MECH 542: CAD/CAM Principles and Practice

Project #2 : NC Part Programming and Verification

Submitted to:
Dr. Jack Chen

Prepared by:
Tianhao Jia(13996012)
Abhinav Sudhir Nair(96584818)
Dhanush Kumar Komalapura Kumar(41850751)

Submitted: April 16th, 2023

Table of Contents

List of Tables

List of Figures

Section 1 : Introductions

1.1 Proposal

The target of Project 2 is to design and verify the toolpath of the part shown in the figure below. The NC programming G-code will be generated by Siemens NX software, and the toolpath of the G-code will be simulated on CutRight software. All of the toolpaths are based on HAAS UMC750 vertical 5-axis machining centre (equipped with b and c axes).

The **first step** of our project is choosing the appropriate tools and fixtures according to the dimensions of the designated part and the operations we would like to implement, which will be done by Tianhao Jia.

The **second step** is calculating the cutting parameters and clamping forces according to the online data that the tool and machine instructions demonstrate, which is the business of Abhinav Nair. We are aiming to fully utilize the machine's power and torque. This step will establish the cutting parameters for the entire process.

The **third step** is to create NX operations and output the G-code toolpath file. This step will belong to Dhanush Kumar. The tools and fixture are selected in Step 1, and cutting parameters are determined in Step 2.

The **fourth step** is to verify the G-code in CutRight, which will be done by Tianhao Jia. This step will focus on whether the G-code can provide the designated cutting toolpath and ensure if there are any collisions between each item.

We also attempt to use the CutRight software to figure out whether the dimensions of our machined part satisfy the tolerance. If the tolerance is not satisfied, another finishing operation might be involved.

1.2 Requirements

1. Choose the tools and fixture in the given website;
2. All operations should be considered in a practical usage;
3. The tolerance of dimensions should be within 0.001 inch;
4. All files should run correctly.

1.3 Constraints

The maximum of spindle is 15000rpm, maximum power is 22.4KW;

The material of stock is Al 6061, the size of which is 4”×4”×3”;

The final tolerance for a dimension is ±0.001”.

Section 2 : Tool Selection

Tool selection in milling operations is a critical aspect of machining, where the appropriate cutting tool is chosen based on various factors such as the material being machined, the type of milling operation, the desired surface finish, and the machining parameters. Selecting the right tool for milling is crucial to achieving high-quality results, optimizing machining performance, and extending tool life.

2.1 Tool calculation and tool assumptions:

In order to machine this workpiece, we need certain tools. To select these tools, we carried out calculations, and for these calculations, we made some assumptions.

Some of these *assumptions* are as follows:

A. The tools are calculated considering all of them are used for slot milling. This is because power and torque required in slot milling is more as compared to other milling operations.
B. We have considered the safety factor to be 1.5 as the material is not too difficult to machine. The loads applied on the workpiece are well-controlled, so there is no need for us to choose a safety factor which is higher than 1.5. The safety factor, however, can be increased to 2 if the part has rough application, or whether the part itself is quite expensive.
C. The tool material we chose is Solid carbide as this material has greater wear resistance, can be used for high cutting speed (not needed in our case), gives higher precision and stability and most importantly, it is cost effective.
D. The power and torque required for roughing operation is more as compared to finishing. So, the tools for finishing were selected in such a way that they would be compatible with the machine as the parameters would be well within the requirements.

For the roughing operation, we carried out calculations and decided to select the tools mentioned below:

A. 1 inch ball end diameter.

B. 1 inch end mill diameter.

C. 0.25 inch ball mill diameter.

The flow for calculating the tools is as follows:

Parameters	**Tools**		
	1 inch Ball Mill	***1 inch end Mill***	***0.25 inch Ball Mill***
Spindle Speed (n)	15000 rpm	15000 rpm	15000 rpm
Cutting Diameter (DC)	1 inch	1 inch	0.25 inch
Axial depth (A_p)	0.1 inch	0.1 inch	0.1 inch
Radial depth (A_e)	1 inch	1 inch	0.25 inch
Number of teeth (Z_n)	2	4	2
Chip Load (F_z)	7.270E-04 inch	3.63E-04 inch	7.27E-04 inch
Feed Rate (V_f)	21.8	21.8	21.8
Material Removal Rate (MRR)	3.57E+04	3.57E+04	8.93E+03
k_{c1}	1700 MPa	1700 MPa	1700 MPa
m_c	0.25	0.25	0.25
h_m	0.0118 mm	0.0059 mm	0.0118 mm
k_c	4750.3 MPa	5649.1 MPa	4750.3 MPa
P_c	2.823 kW	3.364 kW	0.7071 kW
M_c	1.8005 Nm	2.1411 Nm	0.4501 Nm
F_{cut}	31.87 lb	37.89 lb	31.87 lb
F_{clamp}	2294.4 lb	2728.5 lb	2294.4 lb
$F_{clamp\ safety}$	3441.6 lb	4092.75 lb	3441.6 lb

Table 1:Calculations done for selecting the tools

Ball end mills are selected for use in 5-axis machines due to their ability to machine complex surfaces, reduce tool marks, improve surface finish, increase tool reach and accessibility, be versatile in machining different materials, and be flexible in controlling cutting forces.

End mills are selected for milling operations due to their versatility, straight cutting edges, rigidity and stability, efficient material removal rate, cost-effectiveness, and compatibility with different materials and machine types.

On the basis of these calculations and conditions, we narrowed it down to these three roughing tools:

Figure 1:VIPER DVH 1 inch ball end mill

Figure 2: 1 4 FLT EM PREMIUM 1 inch end mill

Figure 3:DiamondBack 0.25 inch ball mill

For the ***finishing operation***, we decided to choose the end mill SECO B38 JABRO TORNADO. The reason for choosing this tool is mentioned later in the report.

Figure 4:SECO B38 JABRO TORNADO

2.2 Selecting the tool holder:

The tool holder plays a vital role in milling operations, as it directly affects the precision of machining, performance of the cutting tool, productivity, safety of the operator, versatility, and overall cost-effectiveness. Opting for a high-quality tool holder and diligently maintaining it are essential for achieving superior milling results and maximizing machining efficiency.

Since the tools we have used range from 0.2 inches to 1 inch, we use an adaptive tool holder with the following specifications:

Figure 5:1-1/2" Shank Diameter x 1" Bore Diameter C-Type Reduction Sleeve

System of Measurement	Inch - ANSI
[ØDMM] Shank Diameter	1.5 in \ 1-1/2"
[ØD] Bore Diameter	1.0 in \ 1"
Type	C
[OAL] Overall Length	3.64 in
[LS] Shank Length	3.375 in
[a] Dimension	2.125 in
[b] Dimension	0.562 in
[W] Width	0.874 in
Coating	Black Oxide
Country of Origin	China
Weight	0.7 lbs \ 0.3 kg
[Ød1] Diameter	1.725 in

Figure 6:Tool specification of the tool holder

Section 3 : Selection of Work Holder

Selecting the right work holder is of utmost importance in milling operations as it directly affects the accuracy, stability, and efficiency of the machining process. A work holder, also known as a work holding device, is used to securely hold the workpiece in place during milling operations to ensure precise and consistent results. It contributes to improved part quality, increased productivity, operator safety, and overall cost-effectiveness, making it a critical aspect of successful milling operations.

Based on the calculations mentioned above, we had quite a few options to select from. But out of these, we selected R96-V75150X.

Figure 7:R96-V75150X work holder

The specifications for the work stand that we use in this project are as follows:

Size	$\mathbf{150}\boldsymbol{mm} \times \mathbf{176.53}\boldsymbol{mm} \times \mathbf{78.2}\boldsymbol{mm}$
Clamping force	**22.0 kN @ 60 Nm**
Max Torque	**60 Nm**
Weight	**10 lbs**

Table 2 : Specification Table for R96-V75150X

As you can see in the specification table, the work holding force is 22.0 kN, which is much more than what we need and would easily accommodate our workpiece as well. The work holder used here is a self-centering vice.

Some of the advantages of using this vice are as follows:

A. ***Self-centering capability:*** The vise jaws can automatically center the workpiece, ensuring even clamping pressure on all sides and minimizing the risk of workpiece misalignment or movement during machining.
B. ***Adjustable jaw width:*** The movable jaws of the vise can be adjusted to accommodate different workpiece sizes, providing flexibility in holding various workpiece sizes within the capacity of the vise.

C. ***Solid and durable construction:*** The medium self-centering vise top tooling is typically made from high-quality materials such as hardened steel, providing durability and stability during machining operations.
D. ***Quick and easy setup:*** The vise allows for quick and easy setup of the workpiece, 5. reducing setup time and increasing productivity.
E. ***Repeatable and consistent clamping force:*** The self-centering feature ensures that the clamping force is evenly distributed on all sides of the workpiece, providing consistent and repeatable results.
F. ***Versatility:*** The medium size of the vise makes it suitable for holding a wide range of workpiece sizes, shapes, and materials, making it a versatile tool for various milling applications.

In order to accommodate the workpiece easily, we also made an extra part that makes it easier to hold the workpiece.

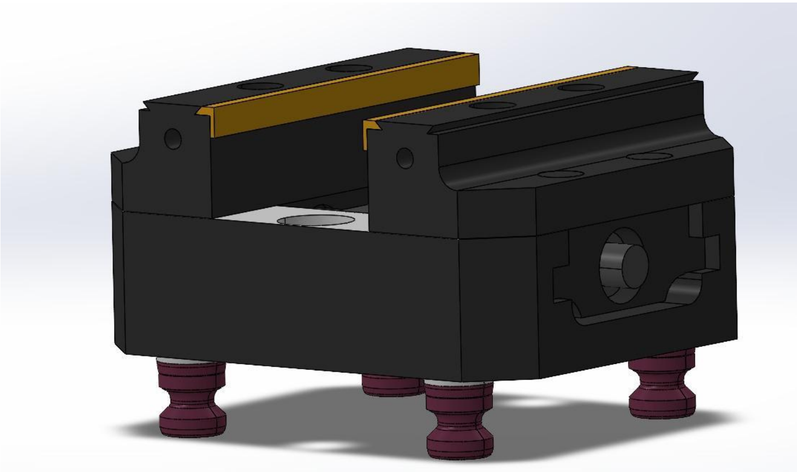

Figure 8:R96-V75150X work holder with extra part (in yellow)

Section 4 : Workflow for Tool path generation in Siemens

This section entails the setup of the NX environment and the various operations used for the manufacturing process.

4.1 Initial Setup

After reviewing the part model, it was imported, along with the vise fixture model (obtained from the 5th Axis website), into Siemens NX. Moreover, as mentioned previously, a stock model was created using the part as the base and then specified as blank. This was created using the bounding box tool option and set to be 4” x 4” x 3”. The fixture was imported into the NX space using assembly models of the fixture taken from the 5th axis. The “fix” constraint was applied to the fixture to make sure it did not move in the modelling space. The part was also completely constrained to the fixture to ensure an accurate analysis was performed. Importing the 5th Axis vise 3D model prior to the programming of the machine operations allowed for the avoidance of obstructions and collisions caused by it. To accomplish this, the “specify check” setting was used when selecting the geometry of the part.

The tools chosen previously were defined in NX. The end mill used in NX was specified using the parameters mentioned in Table 1 and was created for both “mill planar” and “mill multi-axis" operations. The tool numbers defined for this were 1 and 2, respectively. Moreover, the ball mill employed here also uses the parameters defined previously in *Table 1* and is stated as Tool Number 3 for the manufacturing process .

4.2 NX Operations

Name	Time	Toolchange	Path	Tool	Tool Number	Geometry	Method
NC_PROGRAM	02:26:40						
Unused Items	00:00:00						
1234	02:26:40						
CAVITY_MILL_INSIDE	00:21:20		✓	MILL	1	WORKPIECE	MILL_ROUGH
FLOOR_FACING	00:01:08		✓	MILL	1	WORKPIECE	MILL_FINISH
CAVITY_MILL_OUTSIDE	00:36:21		✓	MILL	1	WORKPIECE	MILL_ROUGH
POCKETING_SIDE_OVAL	00:00:59		✓	MILL	1	WORKPIECE	MILL_ROUGH
POCKETING_SIDE_OVAL_FINISH	00:01:00		✓	MILL	1	WORKPIECE	MILL_FINISH
POCKETING_CIRCLE	00:00:11		✓	MILL	1	WORKPIECE	MILL_ROUGH
POCKETING_CIRCLE_FINSIH	00:00:13		✓	MILL	1	WORKPIECE	MILL_FINISH
POCKETING_SIDE_OPPOSITE	00:00:59		✓	MILL	1	WORKPIECE	MILL_ROUGH
POCKETING_SIDE_OPPOSITE_FINISH	00:01:00		✓	MILL	1	WORKPIECE	MILL_FINISH
POCKETING_CIRCLE_OPPOSITE	00:00:11		✓	MILL	1	WORKPIECE	MILL_ROUGH
POCKETING_CIRCLE_OPPOSITE_FINI...	00:00:13		✓	MILL	1	WORKPIECE	MILL_FINISH
CONTOUR_PROFILE_OUTSIDE	00:01:52		✓	MILL_MULTI_A...	2	WORKPIECE	MILL_FINISH
CONTOUR_PROFILE_INSIDE	00:00:45		✓	MILL_MULTI_A...	2	WORKPIECE	MILL_FINISH
VARIABLE_AXIS_GUIDING_CURVES_I...	00:01:17		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_AXIS_GUIDING_CURVES_O...	00:01:23		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_STREAMLINE_INSIDE_FILLET	00:00:53		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_STREAMLINE_OVAL_FILLET	00:00:32		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_STREAMLINE_OVAL_OPPO...	00:00:32		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
ROTARY_FLOOR	00:29:07		✓	BALL_MILL	3	WORKPIECE	MILL_ROUGH
ROTARY_FLOOR_FINISH	00:28:24		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
CONTOUR_PROFILE_TOP_BOTTOM_N...	00:02:09		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_AXIS_GUIDING_CURVES_C...	00:13:39		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_AXIS_GUIDING_CURVES_O...	00:00:32		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_AXIS_GUIDING_CURVES_CI...	00:00:24		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_AXIS_GUIDING_CURVES_O...	00:00:32		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH
VARIABLE_AXIS_GUIDING_CURVES_CI...	00:00:24		✓	BALL_MILL	3	WORKPIECE	MILL_FINISH

Figure 9:Operations created for toolpath generation in Siemens NX

Once the model was fully constrained and the blank geometry was selected, the machine operations were programmed. One major component of the operation programming that was considered was the engagement and disengagement of the cutter at every operation. This was a significant consideration because during the initial design phases of this part, the lack of specified engagement and disengagement paths was causing a variety of issues (discussed in the following section).

In designing the tool path and operations involved in machining the part, a top-to-bottom approach was taken where the tool first machines the top of the part and then the bottom to minimize travel distance. Because the tool starts directly above the part, the first operation done was to cavity mill the bulk material out of the top of the part. Both the roughing and the finishing of the cavity are done one after another since the tool is already in this location. Then the outside slope is roughed, and the oval and circle pockets are roughed and finished. Following this, the tool finishes side milling the stock's outside and then side mills it on the inside slope. From there, the tool is switched to the ball mill, and then the inner and outer chamfering of the edges of the stock is done.

The tool then goes back inside the part to fillet the inside of the stock, and from there, it travels down and performs the fillet of the two oval pockets. The tool then travels to the bottom of the stock until it reaches the top of the neck and then machines or roughs the stock in a circular pattern. This is followed by a finishing process using the same operation. After this, the sides adjacent to the neck are milled to remove the leftover stock. The next step is to fillet the curve connecting the neck and the outside surface of the part. This process had to be thought of extremely carefully, as the orientation of the tool and the milling process would help reduce excess material and keep gouges to a minimum. The importance of this machining method is defined along with the tool orientation for this case in the list below. The last operations to be performed were the chamfers along the sides of the outside walls, which are the two oval and circle pockets. After this tooling process, the part is finally complete, and the tool returns to its home position. In summary, the basic ideology behind the tooling and machining process was to minimize tool movement while keeping the system within the operable travel limits of the machine.

The specific operations used to machine this part from start to finish are as follows (see the Siemens NX file or refer to the above image in ***Section 4.2***):

1. **Cavity Mill Inside:** This roughing operation is used for removing large volumes of material from cavities and cores and is a very suitable operation for machining out the centre portion of material. Thus, it was employed here to machine the inside of the part.
2. **Floor Facing**: Because the end mill is already located inside the cavity of the part at the end of the previous operation, the floor is finished by the end mill using a floor facing operation pass. Floor facing is suitable for basic face milling of planar faces on prismatic parts. The end position of the end mill in this case was set to be above the top of the part to ensure no tool collisions occurred during the following operation.
3. **Cavity Mill Outside**: Cavity milling is used once again to remove the large amount of material located on the outside of the part here. The rough geometry of the part is cut out here, to be finished by a later operation.
4. **Pocketing Side Oval:** Here the first side oval located on the outer face of the part is rough milled. To do this a pocketing operation is used. Pocket milling is recommended for pocket milling of planar faces on prismatic parts, and so is suitable for this operation.
5. **Pocketing Side Oval Finish:** This operation is the finishing operation for the first side oval. A closer final pass is made by the endmill to finish the floor of the side oval. This operation is called immediately after the roughing operation to save manufacturing time, since the endmill is already at this location.

*** Because the operations for the circles and ovals on the outside of the part (operations 6–11) all follow the same logic as operations 4 and 5, they are omitted from this list.

12. **Contour Profile Outside**: Here, the final outer profile of the part is milled using the endmill. To effectively accomplish this, an “auxiliary floor” was defined. This auxiliary floor defines the floor up to which the tool will cut. This is done to ensure a smooth finish is seen around the outside, and so that an excess of material close to the outer fillet is not gouged. This operation is meant for finishing canted walls and so is suitable here.
13. **Contour Profile Inside:** The inside of the part is finished using a contour profile cut once again. Here, it is essential to define the start and endpoints of the toolpaths to avoid tool collisions once again between the tool and the part. The inside floor profile is finished here.

*** The tool is changed here to the ball mill, used for the remainder of the operations (mainly chamfers and fillets) ***

14. **Variable Axis Guiding Curve Inside Chamfer:** The variable axis guiding curve is used to create the chamfers on the top portion of the part. The chamfers here follow a curved paths with complex shapes, and so the variable axis guiding curve is a suitable operation here.
15. **Variable Axis Guiding Curve Outside Chamfer**: Similar to operation 14, the outer chamfer is cut here using the ball mill.
16. **Variable Streamline Inside Fillet**: This operation cuts the inner fillet of the main cavity of the part. Once again here the tool path starts, and end points are essential so that the tool does not collide with the part. The start and endpoints of this path are defined to be much higher than the part. This operation is generally suitable when the flow and direction of a smooth cut pattern is to be specified and so is very suitable here.
17. **Variable Streamline Oval Fillet**: Similar to operation 16, this operation is used to cut a fillet, this time in the oval on the outside of the part. Tool paths start and endpoints are again very important here. This is because when the tool is travelling from the end finishing point on the

curve of the fillet, to the start point of the fillet of the oval on the opposite end of the part, there is a high chance if these points are not specified, the tool will try to travel through the part as is it automatically uses the shortest path to reach the next point of operation.

18. **Variable Streamline Oval Opposite Fillet:** The exact same process for the outer oval fillets is followed here for the oval on the opposite face.
19. **Rotary Floor:** This operation rough mills the neck of the part. The material here was flipped because collisions were initially occurring with the tool and the machine bed when checked in Cutright. Additionally, the initial engagement point of the cutter was changed so that the rotary positions fell within suitable ranges for the Haas UMC-750 machine. This operation is recommended for use in finishing the floors of a cylindrical part and so is suitable here.
20. **Rotary Floor Finish:** This operation finishes off the neck of the part. Flip material was used again because the Haas UMC-750's rotary limits were exceeded. For this case we did not need to specify a start point as the tool can reach the previous point at which the last operation ended without changing the tool axis, however, it was an absolute necessity to specify the return point.
21. **Contour Profile Top Bottom Neck Fillet**: This operation is used to finish the fillets of the neck and finish the adjacent planar faces. From, return, and start points are all very important here to ensure no collisions occur during operation. It was also ensured that the tool was facing the proper side by flipping the tool axis.
22. **Variable Axis Guiding Curves Curve Fillet:** This operations mills the lower fillet on the outside bottom of the part. The necessary cut area was specified and then the guide curve was chosen and "Relative to part" tool axis was selected at a tilt angle of 80 degrees.
23. **Variable Axis Guiding Curves Oval**: This operation mills over the chamfers of the side oval pocket located on the outside of the outer wall. This pocket was chosen as the initial piece to be chamfered because the orientation of the bed for this operation has a similar orientation to the previous operation, and moreover it was the nearest location still needing to be machined, and was thus, chosen.

*** Because the operation for the circle on the outside of the part (operations 24 and 25) follows the same logic as operation 23, it is omitted from this list. ***

Section 5 : Verification of Toolpath

5.1 Siemen NX Toolpath Verification

Using Siemens NX, the verification of the operations was performed using the Analysis option. The verification of the tool paths was checked in NX, using the "verify" option to confirm the presence of the toolpaths. This also ensured that "start," "return," and "from" points were set appropriately. The operations were performed and checked for gouges, and if excess material remained, the methods were redefined by refining the operation.

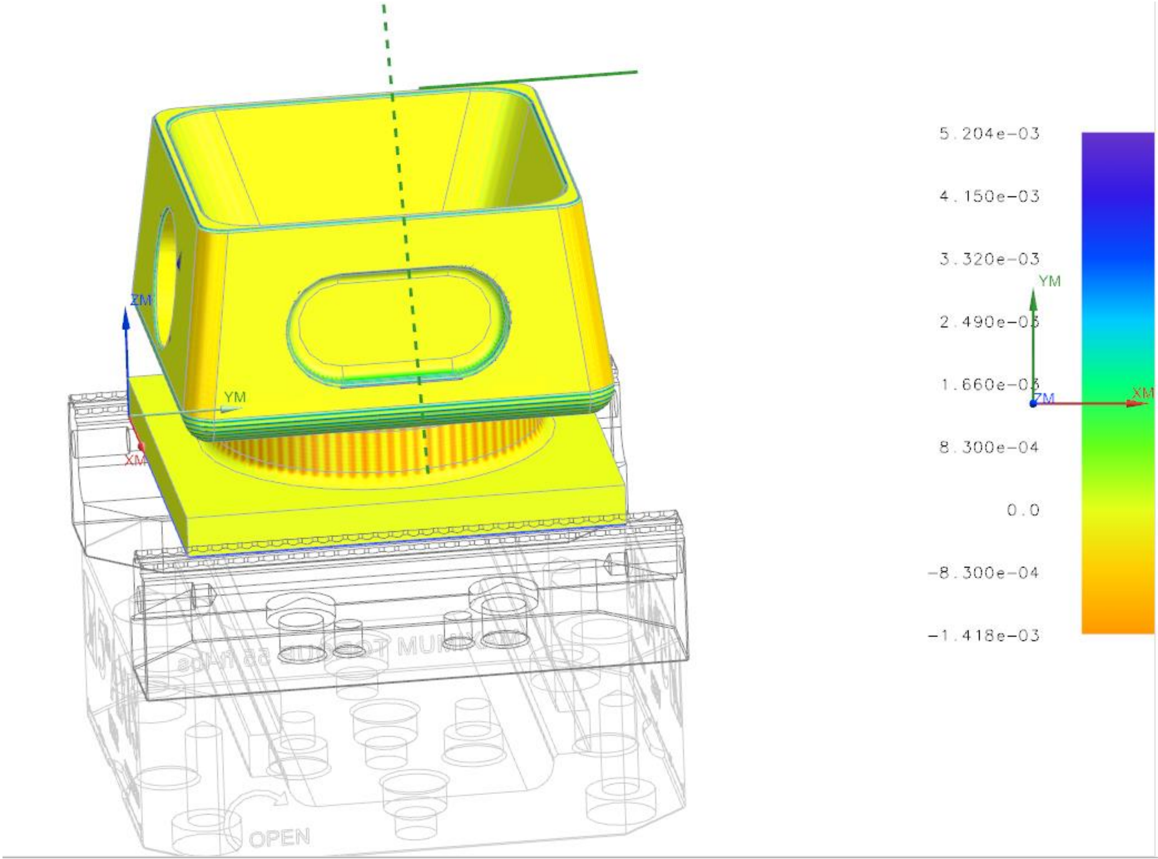

Figure 10:Siemens NX Verification

However, the way in which the tool approaches the cutting surfaces was changed to avoid unnecessary motion. An example of this would be the use of plunge action rather than helical to reach the cutting surface faster and without exceeding the cutting area defined while creating the operation.

5.2 Cut right Toolpath Verification

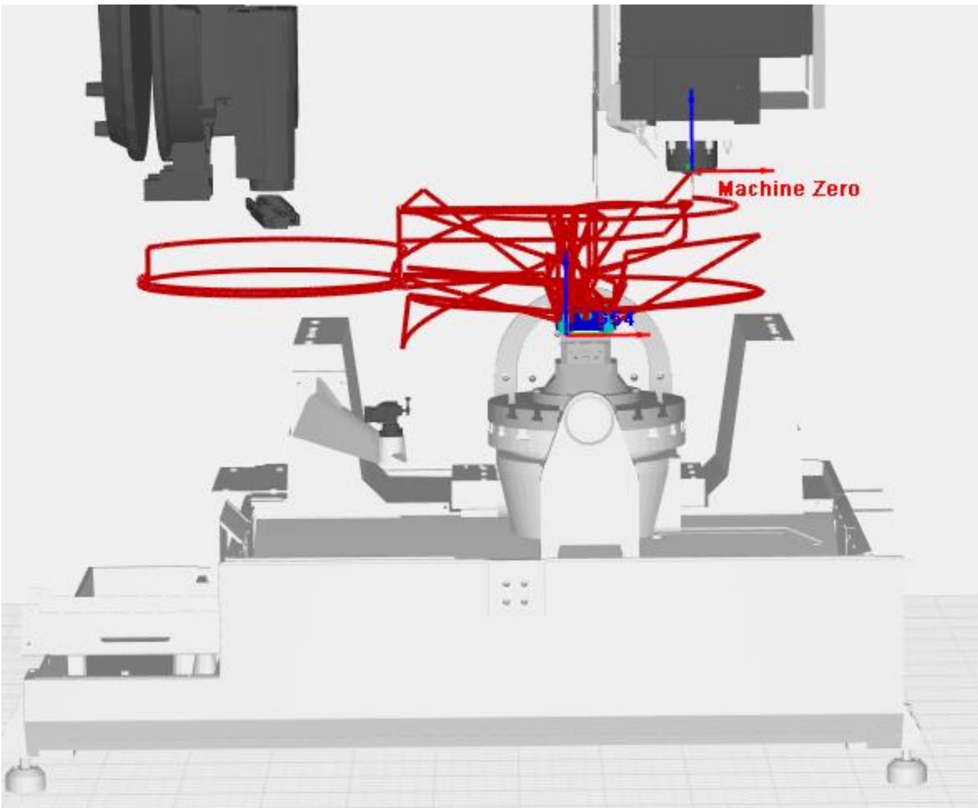

Figure 11:Checking for collision and for any errors using cutright

The verification processes of the NX and CutRight simulations are interdependent on each other. This means the g-code generated in NX must be sent to CutRight, and then the simulation is reset and run. The CutRight simulation, after completing the entire process, was verified for the entire machined part.

Section 6 : Dimensional tolerance verification

The purpose of this section is to simulate and verify the NC programming created in the previous section. Using the CutRight software with a HAAS UMC750 vertical 5-axis machine To simulate as close to reality as possible, NC programs should be properly processed, and the CutRight features, including coordinate systems, tools, and fixtures, should be selected based on real-world applications. Verification is conducted iteratively by identifying the issues and updating the corresponding steps. The general flow of the CutRight verification and troubleshooting are shown in the figure below.

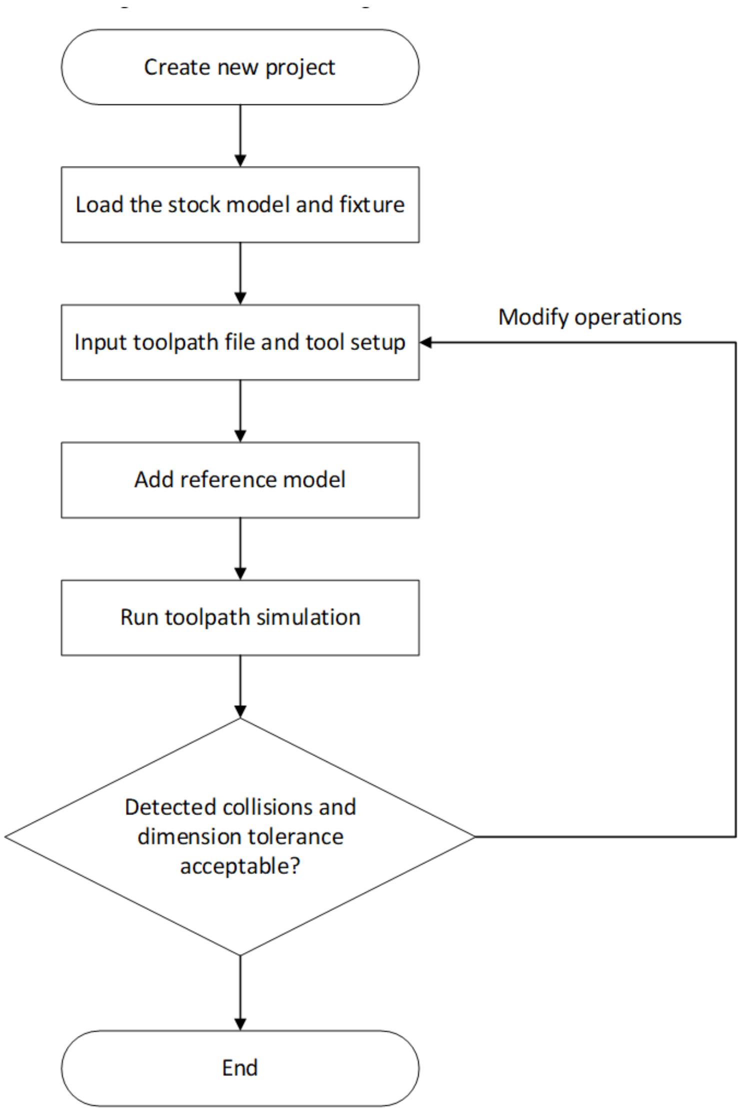

Figure 12:Diagrammatic representation of the work flow followed in this project

In this section, we are going to verify the dimensional tolerance in both CutRight.

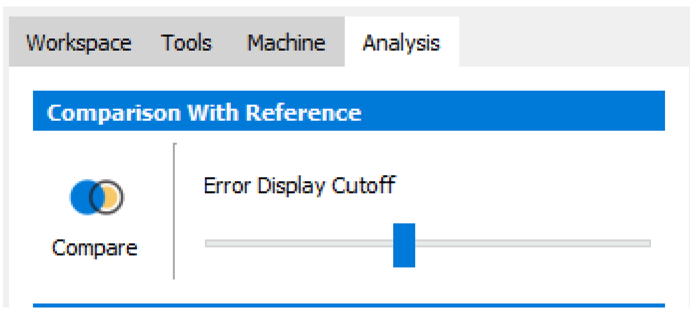

Figure 13:Cutright function Compare for checking error with machined part and original

As shown in the figure below, we can see some ripples on the interior surface of the machined part, where the deviation is 0.0347, which does not satisfy the tolerance of dimensions.

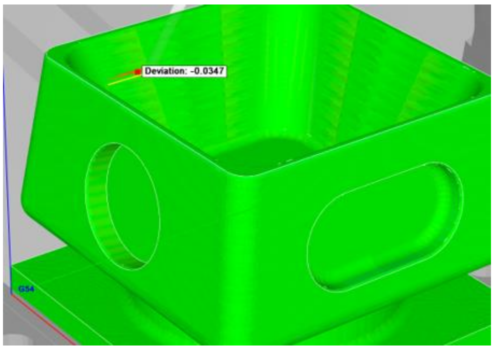

Figure 14:Checking Deviation in CutRight

In order to meet the requirement of tolerance, we added some finishing operations in Siemens NX. We chose a smaller ball mill tool to do these operations. With the aim of obtaining a perfect surface, we set the maximum step of each operation to 0.01 inch. The surface speed should be reduced while the spindle speed is kept at its maximum.

ZLEVEL_5AXIS		✓	✓	CARRIER
ZLEVEL_5AXIS_1		✓	✓	CARRIER
VARIABLE_AXIS_GUIDI...		✓	✓	CARRIER

Figure 15:Choosing for the Z-Level multi axis machining

The dimensions of the added tool are mentioned in the table below.

Order Number: 00022978	**Catalog Number:** 450030-MEGA-T
Publisher product link url:	**等级:** UNSPEC
Coating: MEGA-T	**Hand:** RIGHT
DC: 0.1181"	**DC 1:** 0.1181"
Shank Style: Cylindrical	**DCONMS:** 0.1181"
DMM: 0.1181"	**APMX:** 0.0984"
LSCN: 1.1024"	**RE:** 0.0591"
RE 2: 0.0787"	**NOF:** 2
ZEFP: 2	**ZNP:** 2
FHA: 50	**FHH:** RIGHT
L: 0.1043"	**LE:** 0.0984"
KAPR: 90	**OAL:** 1.5748"
CEDC: 2	**DN:** 0.1063"
GAMF: 18	**GAMO:** 14
GAMP: 50	**LN:** 0.4724"
LN 1: 0.4724"	**PSIR:** 0
RMPX: 30	**WT:** 0.0088"
Material: CARBIDE	**RNA:** 0"
RNR: 0.0004"	**Description:** B38 JABRO Register now!

Figure 16:Tool specifications for finishing operation

Then we generate the G-code file for finishing operations via the given post-processor, and then we append it into our CutRight project for simulation.

In the CutRight software, we do the dimensional tolerance verification again. As shown in the below figure, the deviation between the machined part and the reference part is close to zero, which satisfies the dimensional tolerance requirement.

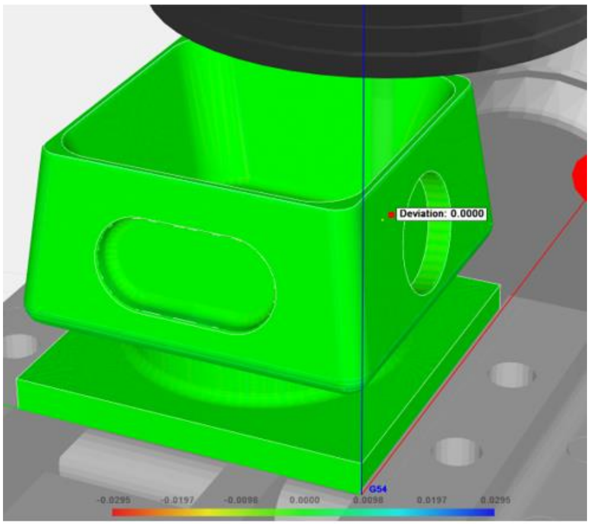

Figure 17:CutRight verification of the workpiece

Here, we finish the whole verification process.

Section 7 : Conclusion

This project was extremely rewarding as it taught us the importance of the tool axis and its location throughout the cutting process, start and return points, and how multiple operations for an object with complex geometry created interact with each other in a (close-to) real-life scenario.

By following the instructions mentioned in this report, we were able to solve the problem statement presented to us. This project helped us understand how we need to select tools and what parameters need to be considered while selecting the tool. Also, based on the tools, the type of workpiece we need to use, and the tools available, the work holder must be chosen. This gave us an insight into the kind of job we should expect when working for a manufacturing firm. The application of Siemens NX for manufacturing helped us understand how engineers work with this software and apply it to real-world problems.