3D PRINTING

CAD — Rohawks 3419 — Lili

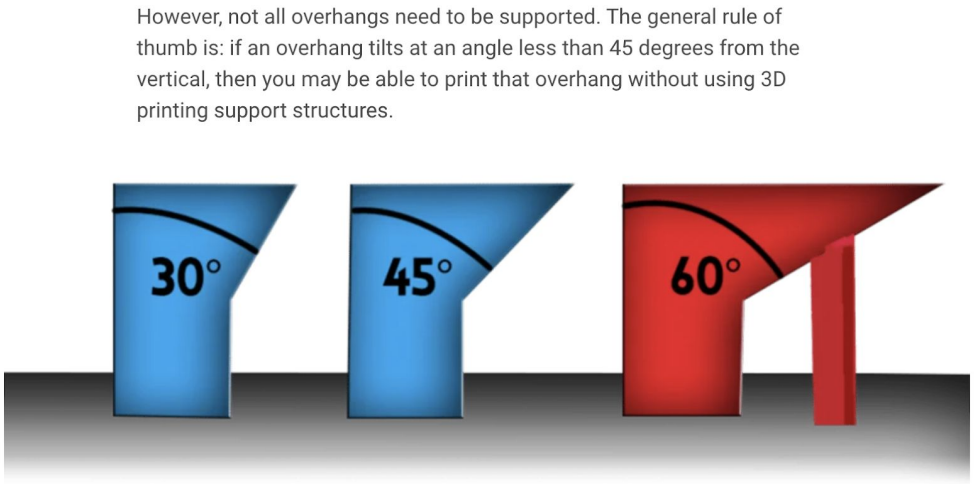
3D printing is the last step in the process of going from idea → CAD → mechanism. If you're involved in CAD at the workshop, I can almost guarantee that you will also be 3D printing things!

CAD FOR 3D PRINTING:

3D printers print in layers, starting at the bottom of a part and stacking them on top of each other. Since 3D printers build from the bottom up, creating shapes like vertical holes or other complex geometries that hang over the layers below them can be a bit tricky—a layer can't be printed on top of *nothing*.

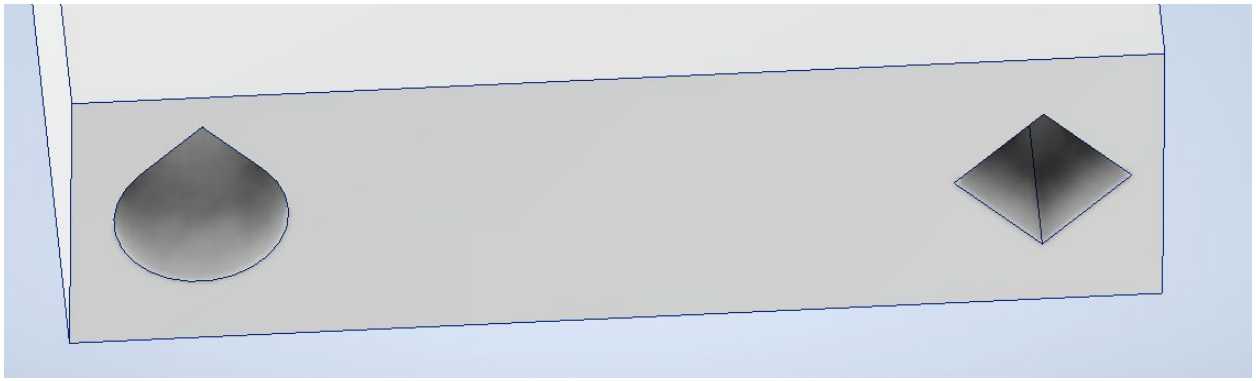
Supports are used by 3D printers to solve this problem. Supports are temporary parts of a print that are used to create surfaces that can be built on top of, if your original design requires printing a layer on top of nothing. They're made in such a way that they can be removed from the part afterwards. A lot of the time, using supports is a necessary evil, but sometimes, we really want to avoid it: it adds time to the build, wastes more material, and can be hard to remove from small openings without damaging the part.

Our 3D printer can handle 45° of overhang. What this means is that, if you think of North as 0 degrees, no layer can extend outwards at more than a 45° angle from the previous layer without needing supports. This makes printing shapes like circles and squares a bit tricky, if we want to avoid using supports.



Nice image from this website

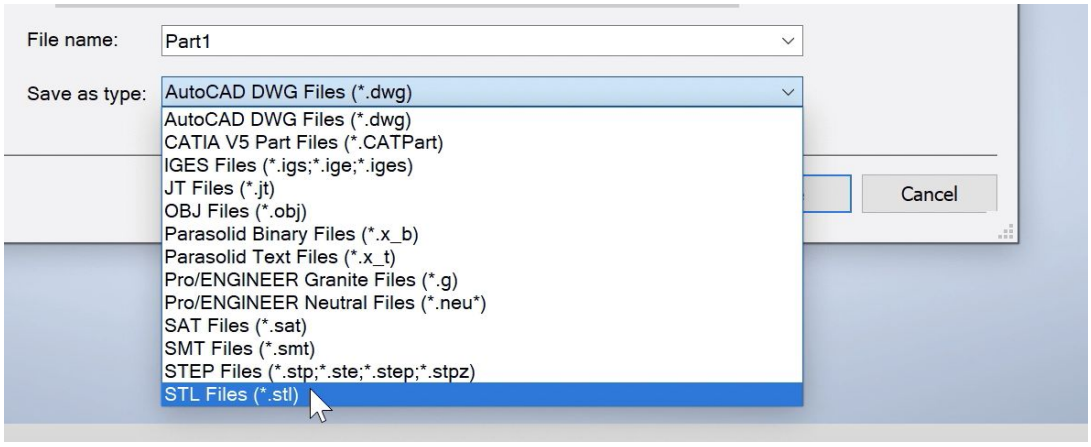
One way that we get around this is by being crafty with our CADs. Instead of normal holes for bolts or square nuts, we can make **teardrop holes** or **diamond shaped square nut holes**. These stay within the 45° overhang limit and save you from needing supports!



Teardrop hole Diamond square nut

IMPORTING YOUR DESIGN:

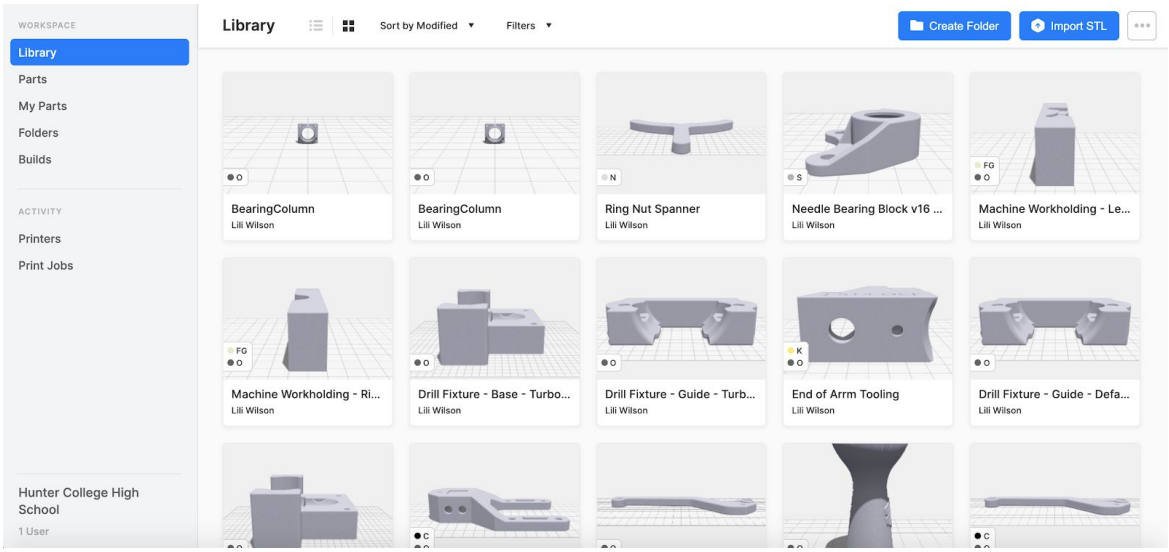
In order to 3D print a part, you first need to have a part CADed that you want to print. After perfecting your design, go to File → hover over Export → Export as **CAD Format**. A window will pop up asking you to save, make sure you save the part as an **STL File.**



3D PRINTING:

Now it's time to actually print! Our team uses Markforged printers, and the site they use to connect to them is called eiger.io. You don't need to make an account right now unless you're curious about checking it out for yourself.

Once the website opens, you'll be taken to the page shown below. Once there, click "Import STL" in the upper right corner and load in your new .STL file.



Once you've done that, your part will open in a new window. Click and drag around the screen to pan around the part. Click any face of the part to flip the orientation of the part on the plane (which represents the printing bed). Usually, you try to orient a part to minimize supports, since printing supports takes time!

In terms of settings, all you'll usually need to change at the workshop is the layer height -- we use 0.2 mm. Other than that, you're good to go! If you click print, you can download a .MFP file (stands for MarkForged Printer :)) and give that to your printer via USB (alternatively, sometimes you can wirelessly print depending on how your printer is set up!)