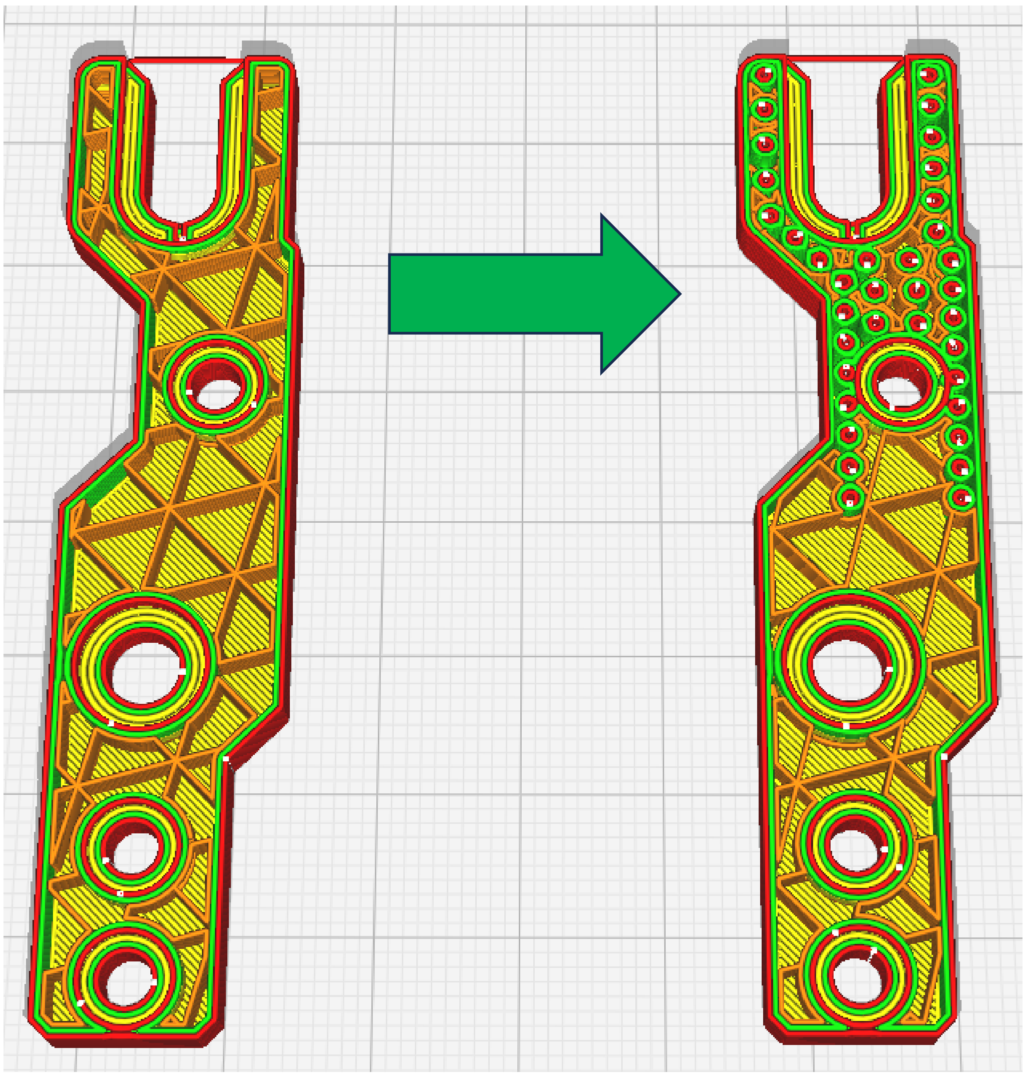
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

Who Cares About Infill ?



Today's Lesson is Sponsored by Raytheon

Contents

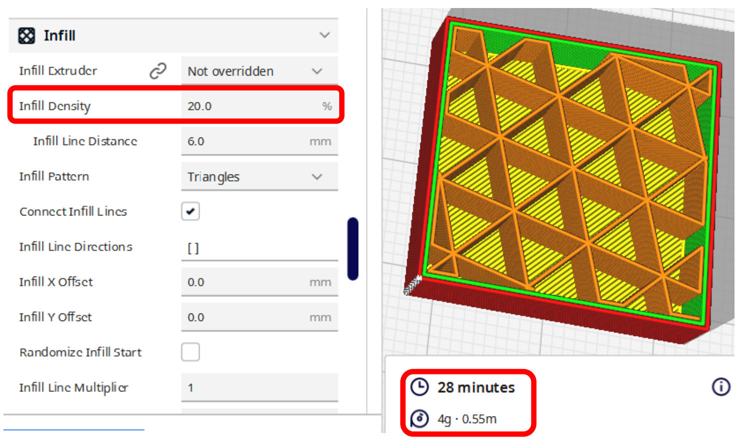
Infill Settings

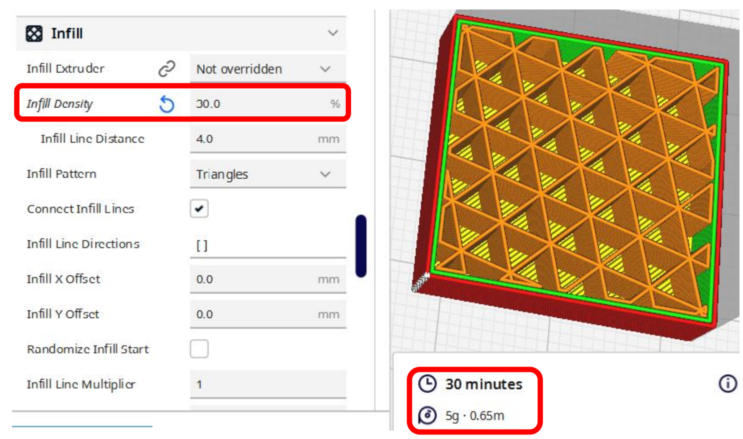
Instead of printing a 100% solid plastic part, 3D printers will “fill” the interior with a pattern of ribs. This both saves printing time and materials (filament) cost. It will also produce lighter parts, however, the weight of the parts is typically not a concern. All surfaces are printed “solid” and thus when looking at a finished part, it wouldn’t be evident that the interior is not solid. The surfaces are also printed with multiple layers and thus the part is often strong enough without a solid interior.

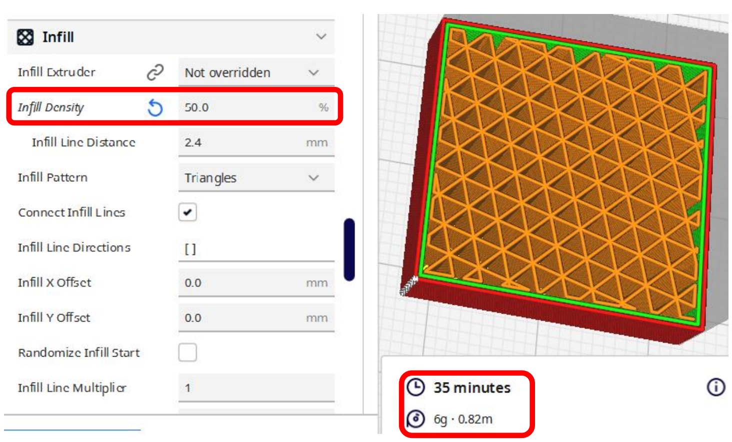
All slicer programs, such as Ultimaker Cura, present the user with many settings for infill. The Infill Density, e.g. 20%, is the most important infill setting, but there can be many more.

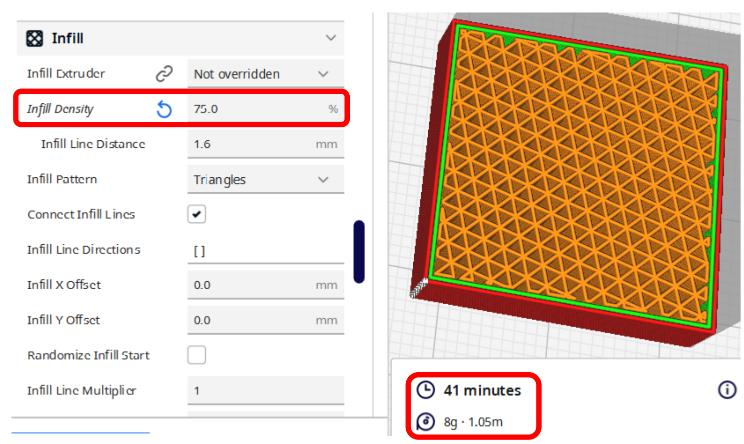
It is common for a Slicer program to have a low default infill density and for Ultimaker Cura, it is 20%. For many prints a low infill density works well and some users may never even change it.

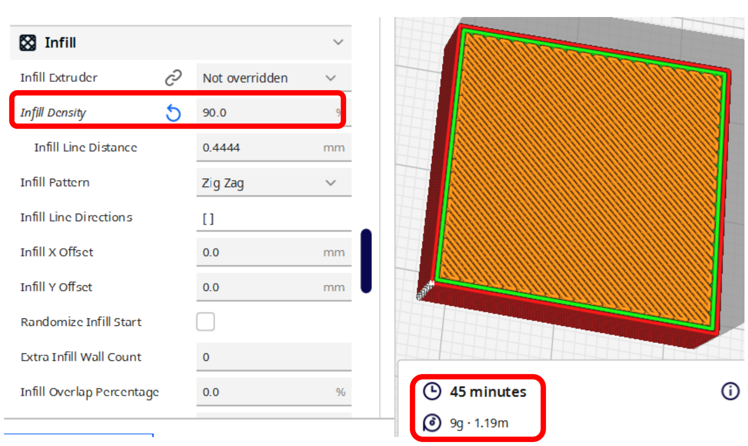
Below are screenshots from Cura using the Preview mode of 1” x 1” x 0.5” high blocks with layer slider about half way up. Starting from the top-left with the following Infill Densities: 20%, 30%, 50%, 75%, 90%, and 100%.

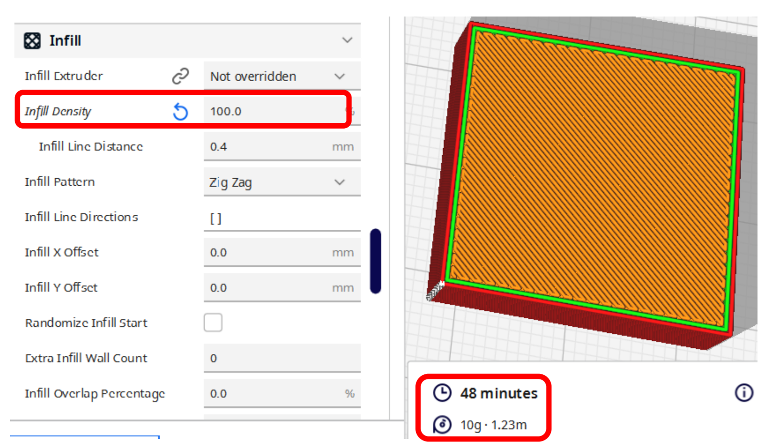












Design For Manufacturing (DFM)

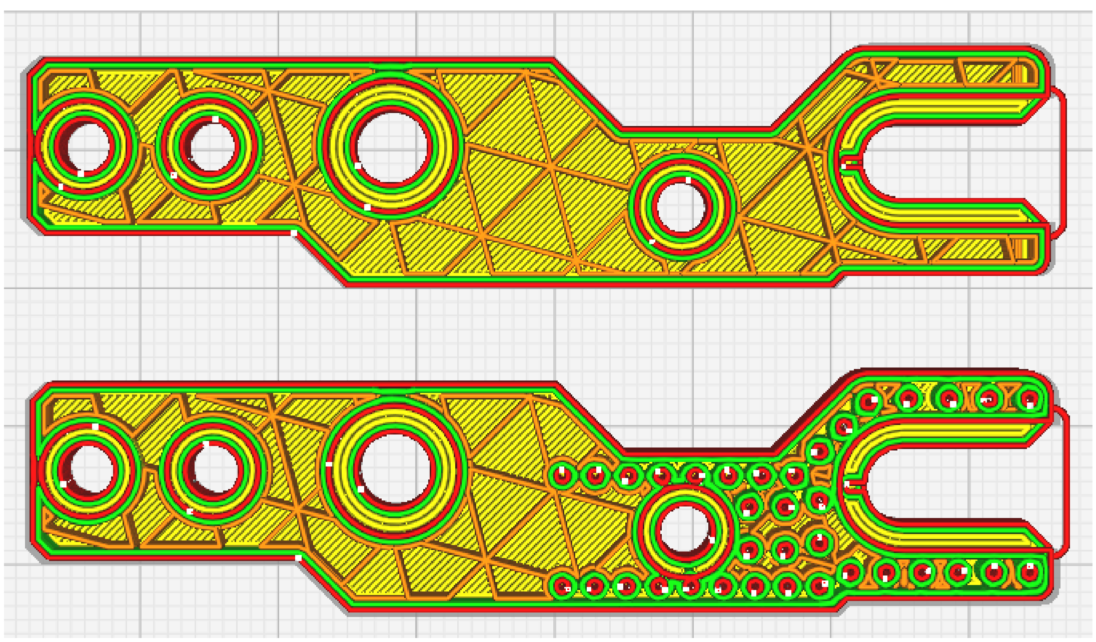
This is an engineering term that relates to optimizing a design for a manufacturing process. This also includes being mindful of potential errors in a process. For example, one may be designing a component that needs added strength due to stresses on the part. For a 3D printed part one could specify that it should be printed with a high infill density, perhaps even 100%.

Now imagine that it gets printed and one forgets to set the infill density. This situation would be mitigated if the part was designed so the infill density does not matter. The build plate of the printer may also be shared with other parts that don't need the high infill density. One can use different settings for parts printed at one time, but this requires more setup time and is another opportunity for an error.

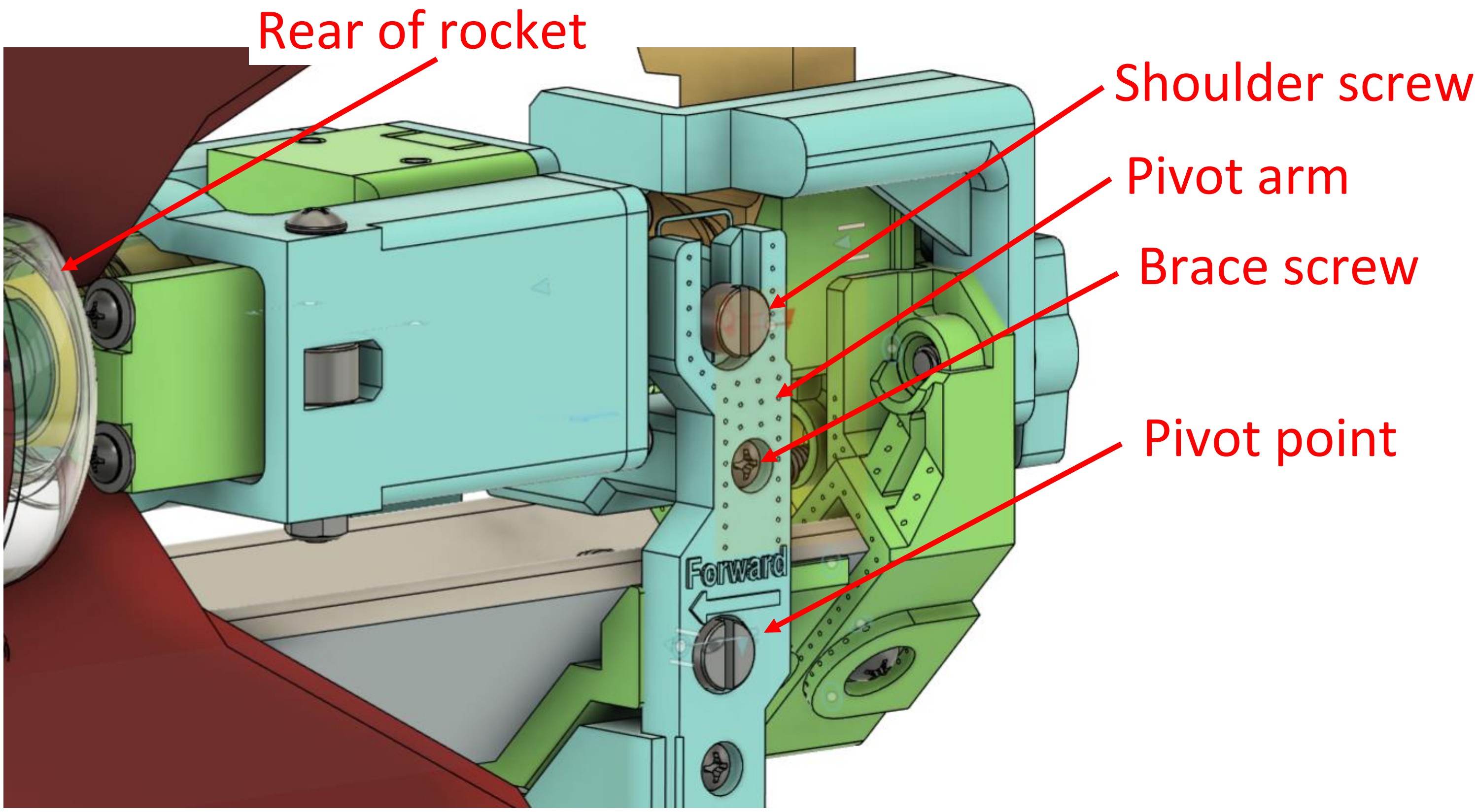
One can see how the print time and mass of plastic increases with infill density. It is possible that a part that needs extra strength, does not need this extra strength throughout the entire part. Here it can be beneficial to design the part to achieve a greater infill density only where needed.

One method to achieve the above involves added small holes to the design. The slicer adds extra plastic for any walls and the circumference of each hole will be a wall.

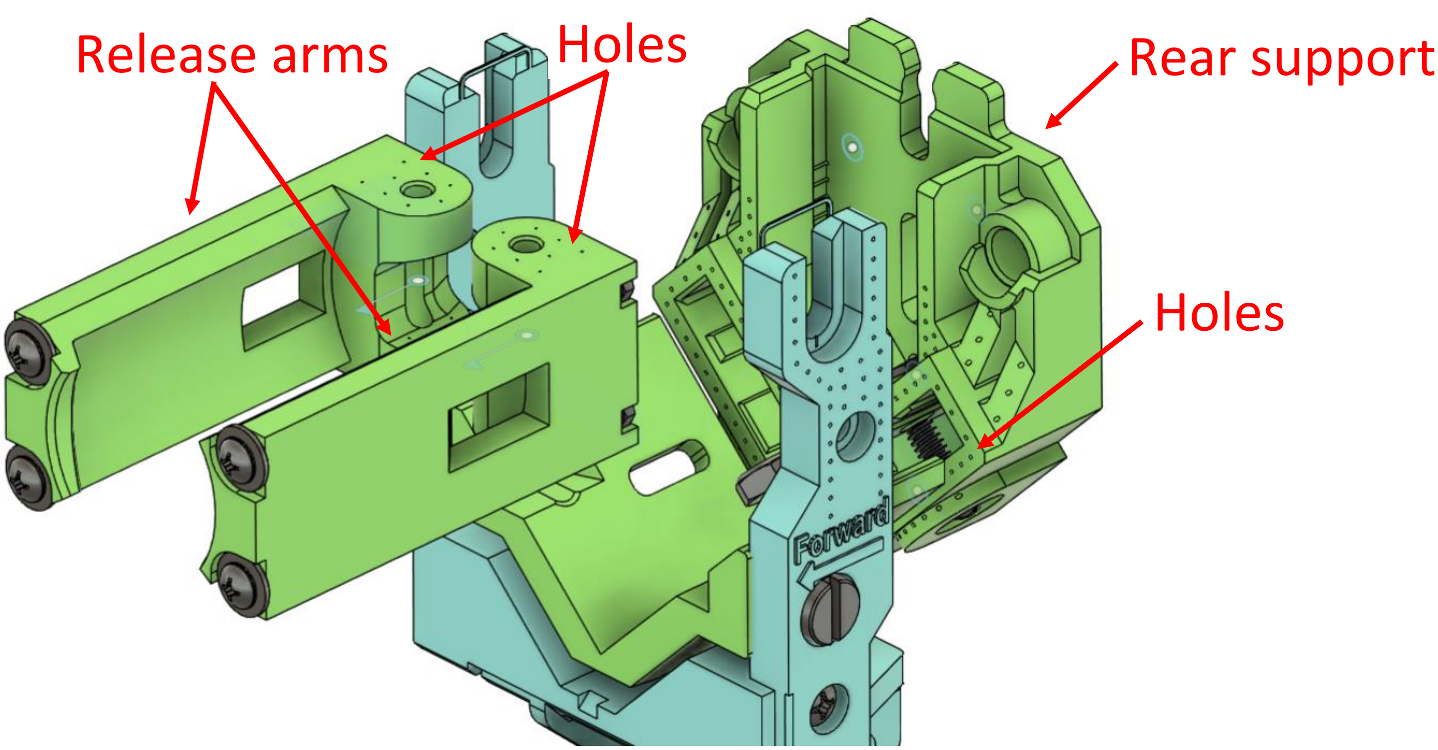
Below is a pivot arm of a rocket launcher shown in the Cura Preview mold at a layer about half way through. This is after slicing using the default infill density of 20%. The bottom part is the same as the top, except for holes that were added in thin sections that are most likely to break. One can see that the wall material added around the holes result in a high infill density. The holes are very small so the plastic missing from the center of each hole is negligible. The left end of this part doesn't require the extra strength and thus there is also a material and time saving due to its lower infill density. This part also has the advantage that it can be sliced for printing using any infill density and it will have the strength where needed.



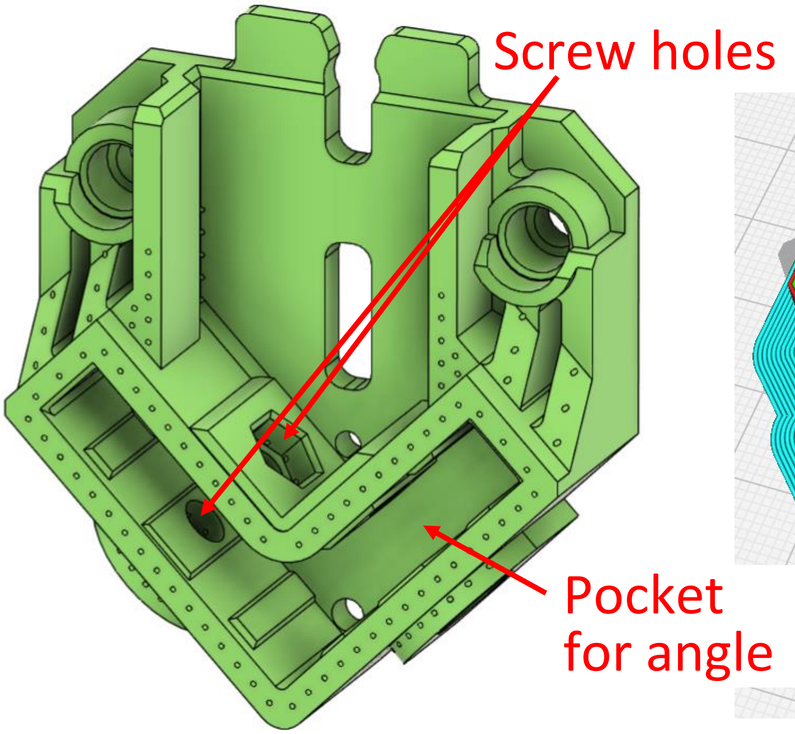
This is a view of the release mechanism of a rocket launcher showing the pivot arm. One can see the small holes around the slot for the shoulder screw and brace screw. Due to required clearances, the pivot arm needs that shape.

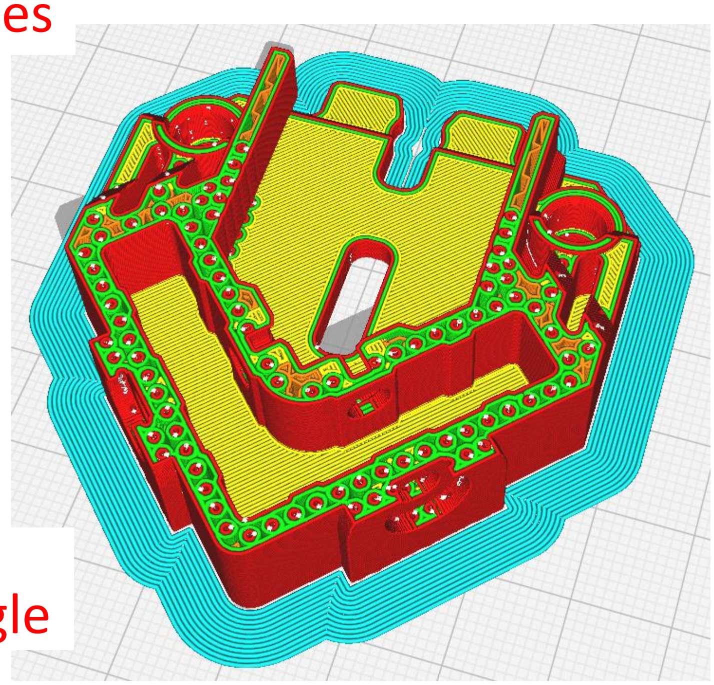


Here is a view with some parts removed. Note the small holes in the **release arms** and in the **rear support**.



Here is the rear support shown in Fusion and Cura (slice through center of part). Most of the holes surround the pocket, which slides on to the angle that supports the rocket and other components. Forces of the top of the support during launch will impart a moment (torque) that will stress the walls around the angle. There will also be shear forces at the holes of the mounting screws.





Here is a release arm with holes visible around the pivot shaft hole.

This part employs another strengthening method where long 4-40 machine screws are inserted through the part. They strengthen the part in two ways. A tensile force parallel to the screws will exist and the part will be most vulnerable at the window. The screws will resist this tensile force. At the end with the screw heads, a force perpendicular to the screw axes, will cause the part to flex. The screws also increase the rigidity of the part. The right picture is a partially exploded view showing the screws with washers added at the screw head and nuts at the far ends.