Arturas Malinauskas

DSGN 345

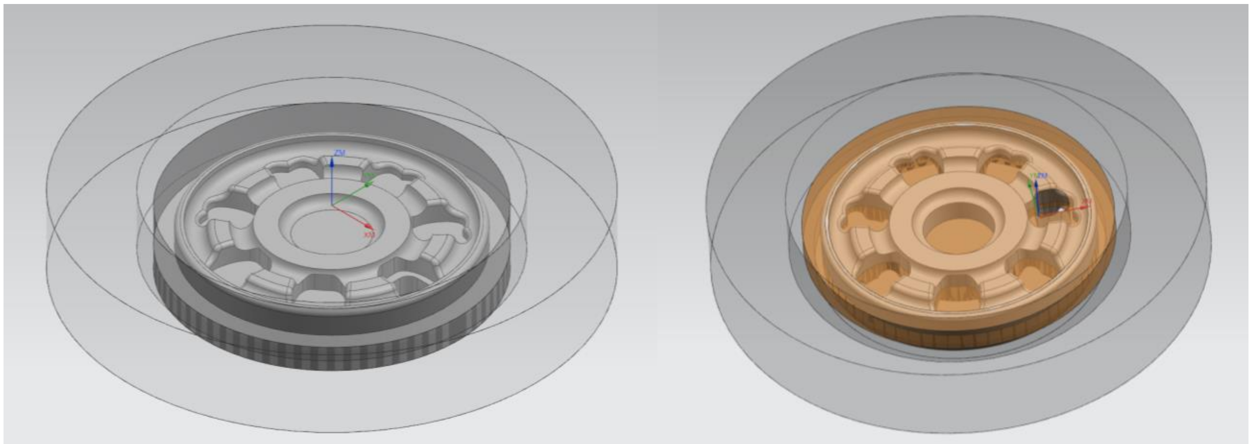
CNC Program to Manufacture a Gyroscope Toy

NX Manufacturing:

Part Design and Assembly Set-Up

The gyroscope design required two setups, one to reach each face of the gyroscope. In NX, I began by modeling my desired final product. I went through three iterations before creating a design that the manufacturing environment could process properly. I wanted a cylindrical object with a set of pockets symmetric about the center to reduce the mass of the object. Removing mass in those areas increases the rotational inertia of the gyroscope. I based the size off a piece of cylindrical stock leftover from an NUSTARS project that happened earlier in the year.

I saved my final part design as two separate files. In one, I modified the geometry to features that are reachable when milling from one side. I used this modified part as my work piece in the manufacturing environment for the first operation. In another assembly, I flipped this part upside-down to serve as the blank in the second manufacturing setup. (Setup1 and Setup 2 in the images below). Both setups included a cylindrical part as check geometry to ensure no tooling made contact with the chuck or the toe clamps holding the chuck to the table.



Left: Setup1, with translucent check geometry, gray blank, and silver part

Right: Setup2, with translucent check geometry, orange blank, and silver part

Tooling

1. 0.75 inch Endmill
2. 0.25 inch Endmill
3. 0.25 inch Ballmill

I wanted to minimize the number of tools used for expediency in manufacturing. Having the need to flip my part, I would have to set zero on each tool twice. I designed my part with the .25 inch end and ball mills in mind to ensure it could be manufactured. I wanted a compromise between the intricacy of the features and the time to manufacture, so .25" seemed like a good size. I used the same tools in the same tool registers for both setups to avoid changing them out.

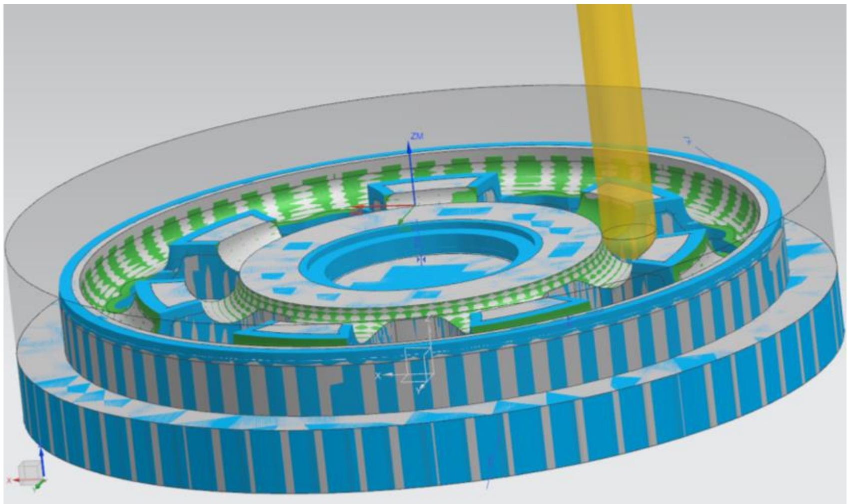
I used a .75 inch endmill as Tool1 to perform my largest roughing operation. I used a .25 inch endmill as Tool2 to do most of my finer geometries. Finally a .25 inch ball mill was Tool3 to take care of the curved surfaces in the part.

Setup 1

Name	Toolchange	Path	Tool	Tool Number	Time	Geometry	Method	Feed	Speed
NC_PROGRAM					01:23:53				
Unused Items					00:00:00				
PROGRAM					01:23:53				
ROUGHING1		✓	EM_0.75	1	00:03:46	WORKPIECE	MILL_FINISH	18 ipm	2000 rpm
SETUP1V2_GYROTOP					01:19:55				
ROUGHING2		✓	EM_0.25	2	00:17:50	WORKPIECE	MILL_ROUGH	12 ipm	6000 rpm
FLATFINISH2		✓	EM_0.25	2	00:09:40	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
PLANAR_PROFILE...		✓	BM_0.25	3	00:02:54	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
PLANAR_PROFILE...		✓	BM_0.25	3	00:04:01	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
PLANAR_PROFILE		✓	BM_0.25	3	00:00:41	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
ZLEVEL_PROFILE		✓	EM_0.25	2	00:00:55	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
CAVITY_MILL		✓	BM_0.25	3	00:07:19	WORKPIECE	MILL_FINISH	12 ipm	6000 rpm
CONTOUR_AREA		✓	BM_0.25	3	00:35:47	WORKPIECE	MILL_FINISH	11 ipm	6000 rpm

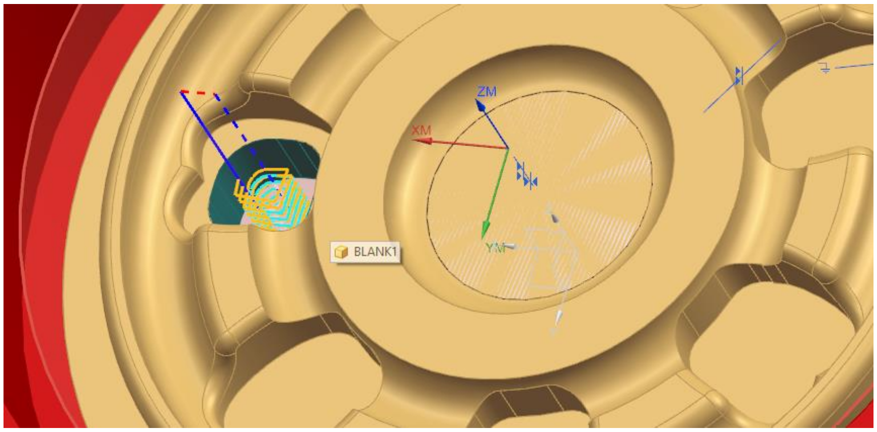
I set the Machine Coordinate System origin to the point at the top of the centerline of the blank. My manufacturing process began with basic Cavity Mill Operations to trim the blank to a more precise size. I oversized the blank to avoid having to process it before beginning my operation. I also wanted to avoid the hassle of using an indicator dial, so I had the CNC take off excess material, so a “perfect” cylinder would be formed regardless of my accuracy in setting up the part. Upon setting up my part, I realized I needed to change the tool working on the outside surface because the toe clamps were closer than 0.75 inches to the outside of the blank.

I used 3 planar profile operations to mill out channels in the gyroscope. I designed them to be the profile of a .25 inch ball mill. This allowed a simple operation to create this geometry, whereas a contour milling operation would have made inefficient tool paths.

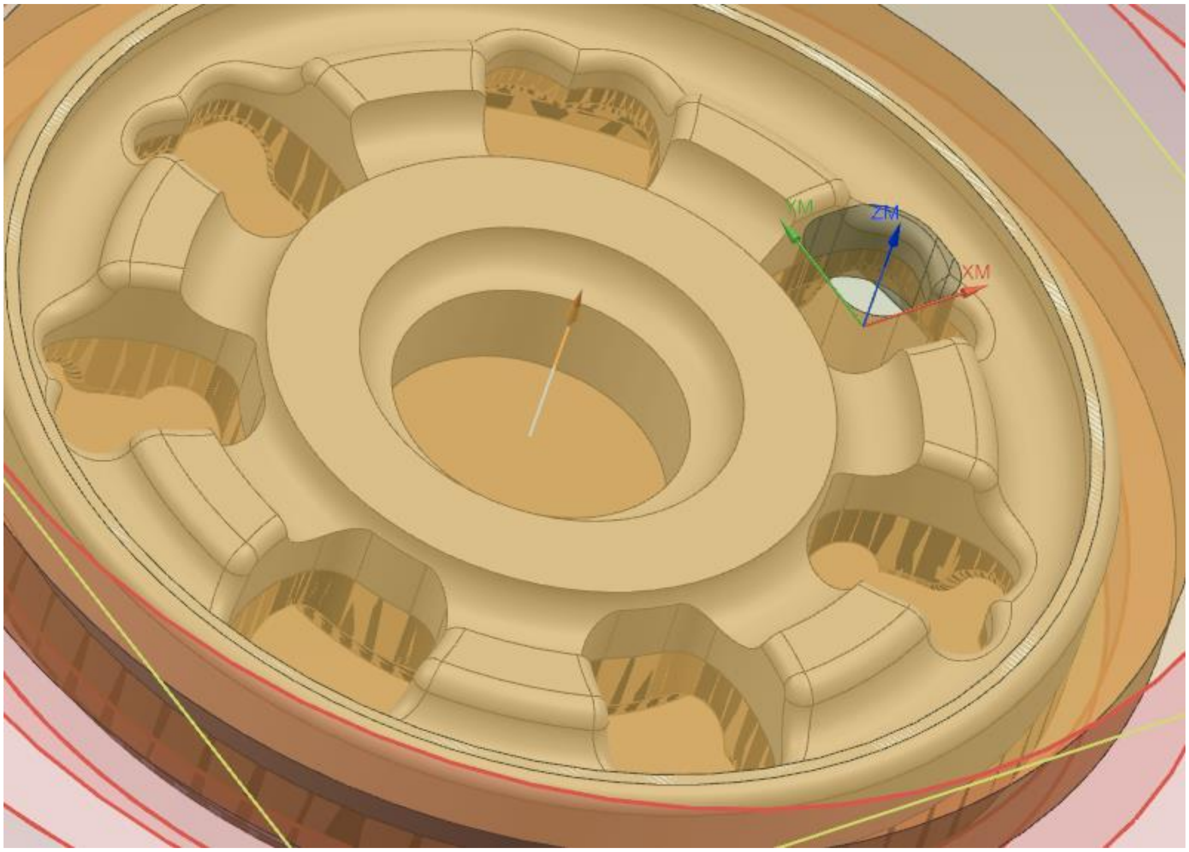


Verification of the ball mill machining the planar profile

I created a single special pocket with a Z-level profile operation to serve as a datum for the second setup. Having a flat on the X and Y axis would allow me to edge find relatively simply and gave a convenient location for the Machine Coordinate System origin in the second setup

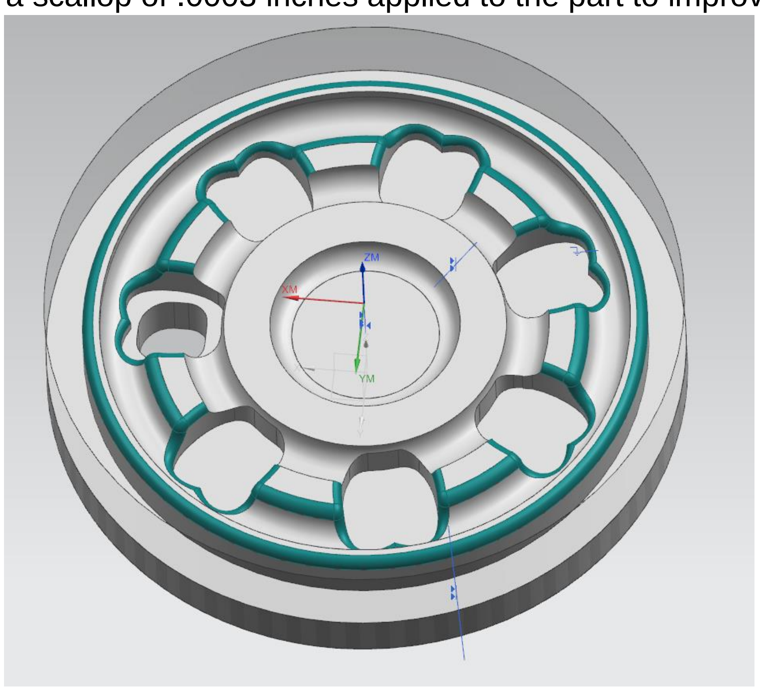


The Z-level profile toolpath



Use of the feature created by the Z-level profile as the MCS in the second setup

Finally, I desired a smooth surface finish. To achieve this finish and smooth curved features, I cavity milled with a ball mill to reduce the time for the final operation: contour area milling. I used a scallop of .0003 inches applied to the part to improve the finish.



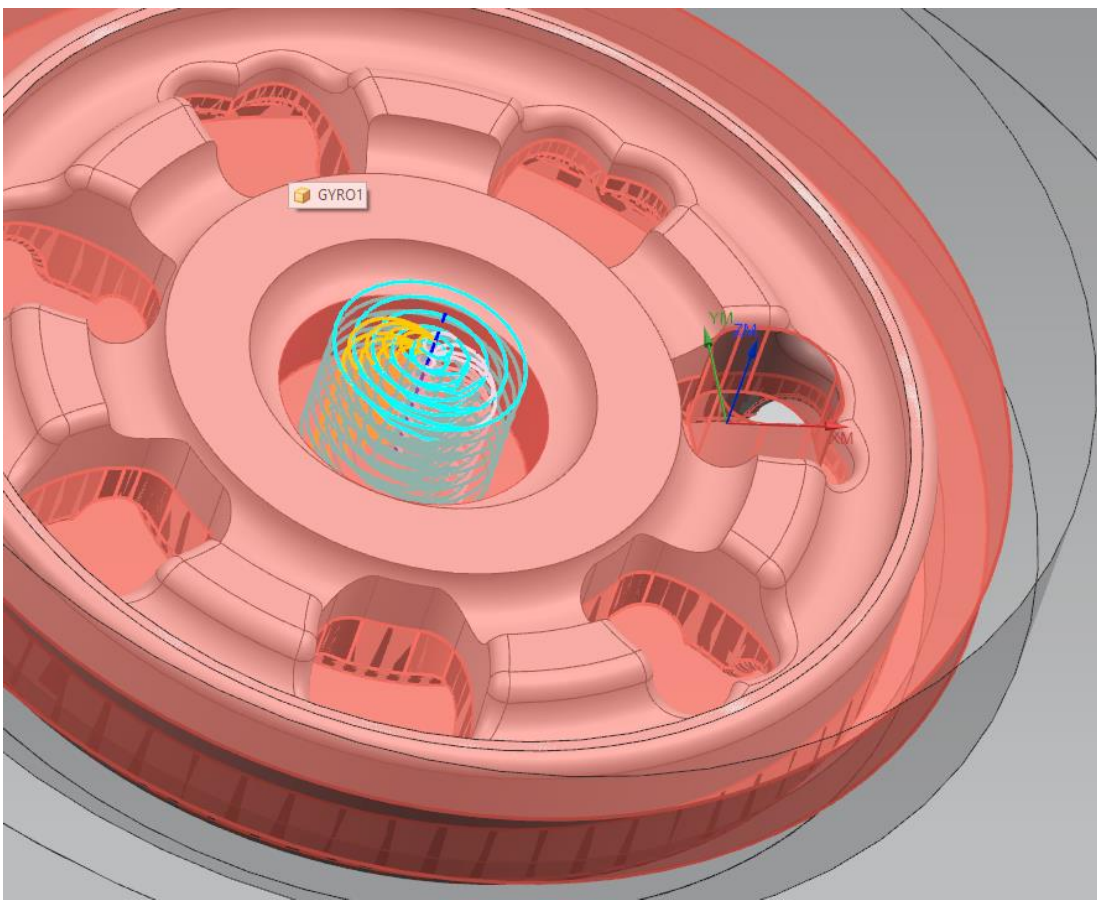
Specified cut areas for the Area Milling Operation are highlighted in green

<u>Setup 2</u>

Name	Path	Tool	Time	Geometry	Method	Feed	Speed
GEOMETRY			01:38:16				
Unused Items			00:00:00				
MCS_MILL			01:38:16				
WORKPIECE			01:38:16				
ROUGHING	✔	EM_0.75	00:01:15	WORKPIECE	MILL_ROUGH	18 ipm	2000 rpm
HOLE_MILLING	✔	EM_0.25	00:08:38	WORKPIECE	METHOD	10 ipm	6000 rpm
ROUGHING2	✔	EM_0.25	00:16:34	WORKPIECE	MILL_ROUGH	12 ipm	6000 rpm
PLANAR_PROFILE...	✔	BM_0.25	00:01:58	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
PLANAR_PROFILE...	✔	BM_0.25	00:02:43	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
PLANAR_PROFILE	✔	BM_0.25	00:00:41	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
FLATFINISH2	✔	EM_0.25	00:23:45	WORKPIECE	MILL_FINISH	10 ipm	6000 rpm
CAVITY_MILL	✔	BM_0.25	00:04:56	WORKPIECE	MILL_FINISH	12 ipm	6000 rpm
CONTOUR_AREA	✔	BM_0.25	00:36:47	WORKPIECE	MILL_FINISH	11 ipm	6000 rpm

My second setup used many of the same operations as before to create symmetric features in the other side of the part. I will only talk about the operations that are altered for the final pass to avoid redundancy.

I used hole milling to create a hole sized for a 0.866 inch bearing (sized with a micrometer) that I took out of a fidget spinner. For the toy to spin nicely, I needed the bearing to fit precisely. I modified the ramp angle and stepover of the operation to form the hole more conservatively and avoid overloading the tool.



The Hole Milling Operation tool path

CNC Manufacturing:

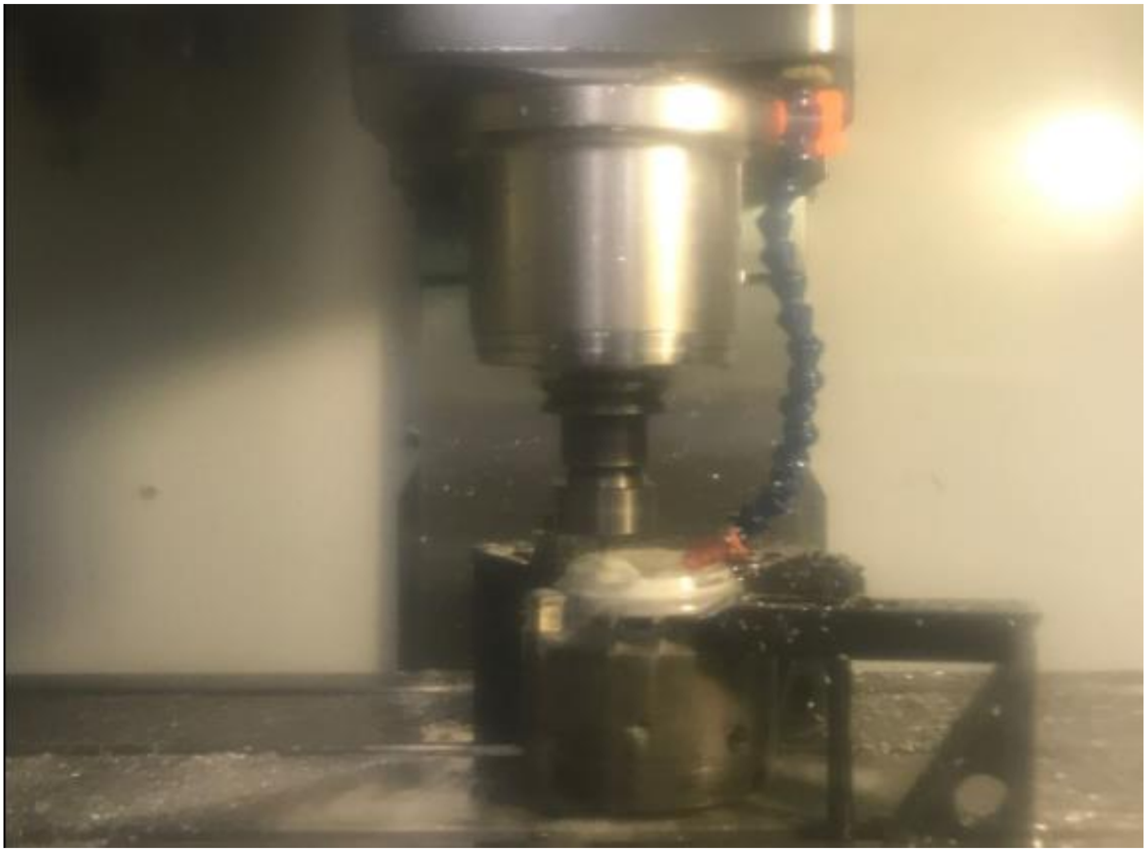
Setup:

To hold my cylindrical stock, I removed the vice from the table of the mill and replaced it with a three-jaw chuck. I secured the chuck to the table of the mill with a set of three toe clamps. I wanted consistency in the position and grip on the part, which motivated use of the three-jaw. After setting up, I added check geometry inside my manufacturing setups to ensure I wouldn't hit anything.

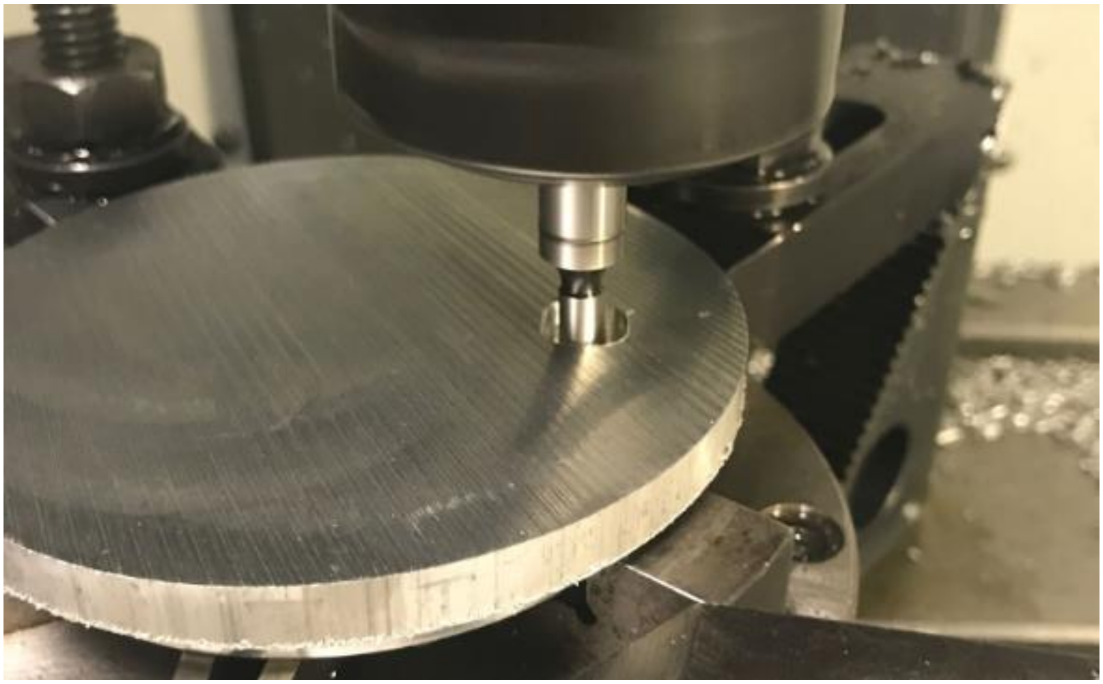
The blank in the improved fixture

I used two short parallels below the part to elevate it above the teeth of the jaws and hold it level. Initially I attempted manufacturing without the parallels, but after the first operation I noticed the chuck jaws could be damaged, so I stopped and added the elevation. I then postprocessed all but the first operation. From there, it was smooth machining.

Machining Progress Photos:


A photo of the first setup running through the grimy door of the HAAS

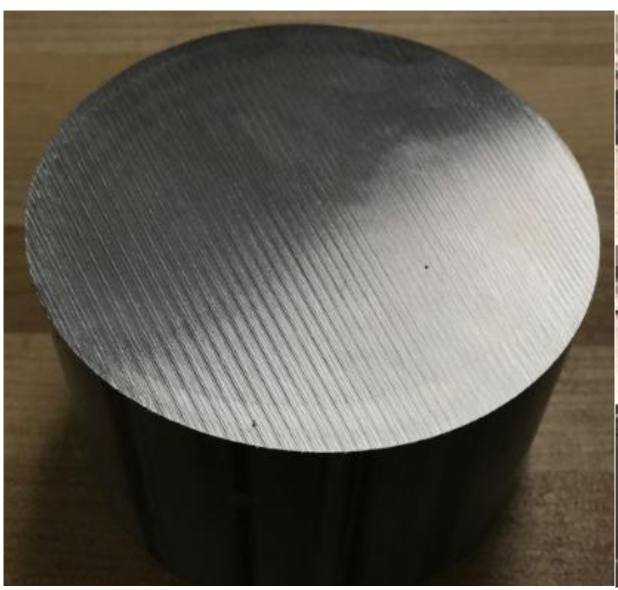

The part after completion of the first program



Edge finding the previously machined reference point

Final Result:

Overall the result is satisfying. Every feature came out symmetric, with a smooth surface finish, and no tooling was damaged. The machined reference worked well. I was concerned about minor angular deviation in the second setup, but I was not able to feel any asymmetry inside the part. There is a slight edge on one half leftover from the first program.



Left: Before intense CNC machining *Right: After intense CNC machining*