circles or use a tear-drop shape to prevent collapsing and severe sagging.
- Overhangs can print at 45^o, but weight and cantilever forces during post processing can lead to failure such as cracking or collapsing. Instructions for FEA simulation of post processing warping can be found in Appendix D.

Printing Best Practices:

- Use MakerBot Print, not CloudPrint. My .printmode file is in Appendix C.
- Scale parts up by 20% in X and Y, and by 26% in Z to counteract shrinkage of post processing.
- Print objects towards to middle of the build plate; printing near the edge resulted in warping.
- Clean the printer's build plate of dust and debris prior to applying Dimafix. **(Use Dimafix)**
- Monitor the first layer of a print to make sure it prints evenly and adheres well to the bed.
- Wait for parts to cool before attempting to remove them from the build plate, as Dimafix's bond strength is related to temperature. It weakens as things cool, making parts easier to remove.

Post Processing:

- **Plan ahead!** Debinding and sintering only happens twice per month and takes a week. The bare minimum cycle time with expedited shipping options is a week and a half, but if parts don't arrive to DSH in time, they will be delayed for another three weeks, so be proactive. Part iterations should be expected to happen at a rate of once per month.
- Handle printed parts with care. The green (printed) state is weak and simply dropping the part could result in features breaking, likely meaning the material is wasted.
- Sand the base of parts flat to prevent cracking. (Design in extra base material for sanding)
- Be generous with bubble wrap when shipping parts. Parts broken during shipping are still processed but will likely be irrecoverable for their intended purpose.
- Machine parts after sintering for support removal, high tolerance features, threads, etc.

Pros	**Cons**
• Inexpensive high-complexity metal parts. • Outsourced processing solution is established and easy to use. • Design consultation available from MatterHackers to increase odds of success. ($200 if added on to a purchase of filament, $500 otherwise) • More resources are available online.	• Part processing and two-way shipping means parts take ~2-3 weeks to return. • Limited dimensional accuracy due to sintering shrinkage. (~±0.01 inches) • Material tends to warp. • Complex parts are only near-net-shape and typically require machining operations for precise features.

Introduction and Background

This document covers my experiences of exploring metal FDM 3D printing. As a high-level overview, one can use this process to produce 3D printed metal parts for a cost that is orders of magnitude lower than conventional methods, though with some drawbacks. For metal FDM, a normal* FDM printer prints a special filament which is 80% stainless steel particles and 20% polymer by volume. The resultant parts are in a "green state", which requires post-processing to turn the parts into near-solid metal parts. Post-processing consists of chemically debinding the polymer component, followed by sintering the metal to remove porosity. These processes are equipment intensive, but there is a convenient outsourcing option to do these tasks (see the Post Processing section below for more info). A consequence of this post processing being outsourced is that the cycle time of producing parts includes a step that takes at least two weeks. Geometry of parts is restricted in similar ways as other forms of metal 3D printing; complex features that require support are "near-net-shape", where subtractive manufacturing techniques are used to finish parts by removing supports and cleaning up areas that require precise features.

* Some specific features are required, such as a new/clean hardened-steel nozzle and a heated bed/chamber.

External Appendices List

Along with this document, a series of files have been organized as appendices to better present findings of the project. The report out should have been shared as a (zipped) folder, which includes the rest of the files, organized as lettered subfolders:

A. A PowerPoint file has been created to include images taken of each test print, which typically included pictures to show warping, bed residue, and general print details. These were placed in an appendix to keep this document much shorter, due to the number of valuable pictures. There is also a folder with all images, including some screenshots from slicing and other useful images.
B. An Excel sheet was used to track prints with details such as their weight, print time, and general notes. Other tabs were added to analyze dimensions of parts before and after sintering.
C. All files related to the 3D printing side have been included. This consists of 3D models as .STL's, as well as .STEP and .f3d files if I made that model myself. There's also a folder of the sliced .makerbot files of each print, and my .printmode slicer settings file.
D. This folder collects useful resources from online for easy future reference. It includes datasheets for the filament and Dimafix, design guides from Forward AM (the producer of this filament) and MakerBot, and links to product pages and tips for success with the filament from MatterHackers.
E. This folder related to post-processing documents. It has the shipping form document I had to fill out for DSH, the post-processing report from DSH, and the processing ticket receipt.

Test Prints

This section goes over descriptions of all test prints, my logic backing the print, and things I learned from it. I recommend reading at least the first paragraph of each print to understand goals of this testing. For information on the final products, refer to the *Final Sintered Parts* section later in this document. Pictures of the parts have been compiled in a PowerPoint file available as Appendix A found within the report-out folder.

Print "0" – Failed Geometry Test

This print was meant to be a self-designed test artifact to evaluate features in a manner somewhat like the NIST 3D Printing test artifact. It used the design guides from the resource appendix as a general reference for acceptable features, but sought to test them by having smaller features, too.

Features being tested included:

- Minimum wall thickness
- Minimum pin sizes
- Slot and hole sizes
- Overhang angles
- Details and text on the sides of parts

I forgot to apply Dimafix to the print bed for this print, and the print peeled up severely after printing only 1mm or so, plus layers were showing some delamination, so I stopped the print.

Dimafix is a special adhesive with temperature-dependent behavior. It is strongly recommended by MatterHackers to achieve successful prints, for good reason as this failed print showed.

This print also showed the nozzle to be "globbing" with a flakey mass of overextruded filament. Future prints primarily showed this to happen during the first layer or due to warping. Online sources say to use a glass bed, but unfortunately this does not seem possible with the MakerBot Method X, which otherwise excels with required specifications to print this filament.

Print 1 – Geometry Test Reprint

This print used the same .makerbot ("gcode") file as Print "0", but Dimafix was applied to the bed before printing.

The corners started peeling up sometime around 60%, but the print was allowed to finish.

When removing the part, some sections of first layer stuck too well to build plate and delaminated from the part. These then had to be scraped off the build plate. This issue reoccurred for many prints, with no settings seeming to help resolve the issue. MakerBot Print lacked settings to fine tune the first layer, which would seem to be the best solution to this problem.

The Excel sheet of Appendix B in the report-out folder includes a tab of recorded measurements from this test part, both before and after sintering.

This test print demonstrated the fragility of “green parts” for this filament (as printed/un-post-processed parts). Between measuring the part’s features and general handling, all 5 pins broke off the base and the 40-degree overhang also broke, about 1/3 of the way up.

I choose to break the cone off to see how much force it took to delaminate. It required moderate pushing and didn’t delaminate quite all in one layer, or along the base layer, so having infill instead of just perimeters (as the pins did) helps. The test was not very scientific but properly testing layer strength is a good idea for later if there is equipment available (a force gauge at least). I think the loads of dropping a part would be enough to risk delaminating a cross sections of ~1cm^2 or less.

More could be done with this idea of testing, but the design needs iteration. The overhangs should be revised such that they are from a more solid wall, like a cliff, instead of the current self-supporting angle.

Print 2 – 20mm Cube (CloudPrint)

Ultimately, the first print was overambitious. I redirected my focus toward simpler for finding settings that worked better. This started with using MakerBot CloudPrint, since I found out that it had a built-in profile for metal filament. The first test model was a simple cube, which was scaled up to have a final size of $(20mm)^3$ post-sinter.

The infill was generated as lines aligned with the X and Y axes. Changing settings such as infill percentage had no effect and seemed to be overridden by the “Solid” profile. The slicing algorithm did not generate an actual solid print, with a slightly larger gap on one end of a layer. Visible gaps can be seen between lines on the final part, and this part ended up 13g (~20%) lighter than the 100% infill cube printed using the desktop slicer app, MakerBot Print. This profile’s default settings interestingly had no roof layers, and the chamber/bed temperature at 60°C, which went against documentation suggesting ~100°C. Perhaps this was for compatibility with the standard MakerBot Method.

During printing, the first layer showed overextrusion from being too close to print bed. The plastic was “globbing out” up onto nozzle. CloudPrint has setting for first layer Z offset which could be used to balance this setting for less overextrusion and still good adhesion, but I stopped using CloudPrint after this one test.

There was also some delaminating around corners within the bottom layers once the print was ~25% done. The print was allowed to finish, but this was another reason that these default settings were abandoned. The first few layers (below the delamination) were stuck to the build plate and had to be scraped away after part removal.

Print 3 – 20 mm Cube (MakerBot Print)

This print was intended to be a direct comparison of the two slicing options, so the same model was used: a cube scaled by the recommended ratios so it should end at $(20mm)^3$.

This print can be distinguished from the CloudPrint cube because it is heavier, and the top surface has diagonal lines.

This print had warping away from the build plate. Dimafix was not working ideally, I think that may have happened because I brushed globs off the nozzle while the print preheated, but I didn't clean the bed after, so there were likely dust and particles from cleaning the nozzle that lessened adhesion. For future prints, I made sure to clean the bed of dust before each print.

I forgot to export this print's .makerbot (gcode) file, so that is missing from the folder. I attempted to reslice the print, but the infill was now along X and Y instead of diagonal at 45°. I didn't change any settings but had restarted the slicer, so this was a minor weird and annoying occurrence.

I allowed print to cool for 2 hours before removal from build plate because I had looked at the Dimafix TDS, which explained how adhesion was reduced at lower temperatures. The part came off easily with a few strands on plate, but not nearly as bad as previous ones that had to be scraped off.

Print 4 – 20mm Domed Cylinder and Cube: Solid Infill

This print is the first in a series of three with the goal of testing non-solid infill for parts. I initially assumed this to be unsuitable for this filament due to the post processing, but the MakerBot design guide reports that 60% infill is generally achievable for reducing the weight, print time, and cost of parts.

I was testing with two objects: a cube and a cylinder with the top half rounded into a dome. Both objects have $(20mm)^3$ overall dimensions. The cylinder helps test adhesion of parts without sharp corners, as well as showing how stair-stepping looks on final parts, and if infill affects the top surface quality on non-planar features. The parts were not scaled to counteract post-processing shrinkage since the final size was not a significant detail, and this saved on material and print time significantly.

As learned from Print 3, I made sure to clean the bed with a wet shop towel and tried to remove dust and lint prior to applying Dimafix to reduce the odds of warping.

I continued using MakerBot Print as my slicer, but I slowed down the speeds a little to hopefully improve print strength.

Both parts still peeled up with minor warping. Both parts were placed near the front edge of the build plate and warped on their front side with respect to the printer. I suspect that uneven heating or convection drafts could be the cause, so I printed closer to the center of the bed in future tests.

Print 5 – 20mm Domed Cylinder and Cube: 66% Thatch Infill

As said above, this is a series of prints to test sub-solid infill. This test used 66% as something slightly above the "official" 60% boundary listed by MakerBot.

MakerBot Print would not update the toolpath to reduce the infill when the setting was the default option of using a linear infill pattern. I found that changing the setting to thatch infill made the software update the toolpath appropriately.

As learned from Print 4, I tried moving parts toward the center of build plate. The two build plates have some cracking and damage in the middle, which could affect prints, but the surface can be replaced if needed as we have spares. This change reduced the warping a considerable amount, but there was still a tiny bit present.

To reduce cycle time and get more tests done in a day, I began trying to swap plates immediately when a print finishes to reduce the reheating time between prints. This method took 5-10 minutes to get back to the target temperature to start the next print, as opposed to a ~20-minute heat up from ambient temperature. I still allowed parts to cool for at least an hour before removing them from the build plate.

Print 6 – Chess Rook

This print was meant as a stress test and a partial comparison to the metal PBF printers that the AMLDC has. It is high detail, in contrast to all previous prints. It goes against the advised aspect ratio of 3:1 (base size vs. height), so it's also testing that guideline. There were sample parts in the online resources that similarly ignored the recommendation.

Here is the link to where I got the model, (which modified it in an unspecified way from Formlabs, the original creator): Rook by alfred_g_c

The original model has an ornate helix and spiral staircase inside the tower, but these features were assumed to be beyond the capabilities of this process. The model was modified using Autodesk Meshmixer to remedy this by making it solid, and therefore printable.

This print still had minor peeling from the build plate, which was noticed around the halfway point. Otherwise, it came out looking quite nice.

Print 7 – 20mm Domed Cylinder and Cube, 40% Thatch Infill

This was the last of the three infill test prints. This geometry is the same as the other tests. The goal was to push the infill beyond the recommended limit, down to 40% and see what happens.

These prints still warped, but only a tiny bit. Less infill probably creates less residual stress that leads to warping, which is another possible benefit. The main reason to print metal though is likely strength, which infill negatively impacts harshly.

These prints stuck to the bed too well, resulting in most of the first layer sticking when removing the rest of each part. This inconsistency between prints is interesting, as the previous infill tests seemed to extrude the first layer slightly too far from the bed, leading to lines that didn't really overlap or adhere to each other. MakerBot Print doesn't seem to offer control over the first layer Z offset like CloudPrint did, but this varying by print is unideal.

Print 8 – Gearset and BASF Pyramid Mimic

This print combined two different tests to make the best use of running overnight. The first was a more functional test that I modeled: a little gearset toy. The second was my replication of a pyramid structure shown in BASF's resources for the Ultrafuse filament.

The gears were created using a Fusion 360 extension called GF Gear Generator. It used the “profile shifted spur gear” option with a shift coefficient of -0.2 for both gears to create some smaller tolerance between them to increase odds of success after printing and sintering. Other settings used were: module = 2mm, teeth = 12t and 18t, height = 8mm, pressure angle = 14.5°.

The cradle to hold the gears is also a functional test by having holes meant to be tapped to an M5 thread after printing, so that screws and washers can hold the gears in place.

The first three layers went down well before I left work for the day. When I came back to work the next morning, I found the print had paused after 4 hours due to the printer detecting a jam. The printer was at least setup to maintain the chamber temperature even when paused. The gear cradle was severely warped away from the bed, so I think the jam sensing triggered due to resistance associated with the glob that developed from having to push down the warped area as it printed over it. I unloaded and reloaded filament and it extruded fine, so I resumed the print.

The print ended up finishing without pausing again. I had partially expected it to pause again due to more blobs forming on the nozzle from the warped area. The blobs would sometimes fall off as crumbs around the prints, but they didn’t negatively affect things.

The final gear holder was ugly with a very warped base layer, but I was able to sand the bottom to be mostly flat so that it can be sent out instead of reprinted. I was worried about delamination of the sides and towers, but they held up to the forces from sanding.

The other parts turned out well. Both gears had minor warping along the teeth and a slight ridge around the perimeter around a third of the way up. This is possibly related to the pause overnight, but it will likely need to be filed after sintering. The pyramid also had some minor warping.

Print 9 – Benchy

This print is a Benchy boat; a famous benchmark for FDM 3D printing. Printing this provided clearer insights on how this material differs from ordinary FDM filaments, as well as acting as another stress test for the capabilities of this material.

Here is the link for the model, which includes a “brochure” of test features under files: 3DBenchy by CreativeTools

This model pushed this filament and demonstrated some of its limits. The most obvious struggle was bridging, which is when a straight line is extruded over a gap with solid structures at both ends. The filament is much heavier due to the metal particles, and therefore sags worse than regular filament. Eventually features over bridges recovered, but there were loose lines that had to be cleaned off the final part in multiple locations around the roof of the boat.

Another area where this test went poorly was the text on the underside of the model. The first layer again stuck to the bed too well, so it mostly delaminated when the part was removed. The bottom of the print was sanded flat for post processing.

Other than those two issues though, the print looked quite good in its green state!

Print 10 – Bevel Gears

One final print was done to get as close as possible to the 1kg weight limit for a single processing ticket. I browsed Printables and found a bevel gear toy that would make a good second functional print. The full assembly requires other parts, which will be printed in ABS to assemble the final product.

Here is the link for this model: Bevel Gear Fidget by Mistertech

I watched the first layer get printed. The first part was messy with some blobs, but the rest were clean. When the nozzle returned to do the second layer, the blobs were absorbed by the nozzle's bigger blob, and things continued printing fine.

The prints turned out looking good, but again they stuck to the build plate too well. All four parts had some area chip away from the first layer. I didn't bother sanding these parts flat though, so that I can see what happens from post processing with a poor first layer.

Slicer Setting Summary

Temperatures	Nozzle = 240°C, Chamber = 95°C (Max for Method X)
Cooling	None (set all fan speeds to 0%)
Layer Height	0.15mm worked great. Parts have ~0.12mm layers after sintering
Shells (Perimeters)	2 (More is detrimental because they tend to delaminate from each other)
Floors	Solid = 0.4mm, Surface = 0.4mm
Roofs	Solid = 0.6mm, Surface = 0.4mm
Infill	Use 100% linear pattern infill for max strength and weight. To reduce weight and material usage, use the thatch infill patter as low as 40%.
Extrusion Multipliers	100% was left for all line types.
Speeds	150 mm/s travel, 15mm/s base layer, 25mm/s perimeters, 40mm/s infill
Supports	None (model them in, if necessary. Do not generate them in the slicer)

Note: MakerBot Print doesn't allow for modifying of other settings. Settings like perimeter overlap and infill overlap percentage would be a good candidate to bump up a few percent if using a different printer and slicer.

Post Processing Notes

Preparation

Be aware of the processing schedule! Post processing only runs twice per month (2nd and 4th Tuesday for each batch's start) and parts need to arrive on the Friday prior to be included in that processing run.

Order processing ticket from MatterHackers early! It says 48 business hours to get a processing code via email, but my experience was much slower (albeit due to the July 4th holiday). I ordered a ticket on Wednesday so that the code should've come in by the Friday a week before the parts needed to arrive. The code didn't arrive and there was a long weekend with Monday and Tuesday off for July 4th. I submitted a support ticket on Monday which wasn't responded to by Wednesday, so I called support to get my code. I still didn't have it on Thursday, so I called support again when they opened, and got an email with the code a few hours later. Parts then had to be overnight shipped due to these delays, otherwise parts would've been returned to me on my second-to-last day of the internship most likely.

Learn from this mistake and buy a ticket further in advance than I did! (If you need to buy one beyond the code that should be included with the purchase of a spool, that is.) They shouldn't expire, so buying it early is a smart move.

This article from MatterHackers is what I followed to prepare my prints: How to Prepare Ultrafuse Metal Parts for Debinding and Sintering

It laid out a six-step process, which I have included a paraphrased list of below. Read the article for more information.

1. Follow the suggested parameters when printing green parts.
2. Clean parts of burrs, residue, and sand part bottoms flat.
3. Buy a processing ticket. (Spools come with one ticket, so this may not be necessary)
4. Fill out a form with part details such as weights and sizes.
5. Safely pack parts for shipping.
6. Ship your package with parts and the form.

Step 2 is the main thing that requires effort after parts have been printed. Here are tips for those processes:

- To remove defects, such as burrs and blobs, I used a regular set of flush cutters and a small file. I found these tools to work great, especially for the print that needed the most work, which was the Benchy due to its poorly printed bridged sections.
- The bottom of some parts were tacky due to the Dimafix. To clean this residue, I rubbed the bottoms of parts with a shop towel dampened with warm water. If any lint from the towel was on the part, I did my best to blow or wipe it off. I did not find fingerprints visible on the parts, but to be safe I attempted to wipe surfaces down with another towel and isopropyl alcohol.
- During sanding, be careful to not break the part, especially if there are any small features on the part, since green parts are fragile!
- Sand parts on top of sandpaper placed on a flat, rigid surface that can get dirty with dust from the prints. I used a corner of my desk, which I then had to clean a sizable powder pile from.

- Attempt to sand by pushing parts in one direction, rather than going back and forth. This helps to keep the base flatter. It's easy to accidentally rock a part back and forth and create a slight curve on the sanded face when moving a part back and forth.
- The rook, Benchy, and spur gear holder all needed to be sanded flat. I did the rook because I was already doing something not recommended with a . The other two parts had terrible bottom surfaces between warping and partial delamination.
- I didn't sand every part flat. Things like the spur gears and cubes have another flat face that they should be able to be sintered on, so they should be fine. Parts like the bevel gears, geometry test, and pyramid had some level of defect on the bottom surface, but I wanted to see how bad the consequences are for a slightly warped bottom surface.

Part Identifier Engraving Key:

I printed multiple parts with identical geometry for the test cubes and cylinders. While they had different weights due to testing infill or differences between the slicer used, I wanted a more discernable differentiation method, so I carved letters into these objects using a pocketknife. A stamping tool set may have been more ideal, but that could also risk excessive damage to the parts.

Here is the key for the letters on the objects:

- Big cubes: C = sliced using CloudPrint (Print #2), P = sliced using MakerBot Print (Print #3)
- Small cubes and domed cylinders: S = solid, M = medium infill (66%), L = light infill (40%). (S, M, L may not have been the best choices with their contrast to the potentially assumed definition of "small, medium, large", but oh well)

Final Sintered Parts

Below are notes about how all the parts turned out after being sintered. Pictures of results can be found in the slideshow of images. This section categorizes parts by purpose rather than chronologically as the prints section did. One note which will not be repeated below is that **the base layer of virtually all parts is not flat due to some amount of warping**.

Settings Tests

Slicer Comparison Cubes

Both cubes were scaled up by the advised 20% in XY and 26% in Z when printing. The parts have dimensions very close to the desired 20mm in all dimensions after sintering. The print sliced in MakerBot CloudPrint had the walls separate from the infill on all four sides along the top face of the print, with dimensions of ~21.4mm in each direction. This wall-infill delamination is yet another reason to disqualify MakerBot CloudPrint as the slicer to be used for metal FDM. The only problem with the MakerBot Print cube is a sparce first layer, but the top layer and sides look great. The dimensions of the cubes and their mass were used to calculate density. The MakerBot Print cube resulted in ~7.4g/cm^3, or ~94% the density of regular steel. The CloudPrint cube had a density of ~5.9g/cm^3 or ~75% the density of steel using the default "solid" profile for metal filament within the slicer. The scale used to measure mass only reported to the nearest gram, so the error margin on a 59g cube is roughly ±1%.

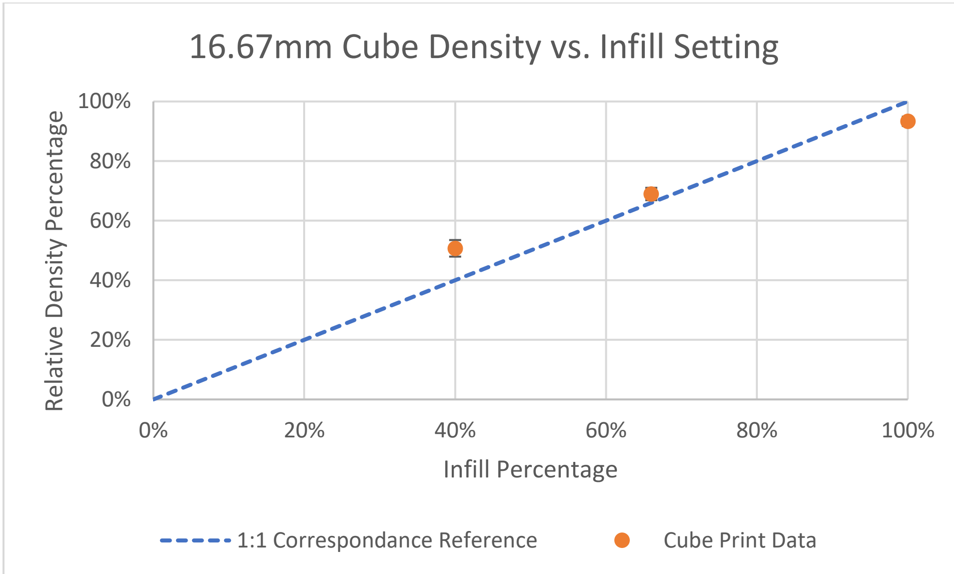
Infill Testing

Infill testing was done with a 20mm cube and 20mm diameter cylinder with the top edge rounded off to a half sphere. These models were not scaled up at all, and thus demonstrate the anisotropic shrinkage if placed next to each other in different orientations. Three sets of prints were done to test 40% and 66% infill, and to have solid references of the parts.

Overall, this test was successful in demonstrating that parts can be printed with lower infill percentages. The test set was too small to say that 40% reliably and safely passes, but both prints here showed no external signs of infill affecting the part during post processing.

The cubes with non-solid infill both had one side's wall separate from the inner wall. I attempted to pry the wall away from the 40% infill part using pliers, but I couldn't do more than bend the segment out around 2mm. The single layer of metal was still surprisingly strong and resilient with is bonds at the corners of the cube. The separation seems to be due to the sharp top corner and could likely be prevented by having chamfers or rounds on the corners. This could also be resolved by modifying slicing parameters such as overlap percentage for walls and infill, but MakerBot Print does not grant control over those. The parameters available in that slicer are mostly limited to speeds, fan power, and extrusion multipliers.

One thing I did with the cubes was calculate their effective density and plotted this compared to infill percentage (see below). The trend should not be taken as more than a loose rule, as larger parts with more infill relative to the solid volume of walls and top and bottom surfaces would likely trend closer to the theoretical density presented by the infill percentage.



After all measurements were taken, I tested out sanding and filing the solid parts to improve their surface finish. The normal surface finish of parts is slightly rough and a matte grey. I sanded one face on the cube, and it ended up with a shiny metallic finish, but I didn't fully sand all layer lines out. A little more work could have done that, but I left them for sake of demonstration. I also used a file and sandpaper to test out cleaning up the domed cylinder. The file worked fast to change the surface from matte to shiny, while the sandpaper was slower but also worked. It was harder to work with the smooth, round surfaces compared to the cube's flat face. Tools like a belt sander would make short work of improving surface finish for non-dimensionally-critical areas.

One final way that these parts were analyzed was by taking the cubes to a mill to create a 3/4 section view to visually expose the infill patterns. The 40% light infill shows clear visible gaps, but the inconsistency of the infill lines is likely due to forces from the cutting tool. The 66% medium infill was initially cut in the wrong orientation, so it has two cuts turning it into a staircase. There are clear lines corresponding to the infill for this cube, but no visible gaps. Theoretically, they should have been ~0.15-0.2mm wide after sintering. I suspect that the gap is present inside the part, but the cutting tool "dragged" material to make it look more solid on the surface. The solid infill reference cube looked like regular solid metal after being cut. There is no pattern or signs of infill present on the surfaces of this part. Wire EDM may be able to better show the infill of parts, but that was not attempted due to the work required for fixing parts in the machine (and Mike being gone at that time).

Geometry Test

This was the first print with this filament, so settings used were not yet tuned on this print, nor was the filament understood. It intended to test small features and limits of the material. The protruding small features all broke. Mostly during regular handling, but the ones that survived that broke in shipping. This allowed me to learn about the weakness of the green state of parts and that small features are very likely to break, so they should be avoided.

One of the other big issues with this part was cracking during debinding and worsening during sintering. The guides in Appendix D list having non-flat bases as a risk for cracking. I would guess the cracking is related to having a first layer more warped than others, especially for this part size. There is a big crack running along the middle of the part, mostly along the interface between walls and infill. This print used three walls, while all following prints reduced that to the recommended two, since wall-to-wall bonding strength is a weak point, as demonstrated by the cube prints.

For design features tested by this model: The slots and holes showed no signs of significant deformation, though the size seemed to lack some precision on smaller features, but measurement accuracy is a possible error source for those. Dimensions can be found in the Excel sheet shared with this report. The thin walls were not extruded as solid and resulted in warping. This test suggests a minimum wall thickness of 2mm. The overhang tests broke on this model, but the bevel gears prints showed that 45° was achievable, while the Benchy print demonstrated the importance of supporting cantilever loads from overhangs.

The shapes on the sides of the part turned out well, with only minor deformation present. The text on the side was sloppier. There was drooping of overhangs from the geometry of the text, making it difficult to read. The best solution would likely be a bigger font size, but changes like a draft angle or deeper embossment would likely improve text legibility of final parts.

Functional Tests

Spur Gearset

The gear holder has a visible crack where the base and wall connect on the side that severely warped, so I was nervous about the wall breaking, but it has held up. The three parts assemble nicely, and the gears run rather smoothly after being broken in a bit. The fit between the gears and holder is a little inconsistent between the two gears due to different final inner diameters. This demonstrates that real final parts would want machining for accurate mating features, but also that this filament is capable of functional applications such as gears. The gears are almost certainly of lower running quality than machined gears and wouldn't be the best option for most engineered applications, but they get the job done with this basic toy.

Another function test accomplished by this part was tapping threads into a printed object. The design was for an M5 thread, but of course that was the only metric tap missing in the shop's tap set. Instead, I tapped the holes to the closest imperial thread: #10-32. The part is slightly marred from being clamped in a vice, but in serious production cases, soft jaws could be used. Tapping went smoothly and screws went in without a problem.

Bevel Gearset

This model shrunk during processing more than anticipated such that the final internal diameter of the gears was closer to 21.5mm than 22mm, meaning that they could not accept a bearing as intended to make the connected fixture of gears. Three of the gears had a wall segment of the inner circle perimeter on the bottom layers delaminate from the inner wall and the fourth gear is missing that segment entirely. The quality of the parts is quite nice, but shrinking more than anticipated makes them unusable as intended, which highlights a risk of this filament.

I also attempted to print the frame for the gears in ABS on the Method X, but the parts didn't turn out well enough to function.

"Torture" Tests

Rook

This model turned out looking great. Compared to the rooks printed on our metal PBF printers, the main difference is the surface finish. The FDM part has a very consistent finish in XY and noticeable layer lines in Z. The PBF parts have less visible layer lines but a slightly rougher surface finish. It also has a significantly rougher finish on the 45° overhang segment. Another difference was that the FDM one had to be modified to have solid geometry, but it would be a good candidate for printing with lighter infill since it is a decorative part.

Benchy

The Benchy unfortunately broke during shipping to DSH. The pillars supporting the roof and smokestack broke apart, which can be seen in Appendix E. The post processing also didn't go too well. During debinding, the front hull of the boat fell forward due to being heavy and unsupported, creating a cantilever force greater than the model could handle. The model changed again during sintering, which had the steering wheel area collapse. Based on how that part sits upon the rest of the boat, it's possible that it broke while being transported for pictures.

The sintered result was a bit of a mess. It looks like the solid infill may have been oriented uniformly parallel to the back of the boat based on the split sections. This would be the worst possible orientation since extrusion lines are more likely to delaminate from one another compared to breaking. Unfortunately, MakerBot doesn't support viewing a sliced preview of saved .makerbot files, so there is no way to determine if this was part of the cause of failure.

All things considered; it isn't particularly surprising to have had this torture test fail. I tested the bonded strength of broken segments by pulling things apart with pliers. I was able to pull off a few small "crumbs" (~5mm x 5mm x 2mm) of the print with a little bit of effort, but I couldn't pull the steering wheel off the base; there seems to be too much bonded area to easily separate it.

Pyramid

This model collapsed under its own weight during debinding. DSH didn't adjust it on the tray at all between debinding and sintering, so it was sintered as a crumbled pile. The pieces managed to bond together during sintering, but they could probably be pulled apart with pliers like the Benchy since its mostly bits of edges and corners bonded. Upon looking a little closer at the promotional image with a similar model, I noticed that that model has tapering arms, so the top is a little thinner and lighter. This model's failure demonstrates the weakness of parts in the brown (debinded) state.

Key Takeaways

General Notes

The one print from MakerBot CloudPrint yielded poor results between low density and poor lamination between walls and infill. Thin features are very fragile and prone to breaking prior to being sintered, but if they survive shipping, they will end up metallic-grade strength. The biggest consideration to make with overhangs is the weight being supported, since some overhanging features that were able to print fine failed during post processing. The stock surface finish of parts is matte with layer lines visible, but parts can be machined or polished like any ordinary metal to improve surface quality.

Shrinkage and Accuracy

The provided scaling factors of +20% in X and Y and +26% in Z gave good results with prints. Dimensions were mostly accurate around the order of 0.005 inches for the sample parts I had, but I wouldn't have confidence in holding tolerances more precise than even 0.01 inch yet, given such limited data. Features that involved even small overhangs had sagging which reduced the Z accuracy. The bevel gears shrank a fair bit more than expected in X and Y. I assume it is due to their ring-like shape with no mass in the center, but it prevented them from being used as intended. Best practices would be to use this process to manufacture near-net shape parts which are then machined where critical dimensions and features (precisely fitting holes, threads, etc.) are needed.

Infill

The models printed with non-solid infill turned out good. Lowering infill would likely reduce strength, so it is probably not a good option for functional parts, but it could be used to save on material costs of display pieces. Infill did not seem to influence shrinkage or part accuracy from my limited testing. Parts printed with 100% infill had a calculated density of ~94% compared to regular stainless steel.

Failure modes of post processing

Part failure during post processing can be categorized into two main ways: Major and minor. Major failure has no possibility of recovering a part, while minor failures may still be able to work depending on the part's application, likely with the need of some extra attention.

The main minor failure modes include delamination between walls and infill of a part and warping during sintering. The adhesion between perimeters and infill is a weak point of extrusion-based 3D printing, and this is highlighted by post processing this material. Parts may be able to be recovered by pressing the walls back against the part, or by machining the affected area away if it is not significant to handle loading. Warping will most commonly occur with overhangs sagging down, but other defects could happen and may require machining to fix parts back to a usable shape.

The main major failure modes observed were parts collapsing or cracking during post processing. A part collapsing indicates that the design was not suitable for the process and that design revisions need to happen for said part. A part cracking is also likely a non-recoverable defect in a part. This can be due to poor infill orientation for supporting the part in its brown state, or due to internal stress concentrations and weak spots (such as the wall-infill interface). Parts with warped bases are more prone to cracking due to cantilever-like forces on the main mass from unsupported areas, which is why sanding parts to have a flat base is strongly advised.

MakerBot Method X Notes

The MakerBot Method X is overall a good printer based on my experience with it, though it does have some quirks.

It has a lot of desired features for printing with metal filament, but it still suffered from parts warping off the print bed at least a little bit every time. MatterHackers recommended Dimafix applied on a glass print bed for optimal adhesion, which the Method X lacks. It *might* be possible to print on a pane of glass placed on top of the bed in this printer. It depends on how it does its initial Z probe, but attempting this would risk stalling motors or breaking the pane of glass, and nothing online indicates this being possible or even people trying this.

One thing to note with using this printer is that I rested the spool on a separate, 3D printed spool holder on the table next to the printer, rather than in the printer's material bays. The filament is much heavier than a normal spool and needs to rotate as smoothly as possible to ensure the extruder can pull filament in.

My main gripe with the ecosystem is the proprietary, locked-down nature of the printer and software compared to the typical hobby setting in which my experience lies, but this is ultimately a rather insignificant detail when it comes to parts getting done.

As far as the quirks go, the biggest one regards the temperature tracking of the printer. First, there is no way to see live values of the nozzle or chamber temperatures on the printer itself while it is printing. One of the screens shows the nominal nozzle temperatures, but seemingly not the live values. Temperatures can be tracked by monitoring the printer through MakerBot print, but that leads to another bit of odd behavior. When the printer is preheating the chamber, the LCD on the printer shows a value for the chamber temperature and target temperature. Both values differ from the numbers shown in MakerBot Print by 15-20ºC for the high temperatures of printing ABS or the metal filament. Both interfaces simply list their reading as the "chamber" temperature, but I suspect that the printer and slicer may be reporting from two separate temperature probes (perhaps one is in the bed?). Still, this discrepancy boggles me.

A printer with a regular heated bed instead of chamber heating (and resultant indirect bed heating) may work better to prevent warping in early layers, but an enclosure to keep some heat in would probably be required. Additionally, the printer would need a hardened steel nozzle upgrade. The lowest cost option that may work is probably the Bambu Lab X1 Carbon ($1,200), which has a capable heated bed, but the chamber temperature maxes out around 45ºC, so its ability to print would still be in question. I have included this printer as an alternative option in the *Cost Analysis* section below.

As it stands, the MakerBot Method X is a sufficiently proven option for metal FDM. Designs could be modified with extra base material for printing to account for warping and being sanded away if absolutely needed. Alternatively, work could be done on finding raft settings to help manage warping.

Slicer Notes and Findings

There are only two options for preparing prints for the MakerBot Method X, and they were ultimately similar, but I found that *MakerBot Print* desktop app was superior to the web based *MakerBot CloudPrint.* Below are notes from my experience, including details on how CloudPrint didn't meet expectations.

One thing to note between the two slicers is that MakerBot Print and CloudPrint profiles are incompatible even though both exported files are saved as a .printmode. CloudPrint gives an error importing a MakerBot Print profile, and Makerbot Print opens the CloudPrint profile but doesn't actually import any data from it, it seems. The profile in Appendix C is for MakerBot Print.

MakerBot Print (Computer App)

MakerBot Print was much more usable than CloudPrint due to issues on CloudPrint's end, but there are still limitations to be aware of.

MakerBot Print has a limited range of settings available for modification. There is enough control to get prints done, but more would have been nice for fine tuning prints. The settings that users have control over are temperature, movement speeds, extrusion multipliers, fan speeds, and physical parameters like infill, perimeters, and floors and roofs. When testing the infill of prints, I found that the infill toolpath didn't update if the pattern was set to "Linear" but changing it to "Thatch" allowed me to properly control infill. I think that issue was related to starting my printmode profile from a "solid" preset.

One of my big complaints with the slicer is that the sliced .makerbot files cannot be reopened by the slicer to review the toolpath. Additionally, compared to Gcode, they are proprietary with non-ASCII data storage, so there is no way to identify settings used for a print or make small changes. Another limitation is that the printable area available in MakerBot Print is limited to the intersection of the two extruders, even if only one is in use (as is the case with metal filament). This may be due to having modified the profile for ABS to print metal, but I couldn't find a way to disable the second extruder.

A nice feature of MakerBot Print is that it has a camera stream of connected printers, so you can monitor print progress from a desk or even from home. This remote access can be used for starting and pausing prints as well.

Ultimately, MakerBot Print is the better tool for slicing metal prints given the two options. It gets the job done well enough, though there is room for improvement.

MakerBot CloudPrint (Browser-Based)

MakerBot CloudPrint had a built-in profile for metal filament, which was the initial draw to trying it out. Selecting metal locked the slicer to using a solid infill profile, and my attempts to change infill settings were overwritten and the same toolpath always resulted. The "solid" infill from this slicer had noticeable gaps in both the preview and on the printed object, and the mass was ~20% lower than the same geometry printed from MakerBot Print, showing that it isn't particularly solid. This profile had no roof layers by default, which likely led to delamination during sintering in its test print.

One benefit of CloudPrint was that it enabled the ability to print using the full volume accessible by the primary extruder when only one extruder is going to be used for a print.

Like MakerBot Print, one of my main complaints is that the slicer cannot re-open sliced files for previewing the “gcode”.

There were more settings to adjust in this slicer, which would be useful if the infill slicing algorithm functioned properly for generating a solid toolpath and allowing control over infill. Due to these errors, **I cannot recommend using MakerBot CloudPrint.** I ended up taking some settings from CloudPrint to improve my slicing profile in MakerBot Print, but I did not attempt to use it again after the first print’s results.

I have also had reliability issues with CloudPrint. Sometimes, it would “white out” in the browser when loading models, and the page becomes unusable. Reloading the page fixes the issue while losing any work I had in progress, but sometimes it just happens again.

Future Tests and Theories

This section contains some ideas that could be explored more in the future. It includes things that I lacked either time or resources to investigate further myself.

- A functional area missed in my testing was larger sized parts. The documentation recommends keeping part sizes within a 4” cube, but bigger should be tested to properly scope out future applications. Larger parts are obviously more expensive due to needing more material, which is why most of my test parts were small.
- Testing of more functional examples and cases would help understand proper uses for metal FDM. I did one functional test which showed promise, but most of my testing was at a more basic level of understanding the material, rather than real applications to Parker.
- Testing complex parts that require supports and machining away said supports is also an important next step for real applications. These supports may need to be manually modeled, but they could possibly use the same workflow as the metal PBF printers. I don’t know enough about said workflow to say for sure.
- Layer heights could be experimented with more. My first print used 0.2mm layer height, but I then changed that to 0.15mm in all following prints, since lower layer heights should reduce internal stresses and warping. Also keep in mind that parts shrink 20% in the Z direction, which affects the final visible layer height. Resources on the filament list a suitable layer height range of 0.1mm to 0.25mm. The things to investigate here would be printability and warping namely.
- Testing the strength of green parts could be beneficial to understanding minimum feature size. It would require printing test coupons and breaking them rather than sintering them. The technical data sheet for the filament lists material strength of sintered parts, but it has no reference for green parts.
- For future part-marking cases, trying to use a stamping tool is probably a good idea. My main concern would be damage to parts given the fragility of the green state, but this option would be a more consistent and convenient option than the manual engraving I did to mark parts.
- Bed adhesion was never something I ironed out to be perfect. Printing with rafts could be explored. MakerBot Print doesn’t support brims. Another option would be more labor intensive but less invasive tweaks such as adding “rabbit ears” corners to prevent warping.

Minimal Investment Cost

Below are two options that could be followed to set up metal FDM capabilities from scratch. The first is the proven method that I used and the bare minimum expenses to achieve that. The second is a cheaper alternative which uses a different printer, the Bambu Labs X1 Carbon, as introduced above. This setup is untested and unproven, but I believe it would be capable.

Experientially Proven Metal FDM Setup (MakerBot Method X)

Item	Price	Notes
MakerBot Method X	$4,900	Marked from $6500. It's unclear if this is due to a temporary sale or permanent reduction
Labs Extruder	$350	Needed for the filament
BASF 17-4 Filament (1kg)	$130	Includes a processing ticket. 3kg options are listed below
Dimafix Adhesive	$25	Needed for the filament
Bare minimum entry cost	***$5,405***	
Optional Upgrades	**Price**	**Notes**
BASF 17-4 Filament (3kg)	$350	Need to buy 2 additional processing tickets
BASF 316L Filament (3kg)	$465	Need to buy 2 additional processing tickets
Processing Tickets	$50/kg	Different tickets for 17-4 vs 316L

Cheaper but Unproven Metal FDM Setup (Bambu Labs X1 Carbon)

Item	Price	Notes
Bambu Labs X1 Carbon	$1,200	See paragraph below about risks
Hardened Steel Nozzle	$15	Multiple nozzle sizes available. The printer ships with a HS nozzle, but the metal filament requires its own nozzle. Contamination with other filaments will ruin prints
BASF 17-4 Filament (1kg)	$130	Includes a processing ticket. 3kg options are listed below
Dimafix Adhesive	$25	Needed for the filament
Bare minimum entry cost	***$1,370***	
Optional Upgrades	**Price**	**Notes**
BASF 17-4 Filament (3kg)	$350	Need to buy 2 additional processing tickets
BASF 316L Filament (3kg)	$465	Need to buy 2 additional processing tickets
Processing Tickets	$50/kg	Different tickets for 17-4 vs 316L
Borosilicate Glass Pane	$37	Cheaper options may be available elsewhere

Notes:

The Bambu Labs X1 Carbon is in a different market segment than the MakerBot Method X, which is a large factor on the price tag difference. It is a *very* capable machine, but if things go wrong, the quality of support provided by Bambu seems to be much lower compared to "industry-grade" 3D printers. There is a large community of people using this printer, so a Reddit or forum post could likely help solve issues, but it is still a key difference to note.

The X1 Carbon lacks a filament storage bay with heating and drying capabilities. This doesn't affect metal printing, but should be noted as a key difference from the Method X. Additionally, the heated chamber can only go up to 50°C to protect the printer's components, compared to the Method X's 100°C.

It looks like people using this printer have modified it to print on glass, which is the recommended surface by MatterHackers' resources, so it *may* even be able to print without warping between that and the regular heated bed. Here's the guide to for using a glass bed: Bambu Lab X1 Clamps for Glass Bed by ZzoyozZ (Note this is a community upgrade and not *officially* supported.)

As part of this setup being untested and unproven, a custom metal filament profile would need to be setup in the software. Values can be referenced from the MakerBot profiles, but it still would require the work of someone who knows a bit about FDM 3D printing already.

Another thing worth mentioning relates to the printer's terms of service and privacy policy. It has gotten some attention from people online with their rights to track/save model data when printing through the cloud service. It seems like there is a LAN-only mode for printing, but regardless, this option may not be the best option for any restricted parts, such as ITAR work, since the models and data are potentially at risk of processing through a Chinese company.

Other Options in Metal 3D Printing

There are likely plenty of other ordinary-tier FDM 3D printer options that would be capable of printing the BASF metal filament. The key things that matter are the option to install a hardened steel nozzle, having a heated bed that can reach at least 100°C, and preferably an enclosed print volume.

There are also specialist FDM printers on the market that are geared exclusively for metal printing, which are a significant step up in cost, and likely in performance. Below are a few options:

- Raise3D has their "MetalFuse" ecosystem. Of that, there is a $10k 3D printer called the Forge 1, which is designed to only print BASF metal filaments. This ecosystem also offers debinding (the D200-E) and sintering (the S200-C) units for in-house post-processing capabilities. I couldn't find clear prices listed for these units, but I found something that suggests it to be in the realm of $100k to 200k.
- Markforged has their Metal X system which uses their own material library, but it is still a filament extrusion-based process. Their line has washing (effectively debinding) and sintering machines available. The price of this system appears to be between $150k and $200k.
- Desktop Metal has multiple smaller-scale 3D printers. Their Studio model uses "bound metal deposition" technology, which is extrusion based like the above options, but they have their own materials. These have been engineered to not require a debinding step, and this option also has a furnace for in-house sintering. From what I could find without requesting a quote, the two machines cost ~$110k. Desktop Metal also has their X-Series printer line with 3 options. These use binder jetting as opposed to extrusion, but there seems to be a wider range of material options as a result. This seems to cost ~$150k as a minimum.

Conclusion

The combination of the MakerBot Method X and BASF Ultrafuse 316L metal filament were found to be a capable solution to making high complexity metal parts at a low cost from my testing. I found success in learning how to print the material and make custom metal parts. I believe that this filament should continue to be investigated, with the next logical test being on a production part case. This would open the window for comparison to the metal PBF printers in terms of capability and would allow for learning about business cases where this would be the best option. This process should be able to be used in a similar manner to the PBF printers, where near-net shape parts are produced and then machined for precision features. The general dimensional tolerance of the process may be slightly lower due to less predictable shrinkage of parts during sintering, but a real part being tested would clarify accuracy limitations a lot more. I tapped holes and did some minor machining on parts I printed, and they seemed to hold up to these tasks just fine.

The largest advantage of this filament is that it provides stainless-steel parts at such a low cost compared to other methods of metal 3D printing. The cost of entry with the Method X is a bit under $6k, while other printers that might be suitable could bring the cost to under $2k. The way that the cost is so low is by outsourcing the post processing of the material (instead of the investment cost in expensive processing equipment), which MatterHackers has made very easy. However, this drastically increases the time required to complete parts. Post processing only runs twice per month, and between shipping both ways and processing, it takes a week and a half at minimum to get parts back. If a part is to be iterated on with this process, there really isn't enough time to make an iteration between receiving a part and the time the next parts would be shipped for the next processing batch. This means development would happen at a restricted rate of one iteration per month.

Printing this material is not without its challenges and restrictions. The biggest challenge which I faced was the tendency of parts to warp away from the bed. This means parts should have thickness added to the base so that it can be sanded flat and allow the rest of the part to be the appropriate size. The Method X's heated chamber helps with the overall printing of the filament, but an enclosed printer with a heated bed may work better to resolve these issues. Other things such as rafts could be tested to try and prevent warping, at the cost of wasted material and added labor in cleaning prints for post processing. The rest of the restrictions from this material mainly relate to constraints that need to be kept in mind when designing parts.

Overall, this process showed a lot of promise towards custom metal part applications where cost is a more important concern than the time it takes to get completed parts. If high-quantity jobs are being looked at, other metal 3D printing options exist with in-house post processing ecosystems, and it would be a good idea to analyze the costs of those options.