The Purple Pneumatics Kit

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The 2022 VRC Season, Tipping Point, introduced the ability to use pneumatics while still being able to use 8 motors. This caused an exponential increase in teams that are using these components. Most teams in Florida are attaching the different pneumatic components with zip ties. After experiencing the difficulties of attaching pneumatic parts to the robot myself, and asking fellow Florida teams (70857B STEMpunk and 6105C Blackout) what they might need, I decided to make a kit. The kit consists of different parts which allow you to snap your pneumatic components into a part that can then be screwed on to a robot.

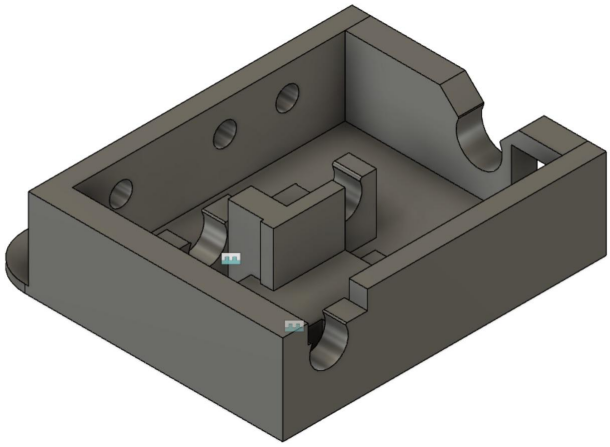
I created a piece that holds both the solenoid and the pressure regulator and then a piece that holds just the solenoid and just the pressure regulator and a piece to hold the reservoir. With these pieces teams would have confidence in the attachment of the pneumatic components to their robot. Our robot only uses one solenoid and one pressure regulator so I 3D designed a part that would allow you to place both the solenoid and pressure regulator together and attach them in the same place. After doing this and printing it, I realized that this was functional for my robot but it would be more valuable for other teams to have these parts available to be attached separately. Other robots might have more than one solenoid. I cut the box in CAD and made it into two pieces. The reservoir piece was designed in CAD but not printed. I have a prototype of the reservoir plate made out of lexan on my robot.

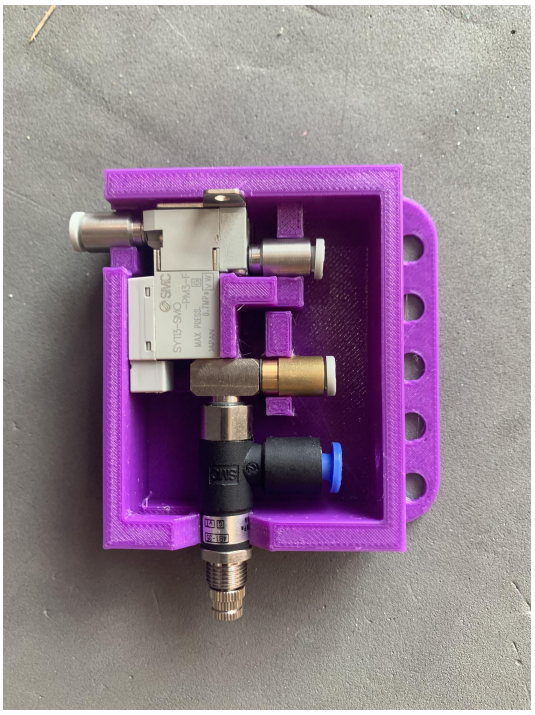
I used Fusion 360, version 2.0.11894 to design these pieces. These are the steps I used when building v.2 of the solenoid part. First I imported the solenoid CAD file, and started a sketch on the XY plane. I used the line tool to create a basic outline around the solenoid about 1mm away Then I used the offset tool to create another outline 2mm bigger. Finishing that sketch I extruded the section between the two outlines at a height of 20mm. I imported the tube fittings that screw into the solenoid and used the join tool to attach them to the solenoid in their respective positions. Next I started a sketch on the sections of the solenoid outline where the fittings stick through. I created a circle that was 8.25mm in diameter which is just a little bit bigger than the fittings in order to take tolerance into consideration. Next I created a rectangle and used the tangent sketch constraints to line up on the equator of the circle. I made another rectangle which I dimensioned to 7.75mm wide. I connected the bottom line to the center of the circle using sketch constraint and made it tall enough to stick over the wall. Next I extruded and cut out the 8.25mm circle and rectangle, only leaving the small section created by the 7.75 rectangle to create a clip to hold the circular fitting. I used the move and copy tool on the cut feature to recreate the same cut for the other fitting. On the solenoid there is a place where a 3 pin wire is connected and I made a sketch using the rectangle tool and cut a hole where the wire needs to go through. Next I created a sketch on the bottom face of the part and drew a rectangle which stuck out 9mm from the piece. I made a circle 6mm in diameter and used the rectangular pattern tool to create a line of circles 12.7mm apart. (The same as VEX aluminum C-Channel spacing) I made sure the circles were centered and spaced far enough away from the wall for a screw to fit. I repeated that process on the side of the piece to create another place to screw the part onto a robot. To finish off the part I added two strips on the bottom to support the solenoid so that it wasn’t being held only by the fittings.

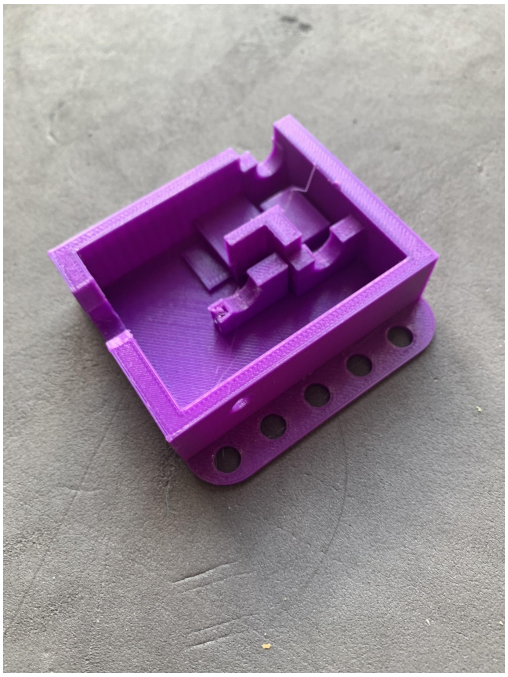
I have used Fusion360 for 6 years now and I always seem to learn something new when making parts. This time I learned that when I use a CAD file of an existing component, in this case the solenoid and pressure regulator, and I want to build a design around it and base a sketch on one of the planes of that component my final design will only download as a .f3z file and not a .f3d file. I only had time to redo one piece (v.2 of the solenoid) and make it a .f3d file. Knowing this, and knowing the format it needs to be presented in, the next time I would still import the CAD file but not base any sketches off of it, like I did for the v.2 solenoid part. I am including the STL files for all the parts in the pneumatic kit even though they were made in Fusion360.

In closing, I would consider this my prototype and if I were to improve this pneumatic kit, I would make all the pieces a little less bulky and add additional holes so they could be attached in more than one direction like I did on the solenoid part.

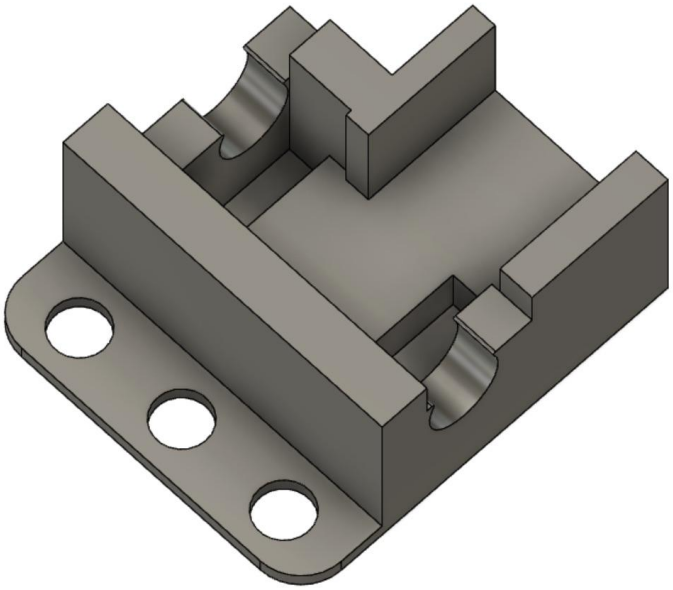
Picture of Solenoid and Pressure Regulator Holder in CAD and 3D Printed.

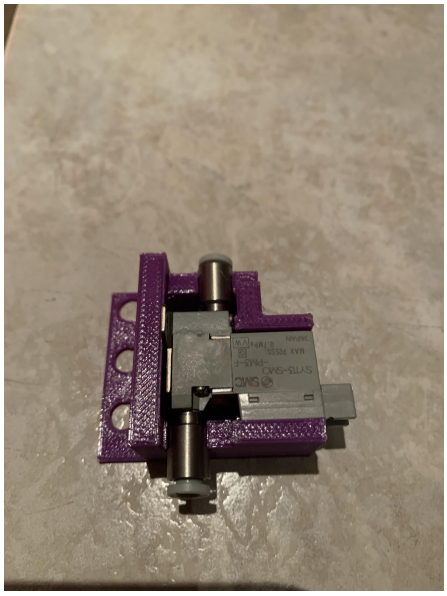


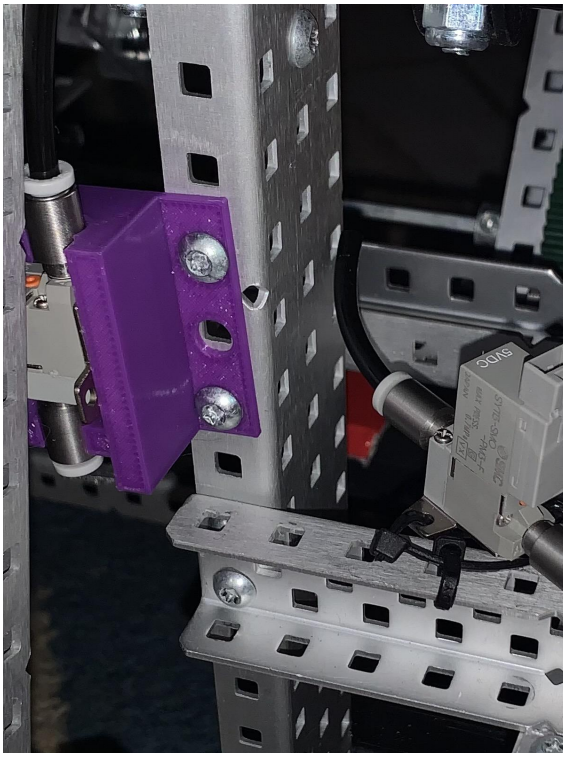




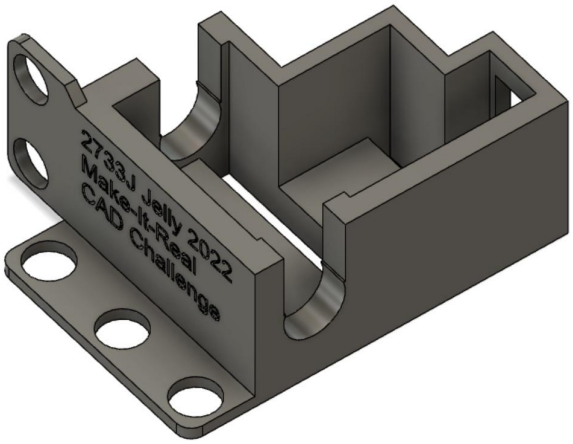
Picture of Solenoid Holder v.1 in CAD, 3D Printed, and a problem and solution showing the piece attached to the robot both with zip ties and in the 3D printed case.



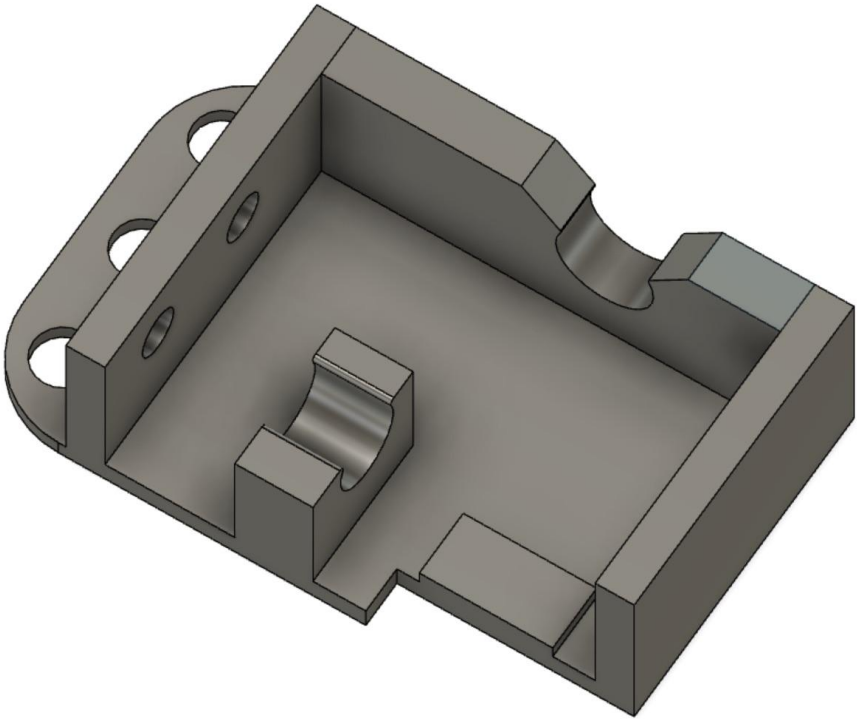


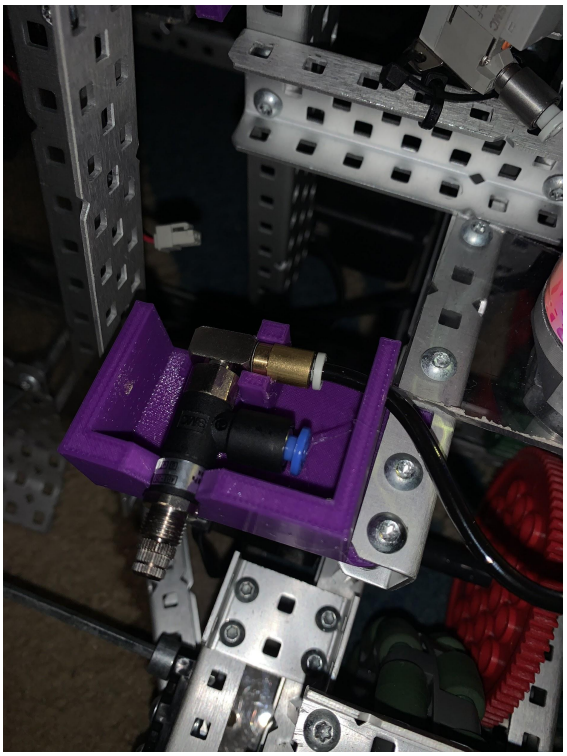


Solenoid part v.2 in CAD



Picture of Pressure Regulator Holder in CAD, 3D printed, and attached to our robot.





Picture of Reservoir Holder in cad and the prototype piece made out of lexan which we currently use on our robot.