The SES Newsletter

Edition 09

January 2020

An Engineer's Perspective

Introduction

Welcome back from whatever festivities were a part of your holidays. Now that we're all relaxed, it's time to get back in the swing and start stressing again! Or better yet, look back fondly on the time you spent with friends and family so it can keep you sane during the dreary winter months.

There were many topics listed at the end of the last newsletter, but just to keep everyone on their toes, this newsletter isn't going to cover any of those. As part of my vacation I was correcting some of my design models and playing with my printer, which got me thinking that I should probably document some of the best practices I've developed when it comes to 3D printing. This is a departure from my usual musings that are more philosophical in nature and addresses the technical aspects of making prototype parts with what is now ubiquitous technology.

Demetri's Corner

There's not much to talk about since the last newsletter since most of the time has been devoted to traveling, napping, cooking, and eating what was cooked. I sent out holiday cards (to a few lucky recipients!) and otherwise just buttoned up business documents in preparation for the upcoming year. And of course printed a bunch of stuff, because, why not?

Today's Subject – Best Practices in 3D Printing

There are so many blogs, posts, guides, etc. covering this topic, it's hard to make an argument that yet one more is warranted. But here I am making that case. When I first got involved in 3D printing with a plywood kit held together with zip ties six years ago, I was armed with a high level of interest and whatever information the internet had to provide. I underwent a concerted effort to learn the capabilities and limitations of the printer, software, use of Fused Filament Fabrication (FFF), and the materials. I made many test prints, calibration parts, printer upgrades, and experiment with various filament materials and suppliers. Eventually I upgraded my printer to a dual head model made of actual metal (!) with an interface screen and enclosure. Throughout this process I regularly consulted the interwebs and my own engineering experience. What I found was that most information was either too basic, too specific to a problem or printer, or focused on aesthetics and approachability of 3D printing. I was looking for more of a textbook for mechanical engineers on how to leverage the technology to make functional mechanical items that could be used as final products or working prototypes. I was disappointed. Now, 3D printing is taught in many manufacturing degrees, but I'm not sure it's addressed in any meaningful way with mechanical engineers, so a gap still exists.

This is not a textbook on 3D printing, but essentially the outline. I'm sure many textbooks exist, and I don't claim to know more than those writers. If my specific approach is seen as a useful way of describing 3D printing, I could be (easily) coerced into expanding on this outline. Any expressions of interest in this regard should be sent to newsletter@sigmaexpertsolutions.com. I'm also focusing exclusively on Fused Filament Fabrication (FFF) as it's the technology where I have the most experience and most of the concepts are applicable through any layer deposition technology. Note that this is often called Fused Deposition Modeling (FDM), which was created by a company pioneering 3D printing. They are the same thing, and I will use FDM here as it's the acronym that I've heard the most and think it's the most well recognized of the two.

The Basics

I'm going to insult the intelligence of some people reading this as most of the information is self-evident if you have any experience with 3D printing. I'll try to be brief, but most of this can't go unsaid as there would be a serious gap in information if it was not provided.

Get to know your printer

All printers are different. They probably shouldn't be, but they all have a character. You need to print a lot of stuff at first that challenges different aspects of capability before you can really dig into making good prints. Taking notes is crucial at this stage, as is testing with changing of only one variable to understand the impact. Here are some specific things I've noticed can vary between printers that should be explored in the initial phases:

- **Nozzle Temperature Control** – The actual temperature vs. sensed temperature varies, as does the thermal mass and ability to maintain the temperature at high feed rates. Coupled with different types of nozzles, amount of pressure from the feed system, and other items, this concept is often overlooked, but understanding how your printer's nozzle temperature is maintained is critical in having consistent prints.
- **Random Vibrations** – Mass is moving around fast and creating a lot of direction changes. The printer will vibrate, even if it is a 70lb behemoth like mine. The idea is to understand how much, under what conditions, and how to mitigate vibrations. Mitigations includes everything from adjusting accelerations, re-enforcing the printer itself, changing components, controlling the direction of fill patterns, etc. The list is endless, which is why the initial experimentation is so crucial.
- **Printer Bed** – The variety of materials used for printer beds is beyond the scope of this discussion, but whatever you have, understand its limitations. Specifically, you need to know how flat it is, best practices on leveling, how to ensure first layer adherence (more on that later), and quirks of the bed heating system (if applicable).
- **Random Airflow and Air Temperatures** – It is often overlooked but airflow within the confines of a printer can have a large impact on printer performance. Most printers have a cooling fan, but it's location and air velocity can cause major differences in print quality. Additionally, printers without enclosures are subjected to external air currents and temperatures that can cause problems. My first printer had no enclosure and I was baffled that some of my prints would come out with bad layer adhesion, when other identical prints were fine. It took me months to pinpoint it to the HVAC intake in the ceiling causing increased airflow and prematurely cooling filament before it could properly adhere to the previous layer.

That First Layer

Not enough can be said about getting the initial line of melted plastic to properly adhere to the print bed. It is almost guaranteed that if there is a problem at this stage the print will fail. And if you're not paying attention, it could do some damage to the machine as it breaks parts off and drags them around while melting random plastic to the hot nozzle. Been there. Done that. I strongly advise against repeating this experience.

Quite a few factors play into keeping the first layer stuck to the bed, but I'll cover some basic categories. Firstly, there is the mechanical adhesion between the plastic and the bed. After that is what I'll call "vacuum adhesion" which happens when your print surface is very smooth. The flatness of the print bed is also a large factor in consistent adhesion. The last major factor is the force the first layer is subjected to during the printing process.

Mechanical adhesion is exactly what is sounds like. The printer is essentially a glorified glue gun, so all the same principles apply. Any surface roughness that gives it purchase will be used to make sure it gets "stuck". On my first printer, I printed on blue painter's tape. The surface, after wiped clean of oil, provided a great deal of mechanical adhesion. This results in a pretty low failure rate, but invariably some traces of the blue tape are left behind on the print. It's also a pretty uneven surface, so it can make the surface of the part less than desirable.

Vacuum adhesion is when you print on something very smooth and non-porous (such as glass). The melted plastic completely evacuates all air between the print and print bed, causing a strong hold between the items. This can be very consistent, but any warping of the print or unevenness of the print bed will destroy this type of adhesion. Additionally, enough surface area must be provided so that there is a strong enough vacuum to overcome printing forces (discussed later).

Print bed flatness is of concern for both types of adhesion, but mostly for vacuum adhesion. Mechanical adhesion can be achieved with enough printer head extrusion, use of a raft, and a rough enough surface. But even then, a badly out of level print bed will result in adhesion on only one area, and a non-flat surface will result in poor dimension control over the first few layers of the print. Both are to be avoided. All that said, without an almost perfectly flat bed, vacuum adhesion is not possible. I've been using glass plates for my prints for years now, and that has done wonders for print bed flatness. There are plenty of other materials out there, but this is my recommendation.

Forces on the print are the reason that we adhere the first layer to the glass. If there was no force, the first layer would just stay where it is. Minimizing these forces is another strategy in ensuring that the first layer stays where you put it. Forces come essentially from three places: nozzle drag, momentum, and warping. Nozzle drag is unavoidable. Increasing nozzle temperature will lessen this force, but almost always the things that result in nozzle drag are caused by factors more critical to printing success, so this is an area where little is to be gained. Momentum is often not recognized as a factor. The style of the printer can be a determinator on whether this is problematic. I personally selected a printer where the bed stays stationary to avoid this force altogether, but often larger scale printers have at least on axis along which the bed moves. This is only problematic for large and tall prints, so it can be mitigated if the item being printed is taken into account and acceleration is decreased as the size of the printed object increases.

www.sigmaexpertsolutions.com/home/newsletter
e-mail: newsletter@sigmaexpertsolutions.com

Warping is a favorite topic of 3D printing enthusiasts. All materials warp, but your goal, in terms of bed adhesion, is to minimize that to an acceptable degree for the duration of the print. For low shrinkage materials like PLA, this is generally not much of an issue unless a lot of material is printed as a solid block. Similarly, flexible materials will not have an issue with this as they impart little force to disrupt the adhesion layer. When prints start getting larger, and the materials are less temperature stable, this becomes an issue. The standard solution is a heated bed. That keeps the first few layers at a higher temperature so they warp less and are a little more conforming as the print grows in size. This drastically reduces the amount of force being put on the adhesion layer. A step further would be a heated enclosure that would keep the entire part at an elevated temperature. I haven't taken it quite that far, but I found that a heated bed in a closed enclosure provides much of the enclosure heating without additional equipment or controls.

Use Decent Software

Most printers come with some software that is either printer proprietary, or freeware. In most cases it's junk. The printer itself is a dumb machine with no ability to self-correct itself (usually) blindly following the instructions you give it. Those instructions come from some software that takes a solid model (or STEP file, which is technically a surface model) and slices it up to create machine code for the tool path of the print head. As we'll talk about further on, your ability to control how this code is generated will be the difference between success and frustration. You need software that gives you as much control as possible. My audience is engineers, so I'm assuming you will have a basic understanding of what you are doing, and you don't want some dumbed down printer specific profile that is great at printing a reasonable approximation of the statue of liberty, but can't make a working threaded connection to save its life. Invest in decent software. I won't endorse anything, but I'll note that I've been using a piece of $100 software for years and it's great. I've also looked at the software integrated into SolidWorks and what comes packaged with Stratys printers. I'm not sure those extremely expensive solutions are any better than my $100 solution, so more expensive is not necessarily better.

Advanced Concepts

Now that we got some basics out of the way and you've gotten to know your printer, it's time to start thinking about how we make functional mechanical parts. After thousands of prints at this point, I have a lot to say on the subject, but I'll try to limit my discussion to some of what I think is less well covered, yet critically important for mechanical design.

Design for Printing

We'd all like to think that we could make anything we fancy, then just send it to the printer. That is the promise, but our equipment is far from there. When you design a part, limitations of its manufacture should be part of the calculus. Just because we have this fancy new technology with the promise of freeing us of these constraints, we shouldn't believe the hype. Just like a milling machine, fabricator, welding robot, sheetmetal brake, and all these other tools, our printer lives in the real world. Some of the basic concepts to consider during the design phase are discussed below.

www.sigmaexpertsolutions.com/home/newsletter
e-mail: newsletter@sigmaexpertsolutions.com

- **Determine the orientation of the part relative to the print bed.** This is probably the most critical part of the whole thing because this can define a lot of how the part performs. The first thought is "how will the first layer go down" and "what is the base of this part". It is possible to just put a raft and support to the part and therefore ignore this consideration. Just be aware of where the supported surface is relative to other components during assembly as this will be the least accurate portion of the print.
- **Determine the orientation of layer lines.** The strength of the part and what portions need to be strong in what axis is a major consideration. Due to the layered construction, FDM prints do not have isotropic material properties. Even with great layer adhesion, the strength between layers will never be the same as within a layer. This is especially critical if any flexibility is needed or repeated cyclical stress is anticipated.
- **Know the limitations of the printer in terms of overhands and bridging.** Armed with your initial familiarity with the printer, you can design parts to take these items into consideration, so you don't need support material, or success of the part is not as dependent on support material.
- **Use support appropriately.** Support is great in that it lets you do overhangs and features otherwise not possible. Much like getting to know your printer, know how it does support and then use good software to select a support system that works best for the material and printer you are using. That said, avoiding support when possible is best so you can: minimizes post-processing, decrease print times, improve accuracy on supported structures, and decrease possibility of interference between the support and part. In some places, the design should absolutely not use support, such as internal voids where it can't be removed, or on small features where post-processing could cause damage. The new craze is dissolvable supports, and they are nice, but I've had mixed experience. Generally the two different materials print differently, so the temperature settings cause strange interfaces, and thermal expansion differences cause issues. I've found using good software with support material the same as the base material generally preferable. Post-processing can be a little more labor intensive, but the surfaces tend to be better and dimensional stability more consistent.
- **Break up the part into an assembly.** Sometimes it's just easier to make the part into several parts and then assemble them (with mechanical fasteners, adhesives, friction fits, or melting) during post-processing. This can ease some of the design issues above such as layer line orientation and support. It can also help mitigate the consequence of failure of one large single part on a long duration print.

Material Selection

When I started 3D printing, the options were essentially ABS and PLA. Since then there has been an explosion of options. And that's only for FDM printing. The new options range anywhere from these traditional materials to exotic flexible filaments, nylon, and fiber impregnated polymers. Don't let all of this distract you though, because the materials are still generally one of four types of polymer with various additives to make them "special". These polymers are ABS, PLA, TPU, and Nylon. I'm not giving you the scientific names because if you start saying them people will think you're weird. You're better off without the knowledge.

www.sigmaexpertsolutions.com/home/newsletter
e-mail: newsletter@sigmaexpertsolutions.com

Of these, ABS and PLA still reign supreme because of their ease of printing and generally good availability. If you need flexible parts, you will use TPU or some similar material, but that's a topic for a whole other discussion. If you need something that you can dye, has the highest temperature resistance, or is slippery, you'll go with Nylon. This is also a relatively rare need.

Ultimately, you're most likely to print with ABS or PLA. They both have strengths and weaknesses. Originally ABS was the filament of choice because the resulting print was relatively functional in terms of toughness and could be easily post-processed with several techniques. PLA was relegated to the "lesser" printers as it did not require a heated build platform, used a lower temperature extruder, and didn't create toxic fumes. But these early versions of PLA were brittle, and the mechanical properties left much to be desired. The landscape has changed, and there are formulations of PLA that retain all of these initially desirable properties (low thermal shrinkage, low melt temperature, and lack of fumes) while improving toughness, flexibility, and even resistance to moderately high temperatures. There are some cases where ABS is still king, such as for items that might see higher temperatures, or repeated flexing of a components is anticipated, but I've found that I'm using ABS less and less. The largest issue is that it shrinks while cooling and properly accounting for the shrinkage during the design and slicing is more trouble than it's worth for mechanical parts when I'm trying to hold tight tolerances.

Now that I've convinced you to primarily print with PLA, there is the consideration of the formulation. Every supplier of filament has a different mix, and even within that trade name, different colors (pigments) make the filament respond differently. I'm sure some chemist could work out the "perfect" filament for each situation, but I'm advocating the brute force approach. Remember that part of getting to know your printer? You'll be going through a lot of filament during that time period. I strongly advise you try multiple suppliers and colors and take rigorous notes. Most suppliers have sample packs that are perfect for this purpose. I have over 20 pages of notes on just filament. 90% of it is PLA! At this point I stock four colors of one specific formulation (and each color behaves differently), one color of another formulation, two colors of ABS, and some flexible filament – just in case. And each has a different profile in my software in terms of printing parameters. If I was asked to use a specific filament for a project not on this list, I would budget in some serious time determining how to print it if I wanted to be proud of the result.

Dimensional Control

This is the last subject I'm going to cover in detail, and it's an important one for mechanical components. Most decent 3D printers have stepper motors with enough discretization to meet the accuracy of mechanical components. The inability to maintain dimensional control is not due to some inherent lack of mechanical capability of the printer, but rather improving its precision and controlling the printing process.

Precision of a 3D printer is just like precision of any other CNC type equipment, but the loads on the equipment are very low, so low cost and strength components can be used to achieve repeatable results. For this reason, most reasonably robust printers can be mechanically tuned to have a fair amount of precision. Since most work with timing belts, threaded rods, and linear bearings, the knowledge and effort required is straightforward and involves ensuring that these running components are in good order and replacing them if they start developing “slop”. Again, due to the low loading, it will generally take thousands of working hours on a decent printer before this becomes an issue, but printers that incorporate plastic parts or low-cost bearings will be subject to wear that will decrease precision.

Control over the printing process is where dimensional control is truly obtained. There are so many variables associated with squeezing a thin line of plastic out of a nozzle in a precise location, and we can't possibly cover them all, but I'll cover some big items:

- **X and Y axis Compensation** – This is an option on any slicing software worth having. It allows you to fine-tune the machine code when extruding the outer perimeter relative to the calculated nominal position determined by the slicing software. Without fail, the squished plastic line will extend beyond the artificial boundary defined by the nominal surface of the modeled part. This results in undersized holes and oversized external dimensions. You may choose to adjust this so that all portions of the part are inside the design envelope, or that the design envelope is right where the “ripple” of the layer lines is averaged out. That is a decision based on how you design parts and the type of post-processing to be used.
- **Z axis Steps** – Selecting the proper step height is very important. The most obvious aspect is that the smaller the steps, the smaller the amplitude of “waviness” caused by layer lines. That seems like a good reason to crank it down to the minimum, but that comes with a very large penalty in terms of printer speed. Additionally, this will cause extremely slow movement of the extruder motor and addition of more heat to the part as it passes over the part more times during a print. On the other extreme, layer heights greater than the nozzle width are dangerous as it requires the printer to essentially “print on air”. Some value between very thin and bigger than the diameter of the nozzle is appropriate. You'll probably want at least a couple of layer heights you normally use; one for when you want very fine layer lines and the other for pretty much everything else. Remember that this will affect the amount of “squish” and therefore your x and y compensation. Another interesting concept is that most z axes are controlled with a course acme thread rod. This means it takes very little stepper motor rotation to affect the needed height adjustment. It is best to select layer height increments that align with the natural step size of the stepper motor. Otherwise you force the software to make some compensations which will show up as a regular pattern in the layer lines. This mostly doesn't affect dimensional control as much as aesthetics and is covered in detail in the interwebs.
- **Extrusion Multiplier** – This may be called something different in your software, but it's essentially how much plastic you extrude based on the nominal. The slicing software assumes that your filament is a perfect cylinder of a diameter you define. It also assumes that the filament is fed at a speed equal to the rate of rotation of the drive cog multiplied by the diameter of the drive cog. If your filament is not perfect (none is) there will be some issue with the total volume of extruded material. If your the linear speed of the filament is somehow affected, there will also be some issue with the total volume of extruded material. As you can see, there is no way that the software can actually know how much plastic is making it through the extruder. The extrusion multiplier is something you will have to experimentally determine, and it is different for different filaments. The diameter of the

filament is the most consequential aspect, so regular inspection of filament diameter is critical in ensuring that the multiplier remains valid. Similarly, linear speed is impacted by how much the drive cog crushes the filament (changing the effective diameter of the cog). Also, the presence of debris in the cog teeth can change the effective diameter and impact linear feed rate. Keeping the drive cog clean should be part of your regular maintenance routine.

- **Print Speed** – The faster you print, the more you stretch the melted filament. If your extruder can keep up with pushing a lot of filament, this may be a non-issue, and up to some speed threshold, this is a non-issue for all printers. That threshold is variable based on the power of the extruder heating element, the clamping force and power of the extruder drive motor, and the melting characteristics of the filament. In general, slowing down the printer vastly improves dimensional control with little sacrifice of print time. 3D printing is already a lot slower than many expect, so why not wait a little longer and get a better product? That beer's not going to drink itself.

Final Thoughts

I've covered a lot of territory in just a few pages, and it's more pages than I intend for these newsletters. I've barely scratched the surface, but I think I've covered some of the big items that will help a moderately informed enthusiast improve their ability to print mechanical items. Some areas that I would have discussed with more space is infill and shell choices, printing for tension vs. compression, snap assemblies, threaded connections, and post-processing. If after reading this, there is an appetite for me writing more on one or more of these subjects, I'll gladly comply if I get requests.

The introduction of 3D printers provides a great opportunity for mechanical designers to explore different design approaches and perform rapid iterative prototyping. The potential headiness associated with being able to print "anything" needs to be tempered by real world realization of the limitations. If we approach this as just one more tool in our toolbox, we won't be tempted to use this hammer to drive a screw.

Your Dose of Aphorisms

I haven't had many opportunities to use aphorisms related to 3D printing, so I'm going to have to get creative. If there is one thing that I want as a theme, it's that a 3D printer is a very powerful tool, but it is just one more tool available and not a panacea. It's valuable to practice using the tool and knowing it's potential and limitations. And as fun as it is to print a Yoda head, it is more satisfying (to me) to print something functional that solves an actual problem (yes, I'm saying that "not enough Yoda heads in the world" is a non-problem). We should search for ways to make the most out of 3D printing and expand beyond downloading premade models of skulls and trinkets. So get out there and create!

A hammer with no nail to hit is just a hammer shaped paperweight.

P.S. You can download some of my useless trinkets on www.thingiverse.com. Search "GrMechEng" (which is my profile name) and behold some truly useless stuff.

The Future

Any comments or suggestions on the discussion in this newsletter or for future newsletters will be welcomed at newsletter@sigmaexpertsolutions.com. The following list of topics is being considered for the future, and any strong opinions on any of the below or additions can also be expressed to the same address.

- Engineering Ethics
- The Role of Automation – Additional Subjects
- How to Run Effective Meetings
- Software development
- The Role of Rapid Prototyping
- Commercial Grade Dedication
- Developing Engineering Products