Lab Report 2

Marwa Al Alawi, Ben Gutierrez, Lily Mueller, Jonathan Sampson

2.008 Spring 2019

I. Machining

A. Blue Body

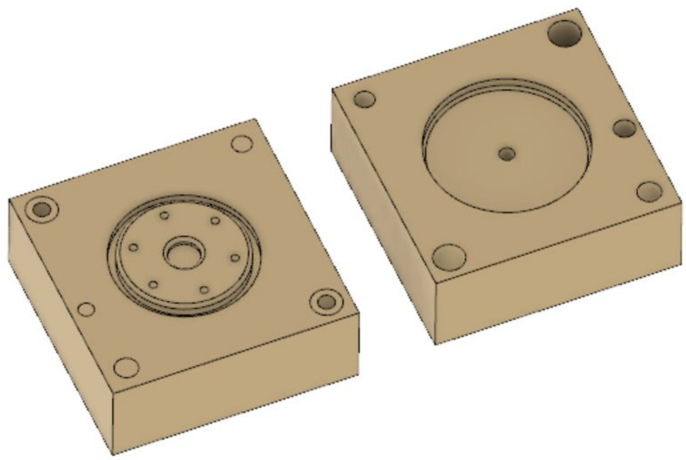

Figure 1: CAD model of blue body mold

The blue body is our simplest part so it was the easiest and quickest mold to make. The core side took 29 minutes to machine and the cavity side took only 5 minutes to machine. The toolpaths for the mold halves were as follows:

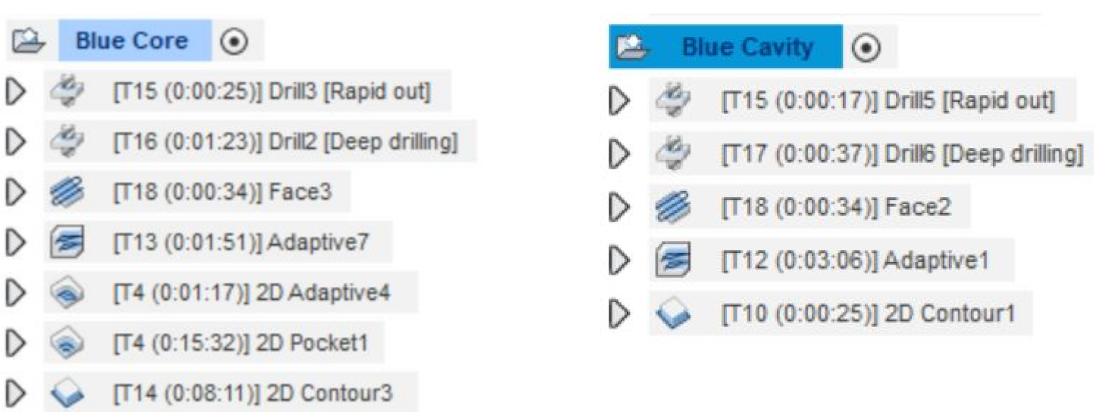

Figure 2: These are the toolpath operations for the blue body mold core and cavity. Both toolpaths begin with center drilling, then deep drilling of either the ejector pin holes or shoulder bolt hole, followed by facing the top of the stock to create a uniform and smooth model top. Next adaptive toolpaths are used as a roughing pass to clear material into the general shape of the mold before finally a finishing toolpath with a smaller vertical and horizontal stepover is used to create a smoother surface finish.

We choose to use 5000 RPM for the spindle speed for every operation because that is the maximum speed of the milling machine. Since the machining time was short for these mold halves with the default feed rates, we used the pre-set feed rate values for all of the operations. In practice we would often slow down the feed rate to 30-40% of the programmed value when just starting an operation to make sure the toolpath was doing what we expected it to do before ramping up to 100% of the programmed feed rate. We were successfully able to machine this blue body mold on our first attempt.

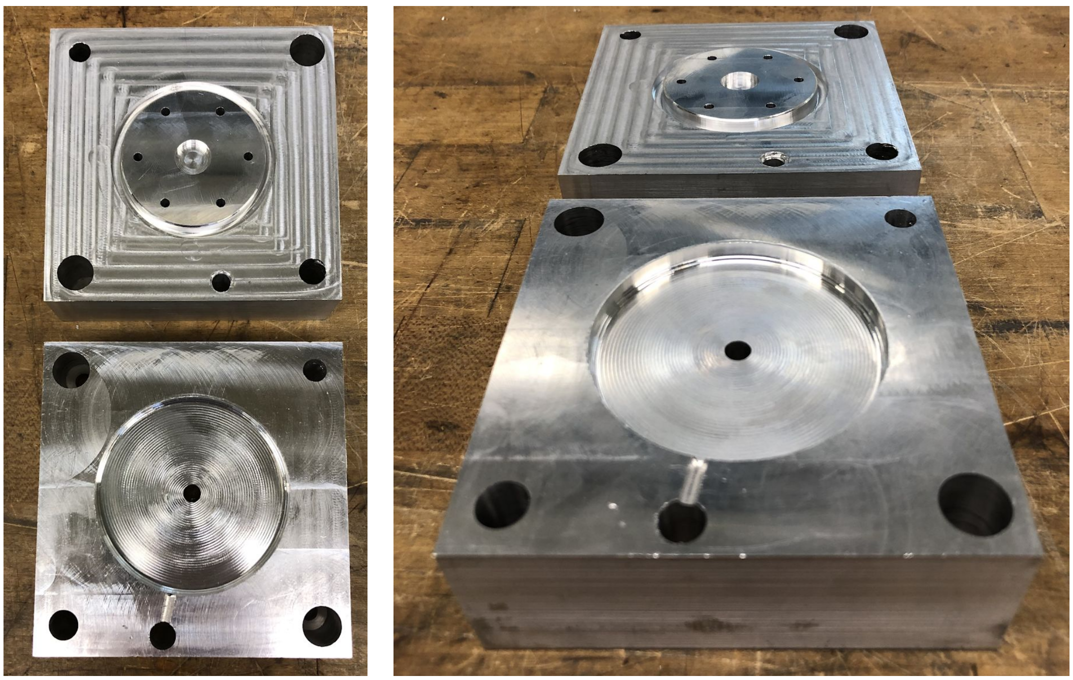

Figure 3: Successfully machined blue body mold core and cavity

B. Yellow Face

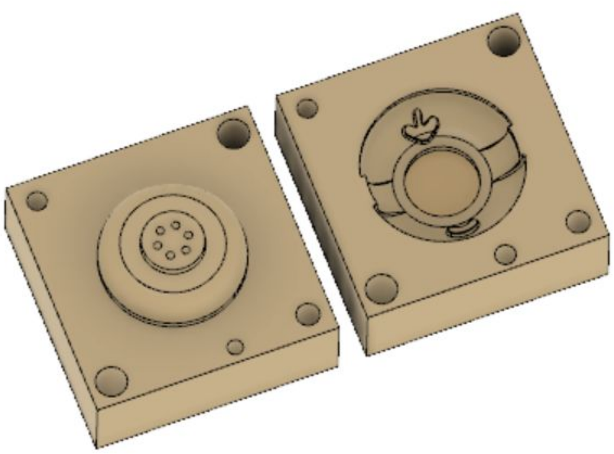

Figure 4: CAD model of yellow face mold

Amongst the injection molded parts in our yoyo design, the yellow face is a complex piece due to the small features of hair and mouth elements in the design. In terms of machining, the core side of the yellow mold was straightforward, and very similar in toolpath to the LMP yoyo mold, taking a total of 1 hour 21 minutes to machine. The cavity side, on the other hand, required a total of 6 hours and 14 minutes of machining. Due to the long machining time, it was necessary to run the first four toolpath operations, as illustrated in figure 5, on the ProtoTRAK mill, and leave the remaining finishing operations to be run overnight on the HAAS.

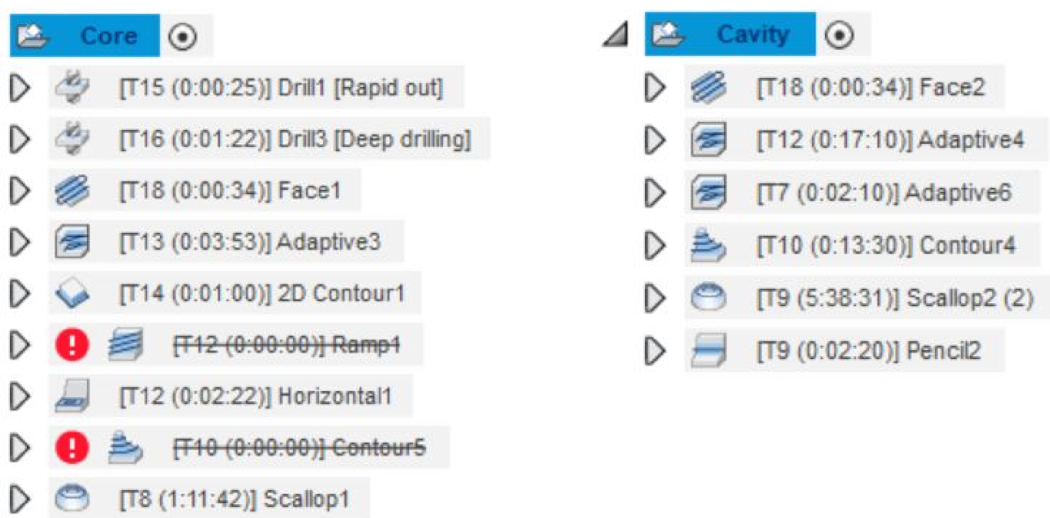

Figure 5: Toolpath operations for the yellow body core and cavity molds. The core toolpath begins with centre drilling followed by a deeper drilling operation for ejector pin holes. Both cavity and mold toolpaths include a facing operation of the stock top to create a smooth top surface. Facing is followed by adaptive passes to remove as much material possible. On the cavity mold, a contour operation is run to remove the majority of material on the hair and mouth features prior to transporting the mold to the HAAS. This contour operation is important to carry out in order to prevent the endmill from breaking on the HAAS. On the HAAS, the finishing scallop (0.002” stepover) and pencil toolpaths are used to reach material between small features, and create a smooth surface finish.

We chose to use 5000 RPM for the spindle speed for every operation because that is the maximum speed of the ProtoTRAK milling machine. The only exception is the spindle speed set for the scallop operation for the cavity mold, which was set to 7500 as it was carried out on the HAAS to speed up the operation. Since the machining time was relatively short for the core mold, we used the pre-set feed rate values for all of its operations. Again, for the scallop operation of the cavity mold, along with the spindle speed change, the feed per tooth was set to 0.00075 in-- much smaller than the default value, to get a better surface finish in a shorter time interval. In practice, we would often slow down the feed rate to 30-40% of the programmed value when just starting an operation to make sure the toolpath was doing what we expected it to do before ramping up to 100% of the programmed feed rate. We were able to machine two iterations of the yellow cavity mold. The first mold iteration did not result in favorable outcomes due to errors in machining and choice of tools. The order of operations on the first core mold was incorrect as the 5° drafting tool had to plunge into a pocket of material rather than simply drafting a vertical surface once all material was cleared. Additionally the toolpath created for the overnight HAAS operation on the cavity did not leave a smooth surface finish and messed up the parting surface of the part by removing material along the perimeter so the part was no longer circular or flush. In our second iteration, the mold was placed on the HAAS, however, after the job was initiated, the 1/16th endmill used for the scallop operation broke when trying to remove material from around the mouth and hair features. This was because endmill attempted to remove large chunks of material around the very narrow areas of the mouth and hair. We revisited our design, and introduced a contour operation on the ProtoTRAK preceding the scallop operation on the HAAS, which removes as much material possible around the two small features of the hair and mouth. Even though the surface finish is poor on our second attempt, the mold halves successfully fit together so we were able to try to injection mold.

Figure 6: Two attempts at the yellow face mold. Our first attempt on the left has order of operation errors on the core and toolpath errors on the cavity. Our

second attempt successfully made the yellow face core however the cavity side surface finish is poor due to the tool breaking during the overnight HAAS operation.

C. Goggle

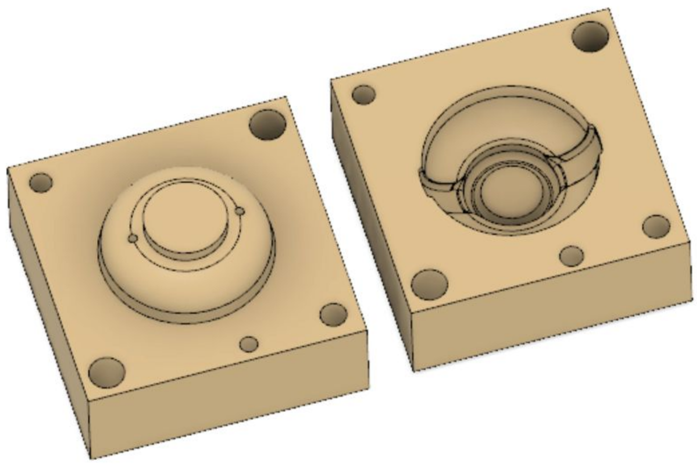

Figure 6: CAD model of goggle mold

The goggle mold went through many iterations before we were able to successfully machine both halves of the mold so that they fit together. The first iteration attempted to use traditional flat parting and shut off surfaces which caused the halves of the mold to wedge when they came together. If there was even a tiny amount of misalignment the mold halves would not fit together, causing us to pivot and pursue curved parting surfaces. In our second machining attempt the surface finish on the cavity side finished poorly despite having a 0.004” stepover. It appeared the adaptive roughing toolpath went too deep and removed too much material. After further investigation we discovered the tool height of the endmill we used for that roughing operation was off by 0.060”. In addition to this, the two halves of the mold did not fit together and there was a horizontal offset between the mold halves so that the sprue hole did not align. We tried to investigate this offset further however all critical dimensions matched the CAD model leading us to guess that we may have incorrectly zeroed the x/y coordinates of the machine or that the stock could have been slightly off. On our third attempt we were finally able to successfully machine both halves of the goggle mold using the lessons we learned in the earlier attempts including using a curved parting plane with a steepest angle of 10° which worked better than a drafted parting plane since the curved surface is symmetric. We also decreased the stepover on the finishing operations to 0.002” to ensure the parting surfaces would close completely to shut off all plastic.

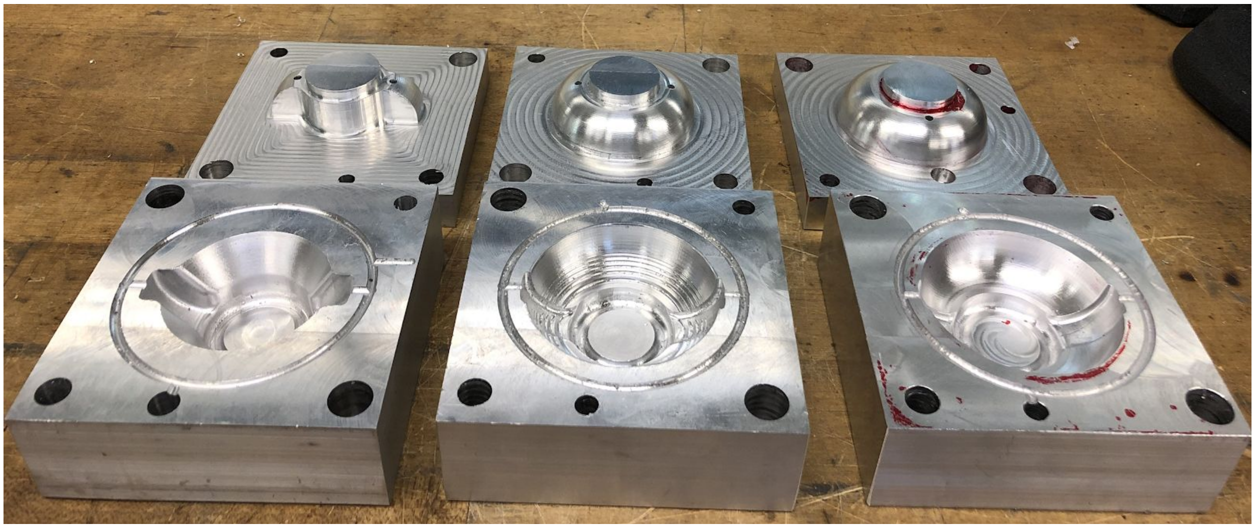

Figure 7: Furthest to the left is our first unsuccessful attempt at the goggle mold. The vertical shut off surface caused the mold halves to wedge together instead of closing fully. In the middle is our second unsuccessful attempt at the goggle mold. The surface finish on the cavity side is poor due to an incorrect tool height which caused the roughing operation to remove too much material. Furthest to the right is our successful goggle mold. There is a curved parting plane so the mold halves close together smoothly. Additionally, a 0.002” stepover was used on the finishing pass on both molds so that the shut off surfaces fit together tightly when the mold is closed.

The core side took 1 hour to machine and the cavity side took 1 hour 40 minutes to machine. The toolpaths for the mold halves were as follows:

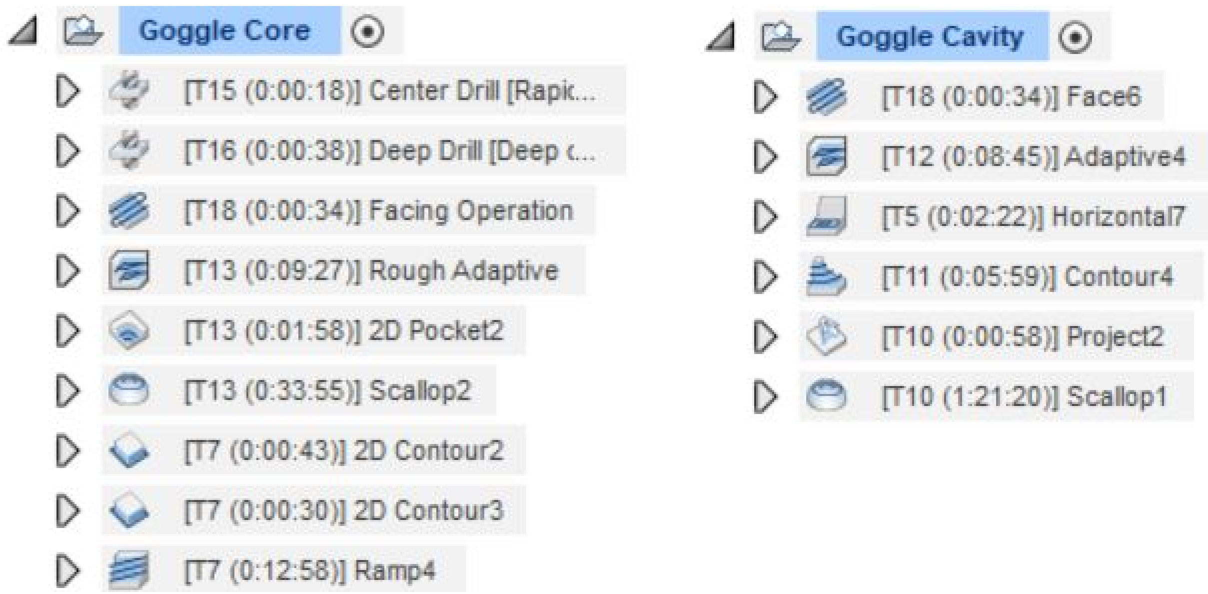

Figure 8: These are the toolpath operations for the goggle mold core and cavity. Both toolpaths follow similar frameworks with an initial facing operation in which 0.1” was cut off the top of stock in order to create flat parting plane. Then an adaptive clearing operation was used as a roughing pass to cut off the majority of the material. Lastly, various finishing operations were used to create a high quality smooth surface finish on all critical surfaces.

We used 5000 RPM for the spindle speed for every operation because that is the maximum speed of the milling machine. We also always used the largest tool that could fit into and cut the necessary areas in order to reduce machining time. For most operations in these toolpaths we used the pre-set feed rate values, however, we did significantly increase the feed rate of the scallop finishing pass on the goggle cavity because this operation was the rate limiting step and it was only removing 0.002” of material. We choose the scallop operation as the finishing pass because it creates passes at a constant distance from another by offsetting them inwards along the surface rather than only doing either a horizontal or vertical stepover. For this operation on the cavity a ⅛ inch ball end mill was used because it was the largest tool that could fit in all of the grooves and the cutting feed rate was changed from the default 15.3 in/min. to be 52.5 in/min. The feed rate was increased in order to reduce the time of this operation from 4.6 hours to 1.3 hours. The material removal rate (MRR) was relatively small due to a very small stepover of 0.002” chosen to create an extremely smooth surface finish. Since the MRR was small, the cutting force was small, hence, we were able to increase the feed rate. We also increased the feed rate of the ramp operation on the core side from 50 in/min to 70 in/min and we increased the scallop operation on the core side from 90 in/min to 120 in/min because these operations were similarly removing a small amount of material so the cutting force was small and allowed us to speed up the feed rate. Since we were not able to get the mold halves to fit together in our first two attempts, we used the negative stock to leave function in Fusion on the core side of the mold to ensure that the halves would fit together. We set it to -0.003” on the top, -0.001” on the curved surface, -0.003” on the ramp/draft. The ramp did not close until we added this setting. Lastly, on the adaptive clearing toolpaths on both the goggle core and cavity we reduced the tool load to 10% from the default value of 40% so we were not overloading the tool (on our second failed mold iteration we broke a tool during an adaptive clearing operation due to a combination of overloading the tool and a lack of coolant fluid).

D. G-Code

```
(1001)
(BLUE CAVITY)
(T10  D=0.125 CR=0.0625 - ZMIN=-0.3305 - BALL END MILL)
(T12  D=0.375 CR=0.03 - ZMIN=-0.471 - BULLNOSE END MILL)
(T15  D=0.1875 CR=0. TAPER=120DEG - ZMIN=-0.408 - CENTER DRILL)
(T17  D=0.234 CR=0. TAPER=118DEG - ZMIN=-1.8483 - DRILL)
(T18  D=2. CR=0. - ZMIN=-0.268 - FACE MILL)
N0010 G90 G94 G17    ] Setting up machine. Sets machine to absolute programming, feed rate
N0015 G20            ] units to  /min. feed, xy plane, and inch mode
(DRILL5)
N0020 M09 Coolant off
N0025 T15 M06   ] Tool change commands if mill had automatic tool change
N0030 T17       ]
N0035 S2000 M03 Spindle speed set to clockwise at 2000 RPM
N0040 G56   Set coordinate system
N0045 G00 X2.5387 Y1.0519   Rapid traverse to this (x,y) position
N0050 Z0.342 Rapid traverse to this z position
N0055 G00 Z-0.058 Rapid traverse to this z position
N0060 G98 G81 X2.5387 Y1.0519 Z-0.408 R-0.058 F8. Set drilling cycle, spot boring, at this (x,y) coordinate drill to this z depth
N0065 G80 Cancel canned cycle
N0070 Z0.342
(DRILL6)
N0075 M01
N0080 T17 M06
N0085 T18
```

Figure 9: Commented G-code from the toolpaths for the blue body cavity mold.

II. Injection Molding

All parts in this report were injection molded using the Engel EC 88 injection molding machine. The machine is rather finicky, and requires considerable patience to operate. However, setting up the machine is the most difficult and tedious part of using the Engel.

A. Blue Body

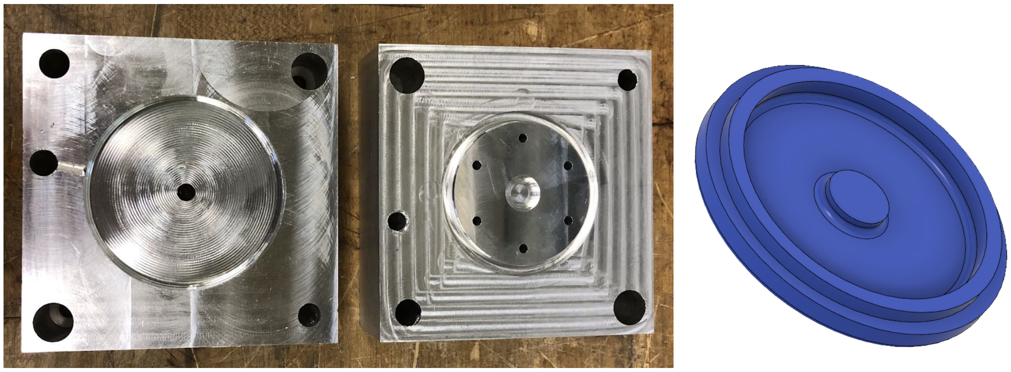

Figure 10: Side-by-side of machined mold for blue body and CAD model of blue body

The blue body was anticipated as the most simple part the team would have to injection mold. It is the most symmetric of the parts, so the thought was that there would be no “out of the ordinary” issues that came up with injection molding. This ended up being a correct prediction, but it ended up being easier because of unexpected reasons.

<u>Machine & Mold Set-Up</u>: Setting up the Engel for this part was relatively simple. Aside from molding parameters, there was not much deviation from the standard set-up required and detailed in the materials provided. However, fastening the molds to the parts that actually interface with the injection molding clamps presented a few issues. The problems came exclusively with ejectors pins, which were either too short or a bit too long. The pins that ended up being used were the 5.250” pins. These were a bit too short, so the parts that came out had non-negligible places where plastic filled ejector pin holes. There was also quite a bit of trouble getting the ejector pins to align with the mold itself, but we couldn’t tell if this was an issue with the holes or the ejector pins themselves.

<u>Gate Placement:</u> The gate just ran directly into the cavity side of the mold. A simple eyeball estimation is how the runner from the sprue into the mold itself was decided on, as the path chosen looks very close to the shortest path between the two areas. Since the part is largely symmetric, there were no features to avoid, or cosmetic surfaces to protect by placing the gate in a different location.

<u>Injection Molding Parameters:</u> We iterated on multiple configurations of molding parameters, but began with the parameters left on the machine from the last person to use the machine. All parts had a hold time of 8 seconds, and a cooling time of 10 seconds.

Figure 11: Attempts #1 and #2 at injection molding. Short shot and dishing occurred in both of these parts, due to shot size being too small (1.50in), and ejector pins that were too long (5.375").

Figure 12: Attempt #3. Increased shot size to 1.75in, and shortened ejector pins. There's less dishing now, but still short shot in the far side (from the mold) of the part

Figure 13: Attempts #4 - #8. Changes are just gradual increases in shot size up to 2.30in, and moving the injection boost pressure up to 1800 psi. There is a weldline that still exists at the far side of the part, but since flashing has occurred, we figured the next change must not have to do with the shot size.

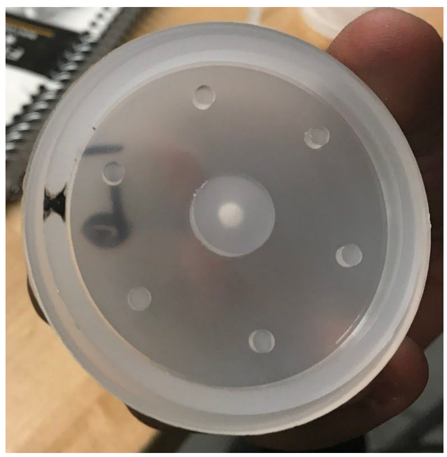

Figure 14: #9. Decrease injection speed profile rate.This allowed the injection speed to stay high for longer, which resulted in flashing and burning.

Figure 15: #10 & # 11. Increased injection boost pressure to 2300 psi, and reset the injection speed profile. No more burn, but weldline still present.

Figure 16: #12 - #15. Increased injection pressure profile from 1299 psi - 1650 psi to 1650 psi - 1800 psi. Ran this configuration four times to allow the machine to “catch-up.” Weldline and flashing still present.

<u>Drawbacks:</u> Obvious drawbacks and defects associated with this first run of blue body parts were the flashing and weldline that were created and left unsolved for the time being. Also, another thing that is not necessarily a “drawback” but a “grain-of-salt” observation, is the fact that we did not have any shoulder bolts with our molds. Obviously, this will be a necessary feature in our final parts, so the parameters detailed here may not hold for the final injection mold.

B. Yellow Face

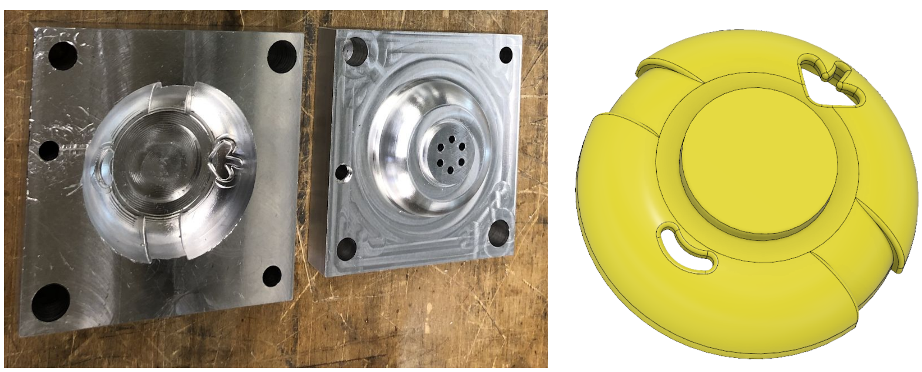

Figure 17: Side by side of machined mold for yellow face and CAD model of yellow face

<u>Machine & Mold Set-Up:</u> We needed to use ejector pins closest to 5.353” for the yellow face mold. We decided to use 5.375” ejector pins along with a 0.02” shim to get the ejector pins as close as possible to flush with the part. We reamed the ejector pin holes and did not have difficulty aligning the ejector pins.

Gate Placement: The runner and gate just ran directly into the cavity side of the mold from the sprue since the yellow face part is mostly symmetric and had no features or surfaces to avoid.

Injection Molding Parameters: For our first part we started with a shot size of 2.20" which corresponds to the volume of plastic being injected into the mold. This shot size turned out to be way too large and we has considerable flash on our first part. We then reduced the shot size to 1" which resulted in a short shot. For both of these first two parts we had the hold time set to 10 seconds and the cooling time set to 20 seconds. For the third part we increased the shot size to 1.2" and reduced the cooling time to 10 seconds because the part is relatively thin. This shot size also resulted in a short shot. For our last attempt we increased the shot size to 1.4" which resulted in flash. Moving forward we now know that the holding time of 10 seconds and cooling time of 10 seconds work well and that the shot size will be between 1.2" and 1.4".

Figure 18: Our four attempts at injection molding the yellow face part. The first and fourth attempt shot sizes resulted in flash whereas the second and third attempts resulted in short shots. We now know our shot size is between 1.2" and 1.4". We had to stop injection molding because the sprue continually got stuck in the cavity side of the mold which forced us to take out the mold and struggle to pry it out.

Drawbacks: With the yellow face, the major drawback was the failure of the part to be ejected from the mold. This may be due to the short depth of the sprue hole on the core side of the mold. There may not be enough depth for the plastic to engage with the threads in that hole, so the plastic continually stuck in the cavity side sprue hole. Another problem we saw was that there was a small undercut created by the misalignment of the cavity sprue hole and the hole on the backing plate that interfaces with the clamp. There was quite a bit of overlap between the diameters of those two holes, creating an undercut between the two parts fastened together, preventing the plastic from releasing from the cavity side of the mold.

C. Goggle

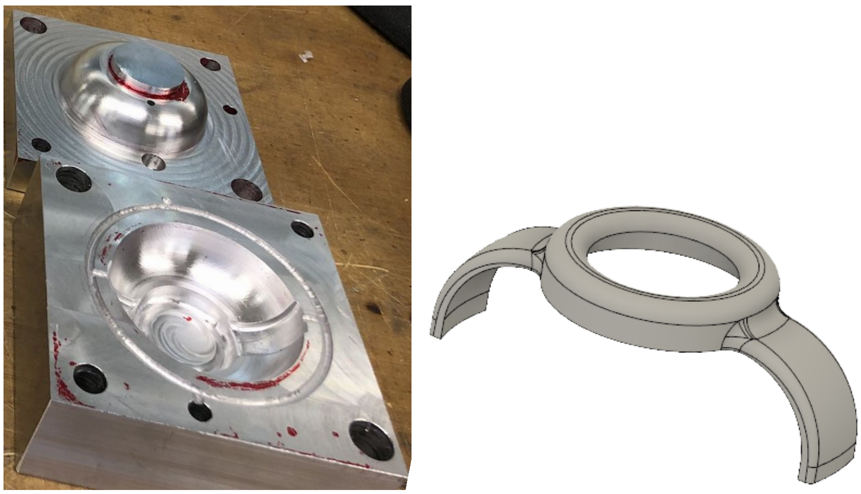

Figure 19: Side-by-side of machined mold for gray goggle and CAD model of gray goggle

Injection molding the goggle piece presented some unexpected challenges. We will cover the injection molding parameters and changes that may need to occur first, and then consider other large hiccups that severely hindered the process of injection molding.

Figure 20: Attempts at injection molding the goggle. Major flashing occurring in the 2nd iteration (left, 1.25 in shot size). Excess plastic removed from 1st iteration (right, 1.50 in shot size)

<u>Machine & Mold Set-Up</u>: Setting up the Engel for this part presented an issue with the mold height. After zeroing the clamp position and finishing the setup for molding, I tried to mold the part by closing the front gate door. However, once I did this, the clamp began to close, but stopped short of meeting flush with the cavity side of the mold. This happened twice in a row. The error message was cleared, and then we simply re-zeroed the clamp position. The clamp closed fully on the third try, but it remains unclear what the reason was for the clamp stopping short of flush with the cavity side of the mold. The ejector pins used for this part were also the 5.250” pins, and they were much easier to align than that of the blue mold.

<u>Gate Placement</u>: The gate runs from the sprue into a runner that traverses around the outer circumference of the goggle mold cavity. This allows both straps at the sides of the goggle to fill first and have plastic meet in the middle. This allows the gate marks to be at the ends of the straps near the base of the yoyo half, minimizing adverse aesthetic effects.

<u>Injection Molding Parameters:</u> Unfortunately, due to ejection mishaps, we were only able to realize 2 goggle parts, where we varied shot size from 1.50in to 1.25in. The other parameters used in injection molding this part were very similar to those used in injection molding the blue part. The injection speed and pressure profiles were identical to that of the blue base. The hold time and cooling time, however, increased from 8 seconds and 10 seconds, respectively, to 10 and 12 seconds. This was not by design, but simply due to the last person to use the machine using the settings. These numbers, especially the cooling time, can definitely decrease. This part will use less plastic than the blue piece, so decreasing the cooling time will increase our production rate. Again, major flashing still occurred in the 1.25 in shot size configuration, but the issues with

plastic getting stuck in the machine made it extremely time intensive to iterate through the different shot sizes and other parameters.

Drawbacks: The major flashing is the most noticeable defect in the goggle part that must be addressed in future iterative processes with the injection molding parameters. However, one of the major features hindering our progress was the failure of the sprue plastic to eject with the rest of the part. When the molds are both clear, the part fills and ejects perfectly fine, but the plastic in the mold cavity sprue hole stays behind. As a result, any molding you try to do immediately after this fails due to plastic not being able to fill the mold because of solid plastic stuck in the sprue hole. Initial hypotheses suggest that it may either be undercutting between the mold and the mold backing part like in the yellow face part. Alternatively, it can be plastic not fully engaging with threads in the core side of the mold, or the sprue ejector pin being too long for the core sprue hole. There was a lot of aluminum cleared from the core side of the mold, so maybe there isn’t enough depth in the hole before the ejector pin fills it.