Injection Molding Project Report

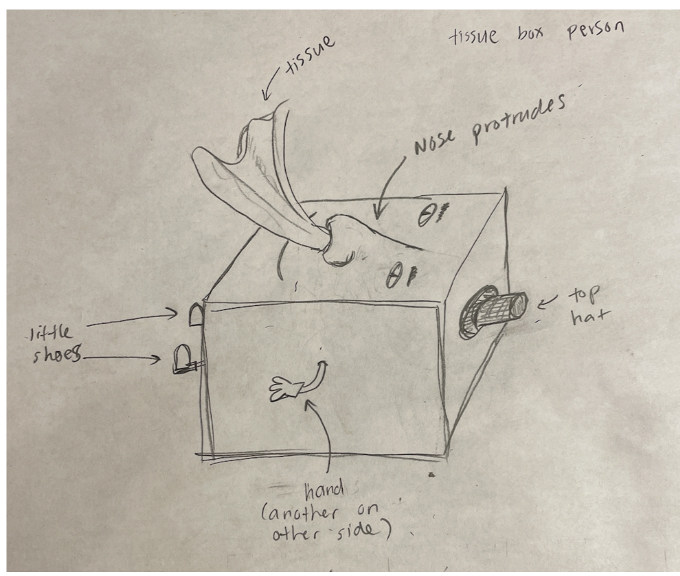
Ana Cuartas, Hannah Huang, Jonathan Li, Derek Wu
3/16/2022
Mechanical Engineering 340-2

Introduction

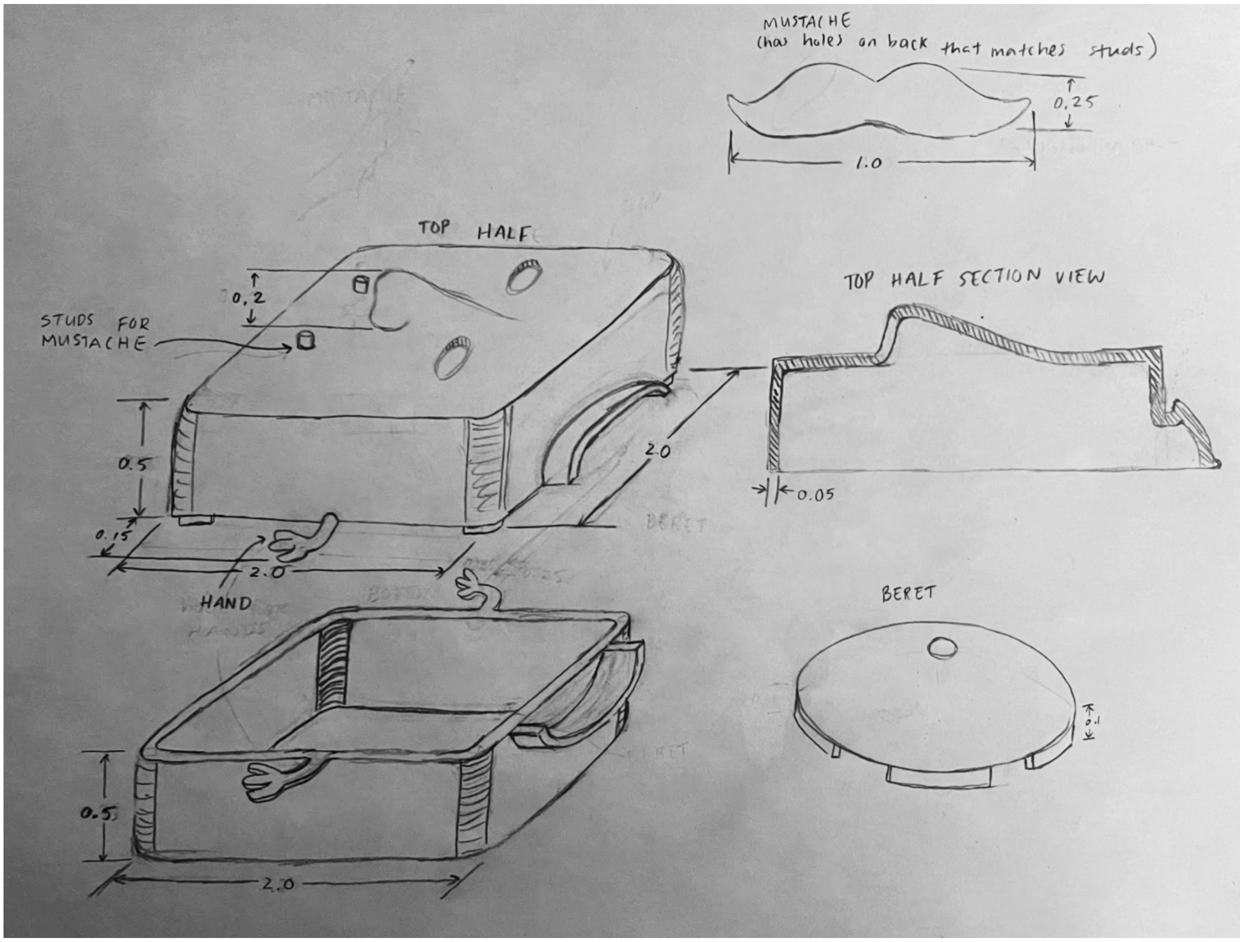
Our project involves computer-aided design, CAM programming, and product development. Our team developed the product, “Box Man”, that involves an assembly of 4 molded plastic parts, the top half of the body, bottom half of the body, mustache, and a beret. This roughly has a 2”x2” wide body that is 1” tall. The development of this product entailed designing a moldable part, which involved a brainstorming session that led to our final selection of the “Box Man” sketch. Next, we selected an assembly method, for which we chose force fits and pin fits. After this came tooling design and fabrication, which will be explained in the review of the mold manufacturing section below, and production of sample parts using the plastic injection molding process, which will be explained in the metrology study section below.

To make our product, we created a solid model of each part, translated those into a mold, and generated a manufacturing program using NX’s CAM software. The NX’s CAM software helps us visualize and program individual operations we would use to manufacture our cavity and core molds. After our program was approved, we manufactured our molds using a CNC mill to create two sets for injection molding. Following injection molding, we measured the dimensions of the mold cavities and injection molded parts. Finally, we assembled the parts into a finished product.

(a)



(b)



(c)

Figure 1: Initial ideas of (a) tissue box man and (b) painter jellyfish. Final idea of (c) a box man wearing a beret.

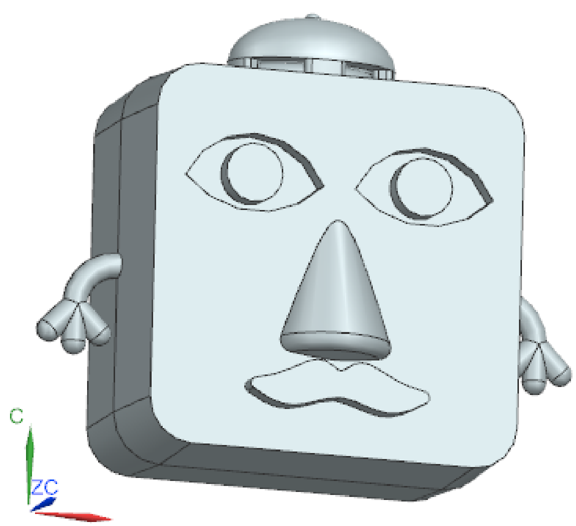


Figure 2: Final assembly.

Review of Part Design

To allow for easier removal of the part from the mold, a draft of 3° was added to both the top and bottom body parts along all the walls. In doing so, the body would require less continuous force to remove from the mold as when compared to having no draft. A wall thickness of 0.1” was chosen, given that the material used was polyethylene and an ideal thickness for this material is between 0.030” and 0.125”. The part was designed with uniform thickness to promote even cooling and reduce shrinkage or other defects. If uniform cooling does not occur, then outer sections will freeze first while the inner sections are still melted. This will cause the outer section to get pulled in as the inner section cools and shrinks, resulting in a deformed part. The residual stress from this phenomenon can also cause warping as the part cools unevenly. A fillet was added to all external sharp corners in the parts, as they would require interior sharp corners in the molds which are impossible to machine. The only sharp corners left were ones along the parting line which can be machined.

A pin-hole interference fit was used to attach the mustache to the face, given the limitations of machining the mold and the location of the mustache. These pins were made to have 1/16” diameters so that machining pins could be used in the molds since machining out pins that small is extremely unreliable. A unique geometry was used on the bottom body part so that it would secure to the top body with an interference fit. The bottom body has 4 L-shaped protruding columns that have slight interference with the interior of the top body so that the parts are fixed during assembly. In this way, no pins or holes were required to assemble these parts so no machining pins would need to be used. To attach the beret to the body of the part, a perimeter interference fit was used, with 4 tabs securing the parts instead of the entire perimeter, to allow for easier assembly and to reduce material usage.

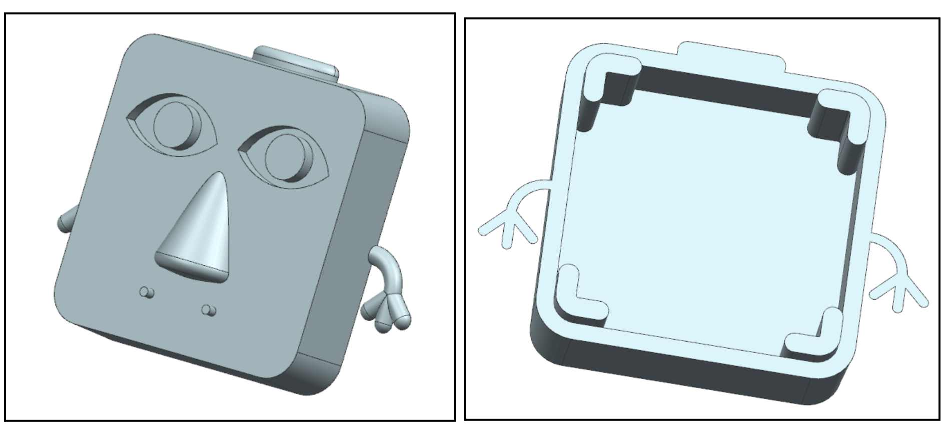


Figure 3: CAD model of top body (left) showcasing pins for mustache attachment and bottom body (right) showcasing L-columns for assembly.

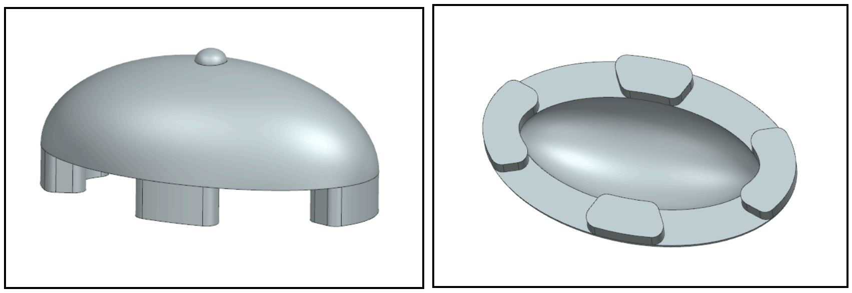


Figure 4: CAD model of beret with right showcasing the 4 tabs for force assembly.

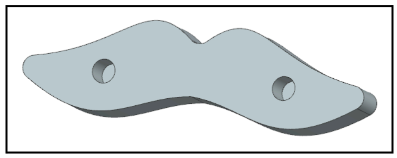


Figure 5: CAD model of mustache showcasing pin holes used for attachment.

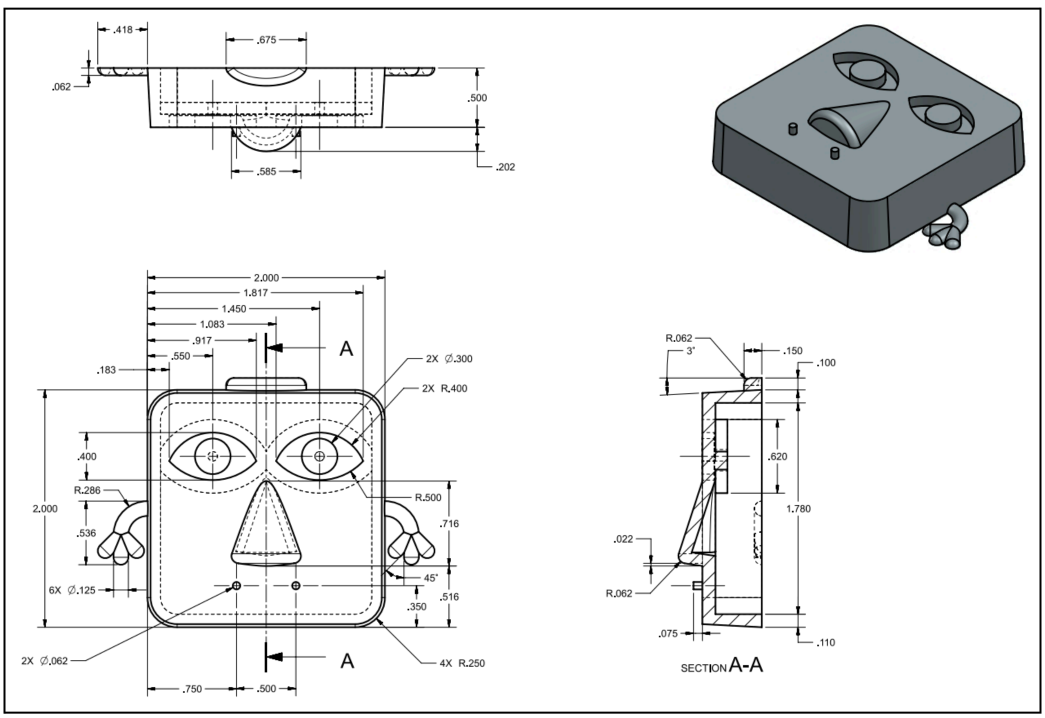


Figure 6: Top-half body part drawing.

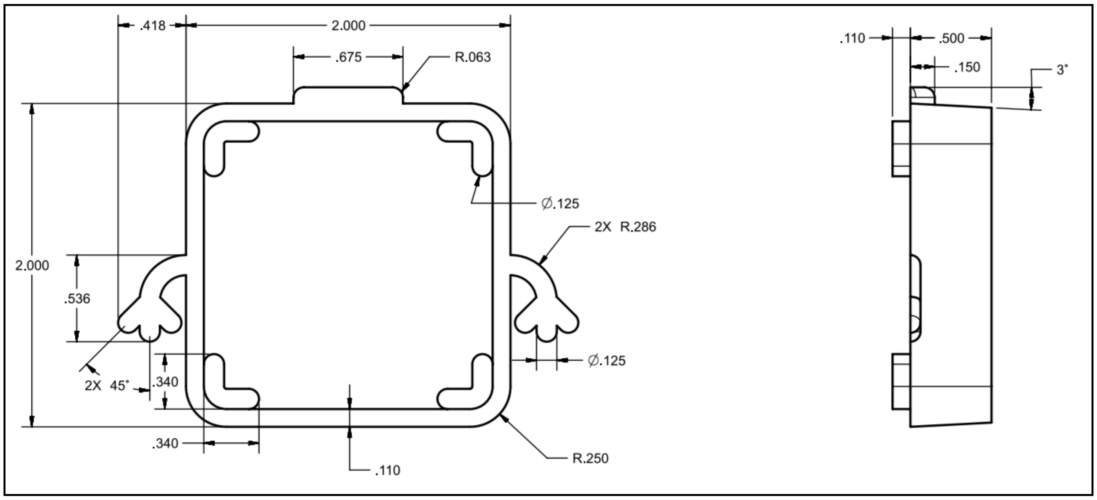


Figure 7: Bottom-half body part drawing.

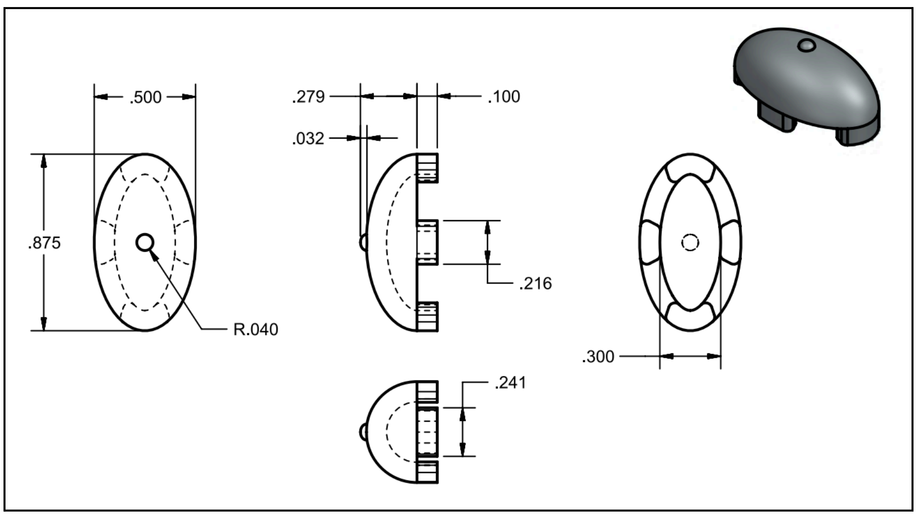


Figure 8: Beret part drawing.

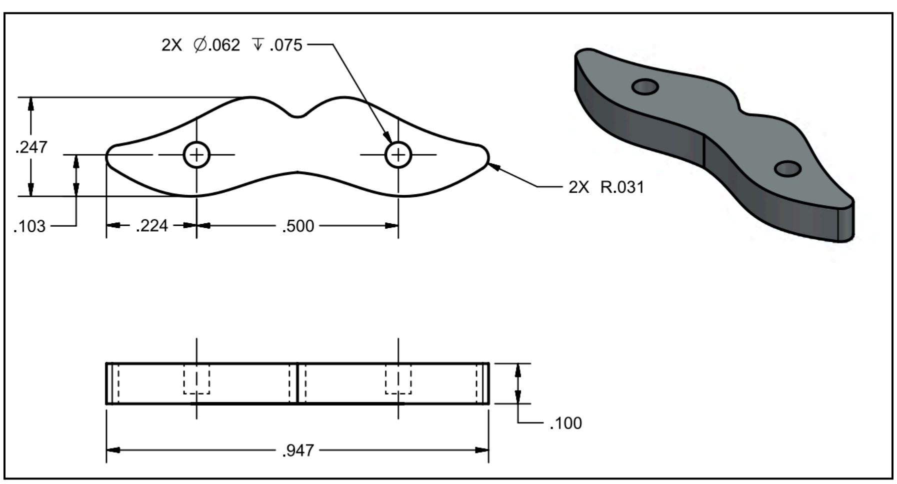


Figure 9: Mustache part drawing.

Review of Mold Design

The location of the parting line was chosen so as to prevent or minimize its visualization on the actual part. This meant having the parting lines along the edges of the parts and allowing the shape to hide this imperfection from injection molding. Parting lines form when the mold blocks used for injection molding are not perfectly aligned or when there is a slight gap between the molds. Therefore, it is extremely difficult to prevent this imperfection, so it is ideal to design around this knowledge. The parts were oriented such that no deep, narrow cavities were present in the molds which would be difficult to machine and make the part nearly impossible to remove after injection molding. Instead, the parts were deliberately positioned to create a cavity and core for each of the molds. Since it was assumed that all parts would shrink proportionally due to the uniform thickness, the parts were left in their ideal sizes which were expected to still fit together after shrinking, with the parts just being scaled down in size. Certain features of the part were designed knowing that only standard tooling diameters and lengths were available. For example, the hands were made large enough to allow a 1/16” diameter ball mill to be used, and the beret was made wide enough to allow the button on it to be milled, given its depth.

Due to our assembly having 4 parts, we arranged it so that the top body and mustache were on one mold, and the bottom body and beret were on the other mold. This was done to ensure that all parts could fit on the molds, given we were only allowed two molds. For the mold with the mustache, the runner to the top body was made much shorter than the runner to the mustache since the mustache would fill significantly faster due to it being a smaller part. A similar methodology was applied to the mold with the beret since the beret is significantly smaller than the bottom body and would fill much faster. This approach was implemented to increase the assurance that all parts of the mold finish filling at the same time, which will help to prevent flash or the part coming out as short-shot. Additionally, the runners were designed so that the placement of the gates would be against a wall to prevent jetting and into the thicker areas of the part so that certain areas do not cool before others. Gates help to direct and restrict the flow of plastic so that the part is filled completely and efficiently.

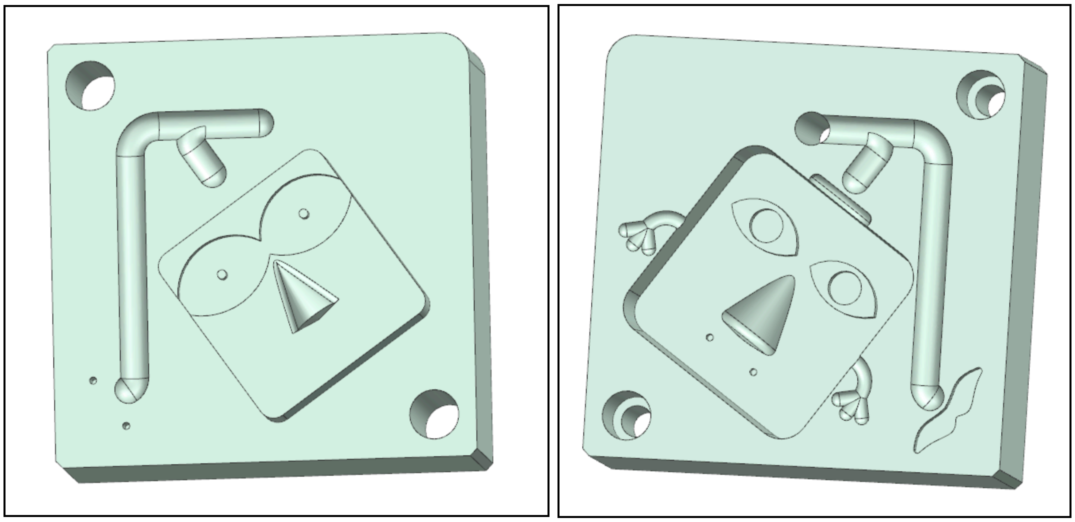


Figure 10: CAD model of molds for top core (left) and top cavity (right) showcasing runner designs and fillets.

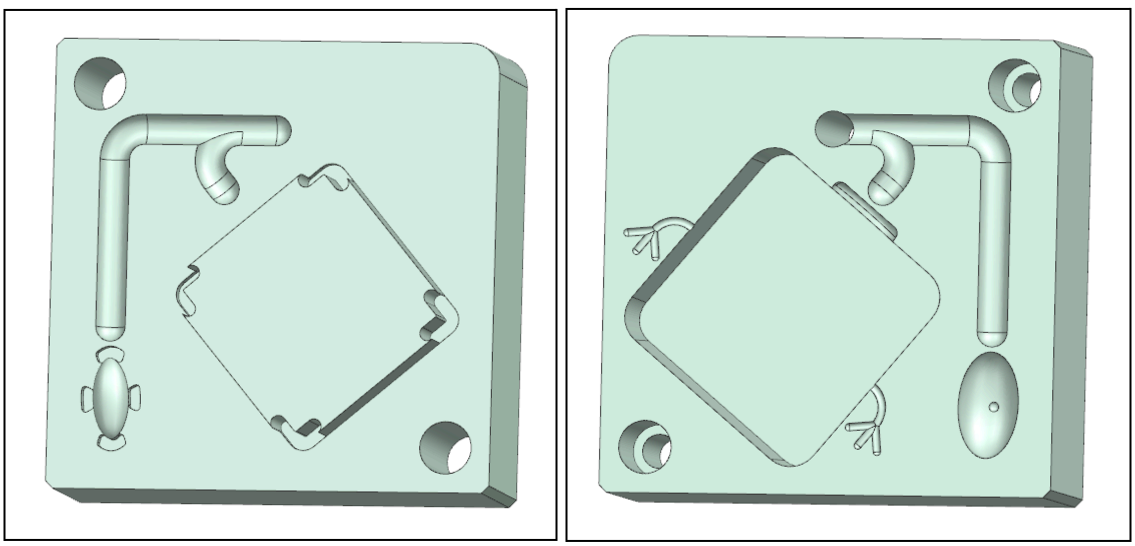


Figure 11: CAD model of molds for bottom core (left) and bottom cavity (right) showcasing runner designs.

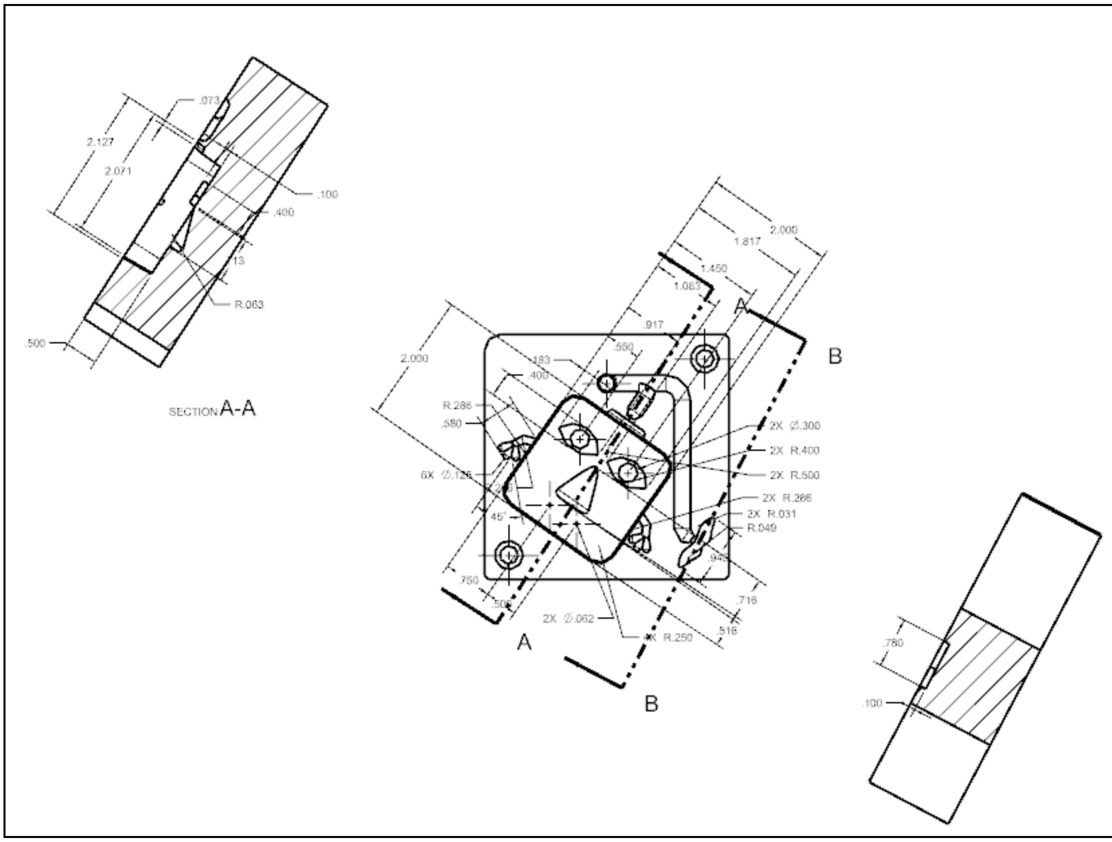


Figure 12: Top Body Cavity Mold Drawing.

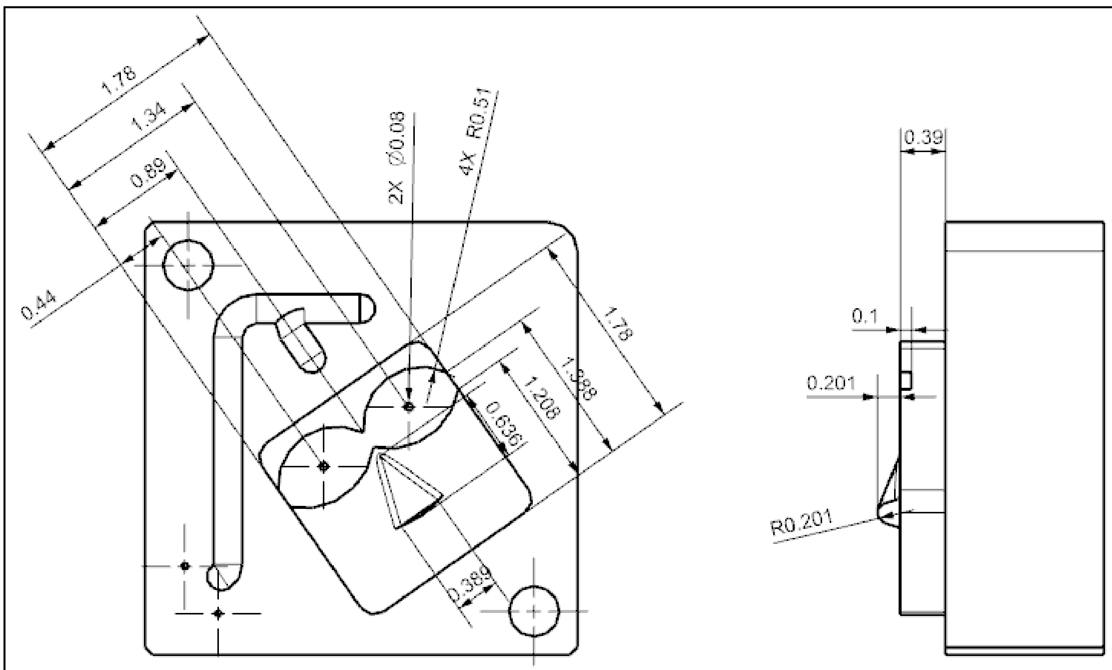


Figure 13: Top Body Core Mold Drawing.

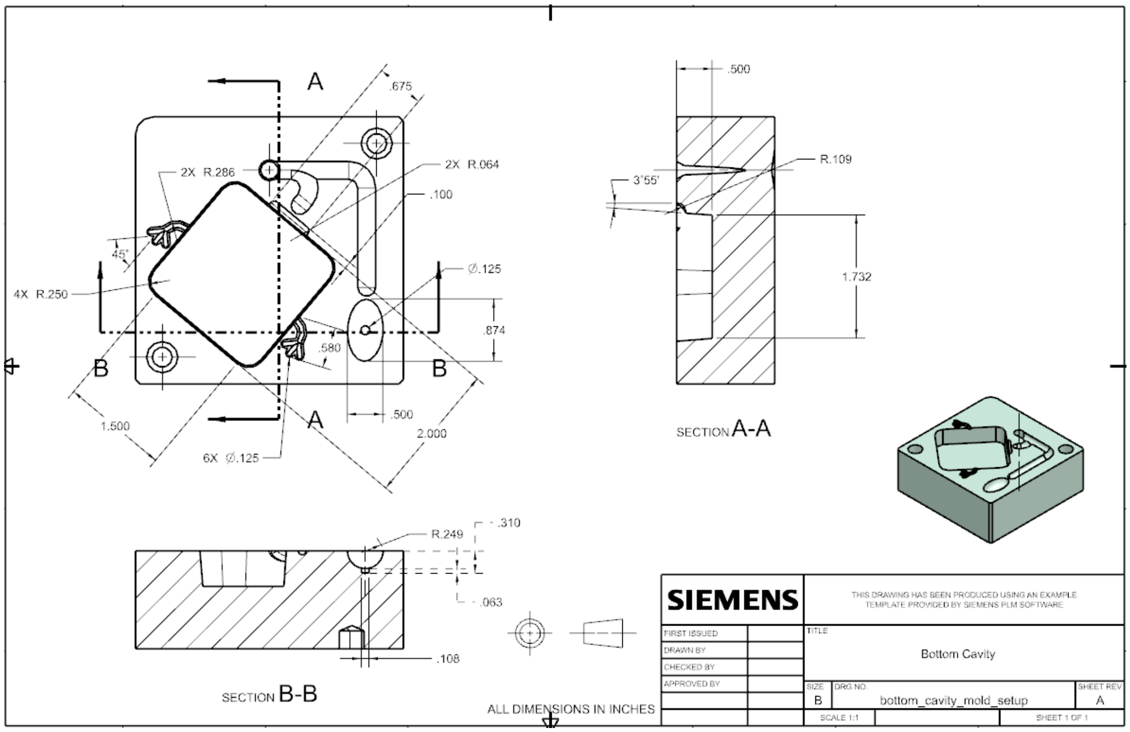


Figure 14: Bottom Body Cavity Mold Drawing.

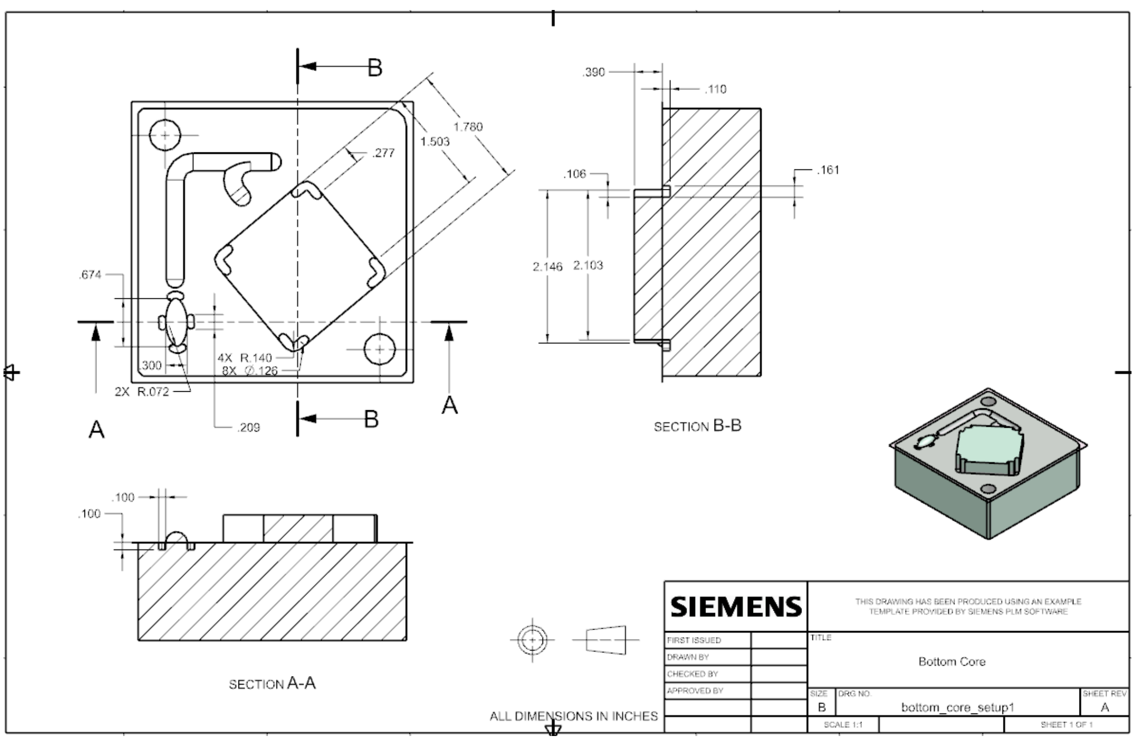


Figure 15: Bottom Body Core Mold Drawing.

Review of Mold Manufacturing (CAM)

Figure 16: Haas Super Mini Mill

To mill our molds we used a Haas Super Mini Mill as shown in Figure 16, where we milled 4 times for two cavities and two cores. In Tables 1-4 below, we outlined the various operations along with our reasonings for each mold.

CAM Operations

Table 1: Top Cavity

Operation	Tool	Method	Depth of Cut	Stepover	Feed	Speed	Reasoning
Roughing 1	0.5 EM	ROUGH	0.1667 in	33% Flat	16 ipm	3056 rpm	To remove materials from the Tissue Box cavity as quick as possible
Finishing 1	0.5 EM	FINISH	0.1667 in	33% Flat	16 ipm	3056 rpm	
Roughing 2	0.25 EM	ROUGH	0.0833 in	33% Flat	12 ipm	6112 rpm	
Finishing 2	0.25 EM	FINISH	0.0833 in	33% Flat	12 ipm	6112 rpm	
Roughing 3	0.125 EM	ROUGH	0.0417 in	33% Flat	6 ipm	10000 rpm	The 0.125” EM removes materials

Finishing 3	0.125 EM	FINISH	0.0417 in	33% Flat	6 ipm	10000 rpm	from the eyes, top of the head, and the mustache
Finishing Taper	0.125 EM TAPER	FINISH	0.0417 in	33% Flat	6 ipm	10000 rpm	Our Tissue Box is tapered 3 degrees, so we used a 0.125” EM Taper to clean off the sides of the cavity
Area Mill	0.125 BM	FINISH	10% Tool	10% Flat	6 ipm	10000 rpm	To finish off the top of the head
Finishing 4	0.0625 EM	FINISH	0.0208 in	33% Flat	2 ipm	10000 rpm	To finish off the mustache cavity
Runner Roughing	0.125 BM	ROUGH	0.0417 in	33% Flat	10 ipm	1253 rpm	To create the runners that are used for injection molding
Runner Finish	0.25 BM	FINISH	0.0000 in	33% Flat	12 ipm	6112 rpm	
Spot Drilling	CENTERDRILL	DRILL	—	—	7 ipm	3000 rpm	To create the holes where we would place the mustache into
Drilling	0.0625 STD DRILL	DRILL	—	—	7 ipm	3000 rpm	
Hand Finishing	0.125 BM	FINISH	0.0000 in	33% Flat	6 ipm	10000 rpm	To create the hand cavities of the Tissue Box
Nose Roughing	0.125 BM	ROUGH	10% Tool	10% Tool	10 ipm	1253 rpm	To create the nose cavities of the Tissue Box
Nose Finishing	0.125 BM	FINISH	10% Tool	10% Tool	10 ipm	1253 rpm	

Table 2: Top Core

Operation	Tool	Method	Depth of Cut	Stepover	Feed	Speed	Reasoning
ROUGHING 1	0.75 EM	ROUGH	0.3 in	20% Flat	18 ipm	2037 rpm	To remove

ROUGHING 2	0.5 EM	ROUGH	0.167 in	33% Flat	16 ipm	3056 rpm	materials from the blank as quick as possible
FINISHING 2	0.5 EM	FINISH	0.167 in	33% Flat	16 ipm	3056 rpm	
ROUGHING 3	0.125 EM	ROUGH	0.0417 in	33% Flat	6 ipm	10000 rpm	To remove materials from the Tissue Box's eyes
FINISHING 3	0.125 EM	FINISH	0.0417 in	33% Flat	6 ipm	10000 rpm	
RUNNER ROUGHING	0.125 BM	ROUGH	0.0417 in	33% Flat	6 ipm	10000 rpm	To create the runners that are used for injection molding
RUNNER FINISHING	0.25 BM	FINISH	0.0000 in	33% Flat	12.0649 ipm	6112 rpm	
AREA MILL	0.125 BM	FINISH	10% Tool	10% Flat	6 ipm	10000 rpm	To finish the nose of the Tissue Box
SPOT DRILLING	CENTERDRILL	DRILL	—	33% Flat	7 ipm	3000 rpm	To create the holes that will be used to place 1/16 pins. The pins are used to create the holes of the mustache
DRILLING	STD DRILL	DRILL	—	33% Flat	7 ipm	3000 rpm	

Table 3: Bottom Cavity

Operation	Tool	Method	Depth of Cut	Stepover	Feed	Speed	Reasoning
ROUGHING 1	0.75 EM	ROUGH	0.25 IN	33% Flat	16.296 ipm	2037 rpm	To remove materials from the Tissue Box cavity as quick as possible
FINISHING 1	0.25 EM 3 DEG TAPER	FINISH	0.16 in	33% Flat	11.0016 ipm	6112 rpm	
ROUGHING 2	0.25 BM	ROUGH	0.04 in	33% Flat	11.0016 ipm	6112 rpm	To remove material from the beret cavity
FINISHING 2	0.25 BM	FINISH	0.04 in	33% Flat	11.0016 ipm	6112 rpm	
RUNNER ROUGH	0.125 BM	ROUGH	0.04 in	33% Flat	6 ipm	9800 rpm	To create the runners that are

RUNNER FINISH	0.25 BM	FINISH	0.00 in	33% Flat	11.0016 ipm	6112 rpm	used for injection molding
TOP HEAD ROUGHING	0.125 BM	ROUGH	0.04 in	33% Flat	6 ipm	9800 rpm	To create the cavity of the top of the head
TOP HEAD FINISHING	0.125 BM	FINISH	0.0005 Scallop	33% Flat	6 ipm	9800 rpm	
TOP HEAD FINISHING 2	0.125 BM	FINISH	10 % Tool	10% Flat	6 ipm	9800 rpm	
BERET FINISHING	0.125 BM	FINISH	10 % Tool	10% Flat	6 ipm	9800 rpm	To finish the beret cavity
SPOT DRILLING	CENTERDRILL	DRILL	—	—	9.8425 ipm	0 rpm	To create our little hole feature of the beret
DRILLING	0.125 BM	DRILL	—	—	6 ipm	9800 rpm	
HANDS	0.125 BM	FINISH	0	33% Flat	2 ipm	9800 rpm	To create the hand cavities of the Tissue Box

Table 4: Bottom Core

Operation	Tool	Method	Depth of Cut	Stepover	Feed	Speed	Reasoning
ROUGHING 1	0.75 EM	ROUGH	0.15 in	33% Flat	16.296 ipm	2037 rpm	To remove materials from the blank as quick as possible
FINISHING 1	0.75 EM	FINISH	0.15 in	33% Flat	16.296 ipm	2037 rpm	
RUNNER ROUGHING	0.125 BM	ROUGH	0.04 in	33% Flat	5.9952 ipm	9800 rpm	To create the runners that are used for injection molding
RUNNER FINISHING	0.25 BM	FINISH	0	33% Flat	11.0016 ipm	6112 rpm	
BERET ROUGHING	0.25 BM	ROUGH	10% Tool	10% Flat	11.0016 ipm	6112 rpm	To create the beret feature
BERET FINISHING EM	0.125 EM	FINISH	10% Tool	10% Flat	5.9952 ipm	9800 rpm	

BERET FINISHING BM	0.125 BM	FINISH	10% Tool	0.001 Scallop	5.9952 ipm	9800 rpm	
CAVITY MILL	0.125 EM	FINISH	0.04 in	33% Flat	5.88 ipm	9800 rpm	To mill out the inner pockets of the Tissue Box
POCKETING	0.0625 EM	FINISH	0.02	33% Flat	2.0046 ipm	9800 rpm	To mill out the outer pockets of the beret feature

Notice that the runner roughing in each case has to use a ⅛” ball mill first before finishing with a ¼” ball mill. This is because the runner is ¼” in diameter, so the ¼” ball mill would perfectly fit in the runner and not be able to leave extra material for a roughing pass. Therefore, a smaller ball mill is used to rough it first.

CAM Feedback and Results

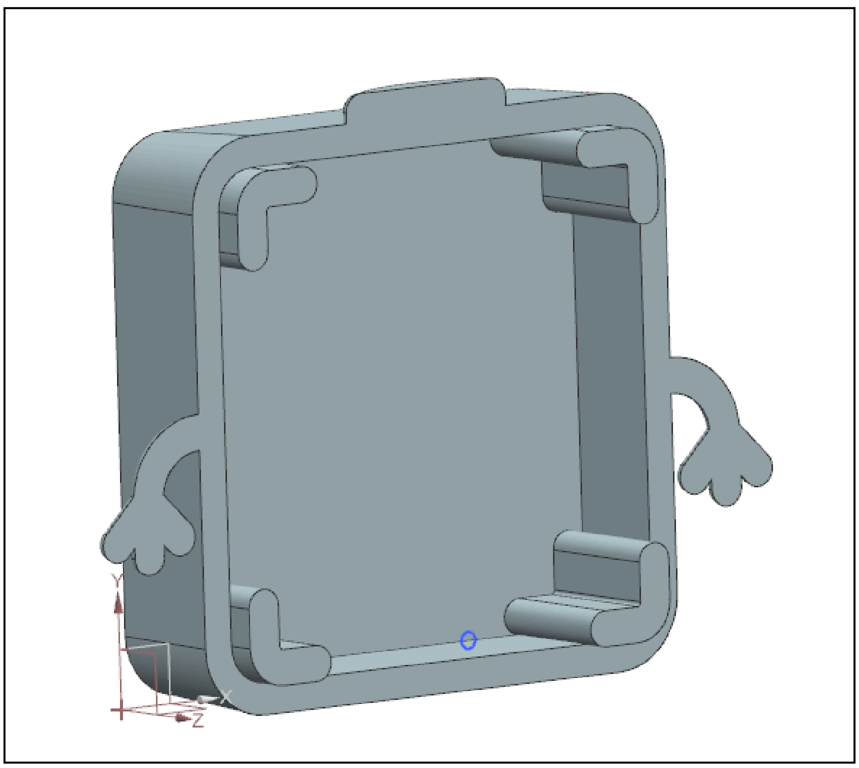


Figure 17: Body Bottom with edge fits.

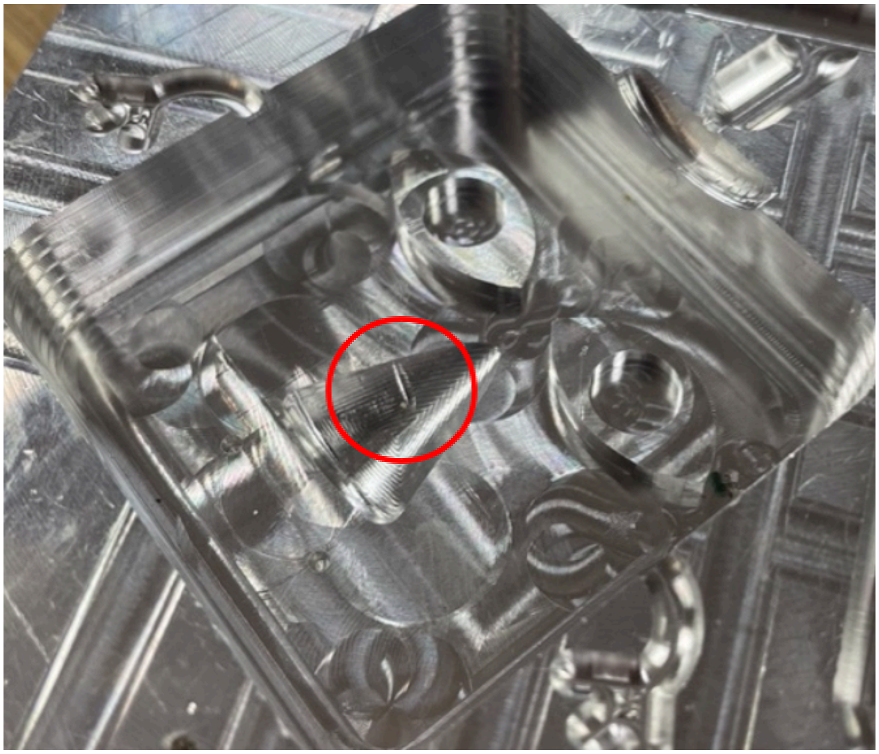


Figure 18: Scratched nose in Top Cavity caused by tool breaking.

Overall, the CAM process work went pretty smoothly. However, we did encounter some issues along the way. When we were milling the Bottom Cavity, a ⅝" end mill was accidentally used instead of a ½" end mill. Another foolish mistake we made was when we reoriented the blank incorrectly for the Top Cavity. For these blunders, we had to re-machine the mold blanks. Outside of these mistakes, we also scratched the nose feature's surface finish in our Top Cavity when the ⅛" ball mill broke as shown in Figure 18. Therefore, in the future to prevent this from happening, we would reduce the feed speed and tool stick-out. Moreover, we also faced issues with the holes we drilled for the pin because they were too small to fit. In response, we simply drilled larger holes to fit the pins and used JB-weld to secure them in the molds. Other than the modifications made to the pin holes, our mold had great surface finish. In regards to future part modifications, after injection molding we realized we should've included drafts on the inner edges of the Tissue Box (Figure 17). Adding drafts to these edges, would help with removing the injection molded part from the mold.

Metrology Study

Our chosen measurement for our metrology study is the height of the top-half of the body (see Figure 19). We used calipers to determine this height, consistently measuring the same corner that is shown in Figure 20.

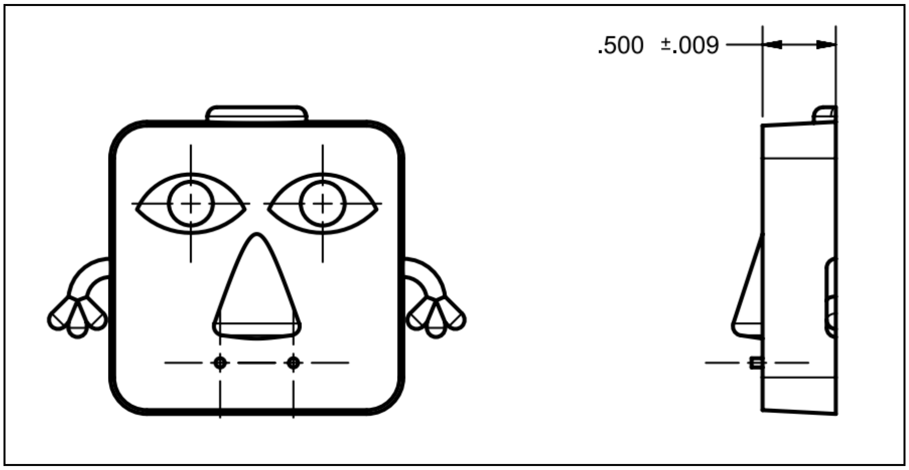


Figure 19: Selected dimension for metrology.

Figure 20: Measuring the height of the top-half body.

Table 5: Production Measurements of Top-Half Body Heights

Trial Number	**Top-half body height (in)**	**Trial Number**	**Top-half body height (in)**
1	0.489	11	0.486
2	0.489	12	0.490
3	0.488	13	0.485
4	0.491	14	0.490
5	0.491	15	0.486

6	0.492	16	0.488
7	0.489	17	0.491
8	0.489	18	0.493
9	0.490	19	0.487
10	0.488	20	0.489
		Average	0.489

We can choose a max angular deviation of $\pm\ 1°$ to ensure we are maintaining at least a 2° draft on the sides but no more than 4° (which are designed to be 3°)

$$tolerance\ =\ 0.500''\sin(1°)\ =\ 0.009''$$

Thus, we set the specified dimension to $0.500''\ \pm\ 0.009''$. The upper and lower specified limits are thus 0.491" and 0.509" respectively.

From Table 5, we can calculate the average $\bar{x}$ to be 0.489. We can also calculate the standard deviation σ:

$$\sigma\ =\sqrt{\frac{1}{n-1}\sum_{i=1}^{n}(x_i-\bar{x})^2}\ =\ 0.002''$$

Because tolerances are for k=3 (99.7% confidence), the upper and lower tolerances are $\pm\ 3\sigma$. Thus, the upper and lower production limits are $0.489''\ \pm\ 0.006''$. The upper and lower control limits are thus 0.483" and 0.495" respectively.

Overall, we can see that our average production height is smaller than our specified target by 0.011". This is a pretty significant difference, and causes our upper and lower production limits to fall outside of the specified limits. However, the production standard deviation is smaller than the specified standard deviation, meaning that our production has good precision but not accuracy.

The average production dimension is likely much smaller than the specified dimension because of the overall shrinkage that happens as the part cools. The deviation is likely small because we let the parts cool around 2 minutes between each run, giving better dimensional stability. Additionally, we made sure to find good parameters that yielded consistent results first, before beginning our sample collection. To achieve our desired measurements, we could design our mold to be slightly larger and account for the shrinkage that occurs.

Conclusions & Suggested Modifications

We are very pleased with our finished products, and looking back on the work that led to this point, we are proud of the steps that we took in developing it and the valuable skills we refined during the process. Going through an entire manufacturing process, from part and mold design, to mold manufacturing, to injection molding, and finally to our metrology study, we learned how tedious the process of product development can be, and how much patience, focus, and agile learning it requires. Using computer-aided design, CAM programming, CNC machining, and injection molding, we produced our “Box Man,” which is composed of 4 molded plastic parts. Altho

Our part design was the most seamless step of our process, as previous skills were practiced in developing this in NX’s CAM software. The mold design was similar, but we did have some complications with sharp edges, assembly methods, and particularly with the shape of the nose for the mold to ensure it came out the way we envisioned. Our mold manufacturing experience was more complicated. We were slowed down by learning as we machined; we wanted to be very careful with setting up tools, double checking the cutting process display, and making sure the tools were referenced correctly in our program. This tedious practice took more time than expected and our setup was a lengthy process. Two of our four mold halves were successful in our first attempt, but we did have an issue with the top core and top cavity. When machining the top core, the nose unfortunately did not come out as intended due to an incorrect tool being used, resulting in a flat area cutting through the cone shape. This issue was addressed, and we had to re-machine that mold half. The top cavity was unfortunately oriented incorrectly when placed in the CNC, so that mold block had to be re-machined as well. The CNC process took many long hours of sitting, waiting, and vigilating to make sure everything was being cut in the way we intended, but it was very rewarding in the end, as our molds came out very nicely. Leading into injection molding was one of the more frustrating parts of our process. We had only two pins to place into our mold, and we thought this would be simple, but that was not the case. This was very problematic, as the pins did not push in all the way, they were difficult to hold vertically when being pressed in, and as we were pulling one out, it broke. This led to us having to use the mill to make a new hole, and then add JB-Weld to fill that hold, setting us back a day since we had to let it sit and dry. Then we used the mill to make a new hole and put in the pin. This was an unexpected setback, as we expected to start injection molding much earlier than we were able to. Injection molding was a very rewarding experience. It took awhile to find the right settings to produce a good product, as we had to adjust the variables of mainly time and pressure, but also sometimes barrel, nozzle, and bottom plate temperature. We varied one variable at a time as we found a part we were pleased with, with good wall thickness, little/no sink or flash, and no other evident flaws. Although this required patience, once the balance of variables was found, the process was easy and efficient. We distributed team members well so that we could do metrology while injection molding, which was very helpful in time management as well. Overall, every step was valuable in a specific way, and they all came together to give us a well rounded skill set in manufacturing.

One problem we encountered during injection molding was lack of material filling on the outside edges of the eyes on the top half of the body. This is because the eyes were deeper than the rest of the “face” in the surface, and had top edges along the other areas of the eye’s perimeters to meet the higher “face” surface, but on the outside edges of the mold the eyes went off the edge, and made an uneven height along the edge, making it difficult for the material to fill in that area well, and oftentimes resulting in a sink in the part. In the future, we would modify this by making the eye’s perimeter even, and having it completely close out inside the face so that the eye does not run into the edge of the mold. Another issue was the filling of the beret. One side of the beret was also not filling in well with material, resulting in a sink area. We realized this was due to how long the runner was from the material inlet to reach the beret, so we would modify this by pushing the beret further in and closer to the inlet to shorten the runner and allow material to reach the part more quickly in filling, which would produce a better part. Besides these two complications, our mold generated parts that we were very pleased with, and our product development was successful.