3D Modeling

The first task in this report was to use SolidWorks to create a 3D model of the Arbor Press P1 (#98471) part seen below in Fig. 1.

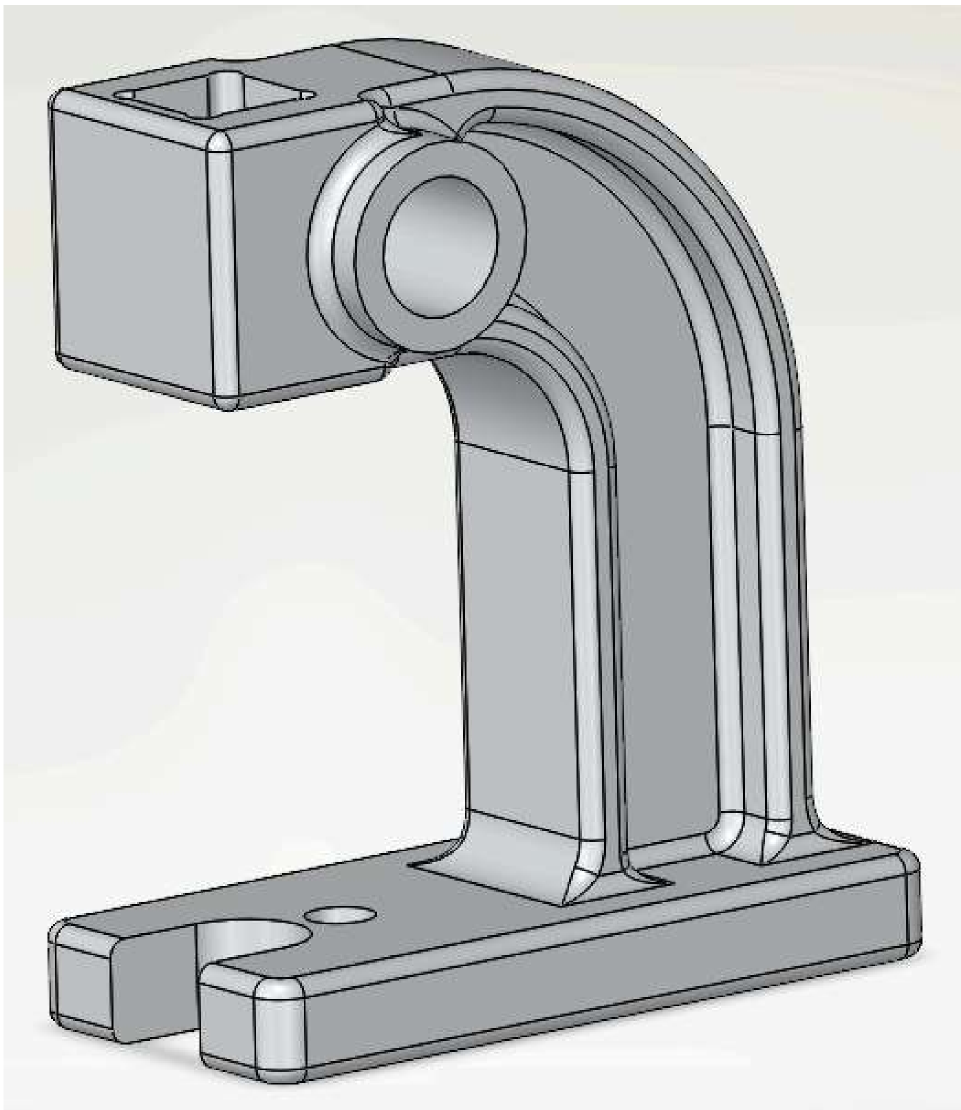

Fig. 1: 3D SolidWorks Model of P1 (#98471)

P1 was redesigned into three more variations: P2, P3, and P4. In P2, the material was changed from Aluminum 6061 to Cast Iron. Because the part was made of a material that required mold manufacturing, 3 degree draft angles were added as seen in Fig. 2 below.

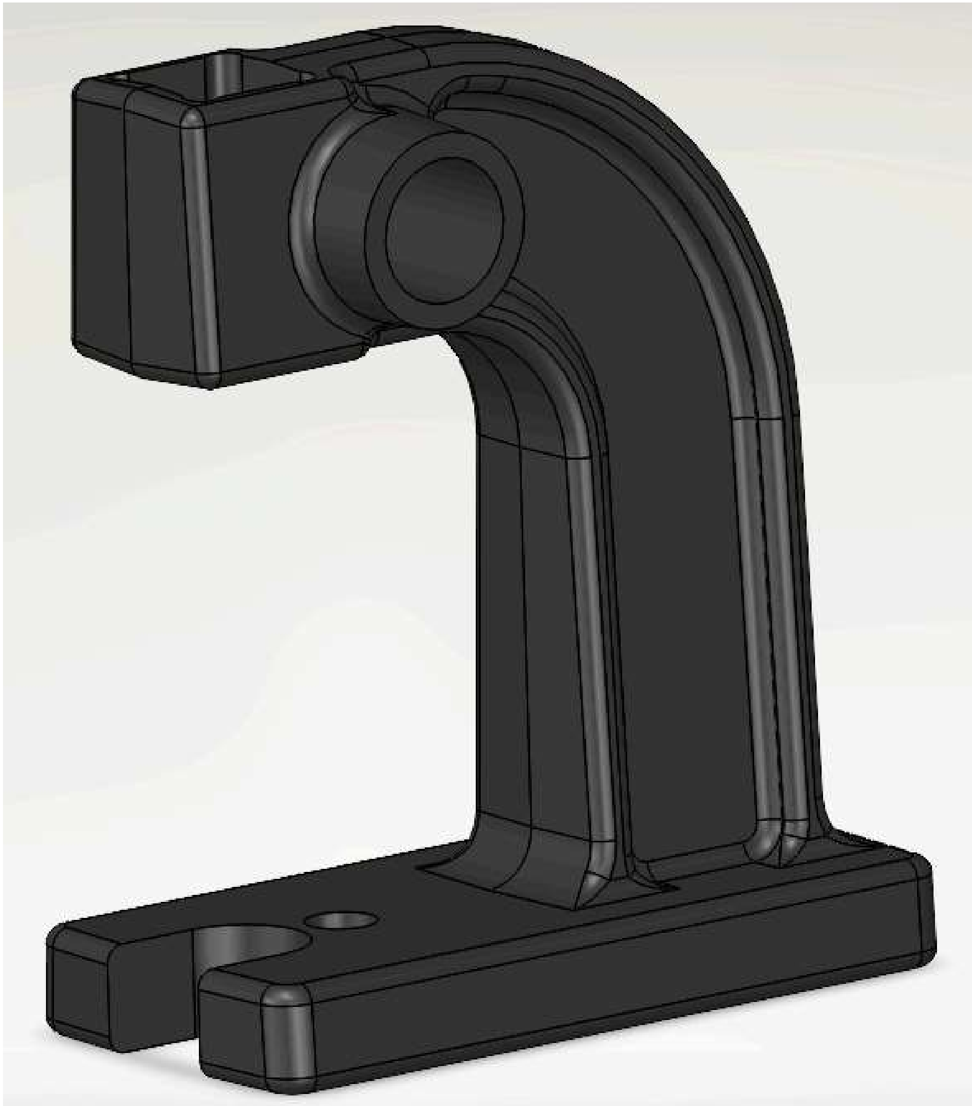

Fig. 2: 3D SolidWorks Model of P2 (#98472)

Similarly, a model was created for P3 (#98473). P3 was an updated version of P1 with a rear spine angled at 85 deg rather than 90 deg. The rear spine was also made half as thin and the rear rib along the spine was removed completely. The material was returned to Aluminum 6061, so no draft angles were included. The P3 model can be seen in Fig. 3 below.

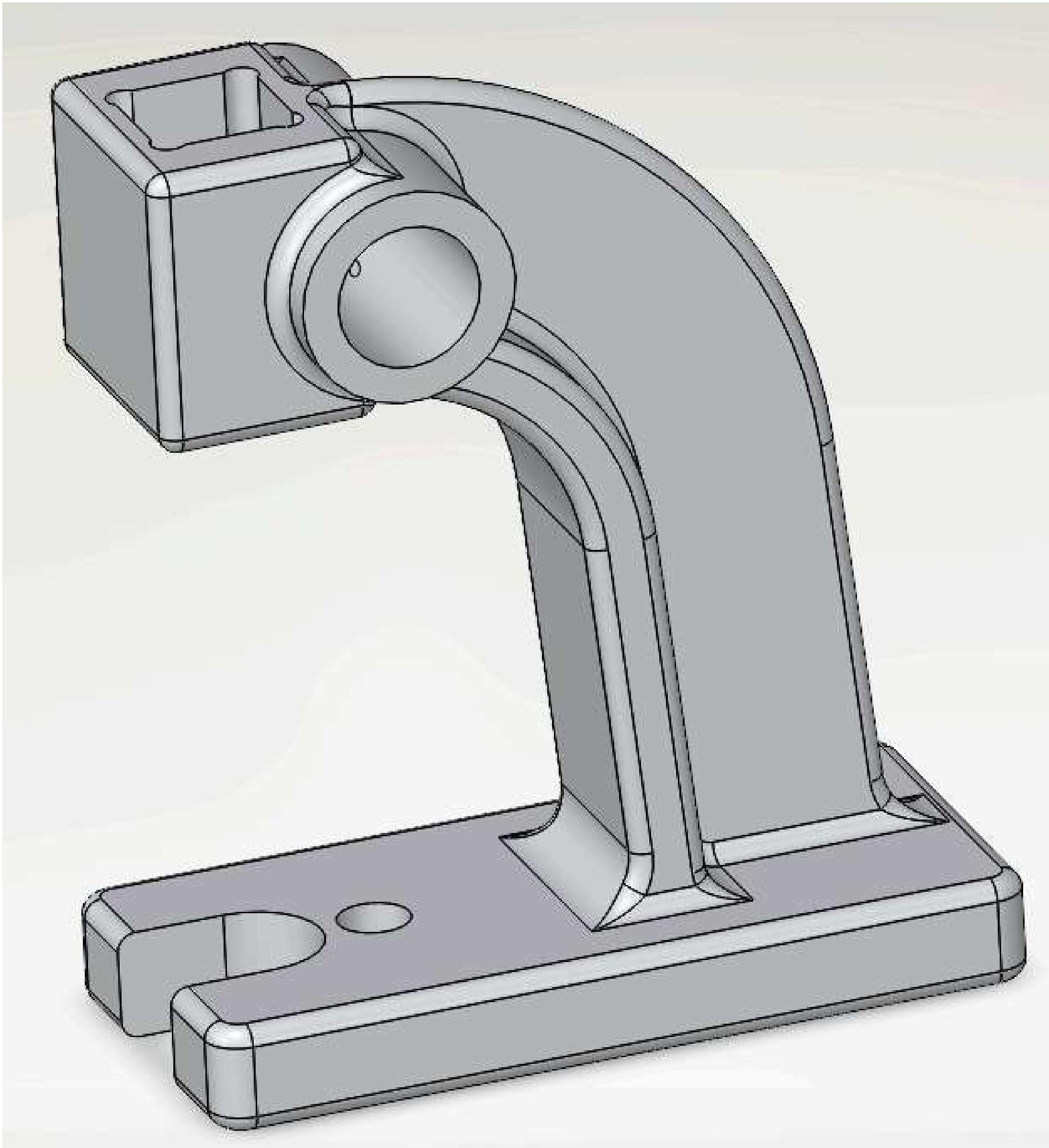

Fig. 3: 3D SolidWorks Model of P3 (#98473)

In P4, P3 was changed to Cast Iron and updates were included to ensure that the part did not fail under load. The only change I made was increasing the thickness of the rear rib. Draft angles were also supposed to be included due to the material change. This design should be updated further, and evidence for this will be outlined in the FEA Studies section of this report. The design of P4 can be seen in Fig. 4 below.

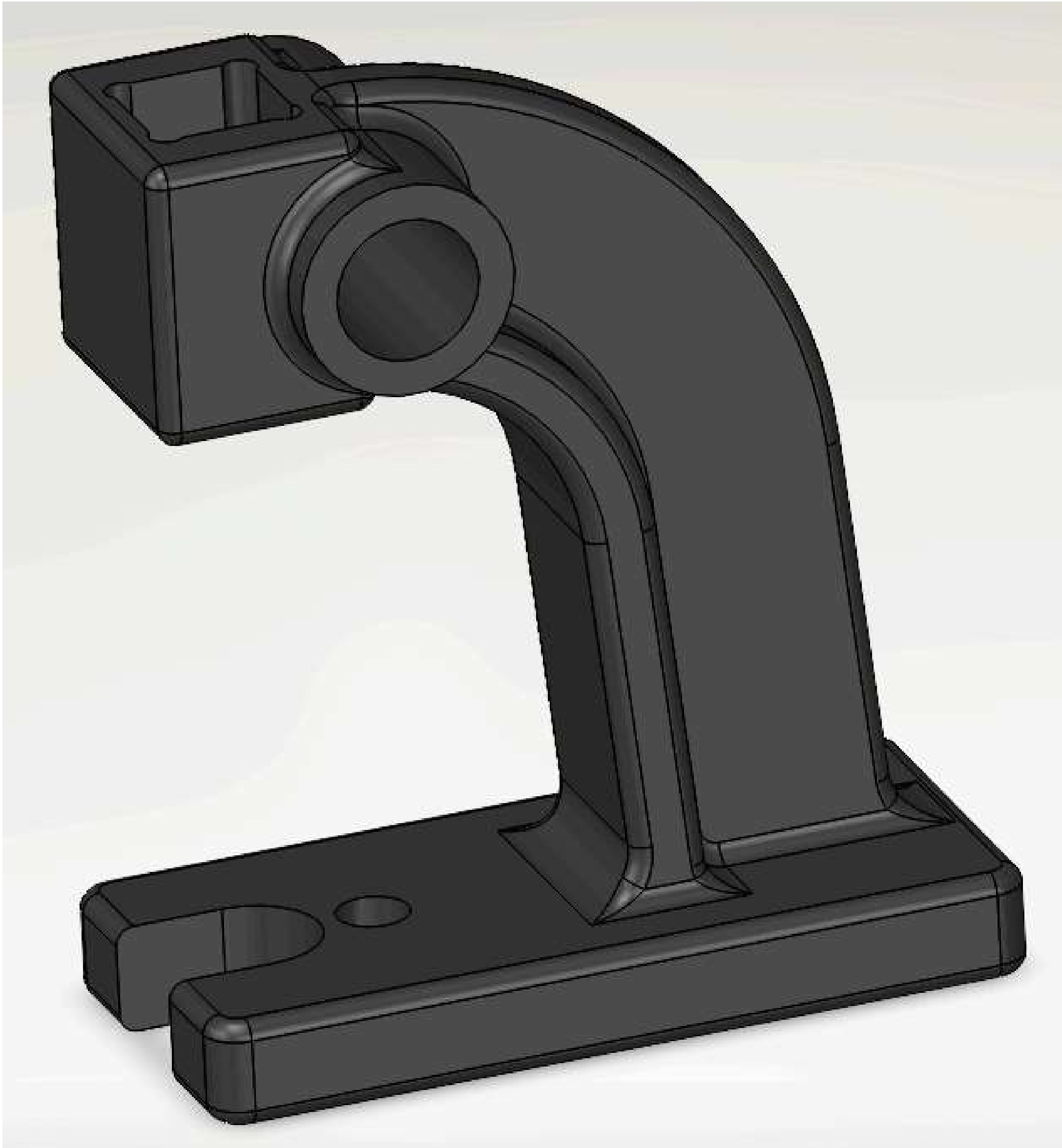

Fig. 4: 3D SolidWorks Model of P4 (#98474)

FEA Studies

A static study finite element analysis (FEA) was performed in SolidWorks for each part. Each part was fixed at its base. A force of 1 ton was applied vertically to the horizontal hole seen below in Fig. 5. Table 1 at the end of this section summarizes the results from all of the FEA studies.

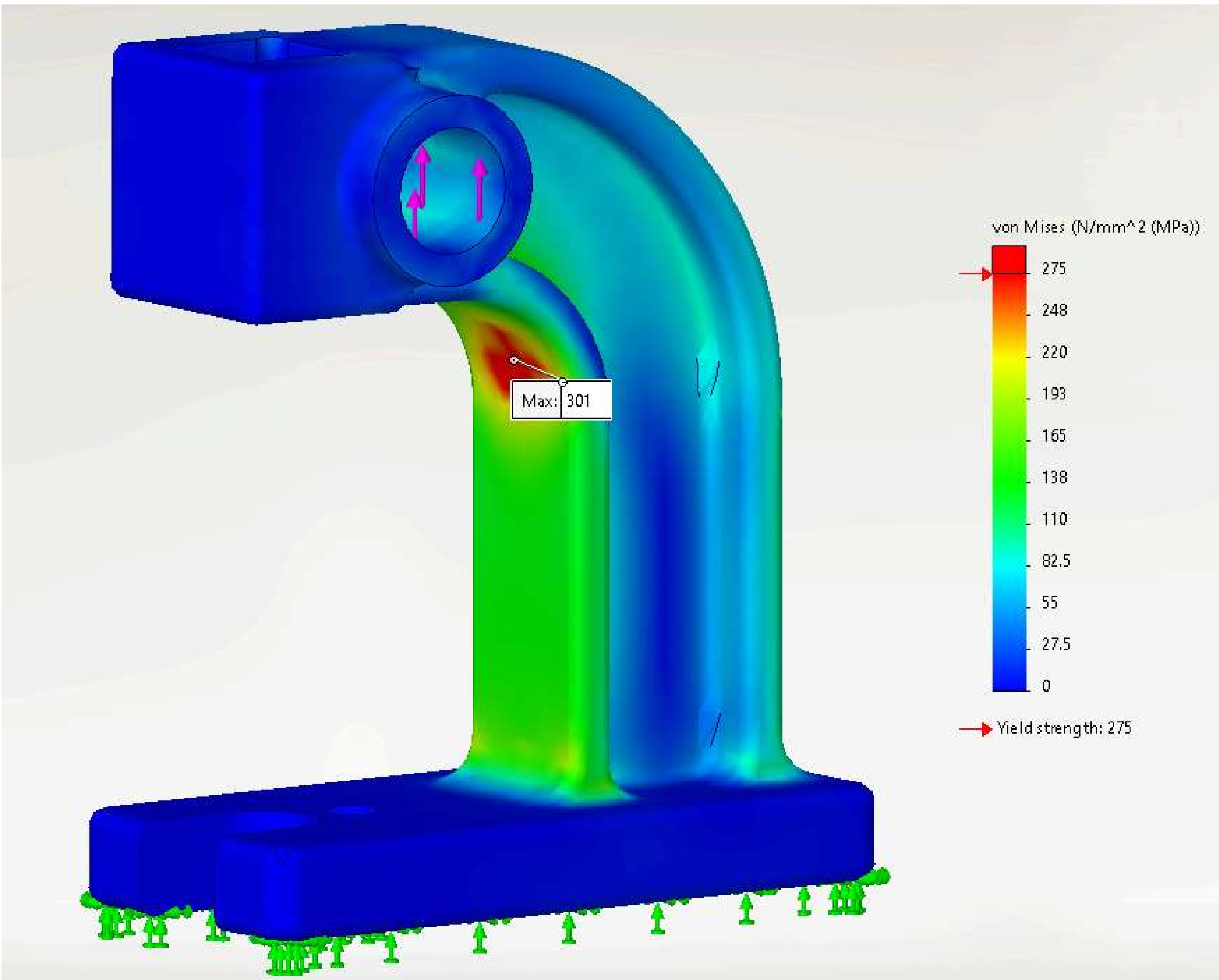

Fig. 5: Static FEA Study of P1

P1 encountered a maximum stress of 301 MPa, which is above the 275 MPa yield stress of Aluminum 6061. A factor of safety report was also generated to visualize the most critically stressed regions as seen in Fig. 6 below. An FOS below 1 indicates failure outside of the elastic region.

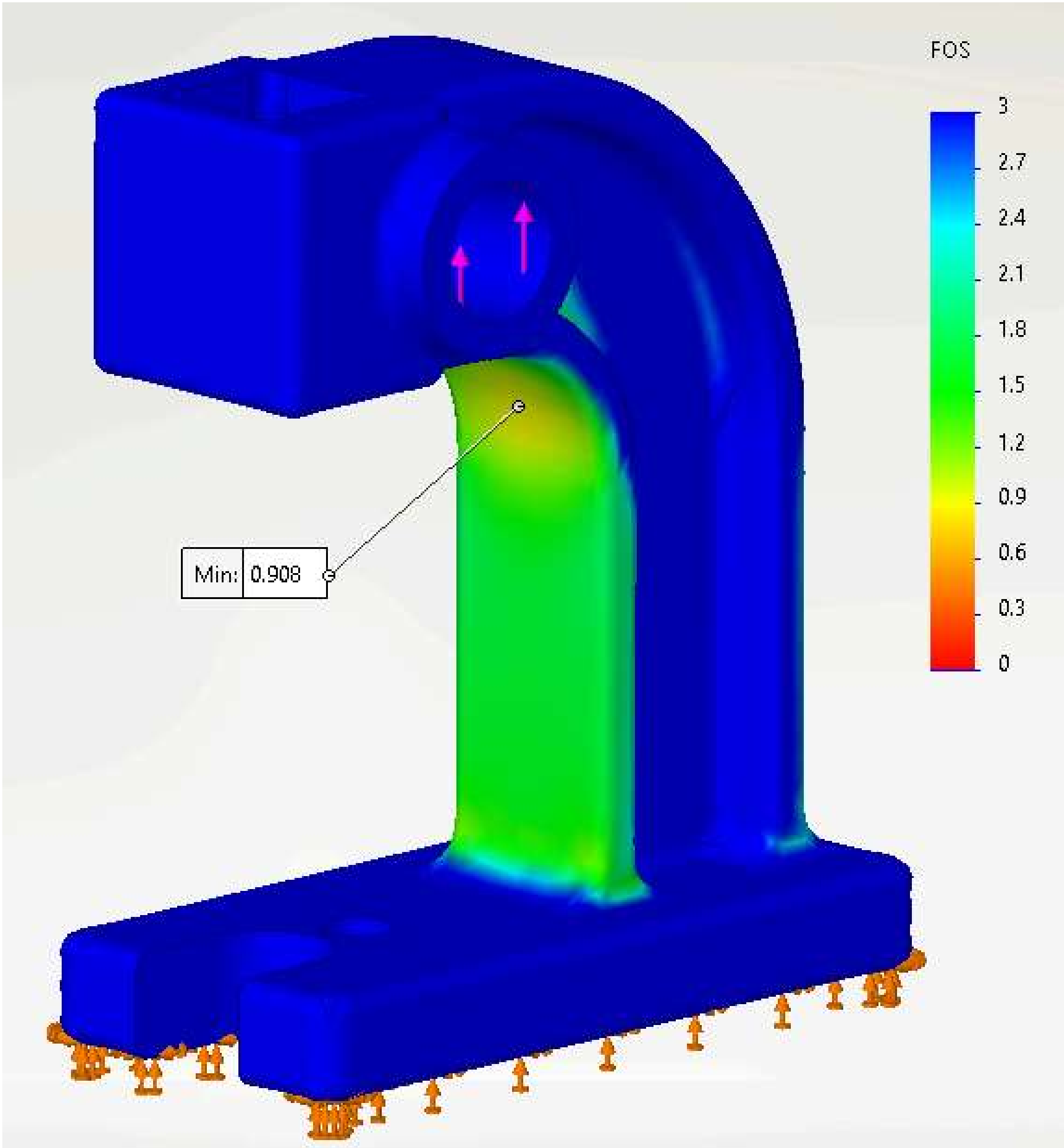

Fig. 6: Factor of Safety Report for P1

Here, the minimum FOS is 0.908, with most of the critical stress occurring beneath the hole where the force was applied. Next, P2 was stressed under the same load configuration as seen in Fig. 6 below.

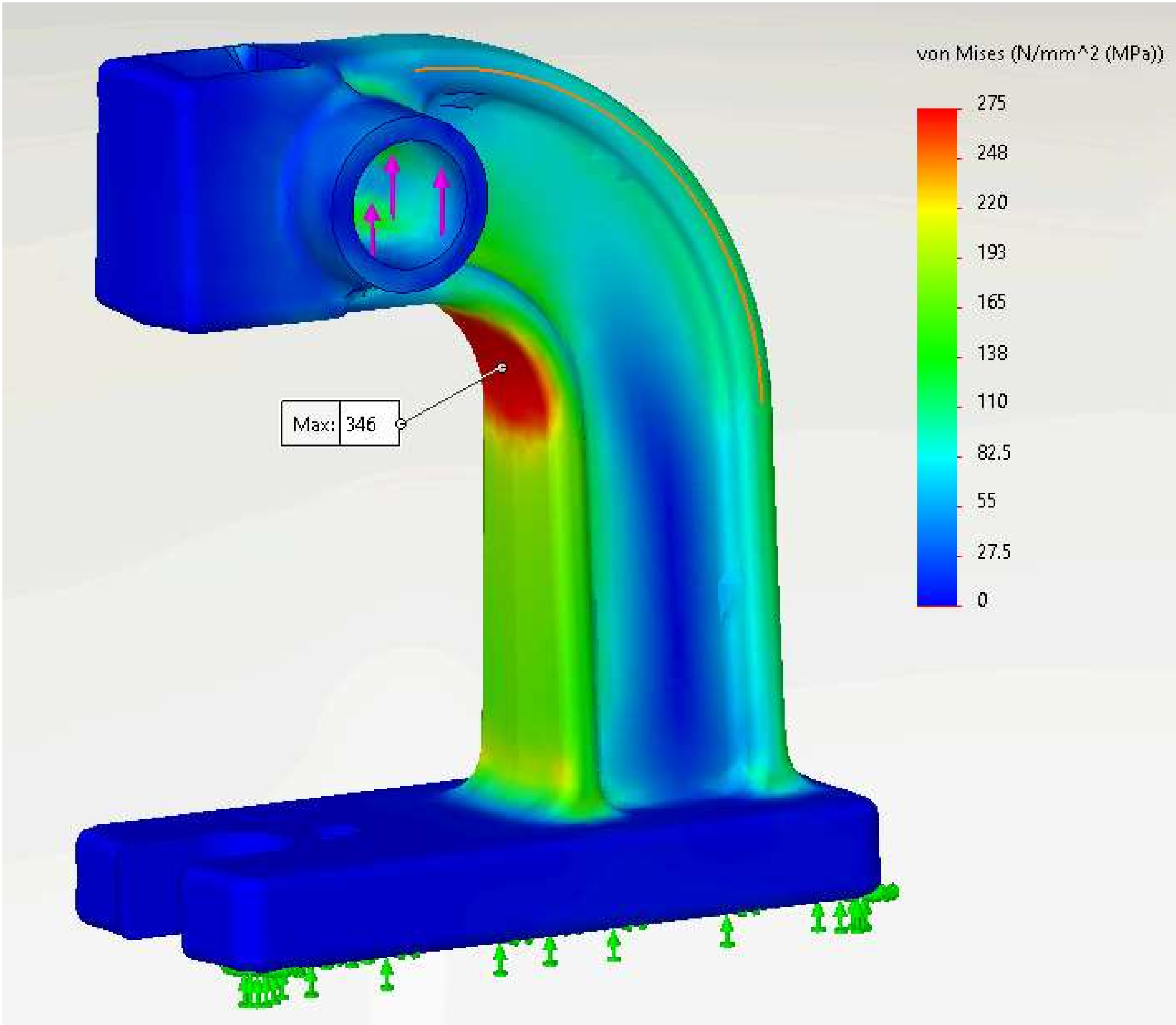

Fig. 7: Static FEA Study of P2

P2 is made of gray cast iron which has a yield strength range of 65.5-420 MPa. Since this range is so wide, the 275 MPa yield stress of Aluminum 6061 was used as a maximum on the scale in each figure to visualize the likelihood of failure. P2 encountered a maximum stress of 346 MPa, which is significantly above the 275 MPa yield stress of Aluminum 6061. A factor of safety report was also generated to visualize the most critically stressed regions as seen in Fig. 8 below. An FOS below 1 indicates failure outside of the elastic region.

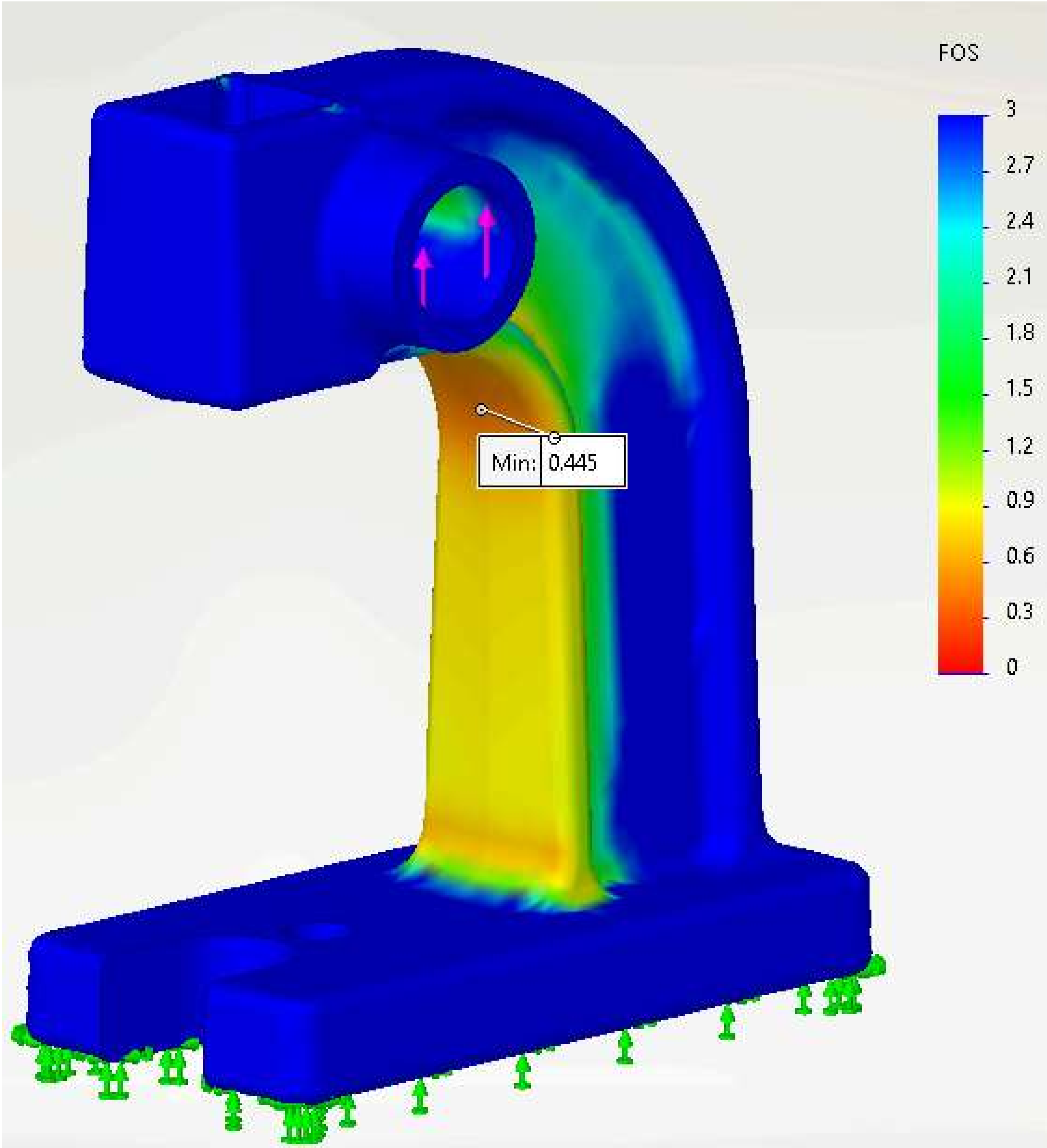

Fig. 8: Factor of Safety Report for P2

Here, the minimum FOS is 0.445, with most of the critical stress occurring beneath the hole where the force was applied. Next, P3 was stressed under the same load configuration as seen in Fig. 9 below.

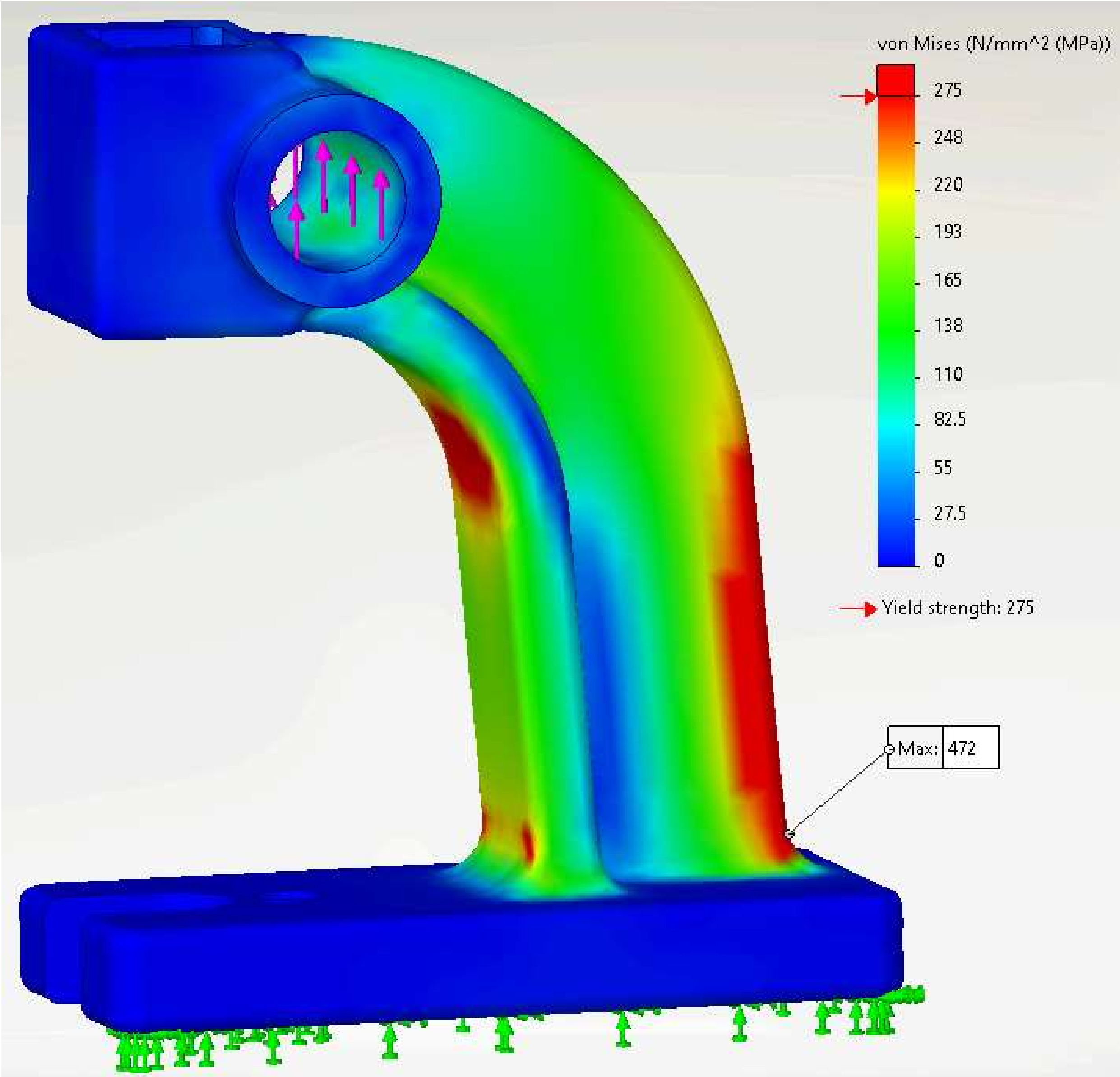

Fig. 9: Static FEA Study of P3

P1 encountered a maximum stress of 472 MPa, which is significantly above the 275 MPa yield stress of Aluminum 6061. A factor of safety report was also generated to visualize the most critically stressed regions as seen in Fig. 10 below. An FOS below 1 indicates failure outside of the elastic region.

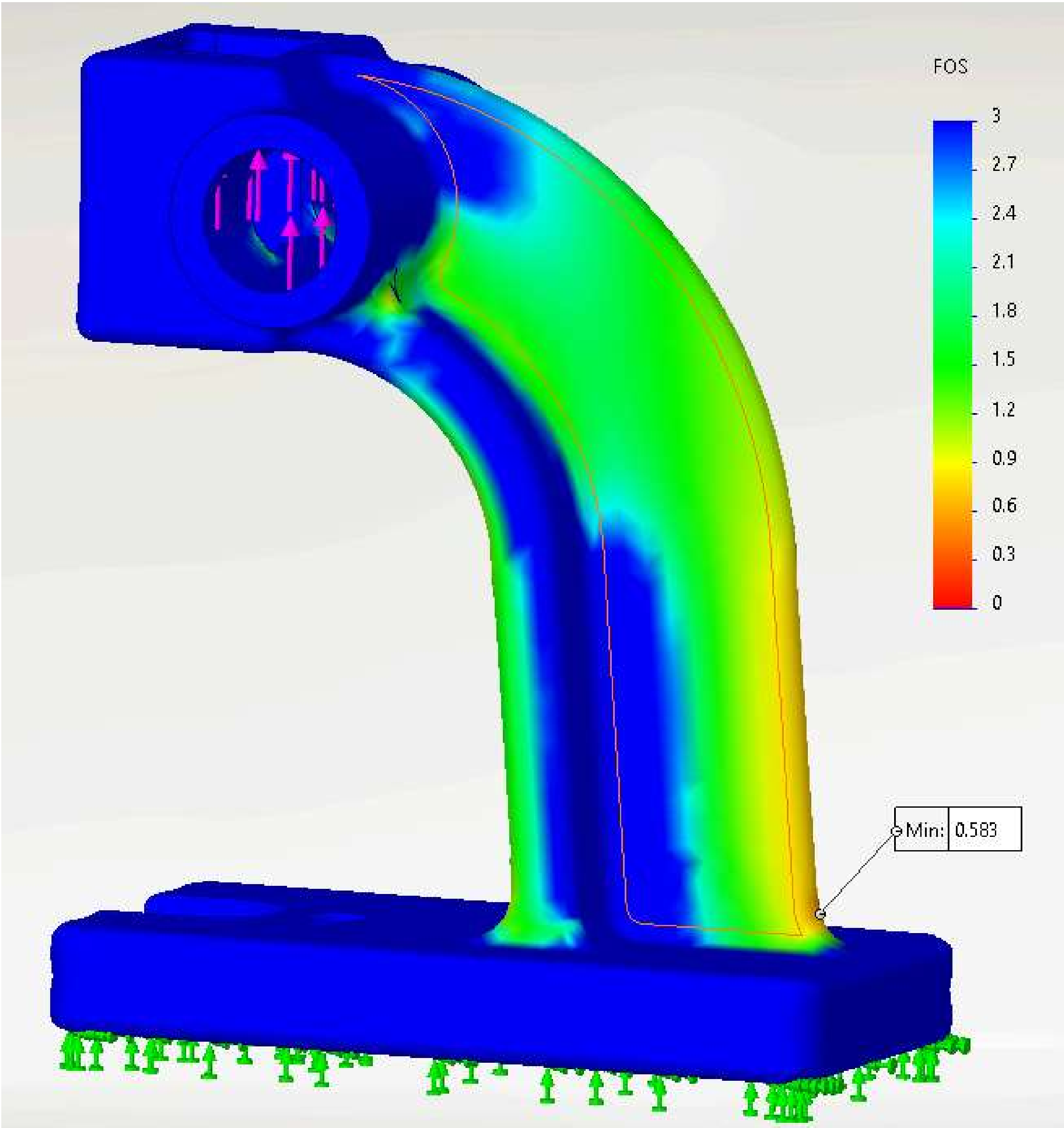

Fig. 10: Factor of Safety Report for P3

Here, the minimum FOS is 0.583, with most of the critical stress occurring on the rear spine. Next, P4 was stressed under the same load configuration as seen in Fig. 11 below.

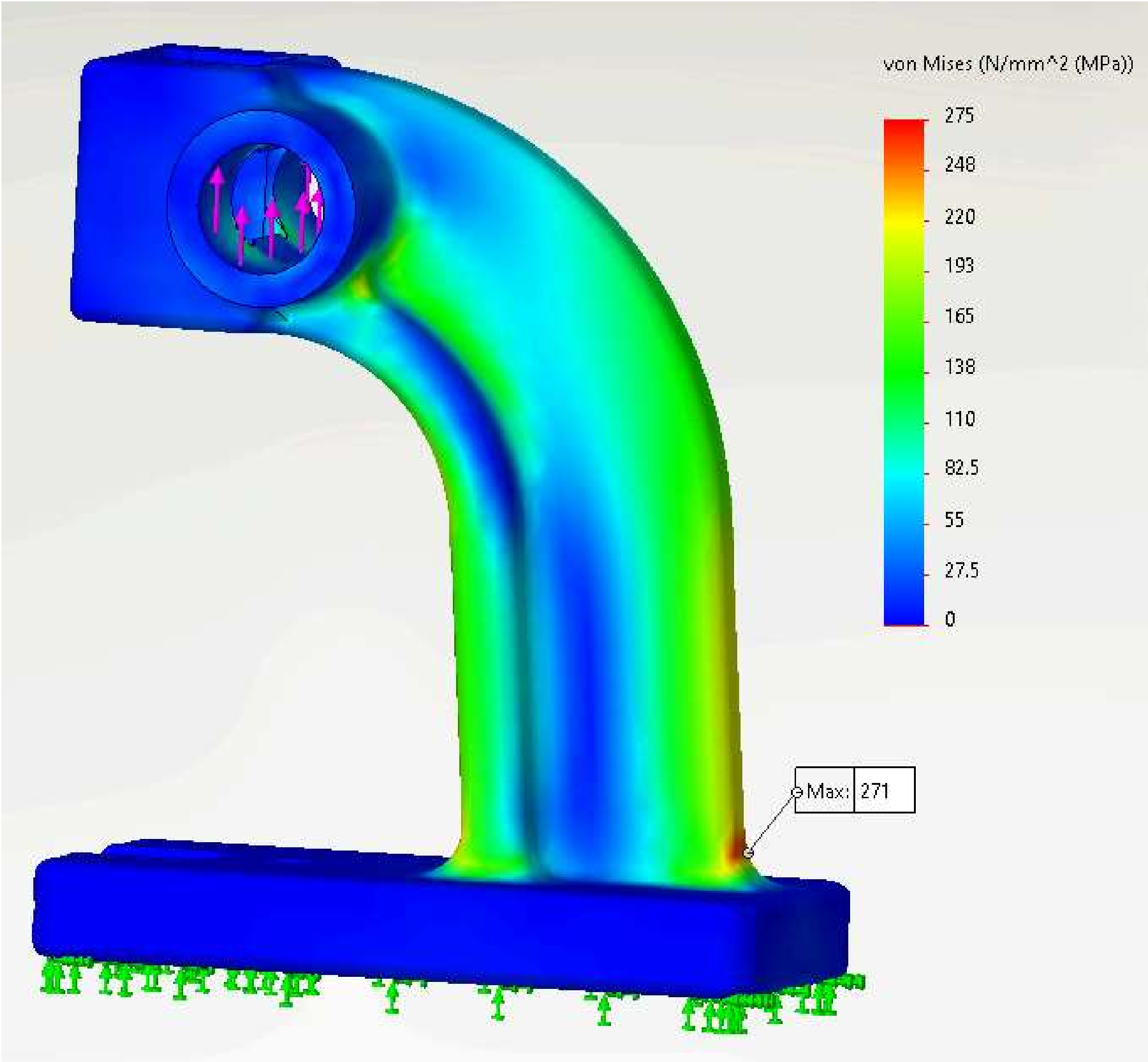

Fig. 11: Static FEA Study of P4

Comparing this gray cast iron part with the 275 MPa yield stress of Aluminum 6061, P4 encountered a lower maximum stress of 271 MPa. A factor of safety report was also generated to visualize the most critically stressed regions as seen in Fig. 12 below. An FOS below 1 indicates failure outside of the elastic region.

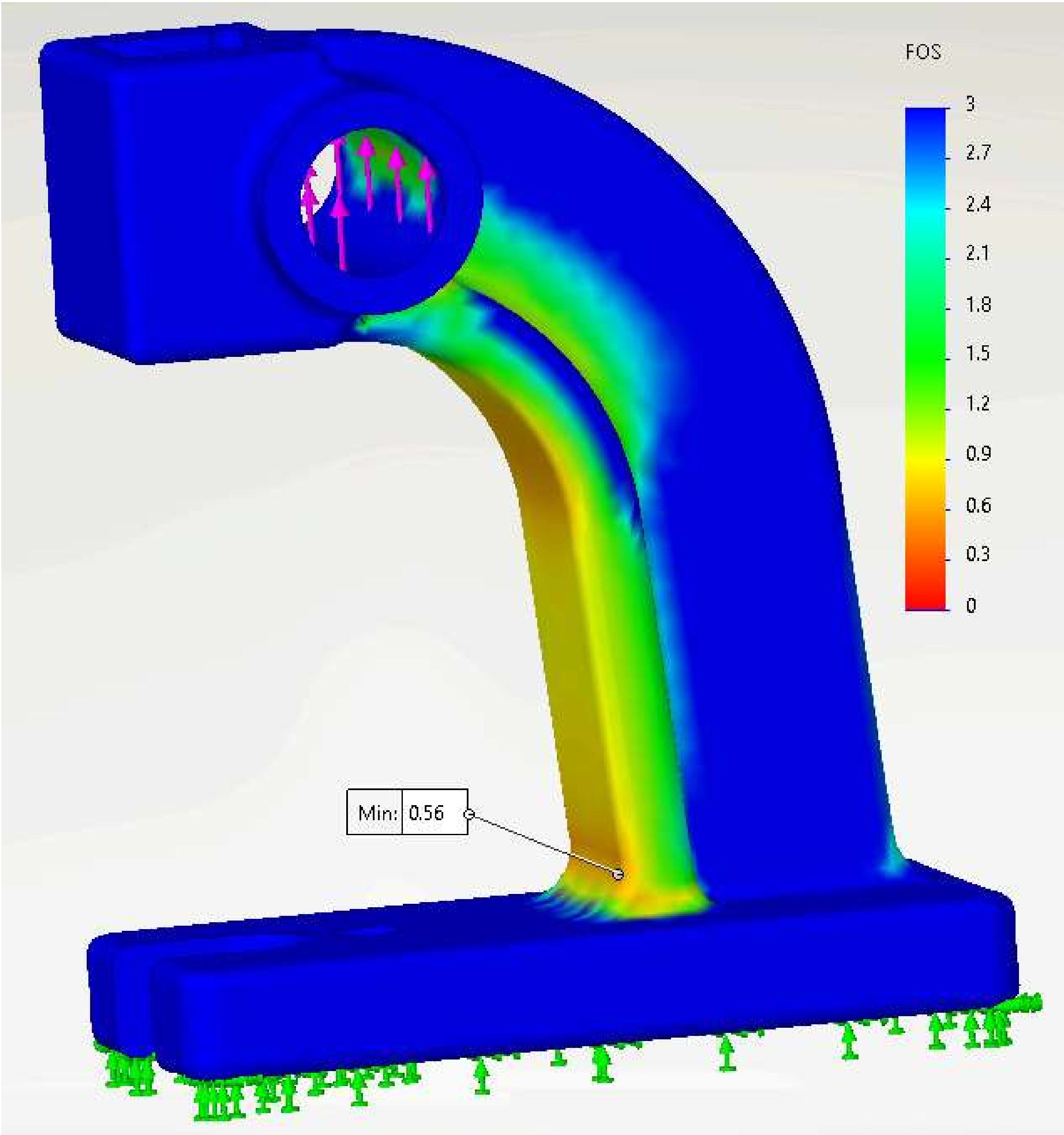

Fig. 12: Factor of Safety Report for P3

Here, the minimum FOS is 0.56, with most of the critical stress occurring beneath the hole where the force was applied. Ultimately, the FOS reports for the gray cast iron did not map critically stressed regions well to the FEA static stress reports, especially in the case of P4. My changes to the P4 design only included thickening the rear spine of the part so that the yield stress was below that of 275 MPa. It would be advantageous to add another rib to the back of the rear spine. Thickening the rear rib, front rib, and the spine would help to strengthen the part. Table 1 summarizes the results from the FEA studies conducted on P1 and P2.

	P1 (Aluminum 6061)	P2 (Gray Cast Iron)	P3 (Aluminum 6061)	P4 (Gray Cast Iron)
Maximum Stress Under 1 Ton Load	301 MPa	346 MPa	472 MPa	271 MPa
Minimum Factor of Safety	0.908	0.445	0.583	0.560

Table 1: Maximum Stress and Minimum FOS for P1 and P2

The results from this table indicate that P1 is a better design than P3 for this loading scenario. P1's maximum Von Mises stress is significantly lower than the P3's, and closer to the 275 MPa yield stress of Aluminum 6061-T6. All parts here failed the FOS test, but P4 did have a lower yield strength than 275 MPa. The aforementioned design changes would be made if the actual gray cast iron yield strength was lower than 275 MPa.

2D Drawings*

Next, drawings were created for each part. Each drawing includes an isometric view in the top right and three orthographic projections. The orthographic projections show the right, top, and front side of the parts. Section view A is projected off of the front view and detail view B is projected off of the top view. Different views are used to better visualize the dimensions and characteristics of every feature of each part. GD&T symbols, including Flatness, Perpendicularity, Symmetry, and Position are used to convey tolerancing requirements of various features. These symbols differ from the title block tolerancing convention to ensure that the corresponding features physically fit their purposes throughout the manufacturing process or in practical post-manufacture use. These drawings can be seen in Fig. 13-16 below.

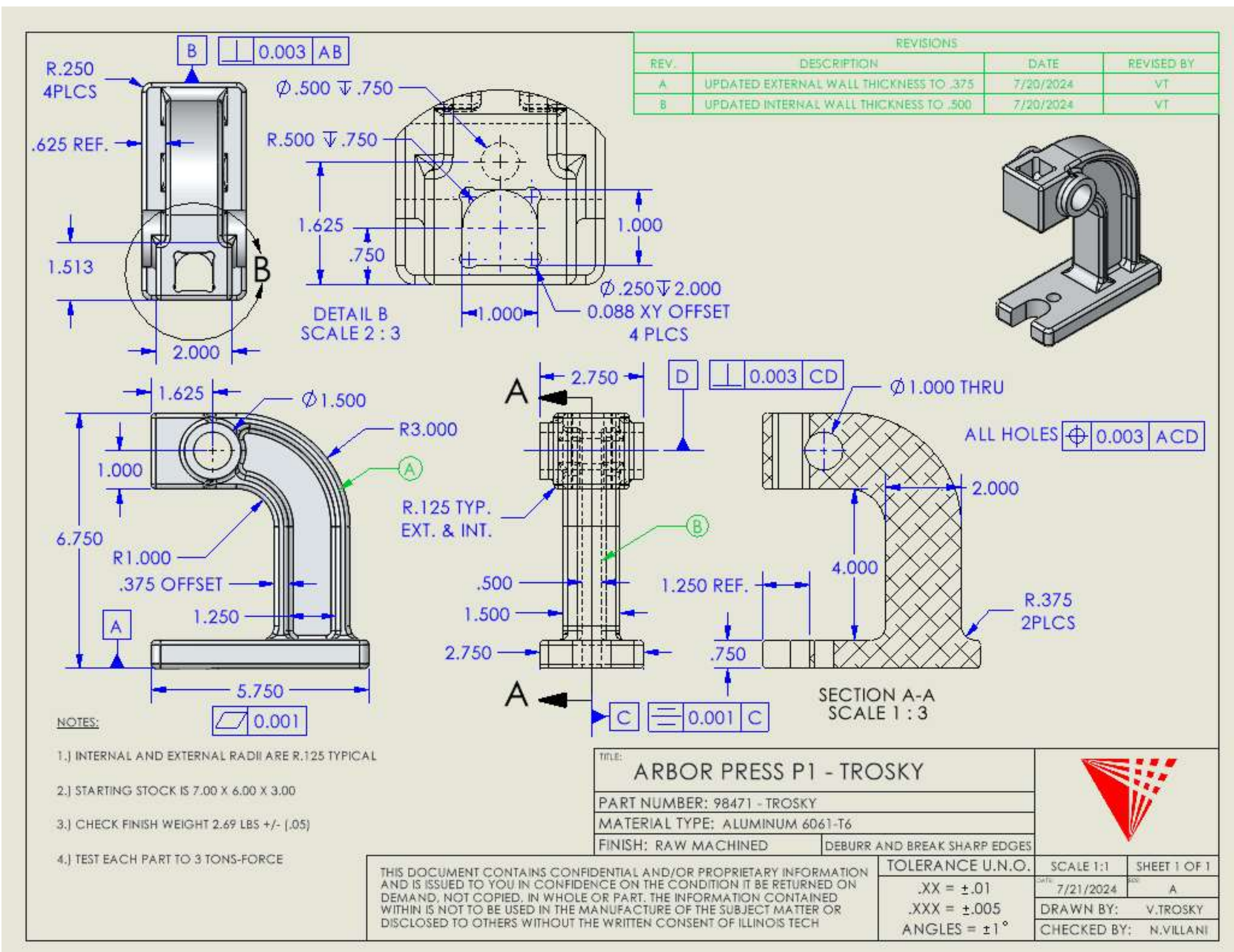

Fig. 13: SolidWorks Engineering Drawing of P1

*Two more drawings were created but were included instead in the 3D Assemblies section for improved readability.

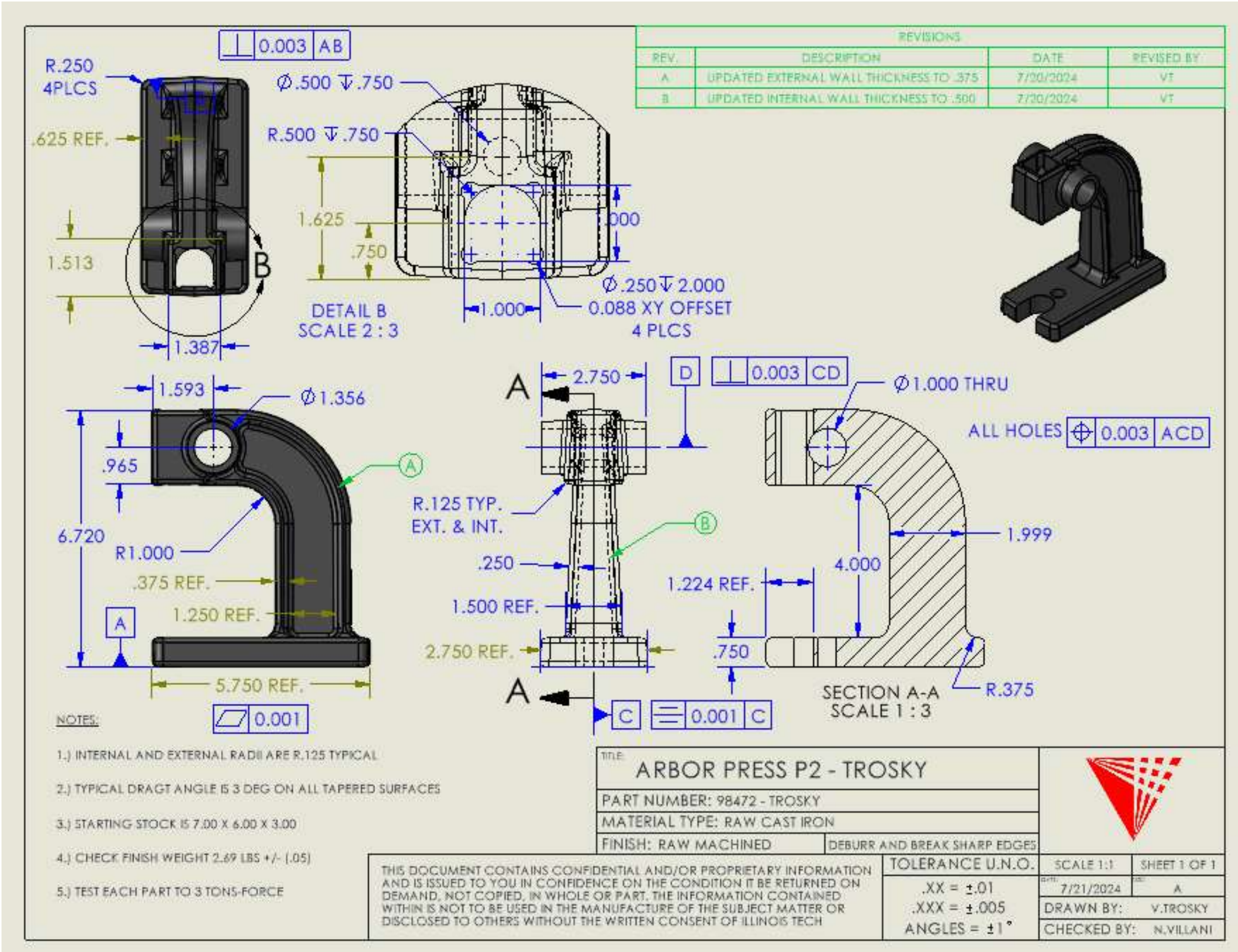

Fig. 14: SolidWorks Engineering Drawing of P2

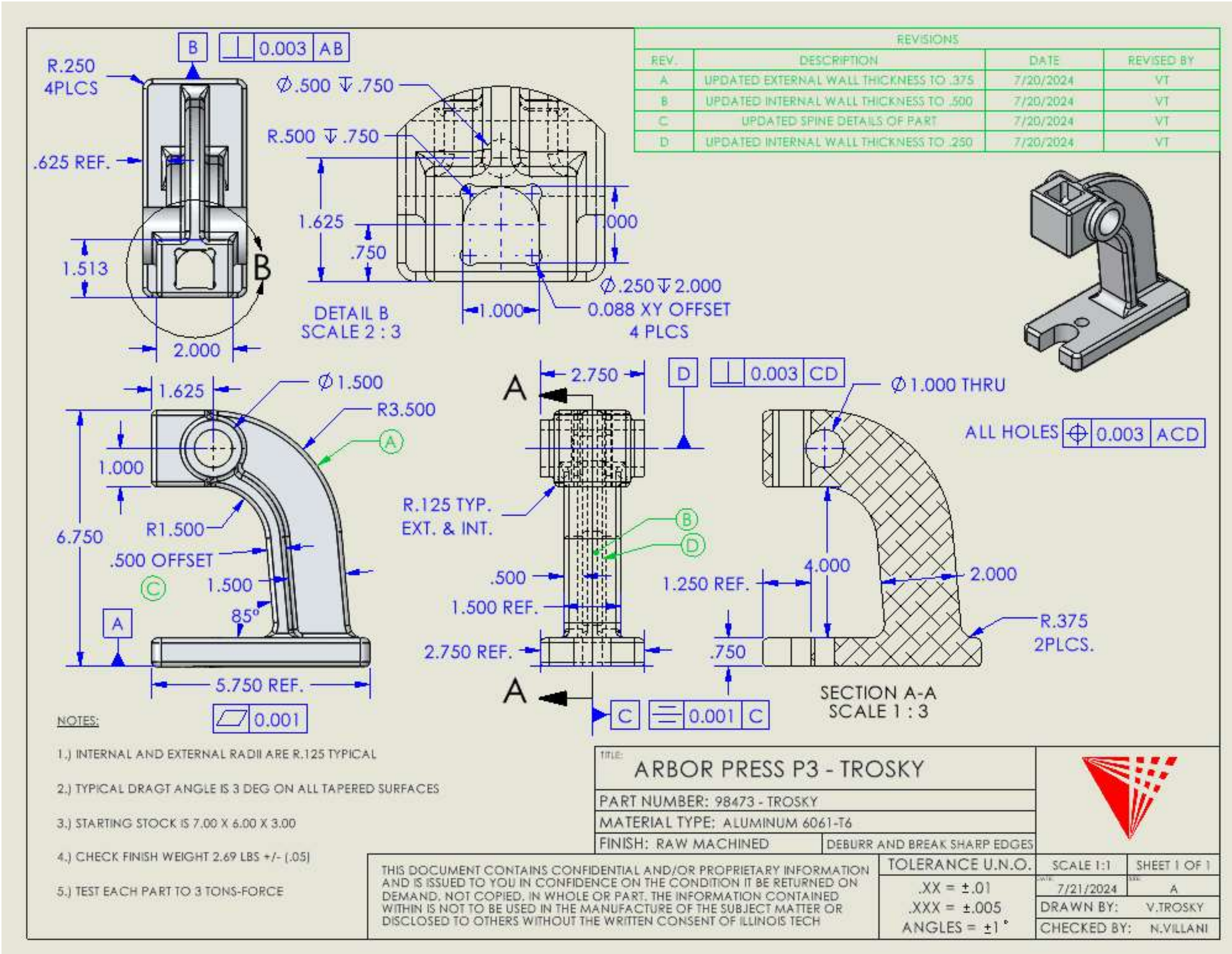

Fig. 15: SolidWorks Engineering Drawing of P3

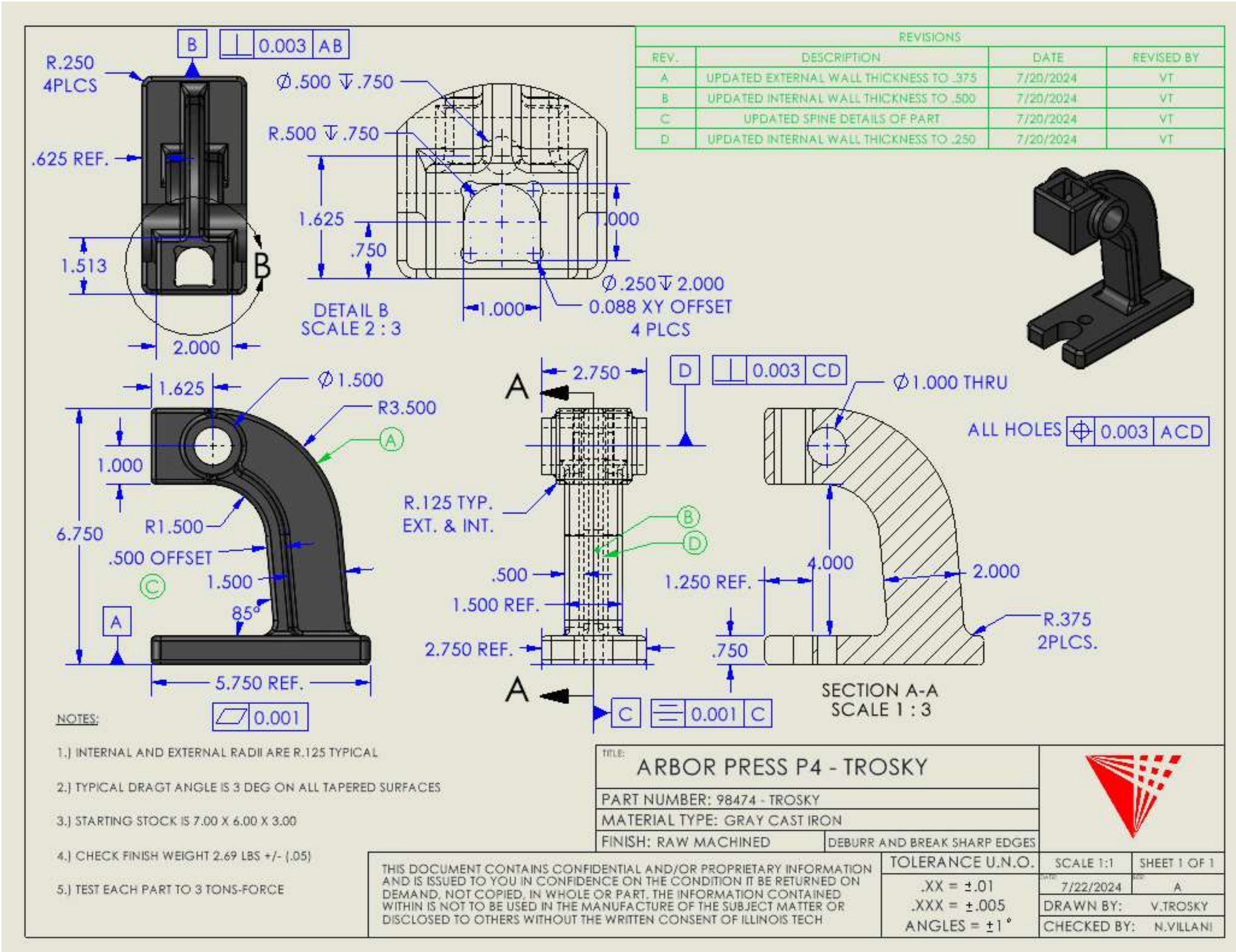

Fig. 16: SolidWorks Engineering Drawing of P4

3D Assemblies

After the drawings of each individual part were created, a 3D assembly and an exploded view of the full arbor press were created as seen in Fig. 17.

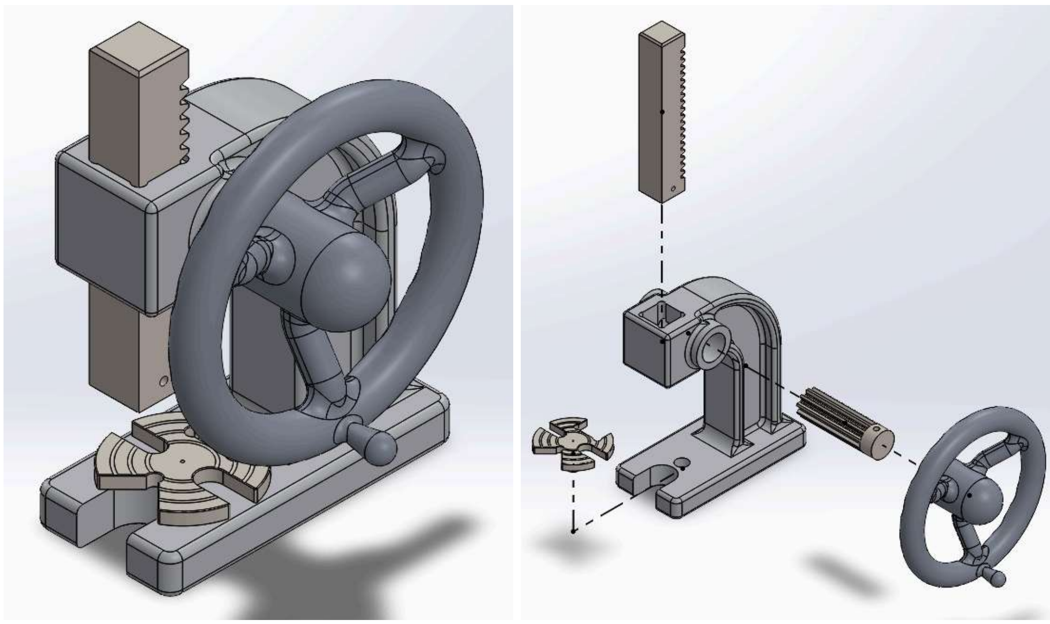

Fig. 17: Arbor Press Assembly (Left) and Exploded View (Right)

The assembly features a mechanical “rack and pinion mate” which moves the rack up or down as the wheel is turned. Another assembly was created of the cavity mold configuration for the cast gray iron P2 part. The assembly and exploded view can be seen in Fig. 18 below.

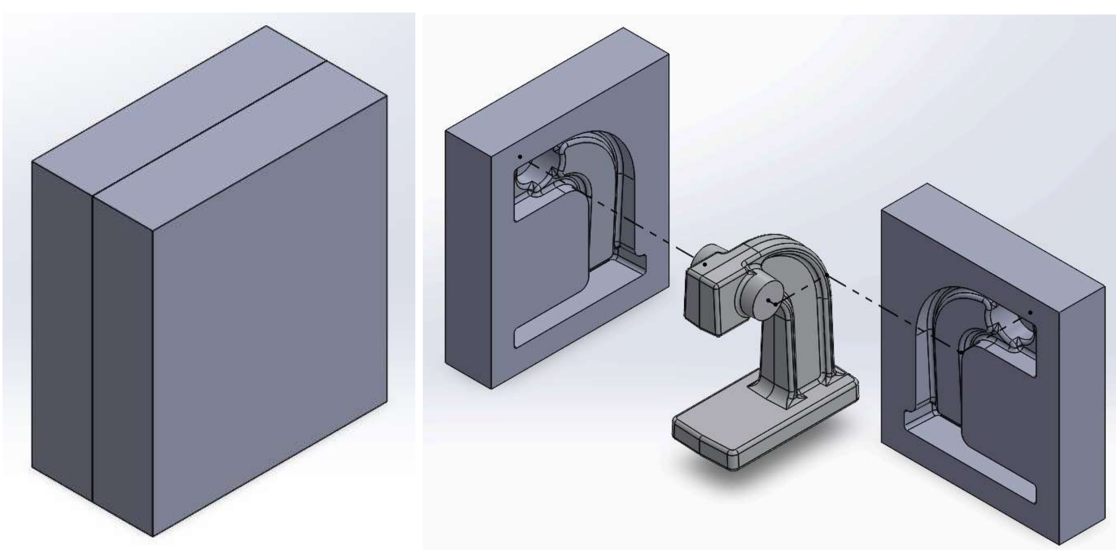

Fig. 18: Cavity Mold Assembly (Left) and Exploded View (Right)

Drawings of the views in Fig. 17 and 18 were created, incorporating a BOM and numbered bubble callouts. The results can be seen in Fig. 19 and 20 below.

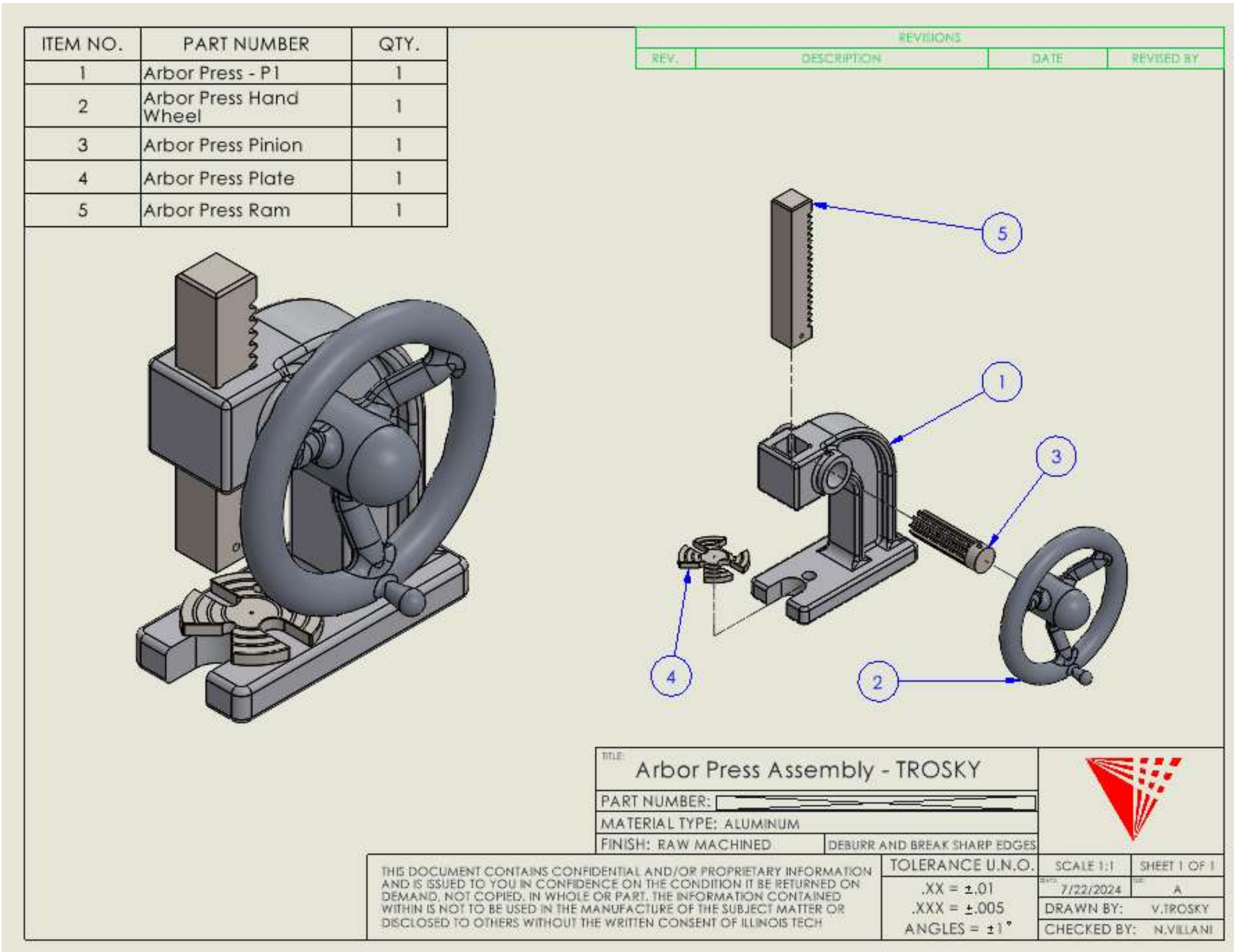

Fig. 19: Drawing of P1 Arbor Press Assembly and Exploded View

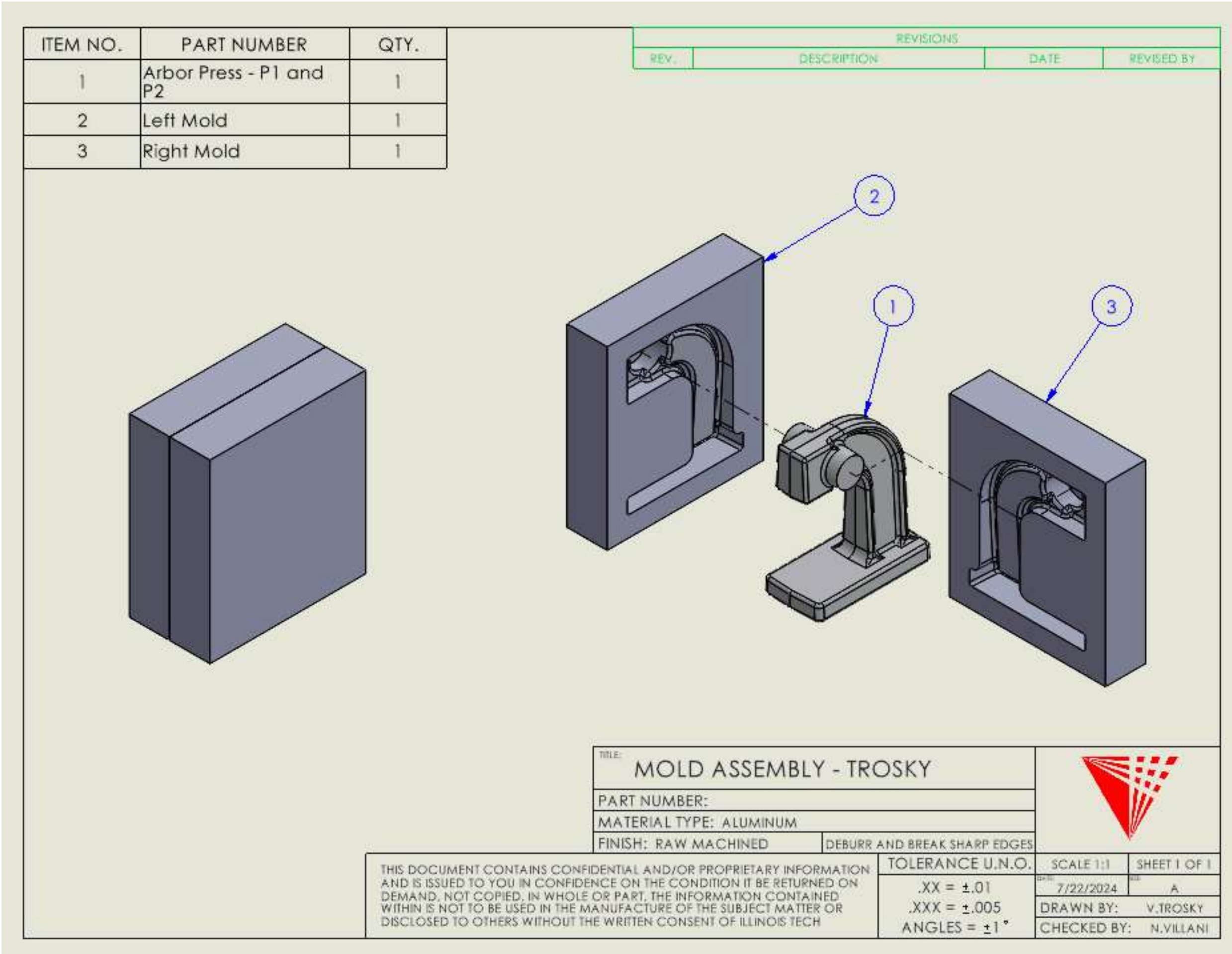

Fig. 20: Drawing of P1 Arbor Press Assembly and Exploded View

Programming

P1 and the Left Mold part from Fig. 20 were imported into MasterCAM to program the G-Code necessary for machining this part on a DMG Mori DMU-50. A 6.00 x 7.00 x 3.00 in. (XYZ) block of stock aluminum was used for P1, and only one operation was performed. The result can be seen in Fig. 21 below.

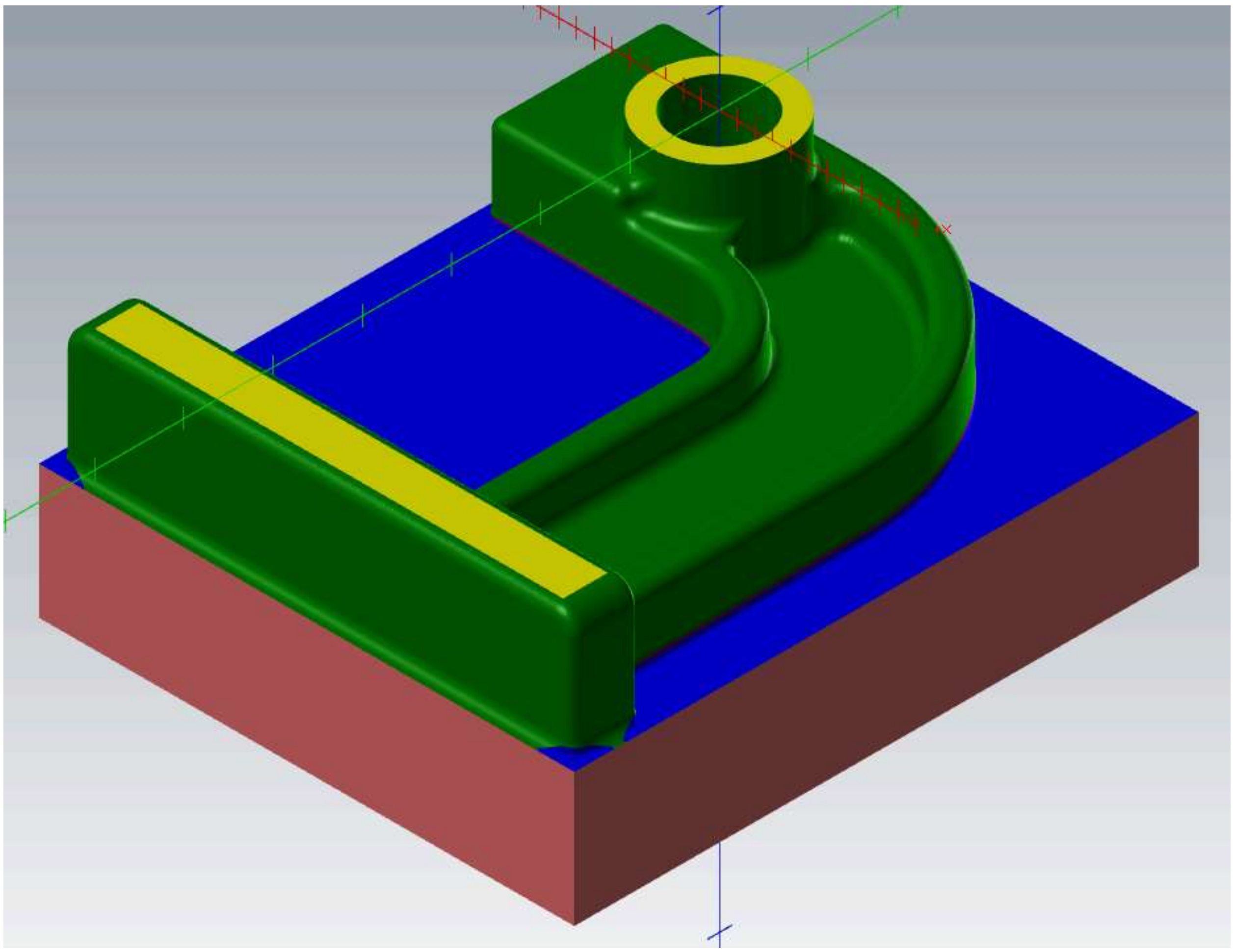

Fig. 21: P1 Operation Simulation

Four toolpaths were used to complete this operation: facing (yellow), Dynamic OptiRough (not colored), Dynamic OptiRest (blue), and Equal Scallop (green). Next, an operation for the Left Mold as seen in Fig. 20 was programmed. While no facing toolpath was used, the operation used the same toolpath progression as P1 but on different contours. The result can be seen in Fig. 22 below.

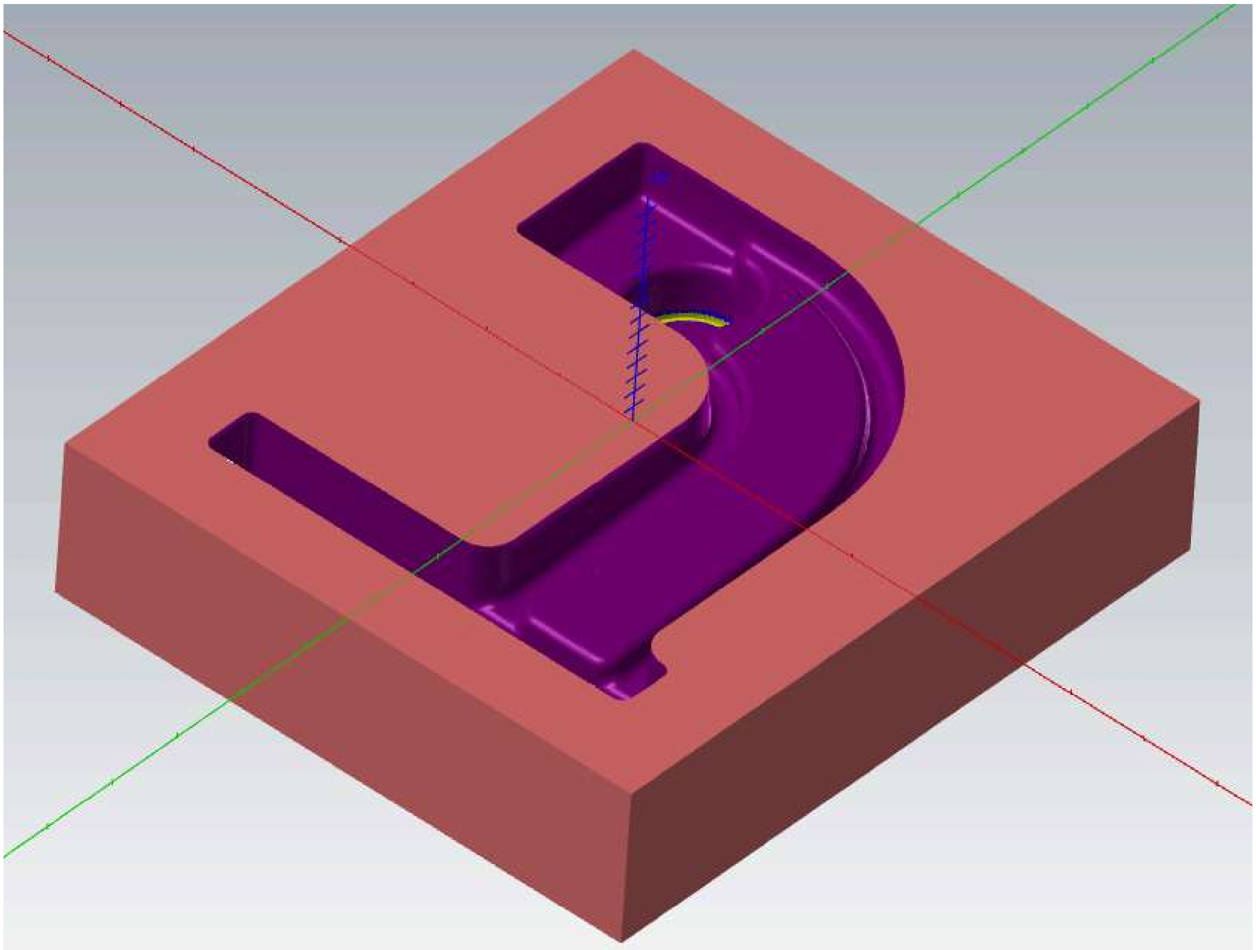

Fig. 22: Left Mold Operation Simulation

Core Mold (Optional)

The final task of the project was to model and program one half of a core mold for the shelled half of P2 seen in Fig. 23.

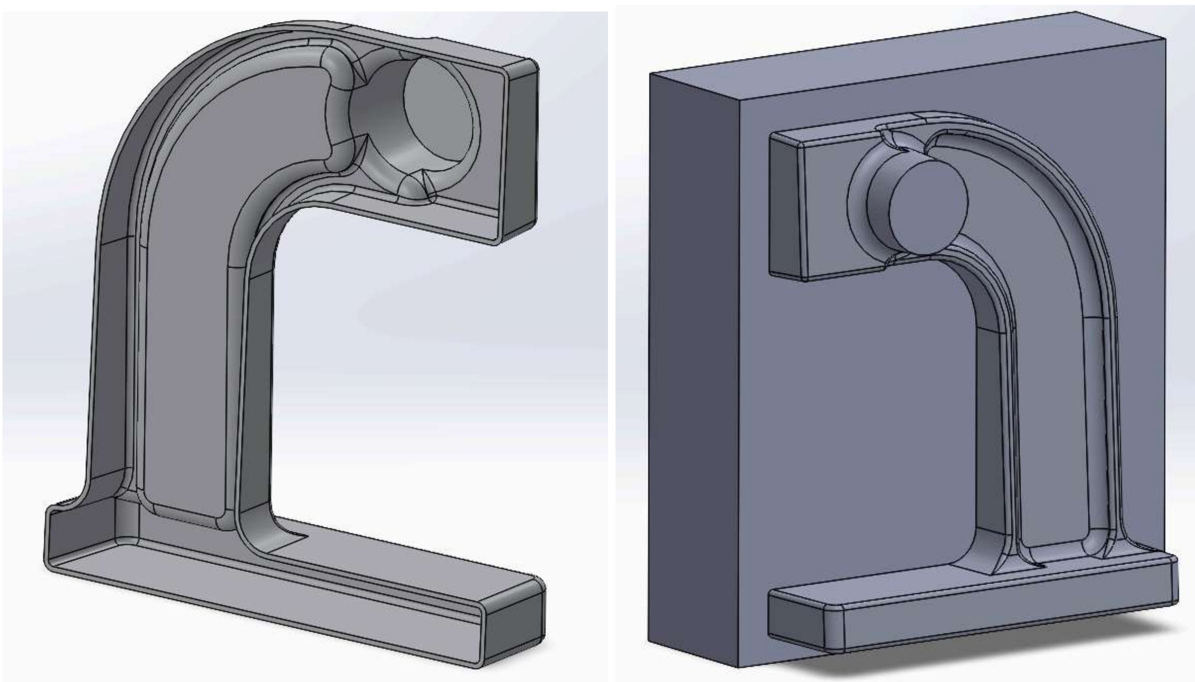

Fig. 23: Shelled Half of P1 (Left) and Positive Half of Core Mold (Right)

The cavity feature in SolidWorks was used to match the shape of the shelled half of P2. Programming the positive half of the core mold in MasterCAM proved more complex due to the 3 degree draft required by the casting process. The result can be seen in Fig. 24 below.

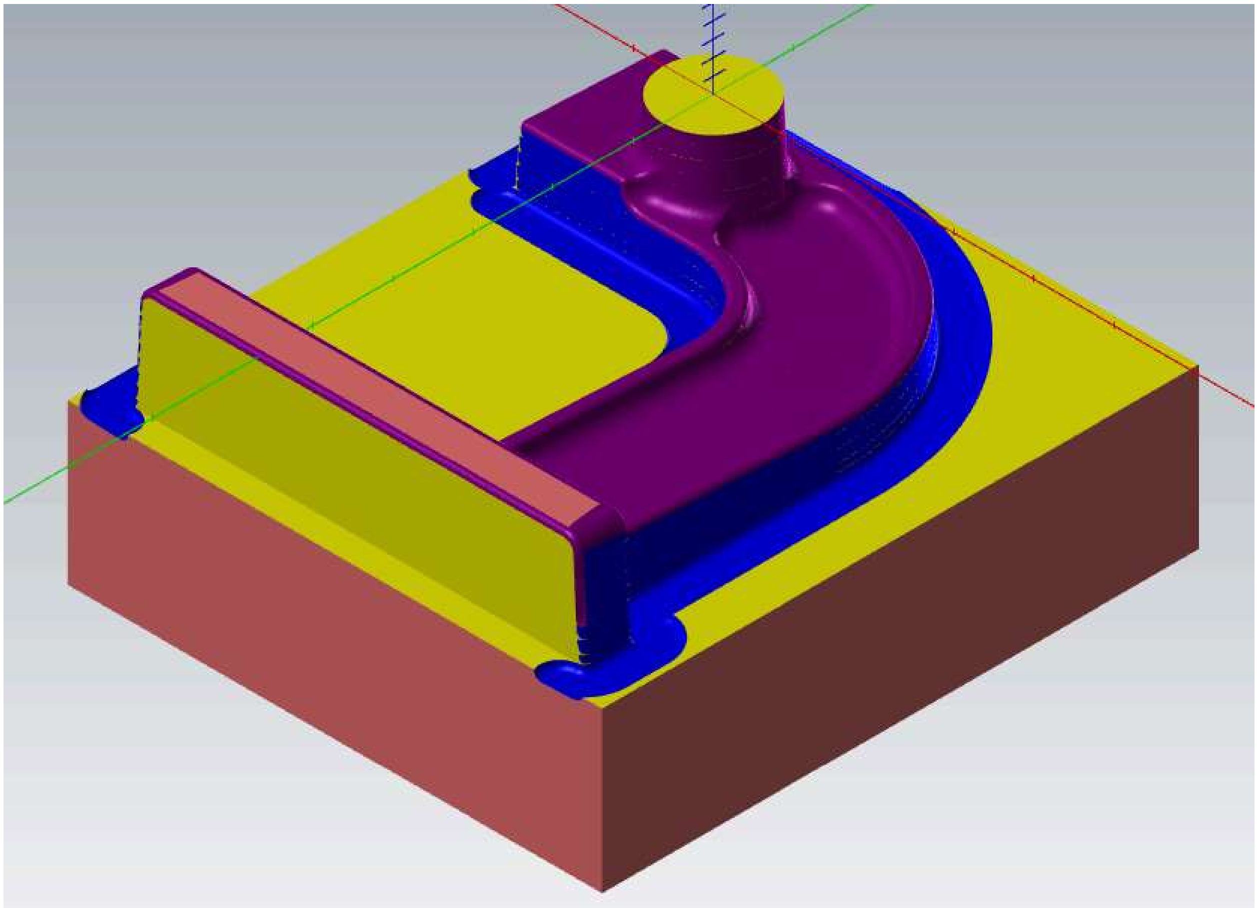

Fig. 24: Positive Half of Core Mold Operation Simulation

While the 3-axis Dynamic OptiRough (yellow) and Equal Scallop (purple) toolpaths were still used, a 5-axis swarf machining (blue) toolpath was introduced to machine the drafted edges. This operation needs revision, as the swarfing operation buried the bullnose endmill slightly beneath the plane it was supposed to remain above.