The WILDCAT

Special Edition Modeling Reference

3D Printing for the Scale Modeler:
With the AnyCubic Photon DLP Printer

By: Michael D. Roof

AMPS #1632

3D Printing for the Scale Modeler:

With the AnyCubic Photon DLP Printer

Mike Roof

AMPS 1632

How did I get here?

Let me say right up front that I'm no 3D printer expert. I'm still a novice at best, so rather than try to lay out some definitive, step-by-step guide for my fellow modelers, maybe a more narrative description of how I got to where I am and what I'm doing might help you find your own path down the 3D printing road. Here is my 3D printing story…

A couple of months ago, I surfed into a discussion on Track-Link about a new 3D printer that seemed to have remarkable resolution at an affordable price. There was a good deal of skepticism among many of the posters – The thing just seemed too good to be true. However, after following the link and watching a video review, I was intrigued. Well, that vid led to some more online research, checking out the manufacturer's website, looking up vendors that had the printer for sale, and finally finding and joining a FaceBook group for owners of the printer. In the end, I was convinced that, yes, the AnyCubic Photon 3D DLP printer was the real deal.

Unfortunately for me, I had no experience with any sort of software needed to design 3D objects. So, while the hardware seemed within reach, I just couldn't justify the expense. There're simply not enough fine scale 3D model printable files available for download on the internet. Without the skills, knowledge and ability to design my own, I just couldn't see that I would get enough utility from owning my own 3D printer. I soon realized that what I really needed was to get educated on 3D design first. I thought if I could do that, then maybe buying a printer wouldn't be totally crazy.

After spending hours researching the bewildering array of available 3D modeling software, I was about to give up. All of the most highly rated programs seemed exceptionally complicated and expensive. It didn't make any sense to me to spend weeks and untold hours learning some professional 3D modeling software when I could be spending that time at the workbench building models. That is, after all, the object of the exercise – to build models, not to learn to be a professional graphics designer. 3D printing needed to supplement my model building and not turn into a new career!

That's when I found out about TinkerCad. TinkerCad is a free, very basic 3D design software for beginners and students that was created by the folks at AutoDesk. TinkerCad was just what I was looking for, something basic, simple and generally intuitive. It has a very nicely structured self-paced tutorial that walks the new user through the main features, and there are a large number of YouTube how-to videos on different techniques to get even more from the program. This is largely because TinkerCad is used by many educators, teachers and schools for various introductory CAD instruction. It's also used by a large number of hobbyists and enthusiasts because it is so basic and simple. In short, TinkerCad was the perfect fit for me.

TinkerCad works a lot like a 3D version of PowerPoint's drawing tools, and I'm very familiar with those being an old "Power Ranger" for years. Tinkercad offers the user a number of basic shapes from boxes and cylinders to domes, pyramids, wedges, etc. There are also a number of other shapes and designs that offer more complex options. Almost all of these basic and complex shapes can be used as either solid or negative space and added or subtracted from each other to build ever more complex designs. The shapes can also be modified by changing their X, Y and Z axis dimensions or rotated about those same axis.

This general design methodology is very similar to traditional scratch building with styrene plastic. Complex structures are understood by the scratch builder to be assemblages of many varied, basic shapes. The modeler scratch builds those complex structures by combining basic shapes and tooling them to modify their dimensions or to add relief, cavities, etc. Subassemblies are created and combined into ever more complex assemblies until the desired object has been created. TinkerCad does the same thing, just with bits and bites and pixels instead of styrene sheet, rod, tubes and other structural shapes. The TinkerCad design methodology will be very familiar and intuitive to anyone with experience with traditional scratch building. The final product of the design effort is "exported" by TinkerCad as an .stl file and saved on your computer.

Now, TinkerCad does have a couple of drawbacks. One of the more annoying things is that it's cumbersome to determine the dimensions between objects / shapes. It can be done, it's just "clunky." Another thing is that the "workplane" is somewhat limited in size, and the user's view is always towards the center. This can make it difficult to view an object from all directions, especially around its edges. Finally, and most significant, TinkerCad has a limit on the file size that it can handle (300K triangles to be exact). This can limit designing complicated and detailed objects. The more parts you combine to make your design, the larger the file grows. The work-around to this is build your object as a series of separate TinkerCad files, bringing the separate sub-components together in another TinkerCad file as .stl's which have exported and saved to your computer. Exporting a TinkerCad design as a .stl "flattens" that design and reduces its file size. Some organization with your own computer files and folders is needed to keep all the various parts of the final design sorted, but it can be done.

One last thing… It's not necessary nor desirable to make multiple copies of an object in TinkerCad to be saved as a large .stl. If you want to print, say, 10 copies of your design, save it by itself as a single .stl, and when you prepare the print file, you can copy it in that file to make as many as you want for that print job. This also applies to models that you create as a "set" of several different objects. Save each discrete part as a separate .stl file and later import them one at a time into the print file to print the entire set in one go.

It took me about two weeks of watching videos and practicing with TinkerCad before I was creating designs that were (to me) acceptably detailed. But, once I was able to create 3D designs of a number of small models that I needed to produce, I was confident that I could make use of a 3D printer, if I owned one. Amazon Prime, here I come… (AnyCubic is an Amazon vendor, so even though it's a Chinese company, I got my printer 2 days after I placed my order.)

As I was waiting for my printer to be delivered, I began to research just how the actual printing process was carried out. At first, this also seemed bewildering. The issue is that there are many, many different combinations of programs that can be used to set up your .stl files for actual printing. There are so many, in fact, that I won't even try to mention any except the one that I use. Suffice it to say, that if you get this far with your own 3D printing journey, you'll discover what's available in the context of your own needs (mostly what works with your printer). As for me, right now, I don't need to try to design organic objects like figures and animals. Basic mechanical designs are all I need, and the path from TinkerCad to my printer is a short one. Maybe later I'll branch out. Just be aware that there are 3D design programs that combine the features of design with preparing those designs for actual printing. You can certainly streamline my process if you have the desire to do so.

As mentioned in the definitions, once you have created an object and saved its .stl file, you will need to prepare that file for use by your printer. Again, after some research, I determined that most AnyCubic Photon owners either use the proprietary Photon Slicer that comes with the printer, or they use the ChiTuBox free slicer software. The inter-web is your friend, and after finding and watching a very detailed, six-part YouTube video on ChiTuBox, I decided that its additional capability was worth the effort to learn it.

BTW, the ChiTuBox software is designed by the same company that designed the AnyCubic Photon slicer. However, that's not all. That same company also makes most of the firmware and mother boards used in the majority of DLP printers, to include the AnyCubic line. What this means is that ChiTuBox works very well with the Photon, so well, in fact, that the AnyCubic Photon is included as one of the default printer settings in ChiTuBox.

As another aside, the six-part YouTube video on ChiTuBox was produced by a gentleman who was provided with a beta test version of the software by ChiTuBox, and he uses the AnyCubic Photon in the videos. I simply setup my copy of the software using the resin and support settings he provides in the videos. These seemed to work perfectly with the AnyCubic clear green resin.

The process is quite simple. Open up your .stl file in ChiTuBox. Study and orient it for printing. Check the way it will print using the ChiTuBox tools and adjust, as needed. Add the supports using the auto support feature. Modify these, as needed. Finally, export the .photon file to your computer and save it. ChiTuBox will automatically use the proper file extension (.photon) if you've set it up correctly for the AnyCubic Photon printer.

Once you have the print file saved, you copy and paste it to a USB stick. I recommend that you buy a new USB 2 thumb drive just for this. The thumb drive that comes with the printer from the factory is notoriously unreliable. You can either copy all the files off the factory thumb drive and paste them into your new drive, or not… Unless you're going to use the AnyCubic Photon Slicer, there are actually no files on the factory thumb drive that you routinely need. You can simply copy and paste the .photon file from your computer to the USB 2 thumb drive (or even export it directly from ChiTuBox to the thumb drive).

Insert the USB drive with the .photon file into your printer, turn it on, and follow the steps in the manual to start printing. It really is as simple as that.

Important points to keep in mind with the AnyCubic Photon.

Follow the set up instructions precisely – take no short cuts! There are a number of YouTube videos (to include a very good one by AnyCubic) on how to set up and level the printer. Level the printer. If you're not sure you've done it correctly, do it again until you are sure.

One leveling tip that I didn't find out until I had watched a bunch of vids is to just loosen the grub screw on the print bed so that there is still some resistance to movement. If you back the grub screw out too far, the ball joint will have so much vertical play in it that when you tighten the screw to lock in the level, the screw will move the ball joint and throw the level out of whack. You'll understand this when you do it. Just remember – loosen the grub screw only enough so that the bed can be moved with some drag or resistance, then level it.

There are two generally acceptable methods for leveling the print bed – the "sheet of paper" method and the "Flint Reed" method. The factory setup video shows the sheet of paper method, and that's what I use. As they say, "your mileage may vary…." Don't just check the side to side level. Ensure that the bed is level side to side and front to rear. You may have to check and slightly adjust it several times to get it perfectly flat before you tighten down on the grub screw to lock it in. Leveling is not hard, but it must be done correctly.

I recommend using some white lithium grease on the Z axis frame and threaded rod before you use the printer for the first time. The frame is an aluminum extrusion, and the threaded rod controls the Z axis movement. Run the print bed down all the way using the "home" button. Apply a little grease on the rod and frame and use a small brush to spread it. Run the bed up some and apply a little more grease. Keep doing this to spread a thin layer of grease all the way up the threaded rod and the frame. Don't use too much, and don't use any sort of oil – just grease. Z axis movement is the main wear issue with the AnyCubic Photon, so start with a well lubed machine to mitigate this. (BTW, I used Shooter's Choice red, high-impact grease on my machine which has a convenient syringe type applicator.)

Before you use the printer for the first time, check the screws around the bottom of the resin vat or reservoir. Check the screws to see how tight the tightest one is from the factory and adjust all the others to about that same, finger tight, torque. Just swag it. The object is to make sure that none of the screws is so loose that resin will leak through the bottom of the vat. When the FEP (the clear screen on the bottom of the vat) is correctly tensioned, it will sound like a snare drum when you tap it lightly with your finger nail.   You want even tension across the FEP, but don't crank down on the screws. Just make them all about the same as the snuggest one from the factory. (AnyCubic provides a pretty comprehensive tool kit with the Photon, so one of the included hex drivers will fit the screws around the FEP.)

Anytime you experience a failed print, drain and filter all of the resin out of the vat. Clean the vat and ensure that there is nothing sticking to the clear FEP sheet on the bottom of the vat. Be gentle with the FEP when you do this. If you don't do this, then the next time you print your print bed will likely crush down on a bit of hardened resin left over from the failed print. This can mar the FEP creating a blemish that will interfere with the UV images from the underlying LCD screen, or worse, poke a hole in the FEP allowing resin to leak out onto the LCD and into the printer's guts. You can reuse the strained resin or keep it stored in the original bottle. (AnyCubic provides a small baggie of paper strainer funnels for this purpose. You can find strainers in the paint sections of most hardware and home improvement stores.)

3D DLP printing is a potentially messy operation. The photosensitive resin will not wash away with soap and water and it will not dry out. Clean up is only possible using a cellulose solvent like isopropyl alcohol (IPA). Environmentally speaking, all paper towels and other waste contaminated with the resin should be exposed to the sun to allow the resin to cure before that waste material is disposed of. Let your conscience be your guide. The critical thing is to always wear disposable neoprene gloves when handling the resin or newly printed objects that have not been cleaned. Once cleaned in an IPA bath, the newly printed objects are safe to handle with your bare hands. IPA is easily found in any drug store. Buy and use the 91% type rather than the 70% "rubbing alcohol" type. "Rubbing alcohol" has been diluted with water and has lanolin added to mitigate harsh drying of the user's skin. Neither water nor lanolin are good for your 3D prints or for cleaning up the resin. Other types of alcohols, such as denatured alcohol, can be substituted for IPA if you find they're less expensive or easier to source.

The "best practice" post printing cleanup is to use two IPA baths, one to get the majority of the left over resin off the object, and the other as a final rinse. Use compressed air (or just blow vigorously) to remove the rinse IPA off the now clean object. Note that the surface of the object will still be slightly soft, so some amount of time in the sun or under an UV light is needed for final curing. I've found that a couple of hours under one of my day-light balanced OTT lights also works for this.

Once the object has cured, removing the supports and cleaning up the attachment points is pretty much just standard, basic modeling skills, analogous to removing parts from molding sprues.

So, that's it. Research your object, design it, prepare the file for printing, print it, and then clean the printed object up. It really is just that simple. Welcome to modeling building in the 21st century!

<u>Definitions.</u>

Fused deposition modeling (FDM). The most common "hobbyist" type of printer. It's the type that uses coils of "weed whacker" string as a printing material. Good for structural types of objects that will actually be put into use. Resolution is generally not fine enough for the detail that most modelers want, at the very best it's somewhere in the .1 mm range.

Stereolithography(SLA). The most expensive type of consumer printer. It is capable of finer resolution and detail than the DLP printer. However, costs are generally about double. Like the DLP, it uses a photosensitive resin as the building material, but the SLA uses laser light to expose and cure the resin.

Digital Light Processing(DLP). This is the type of printer that we'll most closely look at. These are the lowest priced printers that can yield the resolution (~ 5 microns or .05 mm) that most scale modelers will find acceptable. The DLP also uses photosensitive resins as a build material. However, unlike the SLA, the DLP uses a less expensive UV light emitting LCD screen to expose and cure each entire layer of the object.

Selective Laser Sintering (SLS). The type of printer used by many "print on demand" companies, like Shapeways. These are generally considered "industrial" machines although there are "desk top" models available. The big advantages of SLS printers is that the prints are "self supporting" and multiple, inter-locked moving parts can be printed together. These industrial machines also have excellent resolution and the parts are very precise and stable dimensionally. Too bad they cost about as much as an expensive car or truck!

Selective laser melting (SLM). Another industrial type of 3D printer which can produce objects actually made from various metals. These types of 3D printers are almost exclusively "industrial" although some are used in research and engineering labs.

Laminated object manufacturing (LOM). An industrial 3D printing process that uses rolls of material to provide the layers in the printed object. Z axis resolution is fixed by the thickness of the material on the rolls.

Digital Beam Melting (EBM). A more advanced variation on the SLM type of process. It can also produce objects in metals with very high tolerances and strength. Generally used in aerospace and the automotive industries.

*Three most common consumer 3D printer types.

**For a more detailed explanation of how these different printers function, see the following:

https://3dinsider.com/3d-printer-types/

X, Y, and Z axis – "left-right;" "forward-rearward;" "up-down." When designing a 3D object or when manipulating that design to prepare it for printing, the software used will use X, Y and Z axis to define the operations. The 3D printers also move their various printing mechanisms in these same three axis (with the exception of the LOM type).

.stl – "stereolithography" files. This is the file type that you create for a 3D design. It is also the type of file that you need to do 3D printing from. These types of files are created using a CAD (computer aided design) software. All of your 3D designs should have .stl as the file extension when you save them. (Although some 3D design programs will create "proprietary" file extensions, these are generally accepted as .stl by everything else.)

.svg – "scalable vector graphics" files. These are the 2D equivalent of the .stl file. When you're creating a 3D model design, it may be useful to import one or more .svg files into your work to speed up the design of complicated profiles. TinkerCad will accept .svg files as imports.

Slicing – the process of taking a finished 3D .stl design and preparing it for printing. All 3D printers will require that the 3D design is first translated into a number (possibly many thousands) of individual cross-sections, or "slices", that can then be layered, one on top the other, to create the actual printed object. There are many different programs that can perform this function. However, in all cases, the printer ready file must be compatible with the particular printer you are going to use. Thus, you must match your printer to the "slicer" software. Most printers will come with some sort of slicer program. It's important to note that you may be able to source and use some other type of slicer program which may be easier to use and / or offer more useable features than the factory "slicer."

Supports – All SLA and DLP printers create physical objects that are suspended in vats of photosensitive resins. Each layer of the object must be printed against a firm surface to give the exposed and cured resin something against which to form the next layer. As the object builds up, subsequent layers attach to each other. However, the irregular nature of most shapes, with cut outs and overhangs, means that there will often be "islands" that are formed in many layers that are not initially attached to any other part of the forming structure until later in the print. These islands (and other areas) need supplemental supports in order to be printed. Thus, as part of preparing any .stl file for printing, not only must it be "sliced," but normally, it will also need some number of supports added to it. Adding supports and the ease with which they can modified is an important feature of most "slicer" programs (and some 3D design programs).

Hollowing – large objects with thick cross sections benefit if they are "hollowed out" leaving only thinner walls (usually only a couple of mm thick). This can greatly reduce the amount of resin that is used. Hollowing also reduces the physical areas of the cross section to reduce the amount of force needed to pull each printed layer off of the build surface, which in turn assists the machine in achieving accurate z-axis movement (up-down). Finally, hollowing out large objects helps to ensure that the resin can cure thoroughly throughout the entire structure. Generally speaking, objects thicker than about 5-6 mm should be hollowed leaving walls in the 1.5 – 3 mm thick range.

Drain holes (aka vent holes) – If the object is hollowed, then it is usually necessary to add one or more drain holes in it to allow resin trapped inside to drain out and to also allow air to enter to eliminate any vacuum that might be created by resin held in the interior space.

TinkerCad – This is a common, free, 3D design software that is very basic to use. It is considered an entry level design program and is used by many educators for entry level classes into 3D design. This is the program that I use for my own 3D designs. It is a little "clunky," but it can produce some very detailed designs if you're willing to spend the time to do so. There are many, better programs with loads more features. However, the learning curve for TinkerCad is very shallow, and almost any modeler should be able to master it quickly to produce usable designs.

ChiTuBox – This is the "slicer" program that I use. It is designed by the same company that created the slicing software that was included with my DLP printer, but it has many more features and is, IMO, easier to use. It is capable of creating sliced and supported designs that are compatible with my printer. This may not be the case for all other DLP printers, so if you own a different model printer, you may need to use a different slicer.

AnyCubic Photon – This is the model DLP printer that I own. It is made by AnyCubic. It is sold "ready to go" with everything needed to start printing immediately (except isopropyl alcohol for post-printing clean up). The Photon will soon be superseded by a newer model, the Photon X (final designation TBA). I would expect the newer model to be just as capable and user friendly as the current Photon.

Resources.

AnyCubic 3D Printers - http://www.anycubic3d.com/index.html

TinkerCad - https://www.tinkercad.com/ Note that you must register an online account with TinkerCad / AutoDesk to use this program. It is not "downloadable." TinkerCad is used by logging on to your account. Once you log on the first time, do yourself a favor and complete all of the tutorials. You'll be glad you did.

ChiTuBox - https://www.chitubox.com/home.html As with TinkerCad, you must first create an online account with ChiTuBox before you can download the software.

3D Builder - https://www.microsoft.com/en-us/p/3d-builder/9wzdncrfj3t6?activetab=pivot:overviewtab 3D Builder is a MS / Windows software that's free to download and use. It does require Windows 10 or Windows 8.1. I use this program to hollow objects and add vent / drain holes. It has a built in feature that will identify .obj and .stl files that are "broken" with a one-button "repair" command. As with almost all of the software, there are numerous YouTube vids on how to use 3D Builder.

B9 Creator Software - https://www.b9c.com/support/documentation/b9creator-software/download-b9creator-software This software was originally designed to work with the B9 Creations printers and scanners. However, objects modified using it can be exported in the common .stl format and the processed for printing using something else. I have found that it is very easy to use for adding manual printing supports. Adding supports manually allows you position them to avoid details and make it easier to clean up your prints. B9 Creator Software is free to download and use.

Thingiverse - https://www.thingiverse.com/ Thingiverse is an online community of 3D file sharers. It has a large library of files that can be downloaded for free. Note that you can import a .stl from Thingiverse into TinkerCad to be modified and or combined with your own designs. You must also create an online account with Thingiverse and log on to download any files.

3D Content Central - https://www.3dcontentcentral.com/ This is another online 3D file sharing community. 3D Content Central, however, does feature a lot of designs that require a nominal fee to be paid before they can be downloaded. The main advantage here is that the designs available are generally more sophisticated and detailed. This is a very good place to find table-top gaming figures, among other designs. Lots of very nice stuff here, just expect to pay a couple dollars to download a file (although there are a number of free designs, too).

GrabCad - https://grabcad.com/ Yet another online community of 3D designers and file sharers. Here the designs are more "technical" and oriented towards professionals, engineers, lab techs, etc. Still, there are an incredible number of designs available that can be used, modified or combined with other designs.

MakerSpaces - https://www.makerspaces.com/ MakerSpaces are collaborative locations in many cities and towns where members can come and use tools and materials for their own projects. It's an interesting concept, and it offers the potential for a modeler to learn and use tools, like 3D printers, without actually having to buy them for his or her own workshop. If you want to explore 3D printing before you pull the plug, you might consider finding and buying a membership in the nearest MakerSpace. I've thought about doing this for several FDM printer projects that I might want to do. A single, 30-day membership would be much cheaper than buying my own FDM printer just to use it a couple of times.

Online Convert dot Com - https://image.online-convert.com/convert-to-svg One thing that you can do in TinkerCad is import a .svg file and use it in your design. TinkerCad will take the 2D drawing and add "thickness" to it. You can slice and dice these just as you would any other shape file. In order to take a regular drawing and do this, you must first convert it to a .svg, though. Online-convert.com is one of many free sites that will do this. A useful tool to have in your TinkerCad design box.

FaceBook AnyCubic Photon DLP Printer Owners Group -
https://www.facebook.com/groups/AnycubicPhoton/ Almost every 3D printer will have its own "fan club" group on FaceBook. I highly recommend that you find and join the FB group that is made up of owners of your printer model. I have learned a ton from the members of the Photon owners group. Probably any problem or issue that you have as a new owner will have a solution that's already been figured out by someone. Don't go it alone.

Excellent Tip Sheet for the Photon - https://github.com/Photonsters/anycubic-photon-docs/blob/master/FAQ.md?fbclid=IwAR14VT52_OgzYgnXGjUvtsbEtnIQ9_tjDQk2vQT62AWwzbWixNTjkxKQBe8#whats-this-leveling-that-i-keep-hearing-about

<u>YouTube videos and channels.</u>

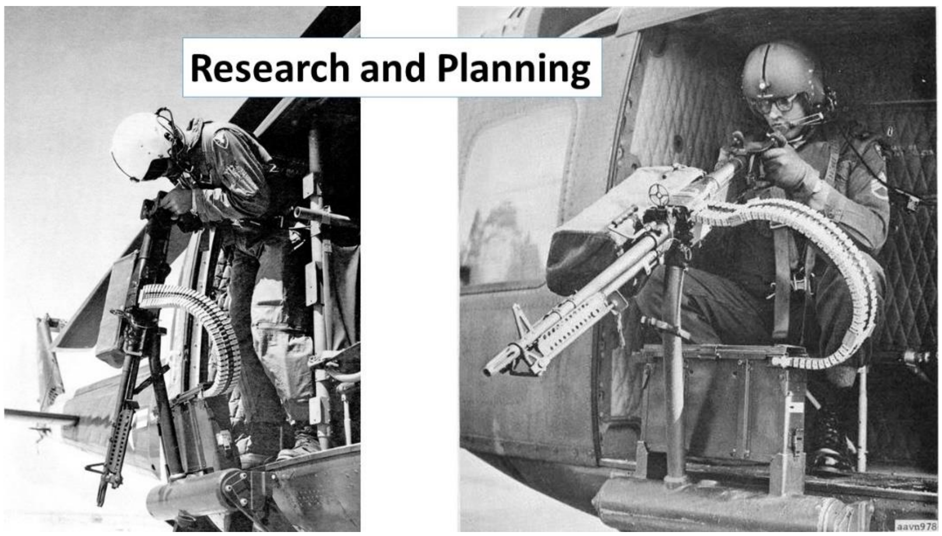
VFX Forge ChiTuBox Tutorials - https://www.youtube.com/user/noobdles/videos?disable_polymer=1 This guy presents a great, six part video on ChiTuBox. I cannot recommend it enough. He takes the viewer from download to printing with the AnyCubic Photon using ChiTuBox as the slicer and support program. (You might need a set of headphones to hear the presenter.)

Chaos Core Tech TinkerCad Tutorials -
https://www.youtube.com/results?search_query=chaos+core+tech+tinkercad This is my go-to

TinkerCad guru. Once you have completed TinkerCad's self-paced tutorials, Chaos Core Tech will show you a number of tricks and tips for getting the most out of what TinkerCad can do. The guy is a little goofy, but don't let that fool you. His "how-to's" are short, sweet and to the point. You won't be sorry you watched them.

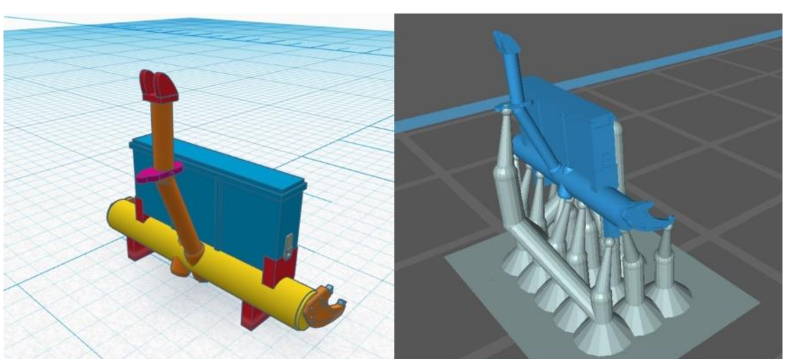
AnyCubic Photon First-Time User Video - https://www.youtube.com/watch?v=bIMwjmVl3Ho This is the factory video on unboxing, setup and using the AnyCubic Photon DLP printer. Watch it.

<u>So, what does it all look like?</u>



The initial step, as with any scratch-building project is to do the research. Here the M23 Armament Subsystem (i.e. the M60D MG mount for a Huey door gunner) is the subject. Numerous photos were collected and some basic dimension data was determined.

The MG mount was designed in TinkerCad using the length of the horizontal tube and an estimation of the width of the ammo box as the only two "known" dimensions. All other dimensions were estimates based on the proportions of the components in comparison to each other.



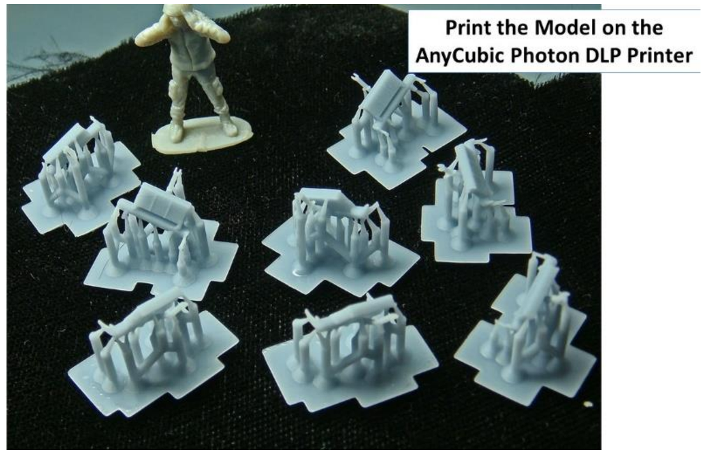
Design and Dimension the Model Using TinkerCad

Add Supports and Slice Using ChiTuBox

After the design was resized to 1/72 scale, it was exported as an .stl.

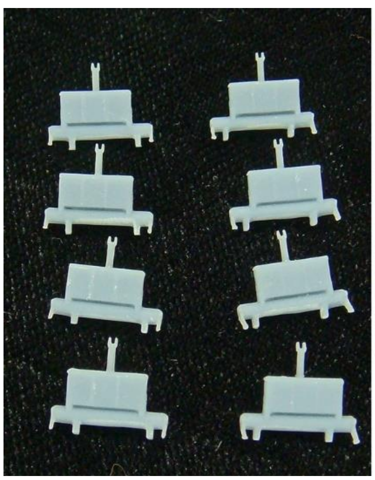
The .stl was then opened in ChiTuBox and prepared for printing by adding supports and slicing. Once it was ready, it was then exported as a .photon file.

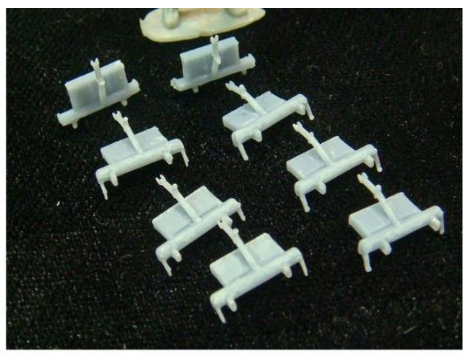
The .photon file was copied to a USB 2 thumb drive which was then inserted into the AnyCubic Photon DLP printer. The mounts were printed using Elegoo Standard Gray photosensitive resin. The print cycle took about 1-1/2 hours.



The prints were removed from the build plate / bed and excess resin was cleaned off using IPA baths. They were then put under an OTT light for about 3 hours to cure.

Once cured, the printing supports were removed using fine cutters, an X-acto knife and a Flexi-file. They're now ready to be added to the Huey "Slicks" on a Vietnam War diorama.





M23 Armaments Subsystem in 1/72 with the printing supports removed. Ready for use!

From concept to ready to use parts took about 2-3 days of discretionary hobby time, most of which was spent doing the research and designing the parts in TinkerCad. The research part of the process would have been the same for traditional scratch-building, and my own estimate is that I spent about as much time doing the design as it would have taken to build one left and right set of the mounts. The 3D design software also allowed me to "mirror" the design so that I was able to create the second mount literally in an instant.

Another time advantage was being able to replicate the design to produce the needed four pairs all at the same time. Printing also ensured that all of the mounts were perfectly identical.

Here are some more examples of models designed using TinkerCad and then printed with the Photon.

Other examples of models designed using TinkerCad and printed on the Photon.