Development of Working Procedures of a 5 Axis CNC Machine

Paulo Ulisses Cunha Morais

UMINHO I 2020

Universidade do Minho
Escola de Engenharia

Paulo Ulisses Cunha Morais

Development of Working Procedures of a 5 Axis CNC Milling Machine

Dezembro de 2020

Universidade do Minho
Escola de Engenharia

Paulo Ulisses Cunha Morais

Development of Working Procedures of a 5 Axis CNC Milling Machine

Master´s Degree Dissertation
Master´s in Mechanical Engineering

Trabalho efetuado sob a orientação de
Dr. Hélder Puga
Dr. Joaquim Barbosa

Dezembro de 2020

DIREITOS DE AUTOR E CONDIÇÕES DE UTILIZAÇÃO DO TRABALHO POR TERCEIROS

Acknowledgement

I would like to start by thanking professor Hélder Puga for all the support and time spent helping throughout this dissertation. I am also thankful for the opportunity, which provided me with an all new perspective of something I really like, the CNC world.

The support of my parents, without whom I would have never been able to det the degree.

And finally, a general express of gratitude to all the friends I made along these years, especially to Eng. Ricardo Martins for all the help, but mainly, for the greatest bus travels.

Declaração de Integridade

Declaro ter atuado com integridade na elaboração do presente trabalho académico e confirmo que não recorri à prática de plágio nem a qualquer forma de utilização indevida ou falsificação de informações ou resultados em nenhuma das etapas conducente à sua elaboração. Mais declaro que conheço e que respeitei o Código de Conduta Ética da Universidade do Minho.

Abstract

The work developed and presented on this dissertation tends to the installation and configurations of a 5-axis CNC machine with the creation of working procedures intended to build a stable workflow that can be employed by any individual expected to use the machine.

Being a large field within mechanical engineering as well as being involved in a large selection of different industrial sectors, the concept of 5-axis machining will be explored to develop knowledge in terms of CAM programming and manipulation/optimization of toolpaths.

The importance/functioning of the transmission of information both from post-processor to the controller and from the controller to the actual machine is also a critical point in this work as they are directly related to the quality of the parts produced.

To accomplish this, the theoretical knowledge foundations regarding CNC machining work were researched, studied, and explained.

Furthermore, the machine model in question (HY-6040 5-axis CNC Router) was meticulously analysed regarding to the machines structure, post-processor, and controller.

Upon assembling all this information, and through the production of some test parts, a permanent manufacture workflow for different machining approaches was established and described.

Keywords: 5-axis CNC; Manufacture Workflow; Machining.

Resumo

O trabalho desenvolvido e apresentado nesta dissertação tende à instalação e configuração de uma máquina CNC de 5-eixos, com a criação de procedimentos de trabalho destinados a criar um fluxo de trabalho estável que possa ser empregue por qualquer individuo que pretenda utilizar a máquina.

Sendo um grande campo dentro da engenharia mecânica e estando também envolvido numa grande seleção de diferentes setores industriais, o conceito de maquinagem em 5-eixos será explorado com a finalidade de desenvolver conhecimentos a nível de programação CAM e manipulação/otimização de trajetórias de corte.

A importância/funcionamento da transmissão de informação quer do pós-processador para o controlador, quer do controlador para a máquina constituem também um ponto critico neste trabalho já que estão diretamente relacionados com a qualidade das peças produzidas.

Para a realização de tal, foram pesquisados, estudados e explicados os fundamentos do conhecimento teórico relativamente ao trabalho de maquinagem CNC.

Para além disso, o modelo da máquina em questão (HY-6040 5-axis CNC Router) foi meticulosamente analisado quanto à estrutura da máquina, pós-processador e controlador.

Após reunir toda esta informação, e través da produção de peças teste, foi estabelecido um fluxo de trabalho de manufatura (CAD/CAM/Maquinagem) para diferentes abordagens de maquinagem

Palavras-Chave: CNC 5-eixos; Fluxo de Trabalho de Manufatura; Maquinagem.

Index

Acknowledgement i

Abstract ii

Resumo iii

Index iv

List of Figures vii

List of Tables xi

List of variables xi

1 INTRODUCTION 1

1.1 Opportunity 1

1.2 Objectives 1

1.3 Dissertation´s Structure 2

2 STATE OF ART 3

2.1 Machinability 3

2.1.1 Workpiece Materials 3

2.1.2 Cutting Tools Materials 5

2.1.3 Operation Conditions 6

2.2 CNC Equipment 7

2.2.1 Axis Designation 7

2.2.2 Multi-axis Designation 8

2.2.3 Multi-axis Components 10

2.3 CNC Milling Work 11

2.3.1 Fixture 11

2.3.2 Manufacturing Project 15

2.3.3 Applications 15
2.4 Computer Assisted Manufacturing 18
2.4.1 Software CAM 18
2.4.2 CAM Workflow 19
2.4.3 3D Toolpaths 22
2.4.4 Post-Processor Controller 24
2.4.5 Programming 26
2.5 Working Protocols for 5-axis Machining 33
2.5.1 Sketch Considerations 33
2.5.2 Positional Considerations 34
2.5.3 Programming Considerations 34
3 MACHINE CONSIDERATIONS 37
3.1 Machine Description 37
3.2 Post-Processor 41
3.3 Machine´s Workflow 47
4 WORKING PROCEDURES 50
4.1 3-Axis Procedures 51
4.2 3+2-Axis Procedures 63
4.3 5-Axis Continuous Procedures 72
5 CONCLUSIONS AND FUTURE WORK 79
5.1 Conclusions 79
5.2 Future Work 80
BIBLIOGRAPHIC REFERENCES 82
ATTACHMENT I – COMMON ALPHANUMERIC ADDRESS CODE 86
ATTACHMENT II – MACH3 MILLING CODES 87

ATTACHMENT III – MACH3 MILLING M-CODES 89

ATTACHMENT IV – FUSION360 MANUFACTURE DESCRIPTION............................ 90

ATTACHMENT V – VISE AND SETUP DEFINITION PROCEDURE 111

ATTACHMENT VI – TECHNICAL DRAWING PART 1.. 120

ATTACHMENT VII – TECHNICAL DRAWING PART 2... 121

ATTACHMENT VIII – TECHNICAL DRAWING PART 3.. 122

ATTACHMENT IX – TECHNICAL DRAWING PART 4.. 123

List of Figures

Figure 2.1 Workpiece Materials ISO Standard [2] 3

Figure 2.2 Toughness and Hardness of Cutting Tool Materials (HW, tungsten carbide; CA, alumina ceramics; CN, nitride ceramics; BN, boron nitride; DP polycrystalline diamond) [4] 5

Figure 2.3 General Axis Designation (a). Right-Hand Rule (b). [1,2] 8

Figure 2.4 Tablle/Table configuration (a); Head/Table configuration (b) [4, 5].............. 9

Figure 2.5 Head/Head configuration [8].. 9

Figure 2.6 Hard Jaws (a) and Step Jaws (b); Milling Vise Components (c) [11]............. 12

Figure 2.7 Soft Jaw [11] 13

Figure 2.8 Fixture System Using a Milling Vise, T-Slots and Step Clamp [12] 13

Figure 2.9 Modular Fixturing [13].. 14

Figure 2.10 Modular Fixturing for Multi-axis Machines [14] 14

Figure 2.11 Nasa´s Orion Bulkhead [15] 16

Figure 2.12 Engine Part Being Machined Using a 5-Axis CNC [16,17].......................... 16

Figure 2.13 Medical Impeller [18, 19] 17

Figure 2.14 Optimized Confinement Chamber Section of a Stellarator [21] 17

Figure 2.15 US Military Drone [22].. 18

Figure 2.16 Simple CAM Workflow 20

Figure 2.17 Different Types of Stock on CAM Programming, (a) stock defined for machining the full part; (b) stock defined to machine two different sections of the same part 21

Figure 2.18 Different Possibilities for Defining the Point Zero-Part............................. 21

Figure 2.19 Part Generated by 2D toolpaths (a) with only horizontal and vertical surfaces; Part Generated by 3D toolpaths (b) with visible curve forms and more than one face machined 22

Figure 2.20 Cut Pattern Morphing Between the Blade Surfaces (a) Path Following a Spherical Cut Pattern (b); both parts presented can be manufactured on 5-axis machines with Head-Head or Head-Table configurations [26,27] 23

Figure 2.21 Tool Axis Control, while the part (a) can be manufacture on both a Head-Head and Head-Table machine configuration, parts with a geometry like the one on (b) are manufactured by machines with Head-Head configuration [9] 24
Figure 2.22 G-code extract, with the header retracted as well as the tool used, spindle speed and the coolant option before the machining movements start. 27
Figure 2.23 LC50-DIGILOG [30] 30
Figure 2.24 Laser Control Systems by Blum Novotest [31] 31
Figure 2.25 Difference between G41 and G42 [33] 32
Figure 2.26 Machine Rotary Zero Point [9] 34
Figure 2.27 Dynamic Workoffset Option [36] 35
Figure 2.28 Rotational Tool Compensation Point [35] 36
Figure 3.1 HY-6040 5-Axis CNC Router 37
Figure 3.2 HY-6040 Axis Movements 38
Figure 3.3 Mach3Mill 6-Axis Interface 38
Figure 3.4 B-axis Rotation (a) -90^0 (b) +90^0 40
Figure 3.5 Model for Axis Orientation 40
Figure 3.6 Mach3Mill Post-Processor 42
Figure 3.7 Extensions Added on Visual Studio Code 42
Figure 3.8 Mach3Mill function onOpen() 43
Figure 3.9 Changes on the Table Parameters 45
Figure 3.10 Changes on the Location of the G43 Codes 45
Figure 3.11 Pass Through 46
Figure 3.12 Post Processor Libraries with the option of saving the user files on the computer where Fusion360 is installed, or on a cloud associated with the user´s account 46
Figure 3.13 Vise Model developed to retract the physical one to be used on the machining process 47
Figure 3.14 Different Fixture Used on G59 48
Figure 4.1 General Workflow of the Working Procedures defined for 3 Axis (blue), 3+5 Axis (orange) and 5 Axis (green) 50
Figure 4.2 Part1 Machined With 3-Axis, with block dimensions of 55x55x55 mm........ 52

Figure 4.3 Part 1 Model and Vise Combined 52
Figure 4.4 Setup for Part 1 with the definition of the work coordinate orientation and origin and also defining the fixture used 53
Figure 4.5 Face Operation Geometry Tab 55
Figure 4.6 Face Operation Heights (clearance; retract; feed; top; bottom) Heights Tab, is this specifi case the retract is coincidental with the feed height................................. 55
Figure 4.7 Face Operation Final Toolpath where it is visible the additional offset given the toolpath to assure that any error committed on the stock placement will not have ant result on the final part.. 57
Figure 4.8 2D Contour Toolpath with 2+2 passes at different heights and with additional offset to ensure the final part dimensions 58
Figure 4.9 2D Adaptive Pocket Selection where the blue area is the one where the tool will pass according to the red arrows pointing directions 59
Figure 4.10 Simulation of the Programmed Part 60
Figure 4.11 Post-Processor Selection 61
Figure 4.12 Part 1 model and final result using the HY-6040 working on 3-axis 61
Figure 4.13 Flat end mill tool used to machine part1, in this image is represented the tool´s parameters within Fusion 360... 62
Figure 4.14 Zero-Part Defined on Fusion 360 63
Figure 4.15 Part 2 (difference between previous part highlighted)............................. 64
Figure 4.16 Pass Through code manually inserted on Fusion360, being later transcribed on the G-code file 65
Figure 4.17 Manual code inserted on Fusion 360 present on the G code file and located before the beggining of a new workoffset 66
Figure 4.18 Setup 2 Established (a); Toolpath from Setup2 (b) 67
Figure 4.19 Part 3 Using 3+2-Axis with block dimensions of 80x50x50 mm................. 68
Figure 4.20 Heights Planes for Part 3 defined for the operations on the top face 69
Figure 4.21 Drawing Created to Assist Toolpath Creation 70
Figure 4.22 Operations Tree of Part 2 71
Figure 4.23 Part 3 Final Result comparation between the model on Fusion360 and the real model produced 72

Figure 4.24 Part 4 with a block dimension of 60x60x60 mm to be machined using 5-axis continuous toolpaths 72
Figure 4.25 Swarf Heights Tab Parameters 73
Figure 4.26 Swarf Strategy Reality vs Simulation 74
Figure 4.27 Part 5 to be machined with 5-axis continuous toolpath and containing a tool change 75
Figure 4.28 Manual Tool Change 76
Figure 4.29 5-Axis continuous 3D flow toolpath represented by the blue lines and being executed by a ball mill 77
Figure 4.30 Part 5 Simulation vs Reality 78

List of Tables

Table 1 Multi-axis Configurations with Axis Designations 10

Table 2 Pros and Cons of the Different Programming Styles 28

List of variables

VARIABLE	UNIT	DESCRIPTION
D	mm	Diameter (component diameter)
f_z	mm	Feed per tooth
f_n	mm/rev	Feed per revolution
n	RPM	Spindle speed
V_c	m/min	Cutting speed
V_f	mm/min	Feed rate
D_c	mm	Cutter diameter
H	mm	Tool height

1 INTRODUCTION

The purpose of this chapter relies on providing the reader a context on why this subject matter was developed, not only the motivation but also the objectives behind it, and how all information is documented throughout this dissertation.

1.1 Opportunity

With the constant evolution of technology, the need for new and more creative solutions for modern day´s problems is a reality. Whether it is the demand to substantially increase productions or the necessity to manufacture parts with very complex designs, multi-axis CNC (Computer Numerical Control) machines can accomplish these requirements.

Multi-axis machines are an indispensable and powerful tool, being heavily used in industries like automotive and aerospace. Nevertheless, controlling full machine motions and continuous work is not a simple task, and the use of advanced CAD (Computer-Aided Design) and CAM (Computer-Aided Manufacture) systems becomes an inevitability.

1.2 Objectives

Upon understanding the potential of multi-axis CNC machines, the dissertation´s purpose involves the installation/configurations of the HY-6040 5-axis CNC Router and the development of working procedures.

These procedures are related to a selected CAM system and are intended to be used, in the future, as educational/informative documents for new machine users.

It will be included all the necessary stages for parts manufacture, which means studying the machine to comprehend its potential and limitations, as well as establishing databases of existing tools and toolpath generation approaches.

1.3 Dissertation´s Structure

The work here presented starts, in Chapter 2, with state of the art divulging all the research made with the intent of establishing a theoretical background to be used throughout the entire dissertation.

The main concepts presented to the reader start with subjects related to the machinability, whether it is workpiece materials, tool materials or even influence of the operation´s conditions.

Notions about the CNC equipment are also studied in this chapter, exploring the interactions and the consequences of the machines components, both hardware and software, workholding solutions available on the market, and CAM systems.

In chapter 3, the machine used on the dissertation is presented, studied, and its' installation/configuration is illustrated, discussing axis configuration, and adapting the post-processor establishing consistent working procedures.

Chapter 3 is essentially the preparation for the next phase of the dissertation. In this chapter all the preparatory configurations are defined and explained and both CAM system and machine are stage set to start manufacturing parts.

To culminate the dissertation, some study cases of different parts, using several machining strategies, are presented and the entire workflow, from the elaboration of the CAM programs to the productions of the parts is presented, respecting the developed working procedures.

Additional attachments were also created to complete the information and create a work base for the future use of the machine.

Since the use of a CAM system is one of the most valuable parts of this dissertation, there was developed an attachment dedicated to the explanation of how Fusion360 works, and how one can define the desired toolpaths using this software.

2 STATE OF ART

With the advances on industries such as medical, aerospace, electrical and military the need for manufacturing complex parts with unique patterns and functionalities increases daily. Combining these needs with the 5-axis CNC (Computer Numerical Control) technology, the potential for opportunities is tremendous. Nevertheless, the multi-axis machine world is complex. Therefore, this chapter is dedicated to establishing the theoretical background needed for the entire dissertations. In this chapter, the concepts and knowledge to understand the various aspects behind manufacturing processes using 5-axis CNC machines will be provided.

2.1 Machinability

Machinability is related to the ease or difficulty a given material has when machined, the understanding of this matter has a tremendous effect when it comes to selecting the suitable cutting tool, attending to the workpiece material, and machining operation.

2.1.1 Workpiece Materials

CNC machines are able to manufacture a wide variety of parts in different materials, and since each material has its own unique characteristics (influence by alloying elements, heat treatment, hardness, etc.), the tools used to cut them will differ in geometry, grade and cutting data.

In order to choose the correct tool, workpiece materials are divided into six groups (Figure 2.1), according to the ISO 513 [1], and each group has its unique properties regarding machinability.

Figure 2.1 Workpiece Materials ISO Standard [2]

The major material group is Steel, extending from unalloyed to high-alloyed material including steel castings, ferritic and martensitic stainless steel, usually presenting good machinability, depending on material hardness, carbon content, etc.

Stainless Steel are materials alloyed with a minimum of 12% chromium. Other alloys may contain nickel and molybdenum. Different conditions, such as ferritic, martensitic, austenitic, and austenitic-ferritic (duplex), create an extensive range of materials. A common factor among all these materials is that cutting edges are exposed to a great deal of heat, notch wear, and build-up edge.

When dealing with Cast Iron, grey irons (GCI) and malleable cast irons (MCI) are easy to machine, however, white nodular cast iron (NCI), compact cast irons (CGI) and austempered cast irons (ACI) are more difficult. All these materials contain SiC, which is very abrasive to the cutting edge.

The Non-ferrous metals are softer metals like aluminium, copper, brass, etc. Aluminium with a Si content of 13% is very abrasive. Usually, high cutting speeds and long tool life can be expected for inserts with sharp edges.

Superalloys and titanium are heat resistant, also including a great number of high-alloyed iron, nickel, and cobalt-based materials. They are sticky, create build-up edges, harden during working, and generate heat. Difficult to cut and reduce the tool life of the insert edges.

Hard materials are steels with hardness between 45-65 HRc, and chilled cast iron around 400-600 HB. Difficult to machine due to the hardness, generating heat during cutting and being very abrasive for the cutting edge.

There is one other group called O (Non-ISO), which represents the thermoplastics, thermosets, glass fiber reinforced plastic, carbon fiber composites, aramid fiber reinforced plastics, hard rubber, graphite. These composites are used with a greater extend nowadays, especially in the aerospace industry.

Even with this division of workpiece materials, sometimes there is no sufficient information to select and order a specific tool. In order to provide a better service, some companies have developed more complex codes to specify workpiece materials (for example, the “Coromant Material Classifications” by Sandvik).

2.1.2 Cutting Tools Materials

The selection of a cutting tool does not consist only in type or dimensions, the material is also something to consider. Possessing basic knowledge of tool´s materials and performance is a valuable aptitude.

The selections is established by the workpiece material, the component type and shape, machining conditions and the level of surface quality required for each operations [3].

Since cutting tools have to endure extreme process conditions, they must have specific properties like presenting high hardness at elevated temperatures, making them resist abrasive wear, and guaranteeing reproducible wear behaviour. High deformation resistance to counteract the cutting edge from plastic deformation under great stresses and temperatures.

As represented in Figure 2.2, which show, for different cutting tool materials, the variation of both toughness (MPa), hardness (HV) and temperature (ºC), it is visible that for high toughness values, reduced hardness is expected, and for increasing temperature values, a decrease on hardness will occur.

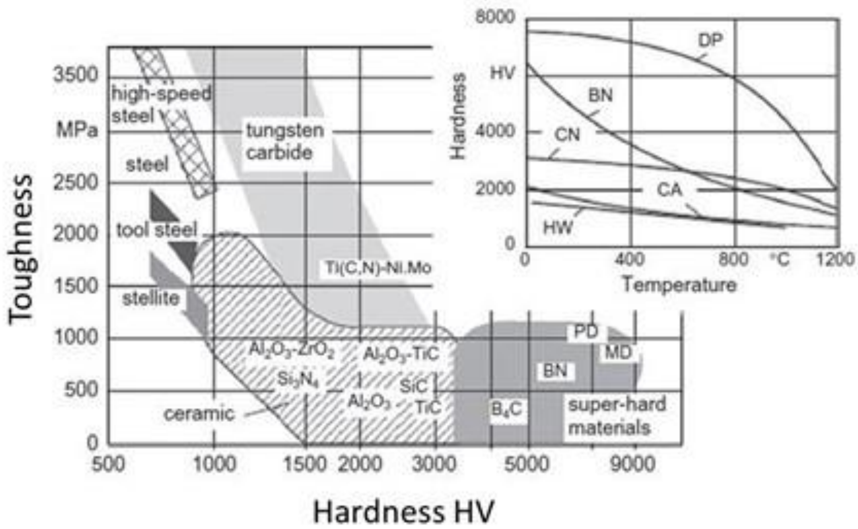


Figure 2.2 Toughness and Hardness of Cutting Tool Materials (HW, tungsten carbide; CA, alumina ceramics; CN, nitride ceramics; BN, boron nitride; DP polycrystalline diamond) [4]

It is also visible, and expected, that different materials possess distinct properties which culminates in specific applications. For instance, high-speed steels (HSS) are a material group with high toughness, offering good wear resistance and generally used for milling both ferrous and non-ferrous materials. On the other hand, tungsten carbide (solid carbide) offers improved rigidity compared to HSS, being extremely heat resistant, and is used on cast irons, non-ferrous materials, plastics and other materials considered difficult to machine.

There are also different types of coating/finishing materials that can be used on the cutting tools. One example is a coating of titanium nitride (TiN) that provides high lubricity, increases chip flow for softer workpiece materials and, the heat and hardness resistance allows for this type of tool to run between 25%-30% faster when compared with uncoated tools [5]. Also, these selections may be perceived as complicated, nowadays tools manufacturers/suppliers have thorough selection algorithms, making it easy to decide on the proper tool for a specific work. There are even systems like “MachiningCloud” [6], consisting of solutions based in industry 4.0 to, upon defining all the relevant parameters of the manufacturing process, generate the appropriate tools and give possible vendors and CAM files of those same tools.

2.1.3 Operation Conditions

When dealing with CNC milling machines, an essential notion is to know the formulas to ascertain suitable feeds and speeds. The values depend and need to be adjusted regarding the machine used, chip shape and/or color or even cutting sound.

Typically, most of this information can be recollected with the tool´s manufactures (websites, catalogues, tool sales representatives, . . .) or even on the CAM systems, since they have sizable libraries with data regarding feeds and speeds. One fundamental aspect is that this information requires adjustments since its commonly based on factors that do not retract the real conditions where the tool will be used, like the maximum spindle speed, the rigidity of the workholding, quality and condition of the tool itself, among others.

Nowadays, if a tool is correctly programmed on a CAM system when creating a certain operation, the user will only need to pay attention to some feeds and speeds values, since the others are automatically derived from the prior ones.

The important parameters to keep in mind and calculated are the Spindle Speed, n (RPM):

$$n = V_c \times 318 \div D$$

The Feed Rate, V_f (mm/min):

$$x = \frac{f_z \times \mathrm{Z} \times \mathrm{n}}{1000}$$

All the remaining feed rates, related to tool movements like spiral diving, approach movements, etc. are defined compared to V_f.

2.2 CNC Equipment

Being computer-controlled machines, CNC equipment is in continuous evolutions, providing even bigger production rates and the possibility of creating extremely complex parts.

Multi-axis machines are the result of this evolution. They are capable of cutting even the most complex of patterns using, usually, only one setup to machine an entire part being thus, used in industries such as aerospace industry, automotive industry, medical industry, energy industry, and military industry.

Throughout the dissertation, the term multi-axis machine will be used when referring to 5-axis CNC milling machine. Although the rem multi-axis machine can be used to mention machines from 4 up to 9 or even 12 axis, the work here developed is dedicated to a specific 5-axis CNC milling machine.

2.2.1 Axis Designation

Standard milling machines have 3-axis with linear motion: XYZ. The 5-axis milling machines have two additional axis with rotary motions according to the configuration presented and motions controlled by the NC (Numerical Control).

The standard axis designation is represented in Figure 2.3 (a), where the primary axis are XYZ, and the second are the rotary axis ABC, rotating around the primary axis, respectively.

There are, however, machine builders that don´t fully respect this designation or simply allow the user to change the machine´s rotational directions. Upon acknowledging this last assertion, the "right-hand rule" shown in Figure 2.3 (b), and accordingly with ISO 841:2011, will be the decisive factor to establish axis designation in the course of this work.

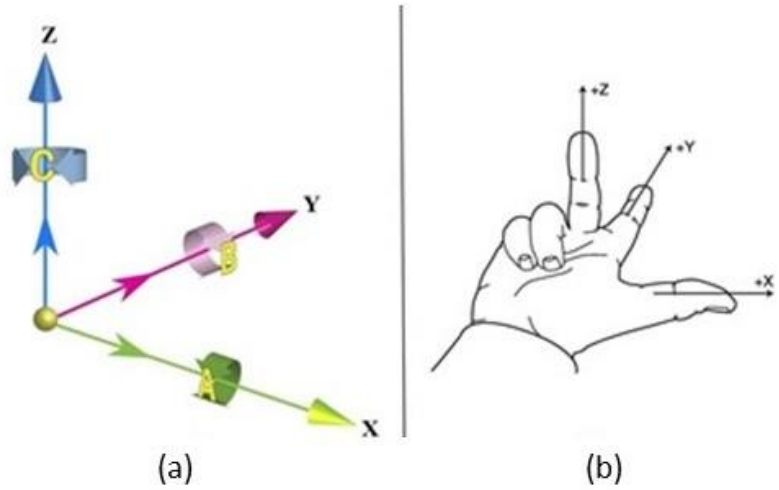


Figure 2.3 General Axis Designation (a). Right-Hand Rule (b). [1,2]

The ISO 841:2001 establishes that: "*The Z-axis is parallel to the principal spindle of the machine"* and *"Where possible, the X-axis shall be horizontal"* [7]. With this in mind, to determine the axis designation and orientation, the user has to simply establish that, using the right-hand, the thumb represents the X-axis, the index finger represents the Y-axis and the middle finger represents the Z-axis. Upon this, first align the middle finger with the primary spindle (making sure the tip of the finger is turned to the spindle cutting direction), align the thumb with the horizontal movement existent in the machine, and the axis designation is determined.

2.2.2 Multi-axis Designation

When working with 5-axis machines is crucial to recognize the type of machine configuration. There are three principal machine configurations, and since the three principal axis (XYZ) are always present, these configurations are established according to the location of both rotary axis. Therefore, when referring to a 5-axis machine configuration labelled Table/Table, it is revealed that bot rotary axis of the machine are located on the table, as represented in Figure 2.4 (a), while a Head/Table configuration has one on the table and the other on the head, as showed in Figure 2.4 (b).

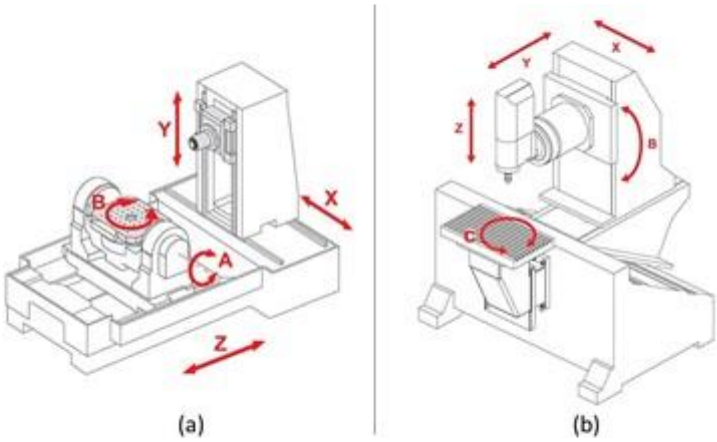


Figure 2.4 Tablle/Table configuration (a); Head/Table configuration (b) [4, 5]

There is also the Head/Head configuration, in which both rotary axis are located on the head of the machine, making it possible to manufacture parts with complex details. In this type of configurations, only one of the axis can execute 360º rotary movement (C-axis) Figure 2.5.

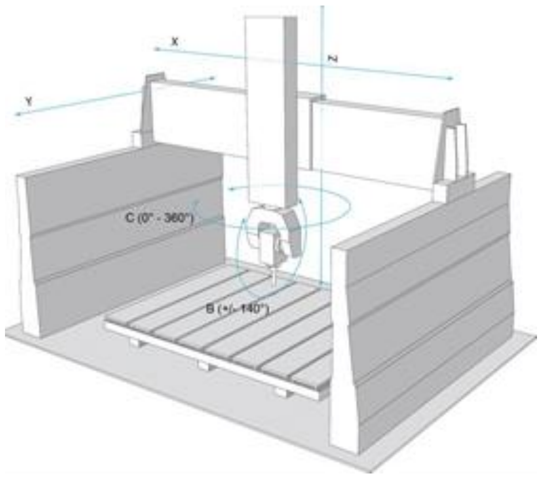


Figure 2.5 Head/Head configuration [8]

In order to simplify the understanding of how the machine configuration affects the axis configuration, the following Table 1 represents the principal multi-axis machine configurations and the implications on the axis designations (the table does not express every machine configuration, just some principal examples to illustrate how rotary axis differ).

Table 1 Multi-axis Configurations with Axis Designations

Multi axis Configuration	Type of Machine	Primary Axis	Rotary Axis	Machine Models
Table/Table	Horizontal	XYZ	AB	Haas UMC-1600-H
	Vertical	XYZ	AC	Haas UMC-500
Head/Table	Vertical	XYZ	BC	Trimill VU 3019
Head/Head	Vertical	XYZ	AC	Dahlih DCM5X-3228

2.2.3 Multi-axis Components

In order to make full use of any machine, it is necessary to fully know all its components and how they work together. This notion applies to multi-axis machines, and it is what dictates what type of operations a specific machine can execute and, consequently, what kind of parts the machine is more suitable to create.

Multi-axis machines are very complex pieces of engineering, however it´s structures can be divided into three major groups: (i) *"The physical properties of the machine"*, (ii) *"The CNC drive system"* and (iii) *"CNC controller capabilities"* [9].

The *"physical properties of the machine"* essentially translate the axis configuration, which has implications on the machine architecture; for example, a machine with a table/table configuration (Figure 2.4 (a)) has a table that allows rotary movements around two axis, while in a machine with head/head configuration (Figure

2.5) the table is immobilized, and a tilting head establishes both rotary motions. This group also incorporates both rigidity and flexibility of the material used to fabricate the machine, important factors to assure stability while machining, the torque and maximum RPM of the spindle, which dictates the limits in terms of speeds. Both guides/slides, rotary bearings, and everything related to the structure itself are in this group.

The second group (ii) *"drive system"*, includes all the servomotors, drive system, the way positioning is controlled and monitored, and the rapid-traverse and feed capabilities. In fact, everything that is responsible for the movements executed by the machine belongs to this group.

Concerning the *"controller"* (third group), this can be called the brain of the machine. Everything from data management, on-board memory size and dynamic rotary synchronization controls represents this group. In essence, this is the group responsible for making the other two groups work synchronized, generating the movements needed for 5-axis continuous machining.

2.3 CNC Milling Work

Before starting to program any part, it is important to understand that the final goal is to manufacture a physical component, and so, understanding how a CNC milling work is set up on the real machine is a precious step when it comes to transitioning from CAM program to the real world.

Defining the correct fixture/workholding, not just dictates time savings, but also increases the safety surrounding the work.

2.3.1 Fixture

Being one of the most important factors for CNC machining, choosing the correct or the most appropriate fixture and/or workholding to use will dictate the number of setups needed per part, leading to different production times.

Before mentioning these components its necessary to define that a fixture is a workholding solution for a particular part. A workholding is any device used to hold a part while machining it firmly [10].

An additional concern is always keeping in mind what type of part(s) is (are) intended to be manufactured. When dealing with sizable production lots, the fixture´s price can be amortized over many parts, and the saved time also represents money saved. In other demands, a part can have very specific details (geometrical), and there will be the need to project a customized fixture.

The most popular workholding solutions being used nowadays is a milling vise, Figure 2.6 (c) which consists essentially in a clamping tool. There are several varieties of milling vises intended for machining different types of parts, but usually the principal components of a vise, are always present performing the same functions.

The fixes jaw is always immobile. The moving jaw moves according to the rotation given to the vise handle, in order to open for part insertion or close to immobilize the part. The vise handle can be removed by merely sliding it off, which is recommended before machining, and the vise stop is a device that allows the parts to be fixed precisely.

Hardened jaws are used along with parallels, which are thin steel plates used to set the vise jaw´s grip length (Figure 2.6 (a)). Step jaws are similar to hard jaws but they include a step, making possible the elimination of parallels (Figure 2.6 (b)).

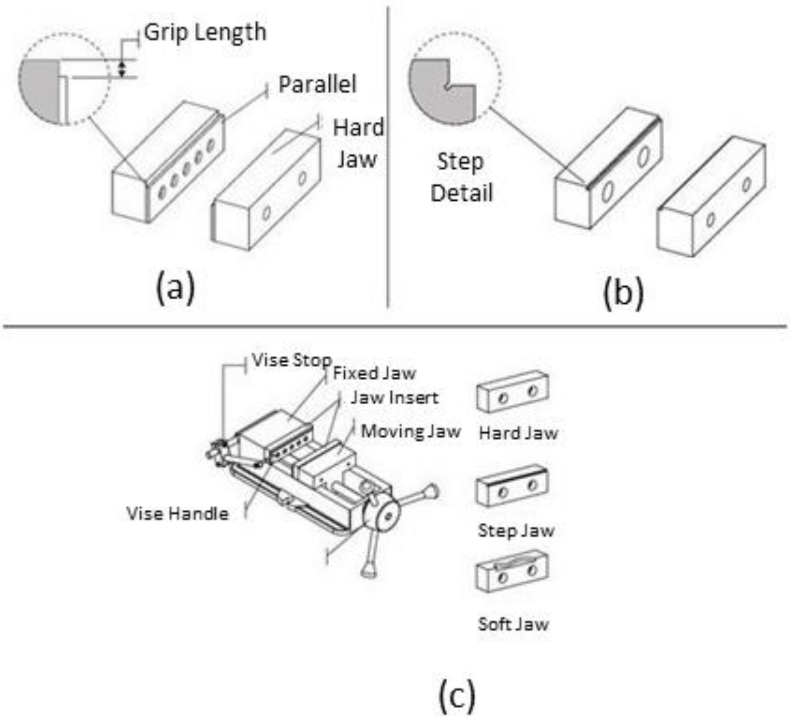


Figure 2.6 Hard Jaws (a) and Step Jaws (b); Milling Vise Components (c) [11]

Soft jaws are essentially blanks of aluminium used to fix parts that cannot use hard jaws, allowing the particularity of being machine with a cut-out of the same irregular shape as the part that needs to be machined (Figure 2.7).

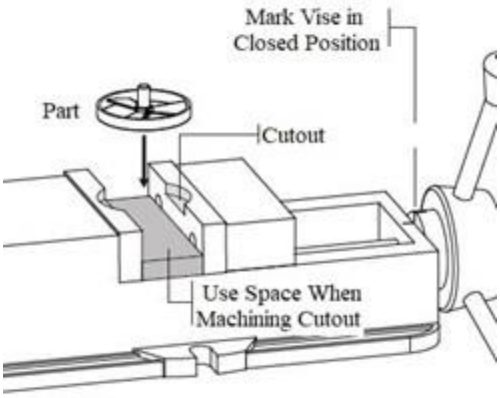


Figure 2.7 Soft Jaw [11]

Being only a workholding solution, the milling vise needs to be bolted to the table. Usually this is done using a fixture structure that includes t-slots and step clamps (Figure 2.8). These tools are commonly used to machine a large variety of parts.

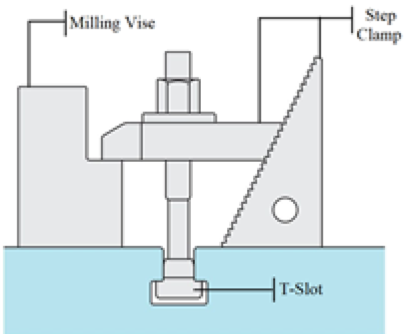


Figure 2.8 Fixture System Using a Milling Vise, T-Slots and Step Clamp [12]

Although this type of system is largely used, when dealing with T-slots there is a significant disadvantage: fixturing the same work holding in the exact same positions (and same orientation) is really challenging. One solution too this dilemma is using fixture plates, which allows for quick and precise fixtures and repeatability. They also present the capability of modular fixturing; meaning that, once installed a fixture plate, there are a collection of other fixture components that can be assembled onto the system, as shown in Figure 2.9.

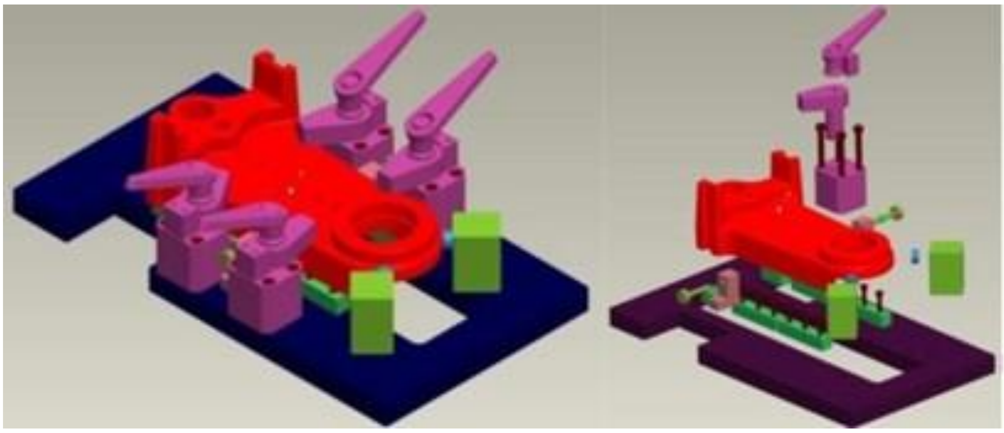


Figure 2.9 Modular Fixturing [13]

The example presented on this figure is one referred in the paper *"Design of 28 Operations, 4 Axis – 360 ^{0} Indexing Milling Fixture for CNC"* [13]. In that paper is explained with detail how it was possible to realize 28 CNC operations (on the red part) using only one setup employing a modular fixture.

The fixture plan designed used 17 tools taking a machining time of 413 seconds to manufacture the intended part (red component on the figure).

Concerning to multi axis machining, and even though every one of the fixture methods presented so far can be used in 5-axis machining (depending on if it will be applied on a 3+2 machining process or five axis continuous machining), modular fixturing has solutions () that surpass all the other methods. In fact, the tables of most multi-axis machines have standardized geometries that make it easier to apply this type of fixture.

Figure 2.10 Modular Fixturing for Multi-axis Machines [14]

2.3.2 Manufacturing Project

The manufacturing project of a part is complex and involves different stages. Gazing at this subject matter as it simply being a manufacturing problem, it can´t be overlooked the fact that the design/geometry of the part was conceived to attain a well-defined purpose.

This means that, although some components may have only an aesthetic motive, most parts (engineering design parts) need to be studied. Pinpointing work surfaces, joints and any specification required is a priority. Although it is usual for all this information to be provided with the CAD files (technical drawing), the date referring to it has to be inputted in the CAM system and it will be the decisive factor when choosing tools, cutting strategies, feeds and speeds on the machining processes.

As seen before, another aspect is selecting the proper fixture/workholding solution for the part(s) in question. It is what dictates the number of setups per part and, subsequentially, most of the manufacturing time. In other words, being able to access and machine the most number of faces on a given part, using the lowest number of setups possible, is always the goal. Other topic that should not be overlooked is the importance of having all the correct data about the tools/devices at disposition. The truth is that there will be occasions when, even though using a specific tool would be perfect for the problem at hands, this will not be possible, whether it is because that same tool is being used on other project or buying it is unaffordable.

Possessing knowledge about the tools/devices that can be used and also having databases about these components on the CAM system will lead to a faster adaptation and creation of new solutions.

2.3.3 Applications

Multi-axis machines allow the manufactures to produce parts with complex geometries, but it also permits to machine parts faster. Being able to produce an entire part with only one setup is a key point when it comes to using multi-axis machines on competitive industries.

There are 5 principal sectors where 5-axis machining is heavily used: (i) Aerospace Industry: the parts needed in this industry are usually very complex and unique, with a high level of details, like the one presented in Figure 2.11.

Figure 2.11 Nasa´s Orion Bulkhead [15]

The image retracts an Orion bulkhead designed by Nasa, and, in simple terms, is a large forged piece of aluminium with a significant number of pockets normal to the surface, which would not be able to be obtained without multi-axis technologies.

The use of 5-axis machines on the aerospace industry is also mainly employed for parts like turbine blades used on plane engines, for example.

(ii) Automotive Industry: being a high production volume sector where each component requires consistency on the specific tolerances of every par (Figure 2.12) and production time is crucial. The use of 5-axis machining is one of the reasons behind this sector´s competitiveness.

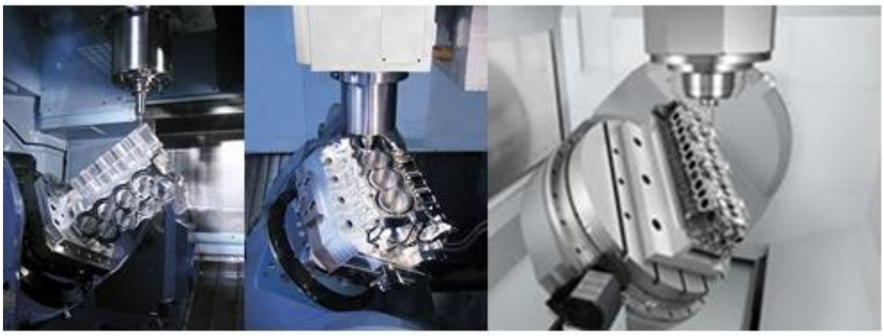


Figure 2.12 Engine Part Being Machined Using a 5-Axis CNC [16,17]

(iii) Medical industry: this sector has an extremely meticulous requirement for quality in order to fulfil the rigorous healthcare standards. Come parts have organic shapes machined on unusual materials in order to be implanted on the patient ().

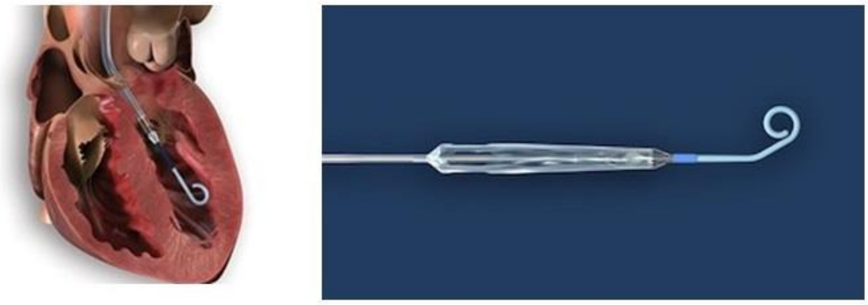


Figure 2.13 Medical Impeller [18, 19]

This medical impeller can be used, for instance, after a heart attack, in order to pump blood on a patient´s heart (more can be read about this specific device on the Science News: "*In a heartbeat: Tiny propeller keeps blood flowing"*) [20]

(iv) Energy Industry: with a never-ending need for innovative components using different materials and complex shapes that make available new or better systems, this sector relies on 5-axis machining. Represented in Figure 2.14 is a section of a confinement chamber of a stellarator, which is a complex device used to store plasma with magnetic fields in order to get a controlled nuclear reaction. Some shapes on this machine are so organic and complicated that the use of 5-axis CNC machining cannot be solved.

Figure 2.14 Optimized Confinement Chamber Section of a Stellarator [21]

One other point about the use of multi-axis machines on the energy industry is that when it comes to green energy, these machines are a big step forward, whether it is on providing parts used on the wind, solar and hydropower, but also when it comes to providing new solutions when researching for new forms of clean energy.

(v) Military Industry: although this industry makes use of the aerospace sector, there are other applications included like submarine parts, high-performance engine parts, smart weapons, sensors and nuclear weaponry (). Multi-axis machining also facilitates and improves when it comes to research, which is a big aspect of this sector.

Figure 2.15 US Military Drone [22]

2.4 Computer Assisted Manufacturing

When dealing with multi-axis machines, there is always a requirement of using CAM (Computer-Aided Manufacturing) systems. The software is a tool with extreme importance and power. Having a good CAM system and the set of skills to control it completely, even if working with not so sophisticated multi-axis machines, can be revealed to outcome the work made with better machines but no so good CAM works.

2.4.1 Software CAM

When approaching a CAM software, the first matter is understanding that the various existing software programs hold on to different roots. This implies selecting the most appropriate CAM system according with the machine and type of work intended. If this matter is not taken into considerations, the CAM software may not be capable of handling its purpose.

Systems with more detailed CAD (Computer-Aided Drawing) functionalities have their roots in CAD software and are superior at parts modelling and large assemblies, allowing a well-detailed parameterization of the work, however their CAM functionalities may have been added later and are often shallow (may not allow multi-axis continuous machining).

Systems with heavy CAM emphasis are great at everything related to toolpath creation. These systems contain tool libraries with associated feeds and speeds for different materials and cutter types. As a replacement for the low CAD abilities, these systems are excellent at importing CAD data from other systems [9].

Systems that excel in both CAD and CAM are an additional option nowadays. Make use of "Fusion 360" from Autodesk [23] as an example, this software uses several interfaces in which one has destined from CAD elaboration with full parameterization. These CAD models can be used on other interfaced intended for CAM, allowing to also create the toolpaths.

Programs like this allow an unique link between both CAD and CAM sections, granting the user to draw CAD components while programming the CAM parts, in order to get improved toolpaths or tool orientations, leading to more optimized CAM workflows. These systems can be referred to as "Integrated CAM Technology", which means seamlessly integrate the CAD design with simultaneous toolpath regeneration, in a connected workflow that can improve efficiency.

One other matter to ponder upon is that a CAM system should be a personalized tool. Creating databases with all the tools, holders and even machines that one may own/use can make programming not only easier but also faster and enhanced.

Linking all these functionalities with detailed setup sheets, possible to create with the CAM systems, specifying workpiece materials, cutting tools and notes per operations, lead to an even greater efficiency achieved by these systems.

2.4.2 CAM Workflow

The CAM workflow defines the sequential processing of design data and its simulations [24].

In reality, the idea of a constant CAM workflow is not a trustworthy notion.

As seen in the previous chapter, different types of CAM systems have different functionalities and interfaces. Therefore, depending on these factors, the workflows will retain some variations, although there are always some basic/crucial parameters that will always be present (the definition of these parameters may change according to the cam system in use even though the purpose is the same).

The arrangement shown next on Figure 2.16 retracts a straightforward CAM workflow, presenting all the common and essential steps of CAM programming.

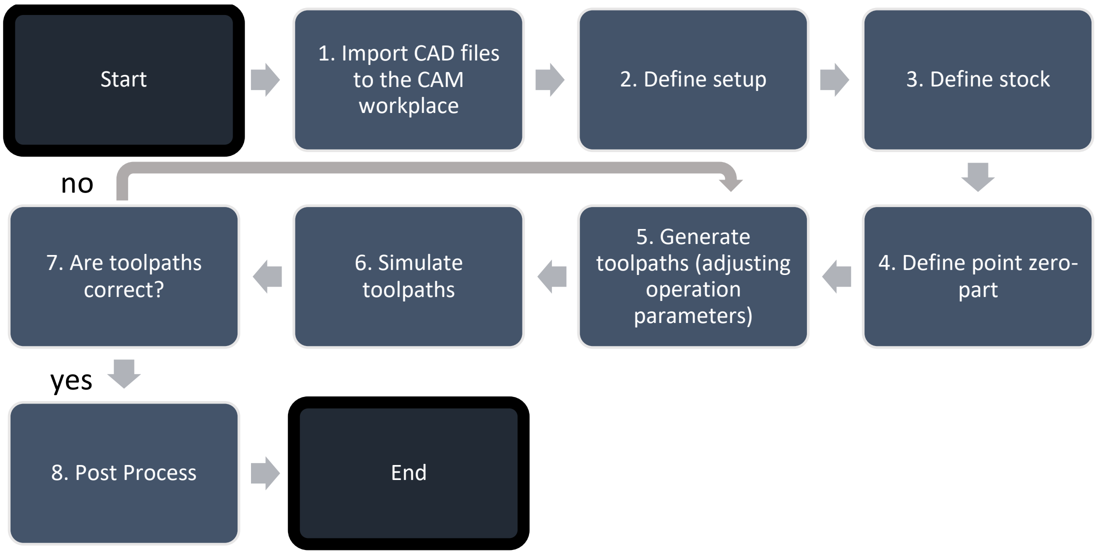


Figure 2.16 Simple CAM Workflow

The CAD files are always the starting point. They need to be imported or loaded into the CAM workplace in order to start programming. With the information concerning the machine and fixtures to be used, the part intended to be manufactured, and the work coordinate system (WCS), the setup needs to be defined (depending on the CAM system in use).

Subsequently, the stock needs to be specified, existing essentially 2 types of stocks: one if the part is going to be machined from zero (Figure 2.17 (a)) and other if the operations takes part on a partially machined part (Figure 2.17 (b)).

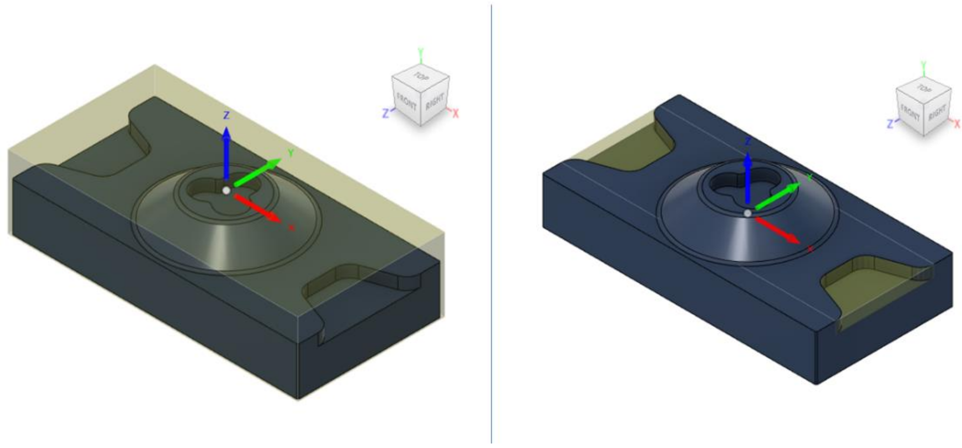


Figure 2.17 Different Types of Stock on CAM Programming, (a) stock defined for machining the full part; (b) stock defined to machine two different sections of the same part

The type of stock defined on Figure 2.17 (b) is very used on casted parts. Usually, this parts need to be machined ether to correct some defaults or to create new geometries that were not viable to produce with the casting process.

Upon this, there must be defined the point zero-part with the axis orientation respecting the axis system of the machine. Depending essentially on the workshop devices owned, if possible to utilize a probe, this point can be located almost anywhere on the stock, however, if that is not possible, and the only tool to be used is an edge-finder (Figure 2.18), locating the point zero-part on an edge or vertex will be more easy and efficient when transitioning from the virtual to the physical world.

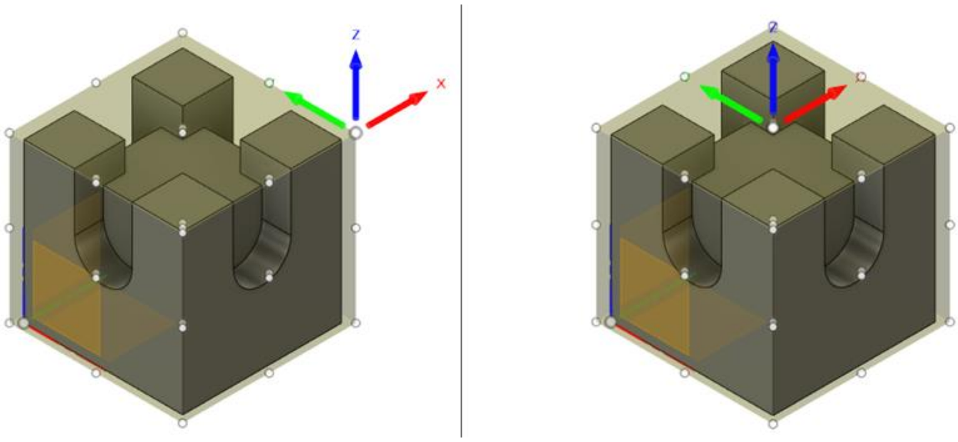


Figure 2.18 Different Possibilities for Defining the Point Zero-Part

Toolpath generation succeeds, and all the operation parameters are included in this phase. More than just defining a 2D or a 3D strategy, it has to be defined the tool used, the feeds and speeds, the use/no use of coolant (if used, coolant type – flood, mist or air), type of passes, the plans offsets, tolerances.

The toolpath simulation will now allow to analyse the work done and comprehend if the need to change anything exists or not (determine if all was correctly defined/programmed and, if not, amend the need parts).

Ending with the post process, this depends only on the existing controller of the machine to be used. This is the point where the G-code is created. Concluding, all the CAM is completed, and the remaining work has to do with the real-world machining/production of the part(s).

2.4.3 3D Toolpaths

Setting off by comprehending how 2D toolpaths differ from 3D toolpaths, the 2D toolpaths are used almost exclusively to machine parts with prismatic forms (all the surfaces are either vertical or horizontal) like represented in Figure 2.19 (a). In contrast, 3D toolpaths are used to machine parts with more complex patterns/designs (freeform part) as shown in Figure 2.19 (b).

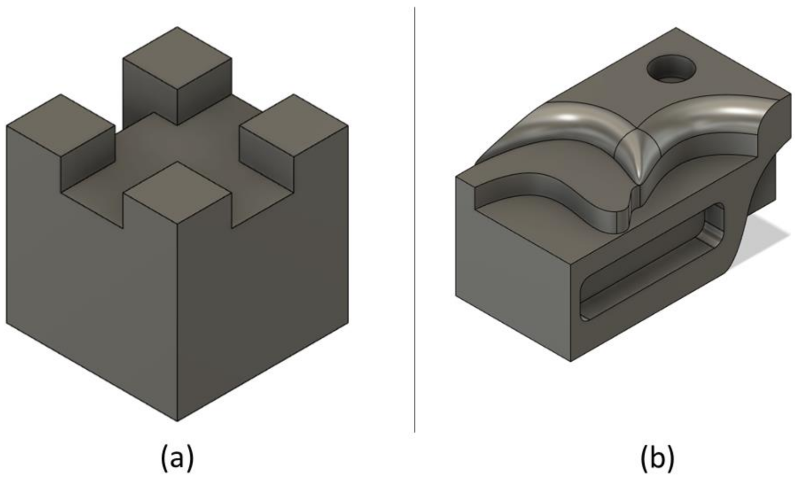


Figure 2.19 Part Generated by 2D toolpaths (a) with only horizontal and vertical surfaces; Part Generated by 3D toolpaths (b) with visible curve forms and more than one face machined.

One other important aspect related to both 2D and 3D toolpaths is that, the toolpaths are commonly divided into roughing and finishing [25]. Fundamentally, roughing creates toolpaths to remove large material volume and as quickly as possible without taking into consideration the finishing surface quality. Finishing, however, uses toolpaths that remove small amounts of material but providing the desired surface quality finishing.

Redirecting now to multi-axis toolpaths, there are three fundamental points that a CAD/CAM system needs to grant user-control in order to create and adjust multi-axis toolpaths assertively: (i) Cut Patterns; (ii) Tool Axis Control; (iii) Tool Tip Control [9].

Cut patterns are defined trajectories where the tool will move along, which can be 2D or 3D wireframe or even complex multi-surface grids (Figure 2.20).

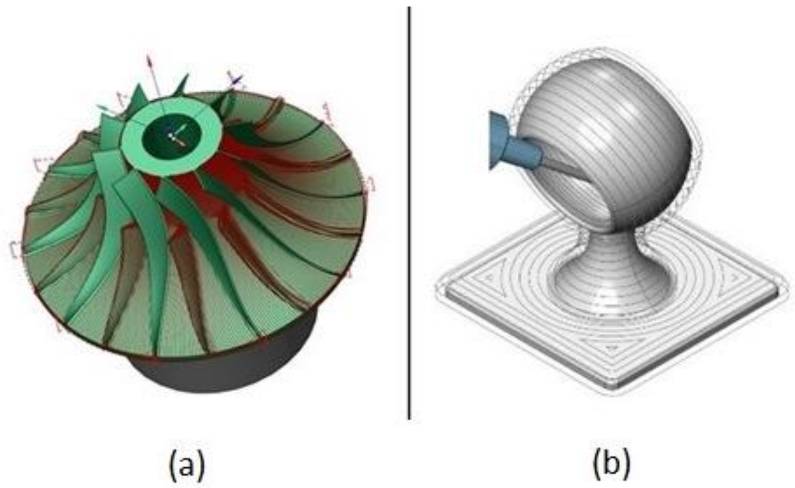


Figure 2.20 Cut Pattern Morphing Between the Blade Surfaces (a) Path Following a Spherical Cut Pattern (b); both parts presented can be manufactured on 5-axis machines with Head-Head or Head-Table configurations [26,27]

Tool axis control comes to play when dealing with parts that possess complex geometry or simply when there is the need to use some tools (example: ball mill, pencil mill). Since the tool follows the cut pattern, manipulating the direction of the tool´s axis is a very important factor (Figure 2.21).

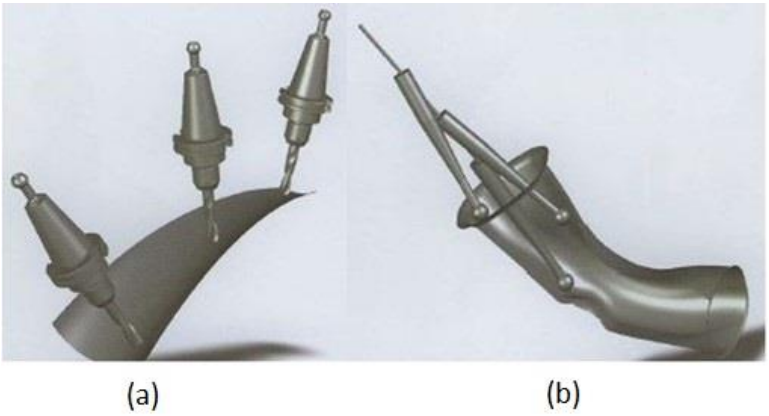


Figure 2.21 Tool Axis Control, while the part (a) can be manufacture on both a Head-Head and Head-Table machine configuration, parts with a geometry like the one on (b) are manufactured by machines with Head-Head configuration [9]

Continuing on Figure 2.21 (a), it is easy to understand that using tool axis control allows to manipulate the tool in order to make its axis normal to multiple surfaces. Looking at the Figure 2.21 (b), it is also visible the possibility to force the axis to change while spiralling down a certain geometry (in this specific case, the geometry represents an intake channel).

After understanding the tool axis control, talking about tool tip control may be perceived has unnecessary, but the truth is that the two concepts are complementary.

Previously, it was stated that a CAD file precedes CAM programming, and every time a CAD file gets translated between different systems, tolerances issues may occur, making it easy for error to be aggravated and leading to a poor quality CAD model (the model may involve many surfaces with gaps between them). This would make the tool axis to flip radically if it tried to stay normal to all the surfaces while traveling.

Fixing these errors is a very time-consuming task. Thus, tool tip control makes it possible to avoid this problem since it allows to target the precise area of the tool tip´s engagement. The combination of these three features makes it possible for full control of the tool movements.

2.4.4 Post-Processor Controller

The post-processor controller comes to play when the entire CAM program is completed, and the only thing left to do is the generation of the G-code files.

Fundamentally, *"CAD/CAM Systems generate 5-axis vector lines along 3D paths. The 3D paths represent the tool motion as it follows the pattern being cut. The vectors represent the individual tool axis directions (IJK vectors) as the tool follows the 3D (XYZ) pattern. Every vector is represented by a line of code, the minimum angular differences, or the linear distances between vectors"* [9]. On the other hand, all this information is transcribed in a generic language (depending on the CAD/CAM system – ex.;NCI) that the machine controllers do not read, which means a translation must be done (this step is called post-processing).

A tremendously vital factor is that every post-processor is associated with a specific machine, calculating the axis motion needed for that same machine to reproduce the CAM vector sequence. It also contains detailed data regarding both physical and computing abilities possessed by the machine, allowing to generate a precise G-code.

When dealing with multi-axis machines, the post-processors have the knowledge needed to verify rotary limits and automatically retract or reposition the axis of the machine. They will warn of 5-axis instabilities, being able to output rapid rotary motions as programmed high feed rates better to control all aspects of a machine´s motions.

An essential factor to recognize is that: *"There are always two possible solutions when a post-processor maps a 5-axis tool´s kinematics"* [9]. The selection of the best option is made by the post-processor, and so, when dealing with multi-axis machines, it is advantageous having a high processing power post-processor since it will result in safe motions and faster G-code management.

Nowadays, the CAD/CAM systems make the post-processing work very simple. There are large databases with almost every post-processor in the market with the option to make changes if needed, leaving the user with the single need to select the proper one, and G-code generation will automatically be done.

2.4.5 Programming

After establishing all cut strategies and parameters, the next task is centred on creating the code that can be read by the machine, in order to perform the movements that lead to the part creation.

Commonly named G-code (or G and M-Code), the official name of this language is RS-274D and it was designed to be as compact as possible. The modern machine tool language is the safest and most efficient way developed to control machine tool motion. Both G-code and M-code related to milling machines are presented, respectively on Attachment II – Mach3 Milling G-Codes and Attachment III – Mach3 Milling M-Codes.

A CNC program is a list of instructions that will be read by the machine exactly as it is written.

The sentences appear on separate lines that are called "block", usually arranged in the following order [11]:

1. Program Start;
2. Load Tool;
3. Spindle On;
4. Coolant On;
5. Rapid to Position Above Part;
6. Machining Operation;
7. Coolant Off;
8. Spindle Off;
9. Move to Safe Position;
10. End Program.

On image Figure 2.22. is represented a small part of a G-code file retracting some of the points presented.

```
(1001)
(T1  D=8. CR=0. - ZMIN=5. - FLAT END MILL)
G90 G94 G91.1 G40 G49 G17
G21
G28 G91 Z0.
G90

(FACE3)
M5
T1 M6
S3000 M3
G54
G0 A0. B0.
M8
G0 X35.573 Y-30.173
G43 Z79.522 H1
Z69.522
G1 Z65.322 F2000.
G18 G3 X34.773 Z64.522 I-0.8 K0.
G1 X32.
X-32.
G17 G2 Y-25.396 I0. J2.388
```

Figure 2.22 G-code extract, with the header retracted as well as the tool used, spindle speed and the coolant option before the machining movements start.

The language uses the alphabet letters as a machine address code, some are even used more than once, with their meaning changing depending on which G-code is presented in the same block. Some codes are modal, meaning they will stay active until being cancelled or changed, other are non-modal, being active only in the current block.

While codes starting with G are called preparatory, being that they prepare the machine for a defined type of motion, codes commencing with M are termed miscellaneous, controlling the machine´s auxiliary options. All the common alphanumeric address codes are presented on Attachment I - Common Alphanumeric Address Code.

As far as developing a code for a CNC machine, there are three principal styles of programming: Manual Programming, Conversational Programming and CAM System Programming [28].

Manual programming, which requires the programmer to know the specific G-code recognized by the machine where the part will be produced, visualize the machining operations needed to make the part and manually write the entire program.

Modern CNC machines have simplified the manual approach, using fixed or repetitive machine cycles, among other functionalities, however, manual programming is a big-time consumer and prone to human error.

Nevertheless, a well-formatted manually written program allows for an optimized machine performance better than any computerized method, For this reason, manual programming is used as a troubleshooting method for code generated in CAM systems.

Conversational Programming style is used in CNC machines that posses conversational programming capabilities (a wizard-like mode that either hides the G-code or completely bypasses its usage).

Using this method, programmers can easily generate a program on the machine in minutes. There are a series of built-in data prompts, allowing to define part geometry, workpiece material, tooling, etc.

Although conversational programming is best suited for simple part geometries, it allows for a toolpath check before production begins, facilitating program editing.

CAM System programming, has seen before, uses CAD/CAM systems to create the programs allowing from toolpath simulation to full machine motion simulation.

This method comes in handy when there are multiple types of CNC machines to program, when working with multi-axis machines, or even when working with complex parts.

To clarify even more the existing programming styles, Table 2 retracts the pros and cons of each style.

Table 2 Pros and Cons of the Different Programming Styles

Manual Programming	**Pros**	**Accomplishes the best machine optimization performance**
		Promotes a high level of "intimacy" between machine and men
		Virtually unlimited freedom in part program development
	Cons	Tedious, time-consuming manual calculations and verifications

		Numerous calculations increase the risk of human error
		An intense process that requires total involvement of CNC programmer
Conversational Programming	**Pros**	Easy to learn and simple to use
		Reduced setup and programming times
		More cost-effective than a CAM system
	Cons	Limited to more basic part geometries
		Unable to support complex toolpaths
		Does not offer the flexibility of a CAM system
CAM System Programming	**Pros**	Automates the programming process for an overall boost in total output
		Virtual simulation reduces machine crashes due to programming errors.
		Allows creation of complex toolpaths
	Cons	Efficient programming management still requires basic manual programming knowledge
		The program output is less efficient than manual techniques

One other point to consider, when transitioning from the virtual world (CAD/CAM system) to the physical world (real machine), some is referent to tool and work offsets.

Every tool inserted into the machine has a different length since it is almost impossible to set a new tool in a holder in the exact same position as the previously placed. For this reason, the CNC machine needs information regarding how far each tool extends from the spindle to the tip. This is performed by using tool length offset (TLO), which is referred to in the G-code by the letter H (most machines have additional information withing TLO referring to the tool´s diameter and/or diameter compensations with the letter D on the G-code).

The most common approach to define the tool limit offset consist of jogging the spindle with the wanted tool from the machine home Z-position to the tool setting point on the machine (this can be, for example, the to of a tool probe) [11].

With the constant innovation on the CNC world, there are currently machines (especially multi-axis machines) with built-in devices to facilitate establishing the tool limit offset, which can be programmed when elaborating the CAM program. The company "Blum Novotest" [29] manufactures these type of systems, just like the one from Figure 2.23.

The LC50-DIGILOG can be assembled to existent machines (products like this are heavily used on the aerospace industry).

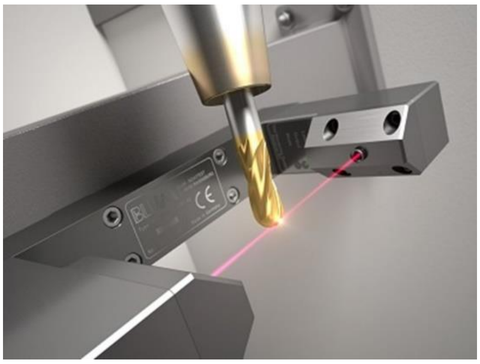


Figure 2.23 LC50-DIGILOG [30]

In reality, these type of systems are so advanced that they can also recognize geometric changes like insert or blade wear, and employing concentricity control, the device also recognizes poor tool fixation and/or contaminations in almost every type of tool (Figure 2.24).

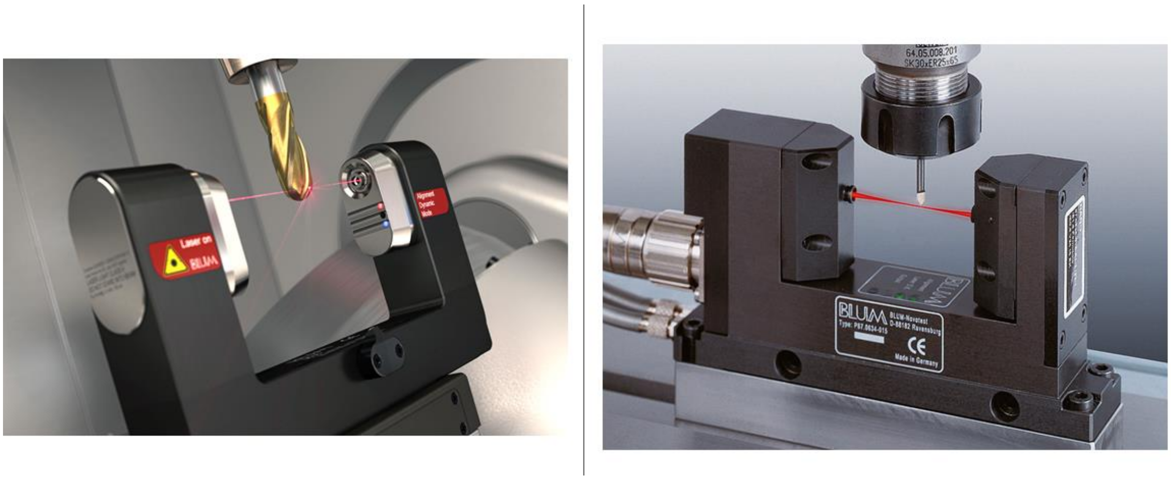


Figure 2.24 Laser Control Systems by Blum Novotest [31]

Furthermore, it is also vital the work offset. Usually when turning on a CNC machine, the axis need to be moved to the Zero Machine Point. Only after this, the machine knows where it is. Following this, and with the fixture placed, using G-code functions from G-54 till G-59 (depending on how many axis the machine has), the work offset is defined. This process is done using auxiliary tools, like edge finders or probes, which are placed on the spindle and moved around the fixed part in order to establish relevant coordinate points, providing the machine with information need to be aware of where the part is.

Other work offsets are presented, which are related to the CAM program and provide information to the tool´s dislocations, but all depend on the work offset introduced into the machine.

One last important concept when it comes to programming is cutter compensation, which is a way to adjust the toolpath at the machine to compensate for tool size, tool wear and tool deflection.

Being a powerful instrument, cutter compensation allows adjusting slight tolerances issues that are detected rather than regenerating all the G-code. Even uncalibrated or poorly aligned tools can be recalibrated [32].

An additional reason to use cutter compensations is that it allows programming the tool tip center, this means that the coordinates in the program do not reflect the actual tool cutting edge coordinates because they are based in tool tip center and not on parts dimensions.

When programming, cuter compensation is employed with a G40, G41 and G42. G41 is used for left climb with a right-handed cutter direction, which means it will compensate to the left-hand side of the programmed path, whereas G42 is used for a right conventional approach, implying the tool will move along the right-hand side of the programmed path as shown in Figure 2.25 (G40 is used to turn cutter compensation off).

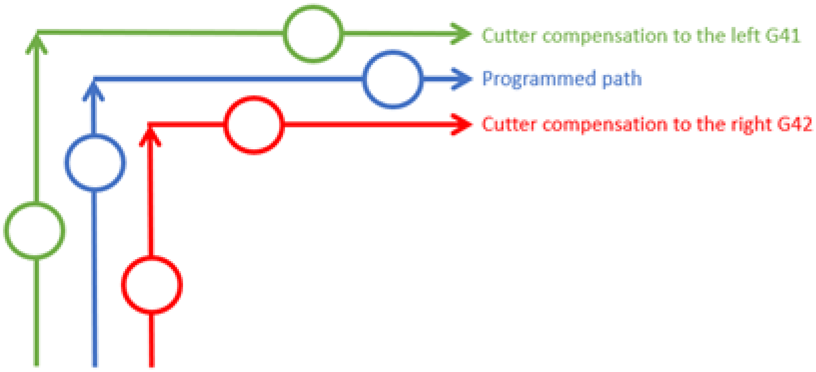


Figure 2.25 Difference between G41 and G42 [33]

While using CAM systems, cutter compensation is defined by operations and there are generally five options: In computer; Wear; Inverse Wear; OFF [34].

Using cutter compensation in computer causes the toolpath to offset from the geometry by the radius of the selected tool, being mostly used in roughing operations. This option is usually the default on most CAM systems.

Applying in control compensation will lead the toolpath center line to follow the geometry. Being and older method, it is normally used when manually programming, saving a lot of time in comparison to calculating offset toolpaths manually. It is usually used when it is possible to combine CAM software with G-code manually written.

Employing wear compensation, the toolpath center line outputs offset from the geometry by the radius of the selected tool just like when using in control. *"The difference is that the diameter in the offset table in the machine control is typically set* *to zero or the small difference between the programmed tool diameter and the actual measured diameter"* [34].

The inverse wear compensation is identical to wear but the programmer only enters the wear adjustment.

If too compensation is not required, by selecting off the toolpath centerline will follow the geometry on center, since there is no compensation.

2.5 Working Protocols for 5-axis Machining

Transitioning to 5-axis machining has many more implications when compared with regular CNC machines. The parts to be made need to be examined on various levels, and the comprehension of both cutting strategies and machine functionalities have a tremendous impact on the work.

Furthermore, the notions presented next are divided with the purpose of providing a better understanding of themselves, however they must be seen and analysed as a whole since they are dependable on each other.

2.5.1 Sketch Considerations

With the perception that multi-axis machines are a lot more complex than regular CNC machines, every step in the process of programming a part requires special care. Commencing with the sketch(es) of the part(s) to be machined, it must be studied, not only regarding the defined parameters but when it comes to 5-axis machining. This means understanding if the part can be fully or partially machined utilizing an indexing approach since it usually if the part can be fully or partially machined utilizing an indexing approach since it is usually more accurate due to compounding error (less axis in motion, meaning working with 2+3 axis).

Indexing also leads to easier and linear programming.

Likewise, it is highly critical to understand and select the suitable available strategies, since some of them are not desirable or even technically capable while using multi axis.

Generally, with 5-axis machining, a sketch must be analysed in a way that the programmer visualizes both 3D model, fixture setups and cutting strategies, these three concepts must be inherent.

2.5.2 Positional Considerations

Referring now to the stage of positioning the stock on the machine, it is crucial to visualize if the machine can reach the desired positions, keeping in mind tha most of the times the rotary movements are limited (axis limits may not allow for the various faces to be reached).

In multi-axis machining, fixturing may be dependent on what options or capabilities the machine and controller have, and one of the most important concepts to grasp is the machine rotary zero position (MRZP), which is, fundamentally, the intersection point of the rotary/pivoting axis as shown in .

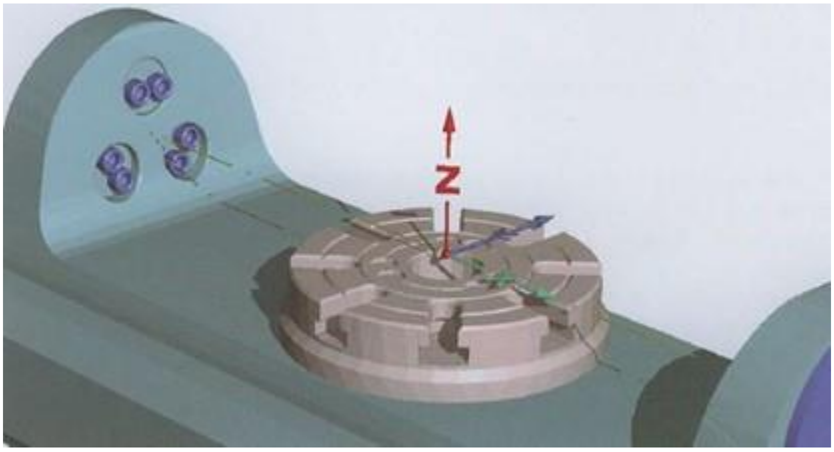


Figure 2.26 Machine Rotary Zero Point [9]

For multi-axis machines that do not possess advanced controllers, this point must be coincident with the program zero point (PZP).

Finding the precise center of rotation is indisputable, the most important aspect to achieve accuracy when working with multi-axis.

2.5.3 Programming Considerations

When it comes to programming, the critical factor to wonder about is the machine´s controller.

Controllers are the “brain” of the machines, and sometimes, even if utilizing the best CAM systems, certain strategies should not be used since the post-processor does not have the capacity to process and command the machine to execute them.

For example, when indexing a part, the programming part can be made from different approaches that will make the job easier or simpler. The most basic method is using multiple offsets (uses separate work offsets for each rotational position or face), but this can lead to considerably longer setup times and, in the worst-case scenario, the number of work offsets available on the machine may be a concern [35].

A more advanced method such as dynamic workoffset (DWO), which makes the controller compensate for any offset in the machine kinematic or the location on the table of the machine (Figure 2.27).

Figure 2.27 Dynamic Workoffset Option [36]

There are even other options like incline plane or tilted working plane, but those require especially good processors to be employed.

Other beneficial features for multi-axis programming are inverse time, which instead of a speed rate, instructs the machine how much time will take to move from a set position to one other determined position, leading to significantly improved surface finishes. Vector programming that allows a more liberally way to control the tool´s orientation. And one advanced method is known as RTCP (Rotational Tool Compensation Point) or toolpath linearization which synchronizes all the axis, including the rotational axis with the linear axis and ensures that the tool tip follows the intended trajectory (Figure 2.28).

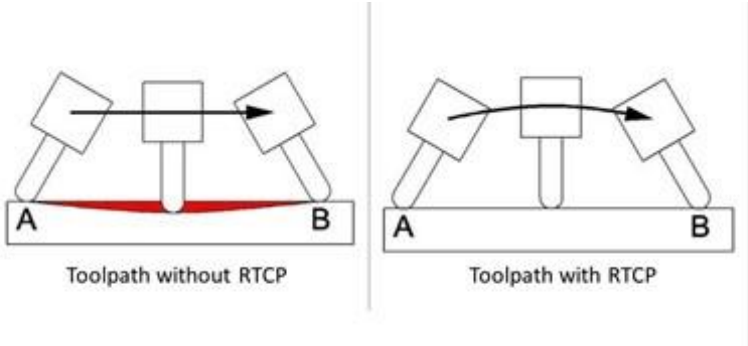


Figure 2.28 Rotational Tool Compensation Point [35]

It is easy to observe that, without RTCP, the tool simply rotates from position A to B and the tooltip travels through a position deeper than expected (area in red on Figure 2.28), while using RTCP the axis dynamically adjust so the tool tip stays along the intended direction.

At this stage of the dissertation, all subjects needed to comprehend before advancing to the real machine work have been aborded. And it can be concluded that, while working with 5-axis CNC machines, it is crucial to first analyse the machine in question and understand clearly what type of machine configuration it is. Knowing and exploring the existent workholding solutions compatible with the machine and which one is the most suitable to the part necessary to manufacture will also make a difference in terms of the quality the parts will present.

One other point is the extreme importance of CAM systems. Everything related with machine movements is programmed on these types of software, meaning that in some cases it is preferable a weaker machine but a very capable CAM system, never forgetting the importance of a programmer with full understanding of the said software.

3 MACHINE CONSIDERATIONS

Before working with any CNC machine, it is important to comprehend both machine architecture and controller. Based on this, this chapter will contain a complete machine characterization in order to establish a proper and efficient workflow.

The post-processor development/edit process will also be presented and explain in this chapter, culminating on a defined workflow for an optimized machining practice.

3.1 Machine Description

The CNNC machine utilized/operated is the HY-6040 5-axis CNC Router, and it has a table/table configuration as showed on Figure 3.1.

Figure 3.1 HY-6040 5-Axis CNC Router

One important observation regarding this machine is that the axis designation does not follow the right-hand rule. Instead, the designation is the one assigned by the manufacturer in which the A axis rotates around Z and the B axis rotates around X.

One other aspect to pay attention to is that, when it comes to the cutting movements, they are executed by the table (with the exception for the Z-axis). This implies that, for instance, when the X-axis is moving in a positive direction, according to the machine´s controller, the cut is occurring on a negative direction.

These considerations presented are extremely important and will be addressed further on this dissertation resulting on the axis respecting the right-hand rule working like it is represented on Figure 3.2.

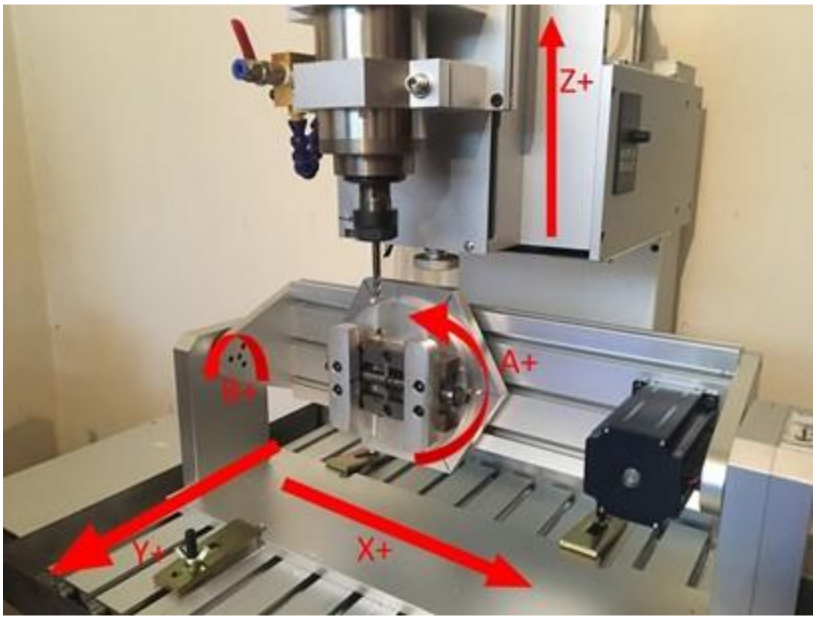


Figure 3.2 HY-6040 Axis Movements

The machine´s controller is Mach3Mill with an interface capable of operating up to 6-axis as shown in Figure 3.3.

Figure 3.3 Mach3Mill 6-Axis Interface

In order to correctly install/setting up the machine, the first thing to do is to align all the axis.

It is important to understand that the machine setup should be done in agreement with the working method intended for the manufacturing process, when it comes to the work offsets definition.

The established setup was created after running some initial and simple parts programs that allowed for a better machine and post-processor comprehension.

Following on the matter, the concept of the machine rotary zero point (MRZP) is the imperative idea to acknowledge. Being this a 5-axis machine with a table/table configuration, the MRZP is the intersection point of the A and B axis and will always be defined as the zero-part point when programming with more than three axis (also for some 3-axis parts depending on user preference).

Fundamentally and on a concise way, the MRZP is set upon assuring the alignment of all axis.

The procedure should be executed utilizing probe routines or an edge finder, the tool must be moved to both minimum and maximum limit of the table along the Y-axis and the middle point of this trajectory should be set to zero.

Upon this, the B axis should be rotated to -90º and +90º (Figure 3.4) and the tool must touch the table in each position in order to register the coordinate values of those positions. Once more, the middle point of this trajectory needs to be determined and now, the exact midpoint between this last one and the zero that was obtained before needs to be calculated and set to zero-Y.

Figure 3.4 B-axis Rotation (a) -90⁰ (b) +90⁰

When it comes to the X-axis, and since the B-axis rotates around this one, it is only necessary to move the tool to the minimum and maximum position of the table, and set the midpoint of this trajectory has the zero-X.

The only coordinate left to determine is the one relating to the Z-axis, and it is located in the center of the bearings that allow the B-axis to rotate (meaning it is at the same level as the imaginary line that defines the B-axis).

Moving the tool to that point and setting the coordinate to zero completes the procedure and the MRPZ that will be used for the workoffsets from G54 to G58 is defined following the constructed model presented on Figure 3.5.

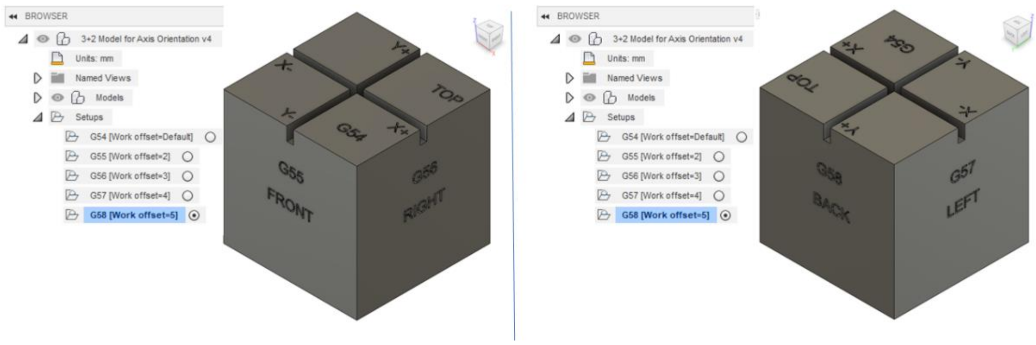


Figure 3.5 Model for Axis Orientation

The correct determination of the MRZP is the major aspect that contributes to the machine´s precision.

Being this a multi-axis machine with two rotary axis, the configuration of the rotary motors is also an important factor that influences the precision, since these are angular axis.

For this machine however, the manufacturer provided the defined values.

Nevertheless, and envisioning possible future changes of motors, the values of these types of motors are introduced in steps per degree and to achieve the said values, firstly, it is necessary to uncover how many steps the motor takes to deliver on complete revolution. After this, and knowing the drive ratio of the rotary axis, the values are multiplied and subsequently divided by 360 (degrees of a complete revolution), resulting on the steps per degree value required.

An additional action for the machine´s setup is to settle the issue presented before and make the machine respect the right-hand rule. The solution to this matter has two stages, the first one is performed on Mach3Mill inversing the Y-axis direction. The second stage is solved by programming the post-processor to attend the machines 'necessities and will be addressed in the subsequent sub-chapter.

3.2 Post-Processor

Knowing the machine´s controller is Mach3Mill, and since the software used for CAM throughout this dissertation is Autodesk Fusion 360, it is needed a post-processor compatible with these requirements.

Although Fusion 360 has an extensive post-processor library, in which the Mach3Mill is located as shown in Figure 3.6, none of these default post-processors files are prepared for 5-axis machines. This comes from the fact that each 5-axis post-processor is built accordingly with the machine´s structures and capabilities, which makes it impossible to have an "universal 5-axis post-processor".

The post-processor has functions for 5-axis that are deactivated, meaning that it is only necessary to activate and adapt these functions. One other particularity concerning this topic is that Mach3Mill post-processor only has functions to activate the fourth axis, requiring the addition of the fifth axis.

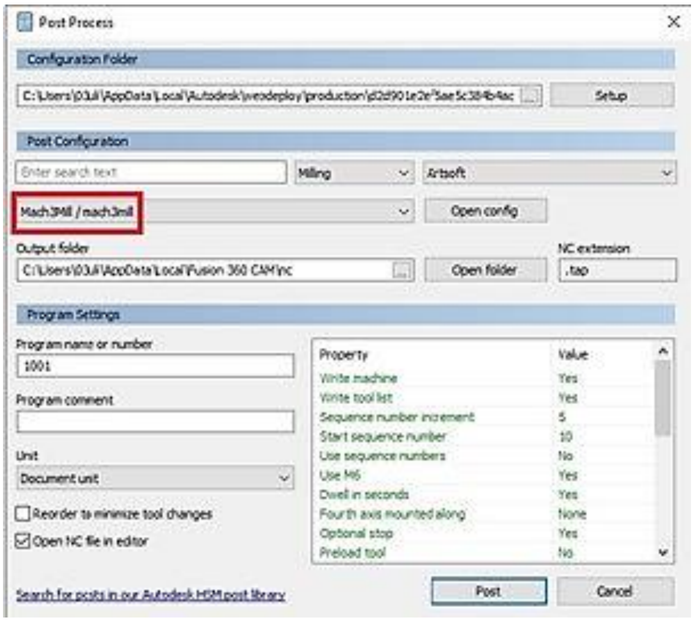


Figure 3.6 Mach3Mill Post-Processor

The post-processors used by Fusion 360 are programmed in JavaScript with a CPS extension, and to facilitate the edit assignment the software Visual Studio Code [37] will be used.

Furthermore, the Visual Studio Code software will also be applied to analyse the G-code files, since this software allows to work in different code languages and it also allows the installation of extensions to simplify G-code interpretation.

The extensions installed and used throughout this dissertation are: *"Autodesk Fusion 360 Post Processor Utility"* which is the one used it comes to post-processor editing practices, and *"G-Code Syntax"* that adds language syntax for CNC G-code and colorization for and easy examination of the programmed lines, both are presented on Figure 3.5.

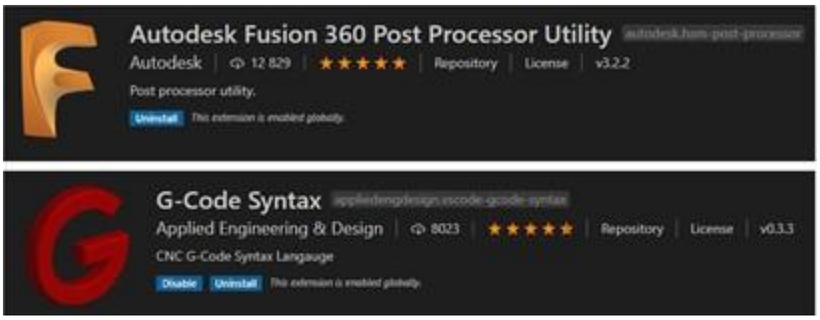


Figure 3.7 Extensions Added on Visual Studio Code

The first line of code to edit is: “function onOpen()”, and just by writing true in the place shown on Figure 3.8 it becomes enable.

```
function onOpen() {
  if (properties.useRadius) {
    maximumCircularSweep = toRad(90); // avoid potential center calculation errors for CNC
  }

  if (true) { // note: setup your machine here . . . Ulisses edited this on 15/07/2020
    var aAxis = createAxis({coordinate:1, table:true, axis:[1, 0, 0], range:[-90, 90], preference:-1});
    var cAxis = createAxis({coordinate:0, table:true, axis:[0, 0, 1], range:[-360, 360], preference:1});
    machineConfiguration = new MachineConfiguration(aAxis, cAxis);

    setMachineConfiguration(machineConfiguration);
    optimizeMachineAngles2(1); // TCP mode
```

Figure 3.8 Mach3Mill function onOpen()

On the same figure, the coordinate specifies the coordinates used in the ABC vectors, the given number will define the letter for the axis: 0 = A; 1 = B; 2 = C.

It is visible that in the post-processor edited, there are both coordinates 0 and 1, which means the post-processor will output values for A and B axis, just like the axis defined by the manufacturer.

The “table:true” specifies that both axis are located on the table, meaning the machine has a table/table configuration. Axis specifies the axis vector as a 3-element array meaning that: [rotates around X, rotates around Y, rotates around Z], and he position where the number one is placed is activated. By examining Figure 3.8 it is concluded that the B-axis (coordinate:1) rotates around the X-axis , and the A-axis (coordinate:0) rotates around the Z-axis just like the machine HY-604 model.

The range is a safety measure, defining the angular range (minimum and maximum) for each axis. The A-axis was defined from: [-360, 360] because this one needs to be able to rotate freely and the B-axis is limited between: [-90, 90] to avoid collision with the tool, spindle or machine structure.

Finishing this point with the preference, this parameter stipulates the preferred working angles, -1 is for negative angles, 1 is for positive angles and 0 if there is no preference. Once more, to avoid collisions relative with the B-axis movement, for this axis it was stipulated negative movements.

The line:” machineConfiguration = new MachineConfiguration(aAxis, cAxis)” is crucial and it is what really creates the new machine configuration defined above.

It is important to referred that, this aAxis and cAxis were created based on the right-hand rule, but in reality, the axis created on the post-processor were A-axis rotating around the Z-axis and the B-axis that rotates around the X-axis, respecting the architecture defined by the manufacturer.

The last step on the edit concerns the TCP (Tool Control Point) mode which only has two different modes: on (defined 0) and off (defined 1). Since the machine in question does not have TCP mode, the same is turned off.

Nonetheless, and even after all the changes made to the post-processor, there was one error that kept transpiring: the tool length offset defined on Mach3Mill was not active.

To further understand the reason to this issue, a G-code file was created using the edited post-processor and it was visible in the header that the line G49, meaning that tool offset is deactivated and a G43 further on the code after the first G0 rapid positioning activating this way the tool offset, leading to a greater uncertainty.

This type of issues results since, sometimes, the machine´s controller interprets the G-code on “its own way”. So, in order to get the post-processor to provide G-code files consistent with the controller, the aspects that didn´t affect tool orientation were changed and tested, and it was concluded that, changing the “table” parameters would also introduce a minor change to the G-code files sufficient to get the machine working properly.

Figure 3.9 shows the changes in which “table:false” is set for both A and B-axis.

```
function onOpen() {
  if (properties.useRadius) {
    maximumCircularSweep = toRad(90); // avoid potential center calculation errors for CNC
  }

  if (true) { // note: setup your machine here . . . Ulisses edited this on 15/07/2020
    var aAxis = createAxis({coordinate:1, table:false, axis:[1, 0, 0], range:[-90, 90], preference:-1});
    var cAxis = createAxis({coordinate:0, table:false, axis:[0, 0, 1], range:[-360, 360], preference:1});
    machineConfiguration = new MachineConfiguration(aAxis, cAxis);

    setMachineConfiguration(machineConfiguration);
    optimizeMachineAngles2(1); // TCP mode
```

Figure 3.9 Changes on the Table Parameters

Basically, the changes inflicted on the G-code files by using this new edited post-processor are referent to the G43 that is now located on the same line as the first G0 rapid positioning. It is illustrated on line 15-16 of Figure 3.10, the difference obtained on the code using the two different post-processors.

```
2   (T1  D=8. CR=0. - ZMIN=5. - FLAT END MILL)
3   G90 G94 G91.1 G40 G49 G17
4   G21
5   G28 G91 Z0.
6   G90
7
8   (FACE3)
9   M5
10  T1 M6
11  S3000 M3
12  G54
13  G0 A0. B0.
14  M8
15  G0 X35.573 Y-30.173
16  G43 Z79.522 H1
17  Z69.522
18  G1 Z65.322 F2000.
19  G18 G3 X34.773 Z64.522 I-0.8 K0.
```

```
2   (T1  D=8. CR=0. - ZMIN=5. - FLAT END MILL)
3   G90 G94 G91.1 G40 G49 G17
4   G21
5   G28 G91 Z0.
6   G90
7
8   (FACE3)
9   M5
10  T1 M6
11  S3000 M3
12  G54
13  G0 A0. B0.
14  M8
15  G0 G43 X35.573 Y-30.173 Z79.522 H1
16  Z69.522
17  G1 Z65.322 F2000.
18  G18 G3 X34.773 Z64.522 I-0.8 K0.
19  G1 X32.
```

Figure 3.10 Changes on the Location of the G43 Codes

Upon this point, the post-processor is ready for the 5-axis work, depending only on the machine´s controller capabilities.

Nevertheless, there was one other edit made on the post-processor with the purpose of enabling one option where the programmer will be able to introduce manual G-code lines from Fusion360.

This edit was created after running some 3+2 programs and verifying that there was an error that needed to be fixed (this matter will be addressed and considered later on the dissertation).

The option added is a pass through and the edit added to the post-processor is visible on Figure 3.11.

```
/**
 Ulisses add this on 15/07/2020. Objective: allows to introduce manual G-code orders from Fusion360.
*/
function onPassThrough(text) {
  var commands = String(text).split(",");
  for (text in commands) {
    writeBlock(commands[text]);
  }
}
```

Figure 3.11 Pass Through

Upon completing this task, the new edited post-processor has to be placed on Fusion360´s library. There are two possible locations within Fusion 360 (Figure 3.12) to allocate the file: the “Personal Post Library” which means the post-processor will be located on a folder inside the installation files of the program and can only be used on the computed that it is stationed, and the “Cloud Posts” that associates the post-processor file with the user´s account, allowing it to be used in any computer after user log in.

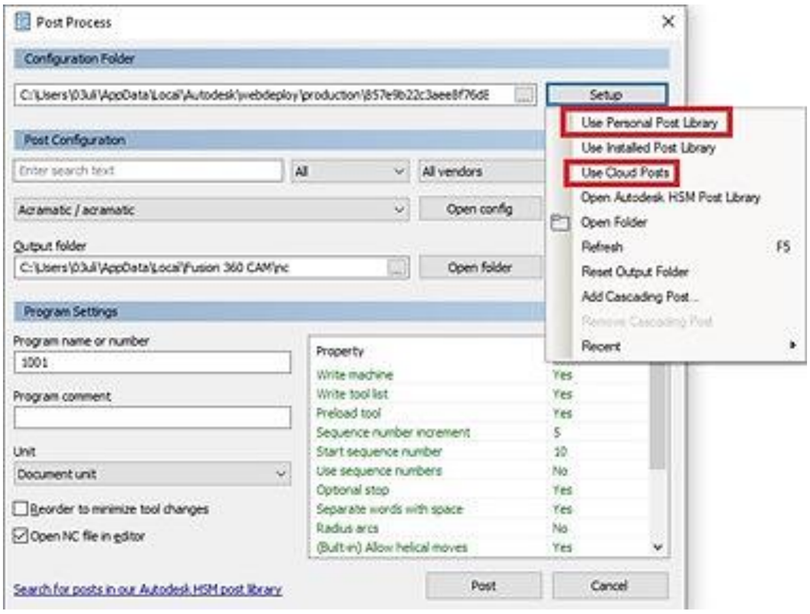


Figure 3.12 Post Processor Libraries with the option of saving the user files on the computer where Fusion360 is installed, or on a cloud associated with the user´s account

3.3 Machine´s Workflow

Acknowledging that the machine´s workflow is a decisive factor when it comes to minimize both preparation times and human error, the workflow created relies on the controller and post-processor features and it is in agreement with the subjects discussed on chapter 3.1.

Since this machine does not possess properties like dynamic workoffset and TCP, there were created five fixed workoffsets: G54, G55, G56, G57 and G58.

These workoffsets have their origin (point (0,0,0)) located on the MRPZ and diverge only on A and B axis values.

The MRPZ also needs to be precisely placed on Fusion 360 so that the part zero-point of the machine, established on Mach3Mill, is coincidental with part zero-point programmed.

Since the machine comes with a self-centering vise, which will be used for the totality of the parts made with multi-axis and even for several parts that require only 3 axis, a 3D model of the same was created on Fusion360 with two additional points as represented on Figure 3.13.

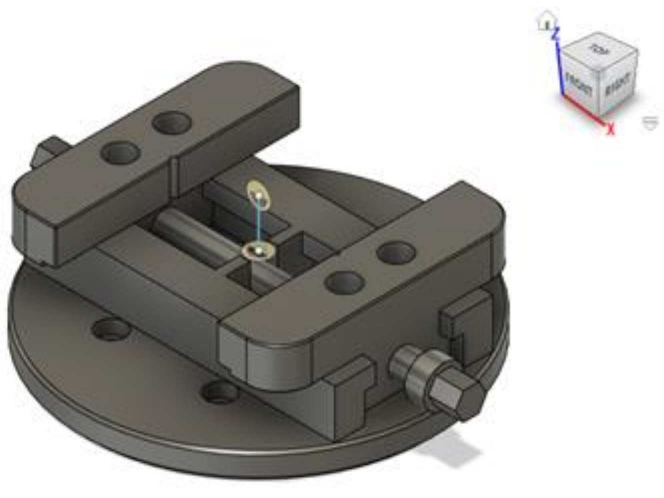


Figure 3.13 Vise Model developed to retract the physical one to be used on the machining process

All measures were thoroughly taken,not only the ones that the define the vise´s geometry but also the MRZP with it´s location represented by the top point (Figure 3.13) allowing the user to select it to establish the zero part-point. The bottom point exits as the contact point between the vise and the stock permitting a totally accurate positioning of the stock withing Fusion 360.

This solution means that the operator will never have to interact with the offset parameters on Mach3Mill, being that the only parameter left to introduce are the tools offsets.

To do that, and considering only the two available/used tools, it is required to program them on Fusion360 and on Mach3Mill, which can be done following the Attachment V.

The main notion regarding this matter is that, since the machine does not possess an automatic tool change system, the operator performs this action. Thus, to minimize errors, the procedure established defined that the inserted tool must always have a length below holder of 40 mm.

There is and additional workoffset (G59), and this is left dedicated to 3-axis work that requires a more rigid fixture solution (Figure 3.14).

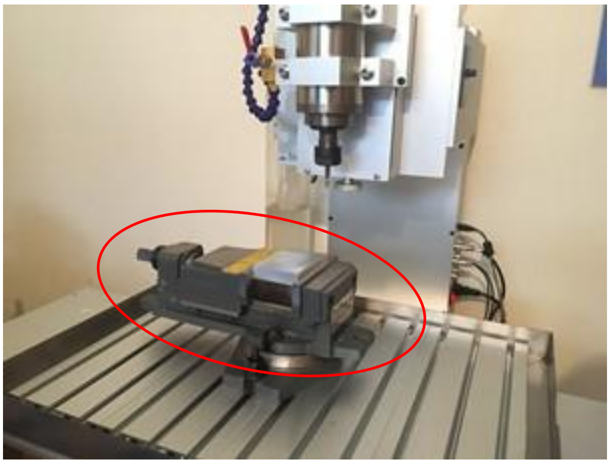


Figure 3.14 Different Fixture Used on G59

The zeros of this workoffset are defined for every new part according with the user´s preference, being the only requirement that it matches the part zero-point defined on Fusion360.

To conclude, the machine follows a table/table configuration being that the A-axis rotates around the Z-axis and the B-axis rotates around the X-axis. The post-processor to be used is defined and programmed, allowing the user to even had manual G-code lines from Fusion360 which will be the software used for CAM. The fixture solution relies on a self-centuring vise from which was created a 3D version and the MRPZ was defined as well as all workoffsets needed. Everything is now set to start manufacturing parts.

Upon the work presented on this chapter from where resulted the study and comprehension of the machine configuration and the creation of the workholding solutions and tools available within Fusion 360, and together with the programmed post-processor, the machine is ready for the manufacture if some parts.

4 WORKING PROCEDURES

Directing now the work for parts creation, this chapter is dedicated to the complete working procedure of different types of parts, exploring both 3, 3+2 and 5-axis continuous machining routines (Figure 4.1), as well as the G-code files and the parts manufacture with study cases concluding on the final results, obtained on the dissertation.

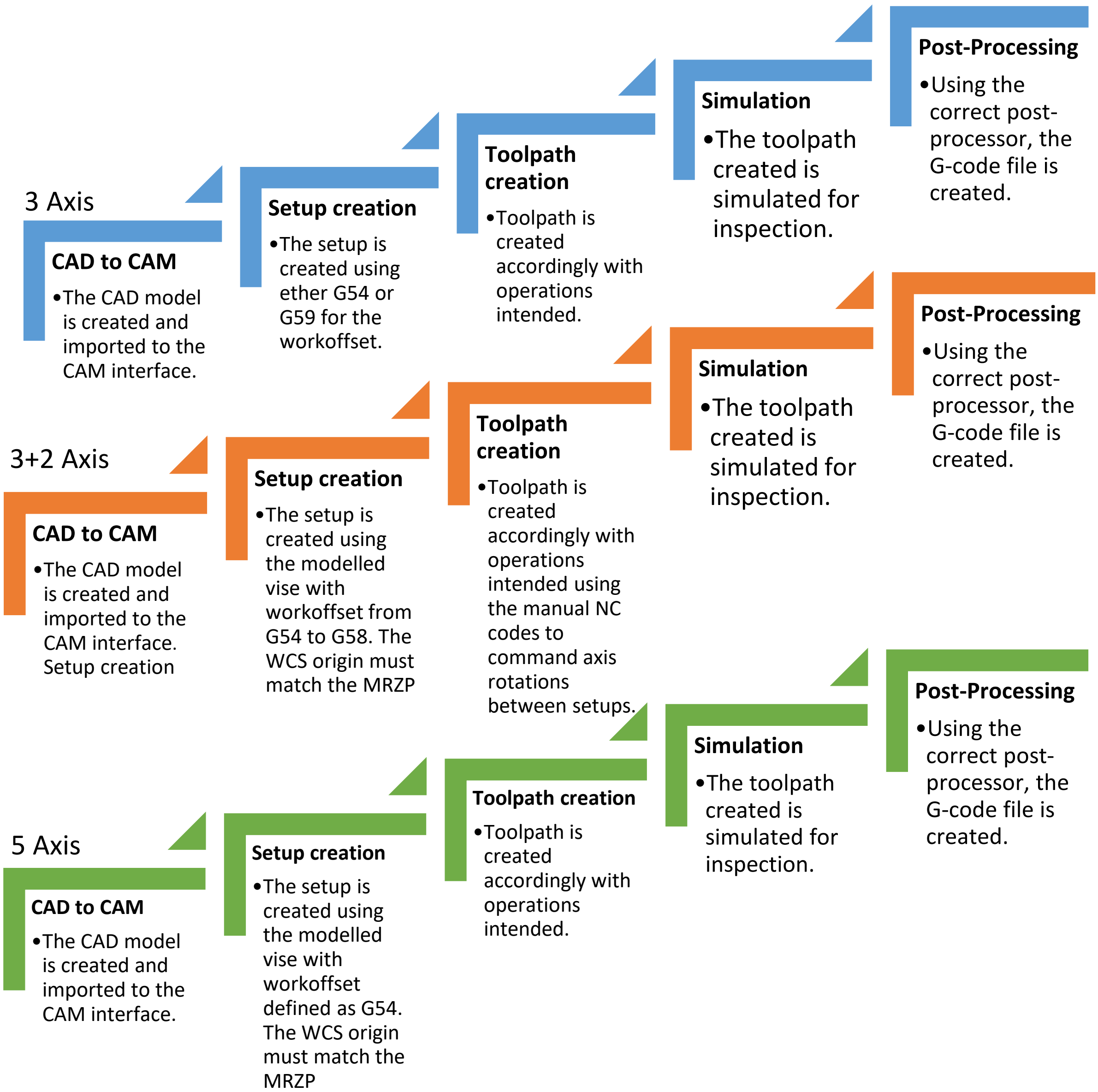


Figure 4.1 General Workflow of the Working Procedures defined for 3 Axis (blue), 3+5 Axis (orange) and 5 Axis (green)

This chapter should be analysed simultaneously with the Attachment IV and Attachment V that constitute both Fusion360 features study and an altogether manufacture workflow for the HY-6040 machine.

4.1 3-Axis Procedures

Before starting the CAM programming process, there are a few matters to be disclosed, the Fusion360´s licence used on this dissertation is an educational license, this means that all software features are fully available for use but, although most parts of them are already activated, if introduced the command: “CAM beta mode /on” on the “TEXT COMMANDS” panel of the software, there are some additional strategies that will be empowered.

Likewise, there are some extensions and add-ins that can be installed and used to facilitate the work. Upon this, all the work here presented is done with Fusion360´s manufacture extensions purchased and with the add-in “Visual Styles” that allows the user to change to wireframe view while working on CAM programs and while simulating cutting strategies leading to a better examination of the toolpaths created.

Additional information is that the workpiece material used for every part is extruded polystyrene (also known as roofmate). The reason for this choice is that, with this material, and since most of the parts were developed to study the machine capabilities, if any error that could potentially arm both machine and operator occur, the material is soft enough to avoid damaged, while being tough/rigid enough to be machined without deformation.

Acknowledging all this concerns, the programming process can now start.

The first part created has a simple geometry has represented on Figure 4.2 (for more detailed information the technical drawing of this part is presented on Attachment VI) that allows the employment of a few cutting strategies.

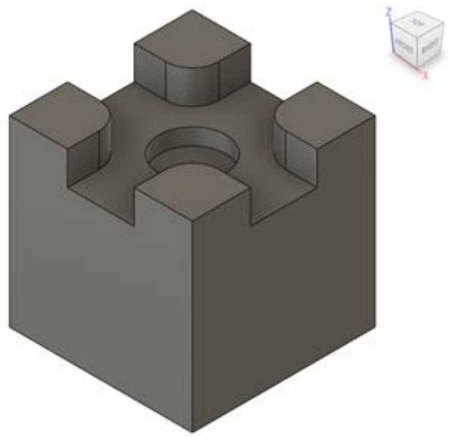


Figure 4.2 Part1 Machined With 3-Axis, with block dimensions of 55x55x55 mm

Upon having the 3D model of the part on Fusion 360, it is important to have the file saved in the same folder as the vise model. Only after this last step is completed, can the vise mode be imported into the part´s workspace.

This process, in conjunction with the procedure to accurately place the part on the vise, are thoroughly described in Attachment V and so, will not explained here. Nonetheless, this is the initial step of almost every program presented in this dissertation and it is and extremely important factor like it was clarified on chapter 3.3.

Working now on the Manufacture workspace, the first step for every CAM program is the setup definition, and by examining the changes on the model (Figure 4.3) it is understandable that the software is processing both model and vise as a one-piece part.

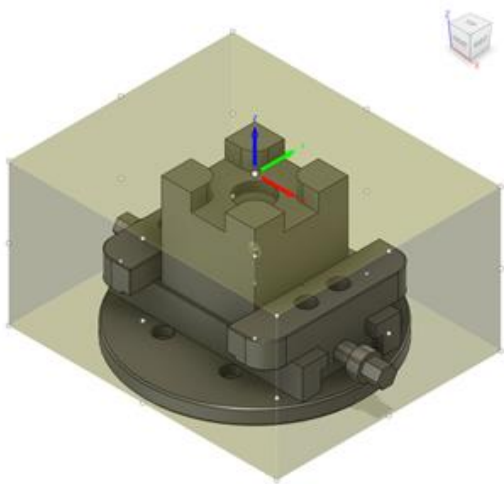


Figure 4.3 Part 1 Model and Vise Combined

This can be changed in setup definition, establishing that the model as being the part, and the fixture the vise. After this the MRZP can be selected as the WCS (work coordinate system) Origin and the axis should be correctly directed. The stock can now be created and, for this part, the stock was created with the relative size box mode, which allows to had material to the model part, ending up with 1.5mm stock side offset and a 2mm stock top offset. This means the stock for this operation has the following dimensions: width (x) and depth (Y) of 58mm and the height (Z) of 57mm.

Being this a 3-Axis operation on the top face of the stock, the WCS offset should be set to 1, meaning that when the G-code file is created, the post processor will generate the code with the G54 offset (Fixture 1 on the machine).

The model should now be displayed as seen on Figure 4.4, meaning the setup was correctly created and everything is prepared for cutting strategies creation.

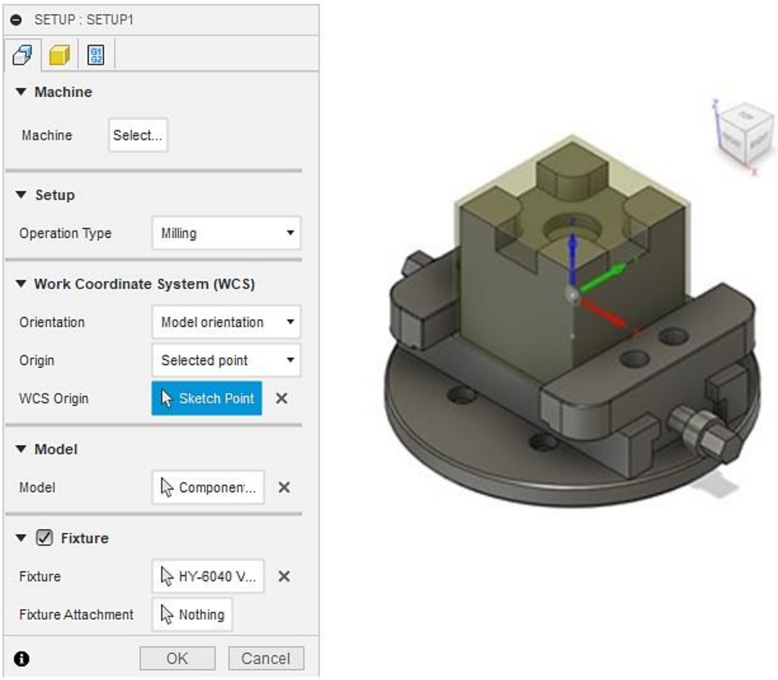


Figure 4.4 Setup for Part 1 with the definition of the work coordinate orientation and origin and also defining the fixture used

The first two operations will be roughing operations, the first one will be a face cutting and the second a 2D contour both destined to reduce the stock dimensions to match the model ones.

Proceeding with the line of thought, the face operation under the 2D panel must be selected and the operation´s customization menu will open.

The first tab of this menu is related with both tool as well as feeds and speeds. When it comes to the tool, there was created a library for the spindle and tools that belong to the HY-6040 machine and that will be used throughout all manufactured parts. This means that the tool selected may not be the proper/desired one, depending on the type of operation, due to the reduced number of tools available.

For this operation, the tool selected is a standard flat mill with a diameter of 8mm and regarding the coolant, the selection here won´t result in anything since the coolant circuit on the machine to be employed is not controlled by Mach3Mill but by an outside system.

Directing the attention now to the feeds and speeds segment, spindle speed and ramp spindle speed will both be set to 3000RPM, cutting feedrate will be set to 1500mm/min and the remaining parameters related with cutting feeds and speeds will automatically be recalculated and defined by the software.

The additional feeds and speeds parameters are related to non-cutting movements; however, these values have a tremendous impact on machining times, even more when considered the tool selected for the 2D face operation. Upon acknowledging this, lead-in and lead-out feedrate should be set to 1500 mm/min. and ramp and plunge feedrate to 1500 mm/min. Once again, the remainder of the parameters will automatically be recalculated by the software.

The reason for the feedrate values being all set to the same number is that this is the highest feed rate the motors can work at. If anytime the person creating the CAM program to a value greater than this, Mach3Mill will read the established parameters but will only work at the maximum capacity (1500 mm/min).

Still, throughout this document, the feedrate values assigned on the programs will not take into account this aspect, in order to better expose/explain the programming dynamics one must have while using Fusion360 for CAM.

The second tab (geometry) automatically recognized the part contour, which means, for this type of operation, there is no changes needed as showed in Figure 4.5.

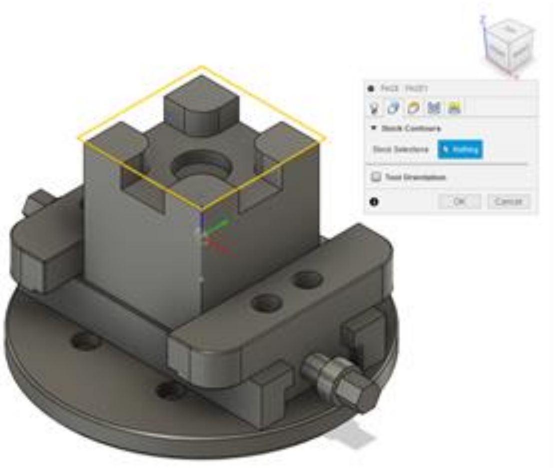


Figure 4.5 Face Operation Geometry Tab

Moving to the next tab (heights), this is where Z height limits are set, represented on Figure 4.6.

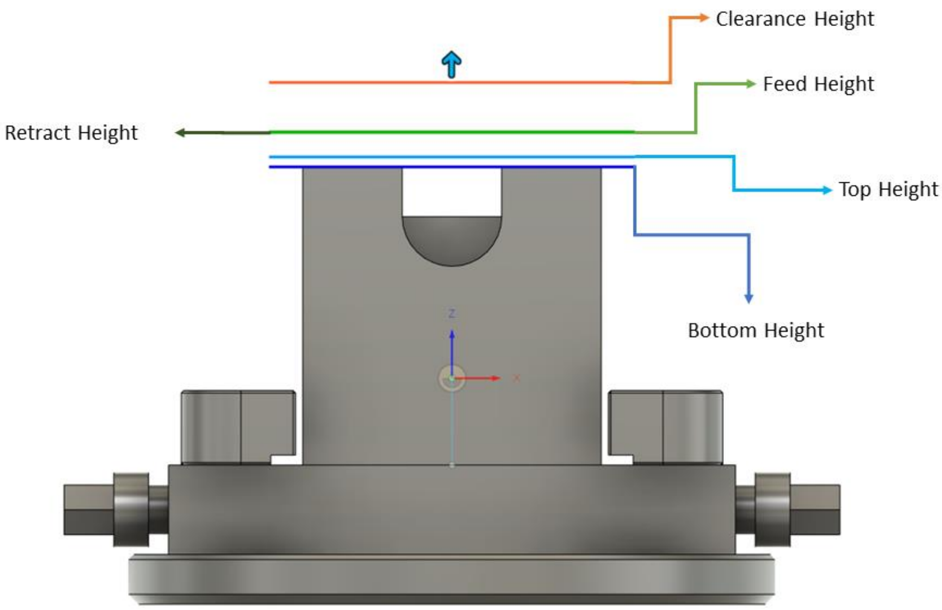


Figure 4.6 Face Operation Heights (clearance; retract; feed; top; bottom) Heights Tab, is this specifi case the retract is coincidental with the feed height

There are essentially five height planes for customization: the first one is the clearance height and it specifies the height to each the tool tip will rise when the operation is over, although when working with only 3 axis this parameter just needs to be set above the stock height value, when working with multi-axis this is of extreme importance to make sure the tool does not collide with the table or vise.

The second plane refers to the retract height and establishes the Z value to which the toll will rise after every pass, however this specific parameter may not be involved, depending on further cutting options selection.

Progressing with the third option: feed height, here the height to where the tool rapid moved before changing to feed/plunge and enter the part.

The last two constraints, top and bottom height set the cutting domain in Z, meaning the machine will only cut material between these two plans.

Establishing all these values can be done is various ways, but usually it is easier to set top and bottom heights first, as selection, which allows the user to select the faces or edges to define the planes. The remaining values can be set bases on the first two.

Proceeding with the passes tab, this is where all factors related with the passes, just like the name indicates, are set. Attending to the work condition this tab will be used to combat possible errors that may or may not appear, due to the positioning of the stock on the machine.

Since it is not possible to use probe routines or even an edge finder to assist in the correct positioning of the stock, the mase is measured after being cut and placed by hand onto the middle of the vise. So, by creating a relatively larger pacing toolpath, and later the same for the 2D contour toolpath, it is assured that all the stock is machined correctly even though this also results in bigger machining times.

Upon this, pass extension is set to 2 mm, stock offset to 4 mm and stepover to 6 mm. Both ways cutting is activated, as well as multiple depths, creating a 1 mm maximum stepdown.

The last tab is related to both linking and leads/transitions movements. Since a face operation is a simple operation, on this tab the only parameter edited is the transition type, being set to shortest path, which makes the tool to rise to the previously defined feed height before engaging on the material cutting movements, like it is shown on the .

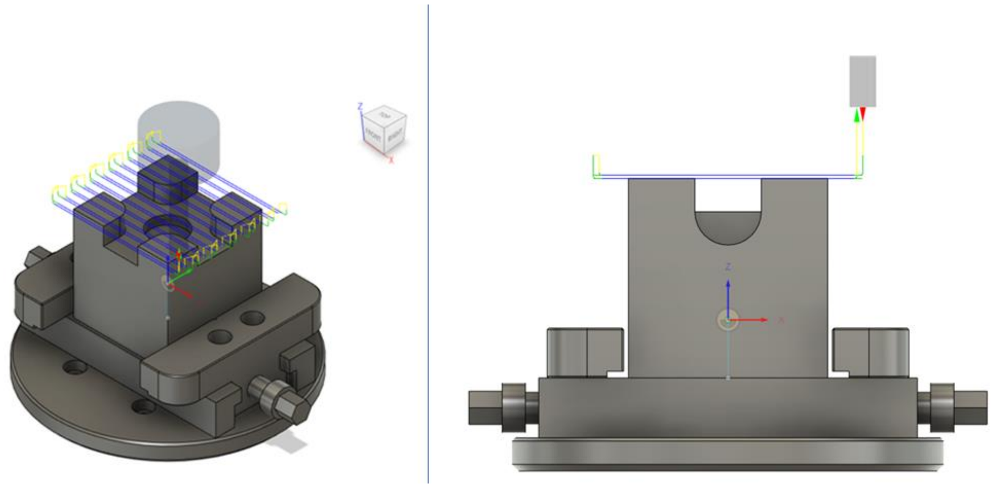


Figure 4.7 Face Operation Final Toolpath where it is visible the additional offset given the toolpath to assure that any error committed on the stock placement will not have ant result on the final part

Preserving both tool and feeds and speeds parameters from the 2D face operation, the following step is the creation of a 2D contour strategy.

For this, it is important to select, on the geometry tab, the model contour on contour selection, and to activate the stock contour so that the software knows from where to start and finish the cut.

Being the tool height a limiting factor, the part will not be entirely contoured, but only an height of 20 mm. This is defined on the heights tab.

On the passes tab, roughing passes with 3 mm of maximum stepover and a number of 2 stepovers should be specified, as well as multiple depths with a maximum roughing stepdown of 100 mm. This allows compensating for stock positioning errors, just like in the previous operations.

To finish this operation, in the linking tab, the option keep tool down must be enabled, eliminating unnecessary lifting movements and resulting the toolpath displayed on Figure 4.8.

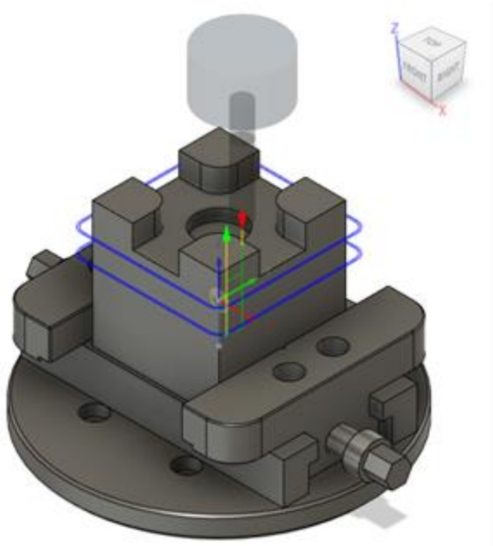


Figure 4.8 2D Contour Toolpath with 2+2 passes at different heights and with additional offset to ensure the final part dimensions

Upon this point the stock has the same brute measures as the part, and the only two operations left are the ones that will define the top face geometry.

The first of these operations is a 2D adaptive, and both tool and feeds and speeds will remain equal to the previous two operations. On the geometry tab it is crucial to select the pocket selection that delimit the area intended to be machined.

Being this a 2D cutting strategy, Fusion360 needs the user to provide all the information regarding the geometry (when using 3D strategies, the software automatically recognized the geometries) and the blue area that appears between the red arrows translates the area to be machined, which can be viewed for this part on .

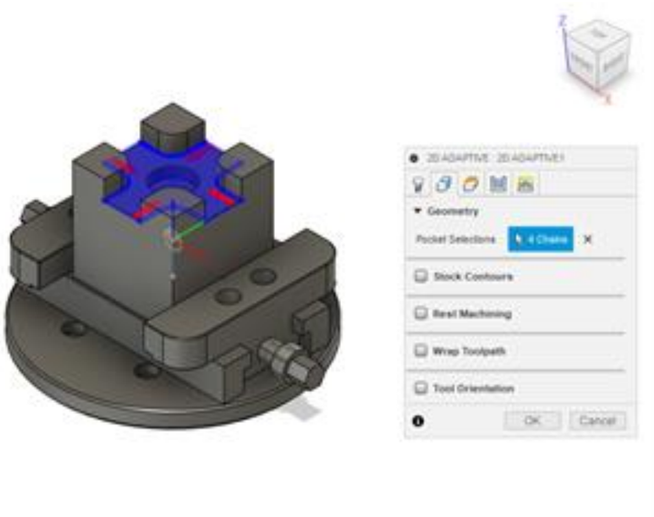


Figure 4.9 2D Adaptive Pocket Selection where the blue area is the one where the tool will pass according to the red arrows pointing directions

On the heights tab, top height should be set as model top since the previous face operation levelled the stock to the part height, and the bottom height must be set as selected contour(s).

On passes the both ways box should be enabled, allowing the tool to cut material in both directions. Multiple depths ought to also be enabled, and the maximum roughing stepdown set to 2.5 mm.

If a finishing pass is intended, the option stock to leave should be enabled, however, for this part in particular, stock to leave will be disabled and there won´t be any finishing passes.

To complete the strategy creation, on linking, the ramp can be changed to plunge which will reduce machining time compared to the default option helix, and since there are no constrains to limit tool movements (the borders of the part are not obstructed and the tool is able to penetrate the material from the sides), this is the best option.

The last operations will create a circular pocket, the tool will have to descend in a spiral movement. Upon descending the toolpath with get wither, cutting the material and defining the geometry, but this descending movement must be done with a higher feedrate than the cutting material. Therefore, spindle speed will remain on 3000 rpm, ramp spindle will be set to 5000 rpm and both cutting feed rate, lead-in feedrate and lead-out will remain at 3000mm/min, but ramp plunge feedrate will be set to 4000 mm/min.

On the second tab, and being this a closed geometry, it is enough to select bottom circle, which will lead to a correct definition of the heights plane by the software.

The passes tab is used to enable multiple depths and establishing a 2.5 mm of maximum roughing stepdown, and on the linking tab, the default configuration with and helix ramp type is the correct option for this particular operation.

The program is now complete and after running the simulation is visible not only the final result of the part (Figure 4.10) but also the verification that no collisions occurred.

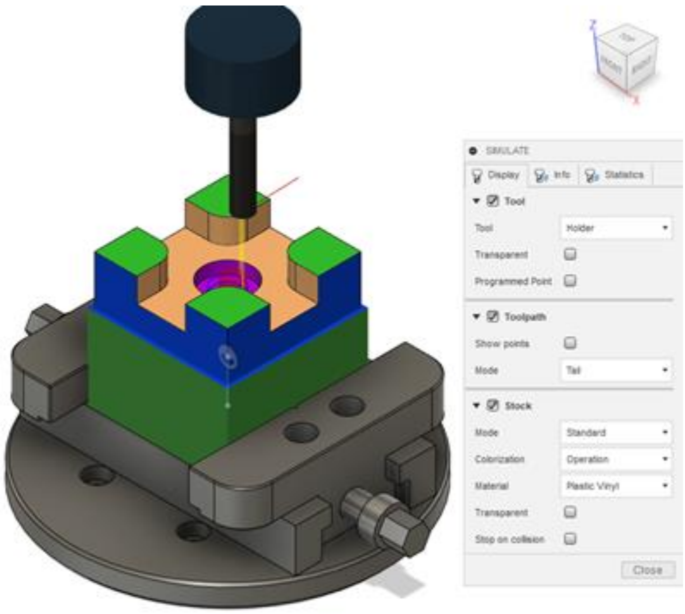


Figure 4.10 Simulation of the Programmed Part

To conclude the work on Fusion 360, the final step is to generate the G-code using the proper post-processor and selecting the entire setup, otherwise it will only be created the g-code of the selected operations. The Figure 4.11 highlights these important aspects.

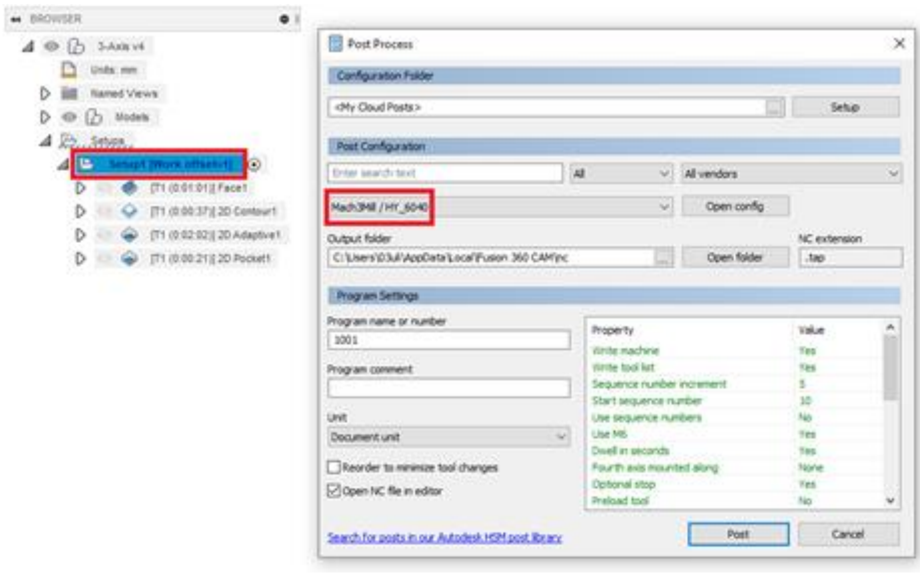


Figure 4.11 Post-Processor Selection

The Visual Studio Code will now open the G-code file and by analysing the first lines, it is possible to observe that the machine will activate the G54 working on Fixture 1, and with a “G0 A0 B0” assuring the axis alignment.

The only actions needed now, after placing the stock on the vise machine and locking it, is to defined the tool offset on Mach3Mill. To do this tool offset should be activated on Mach3Mill, Tool No. must be set to 1, which is the number of the tool programmed on Fusion360, the tool length below the spindle should be measured and the value introduced on the ´Z´ Offset dialog box.

To produce the part just load the G-code on Mach3Mill and press Cycle Start, the part will be manufactured, and the result can be seen on Figure 4.12.

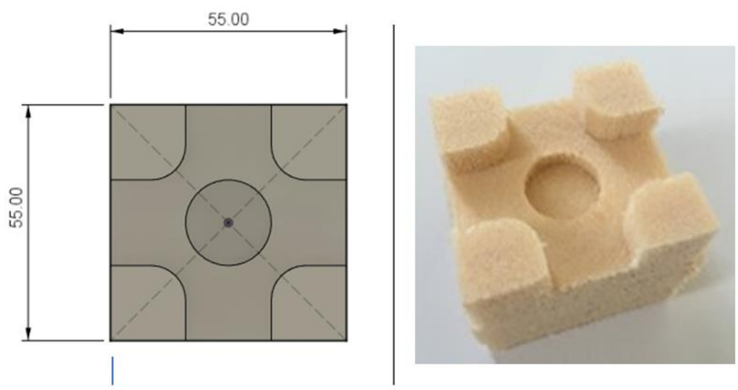


Figure 4.12 Part 1 model and final result using the HY-6040 working on 3-axis

The tool used for this part is represented next on Figure 4.13 and its parameters, being a flat end mill with an 8mm diameter.

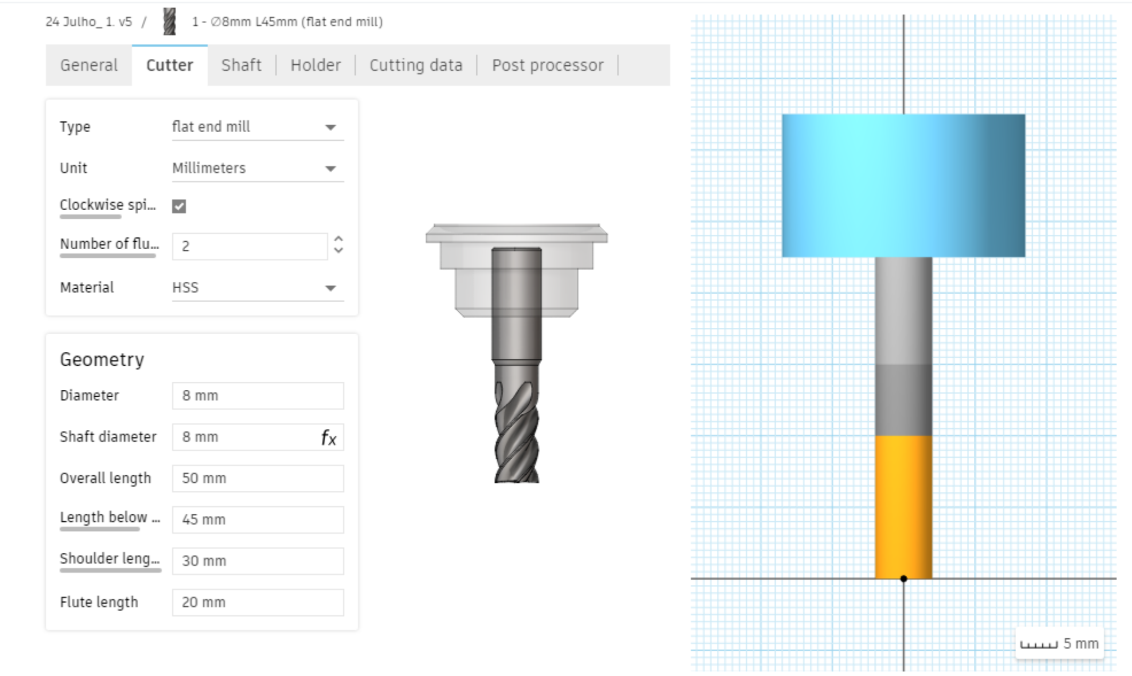


Figure 4.13 Flat end mill tool used to machine part1, in this image is represented the tool´s parameters within Fusion 360

While on 3-axis procedures, and to machine materials like aluminium, the work will be done using G-59 and with different workholding solutions. The table with the 4th and 5th axis is removed, and a more rigid vise is positioned and aligned onto the table.

The use of the programmed vise is now obsolete, in fact there is no need to use any fixture structure inside Fusion360, the only aspect to ponder is that, the zero-part (Figure 4.14) point must be defined in the same place where G-59 zero is established.

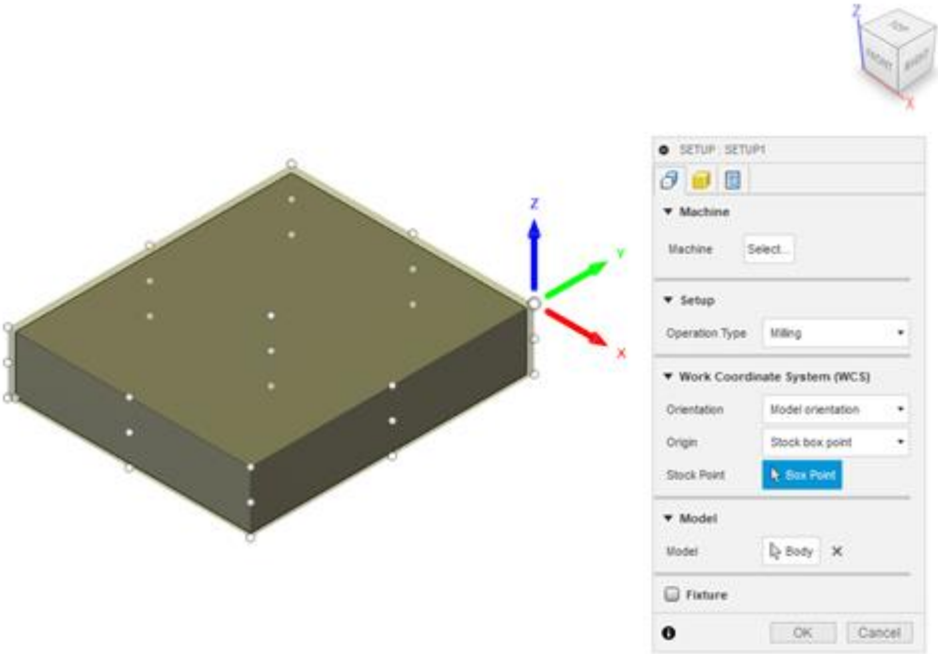


Figure 4.14 Zero-Part Defined on Fusion 360

To make this procedure even simpler, the zero-part point should be defined, if possible, on one of the stock´s vertex.

4.2 3+2-Axis Procedures

The 3+2 axis work will be presented in two study cases. The first is the part manufactured previously on chapter 4.1 but with two new changes. This allows understanding how reliable it is to machine existent parts on this machine, either to correct some details or to make changes to the previous geometry. The second will be a part produced fully with a 3+2-axis motions.

The first part, is represented on Figure 4.15 (technical drawing on Attachment VII), and since there is no probe or edge-finder available, to avoid displacement errors, the part will be created after the manufacture of the previous one, without moving it from the vise.

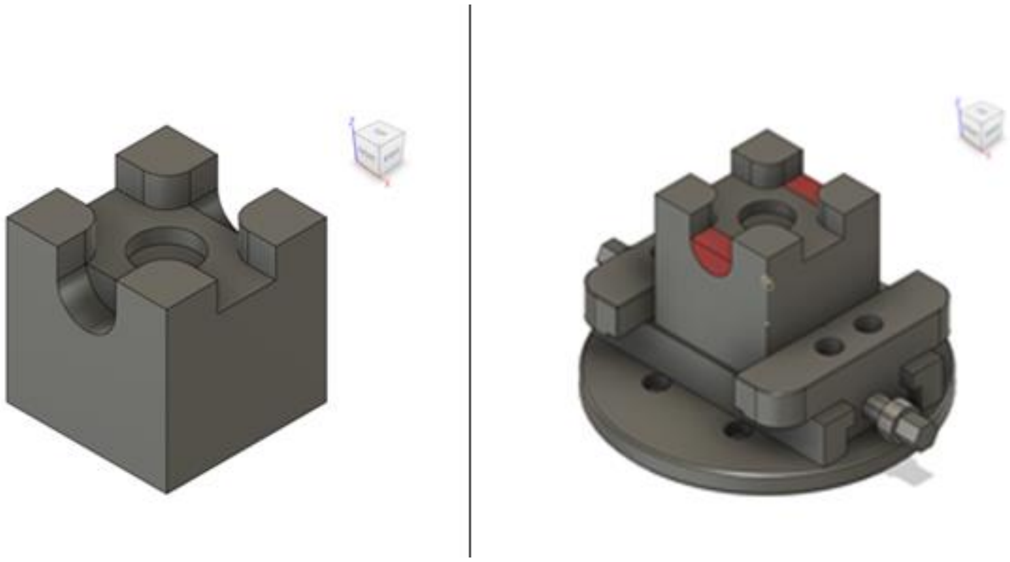


Figure 4.15 Part 2 (difference between previous part highlighted)

Consequently, being that the program of the original version already exists, it is possible to program the new part based on the former. To do this, first, the final model must be created and the material that is removed between operations should be created as a new body (red material on Figure 4.15). There red bodies will be used to define the new stock.

Before starting to program the new part, there is a crucial matter related with the use of the 4th and 5th axis. The next operation will be made on Fixture 2 (G-55), this means the B-axis (Figure 3.4 (a)) will need to rotate exactly-90 degrees before the cutting strategy begins.

One of the problems with the machine´s post-processor is that: only the first fixture of any part is set to their point-zero position (A0 B0), and since this program is the continuation of the previously 3-axis one, the machine will not rotate the B-axis even if the new setup is proper defined on Fusion360.

To overcome this issue, it is necessary to make use of the code lines added to the post processor on chapter 3.2. The process is simple, and it is necessary every time an A or B-axis motion is needed on 3+2 strategies.

To begin, it´s required to suppress every strategy on the setup one, created for the previous part. Upon this, all that is needed is to create a manual NC with a pass through type and write: “A0 B-90” on the dialog box, just like represented on

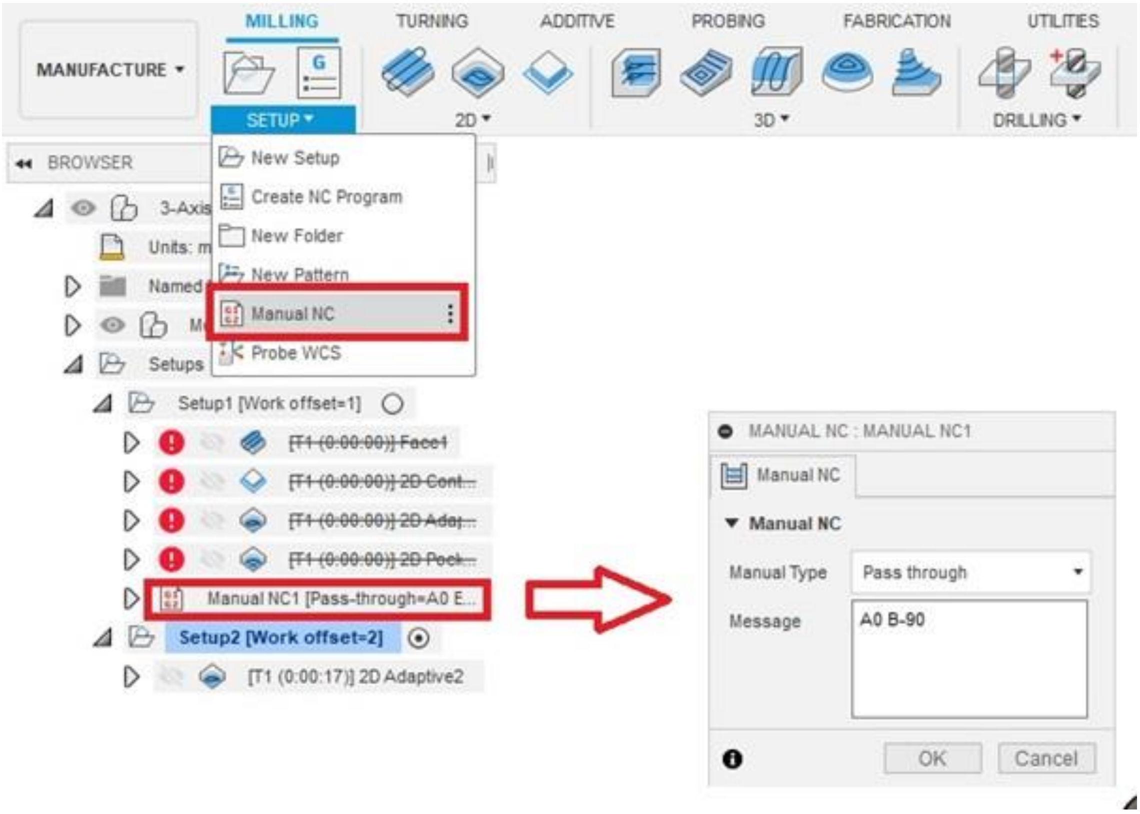


Figure 4.16 Pass Through code manually inserted on Fusion360, being later transcribed on the G-code file

This process adds a manually G-code line on Fusion360 to the G-code file, and the location on the CAM program to do this is after every new setup that demands an A or B-axis rotation. This is one other reason why the understanding of the model presented on Figure 3.5 is crucial.

The result of this on the G-code files is represented on Figure 4.17 and it can be noticed that the insert code: “A0 B-90” appears right before a new workoffset (G55 in this case) is activated, going accordingly with the work procedure established.

```
X5.665 Y0.651 Z23.075
X5.632 Y0.673 Z23.322
G0 Z64.522
A0 B-90
G28 G91 Z0.
G90

(2D ADAPTIVE2)
G55
G0 G43 X-3.155 Y34.055 Z52.5 H1
Z42.5
Z26.6
G1 Z25.8 F2000.
X-3.162 Y34.053 Z25.696
```

Figure 4.17 Manual code inserted on Fusion 360 present on the G code file and located before the beggining of a new workoffset

Upon concluding this step, the new setup referent to the G-55 workoffset can be created, being the setup tab configured exactly the same way as in the previous setup with the only exception being the axis orientation. On the stock tab, the “from solid” mode must be enabled, and the red areas must be selected. This informs the software which geometry is intended for machining.

On post process it is imperative to set WCS offset to 2, making the post-processor know that Fixture 2 (G-55) is active.

The new setup result must correspond to the one displayed on Figure 4.18 (a).

The cutting strategy may now be created, and for this geometry, the adequate strategy is a 2D Adaptive. The tool is once again the flat mill with an 8 mm diameter, and the feeds and speeds set are spindle speed and ramp spindle speed to 3000 rpm, with a cutting feedrate of 3000 mm/min, and the remainder of the feedrates set to 2000 mm/min.

The geometry selected must coincide with the semi-circular geometry of the stock leading to a correct default configuration of the heights tab. On the passes tab, stock to leave should be disabled and multiple depths enable and define maximum roughing stepdown to 2.5 mm. Since the stock has a 10 mm height, the previous configuration will lead to 4 passes.

To finish this operation, the ramp type on linking should be set to plunge, resulting on the toolpath illustrated on Figure 4.18 (b).

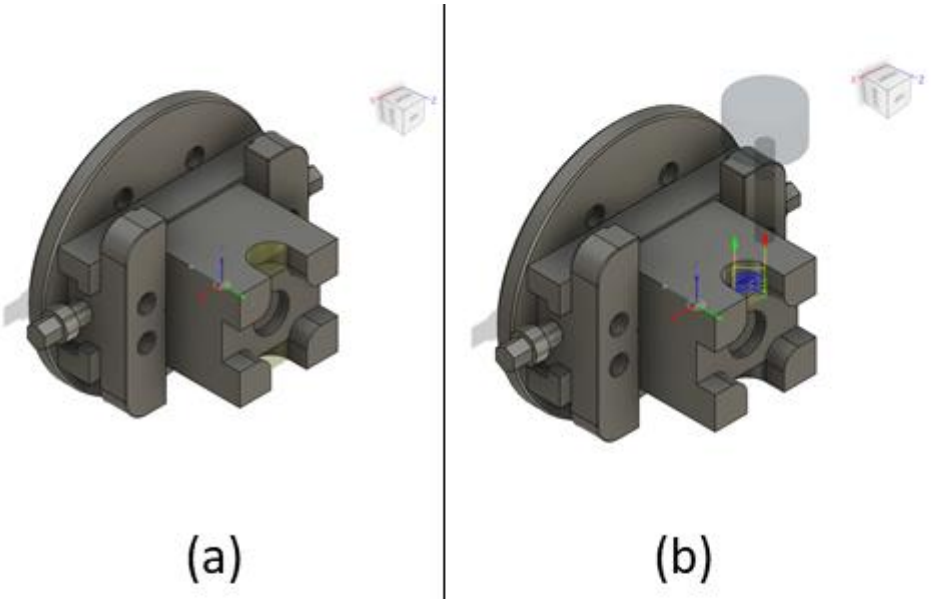


Figure 4.18 Setup 2 Established (a); Toolpath from Setup2 (b)

At this point, and since it is established that the B-axis should work in the negative side to avoid collision, the A-axis must be rotated 180 degrees. The same way as before, creating a manual pass through operation and typing: "A180 B0", the workspace will rotate and the G-58 workoffset will be facing the top.

In case any doubts come from this explanation, it is important to understand that before all this motions, the original positioning of the part is the same as represented on Figure 4.8.

The creation of the new setup is just like the previous one, changing the WCS offset to 5, enabling the Fixture 5 (G-58). The cutting strategy is created exactly like the former and the only aspect to establish now is at the end of the program to create a new manual pass through with: "A-180 B90", which will rotate the table to the G-54 position, making it easier and safer for the operator to remove the part.

Proceeding now to the second part, this one will be created from a 54x54x82 mm stock and the geometry is presented on Figure 4.19, a technical drawing is presented on Attachment VIII.

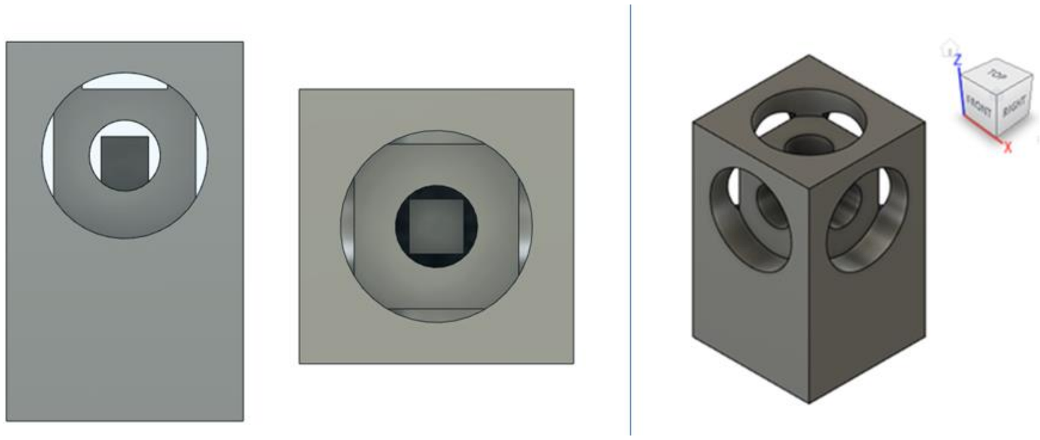


Figure 4.19 Part 3 Using 3+2-Axis with block dimensions of 80x50x50 mm

This part was designed not only for a 3+2-Axis demonstration, but also to explain additional features of Fusion360, where the user can migrate from the manufacture and design menus, creating operations to delimit the toolpath. Being that the heights of the part do not allow for 2D contour operations without collision, all faces will obtain 2D facing strategies.

Starting by placing the part onto the vise following the same procedure as explained before, the first setup created will be for the top face corresponding to the G-54 workoffset, which means that, when creating the first setup. The WCS offset must be set to 1.

The cutting strategies can now be created, and every face will be produced with three strategies, so the parameters for each operation will be explained only for this setup.

Starting with a 2D face operation, the tool used (the tool will be the same for all operations) is the flat mill with 8 mm diameter, and spindle speed will be set a 3000 rpm and ramp spindle speed to 5000 rpm, every type of feedrate must be set to 2000 mm/min for this operation.

Since this 2D facing strategy is relative to the top face, both geometry, heights and linking tabs can be left with the default parameters and on the passes tab, multiple depths should be enabled with a maximum stepdown of 1 mm and pass extension must be set to 1 mm, stock offset to 4 mm and stepover to 5 mm, to compensate for stock positioning deviations, following the explanation given on chapter 4.1.

The cylindrical geometries will both be created using a 2D pocket strategy. Firstly, the on with the greater diameter, persisting with the 8 mm diameter flat mill, spindle speed and ramp spindle speed will mutually be set to 4000 rpm, selecting a cutting feedrate of 1500 mm/min and setting all the other feedrates to 3000 mm/min. In geometry, the top circle should be selected and on the heights tab, the top height should be set to model top and for the bottom height, employing the selection mode, the bottom circle should be selected. At this stage the model would match the one represented on Figure 4.20.

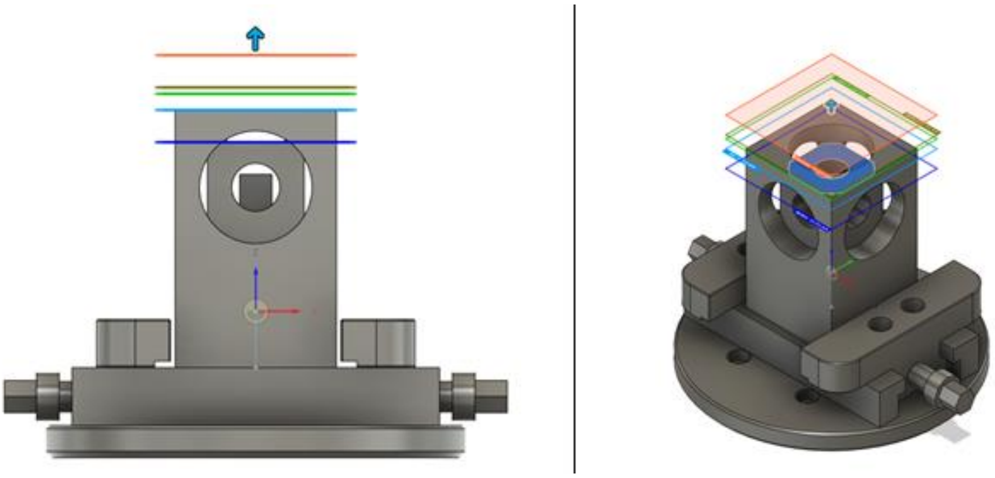


Figure 4.20 Heights Planes for Part 3 defined for the operations on the top face

The next geometry will be set with the same parameters as the previous, with the only change being set on the heights tab. The fact that this is a different geometry will change both top and bottom heights, nevertheless, being this the last operation before having to rotate the B-axis -90°, the clearance height must also be changed to 25 mm above model top. This is the position to where the tool will be lifted allowing now for a safe B-axis movement.

This way, by creating the manual pass through with: "A0 B-90", the first setup is finished, and by creating the new setup for the G-55, the operations on the front face of the part can start.

As stated before, the tool being employed does not possess enough length for a 2D contour, resulting on the creation of a 2D facing operation for each face. The problem is that, when selecting the intended face to machine, Fusion360 will create a toolpath along the entire face, resulting on collisions and damaged. To overcome this, it is possible to switch to the design interface, and draw a sketch to delimit the are meant to be machined. For this part, the drawings elaborated are illustrated on Figure 4.21, and the only requirement was that they should surpass the cylindrical geometries.

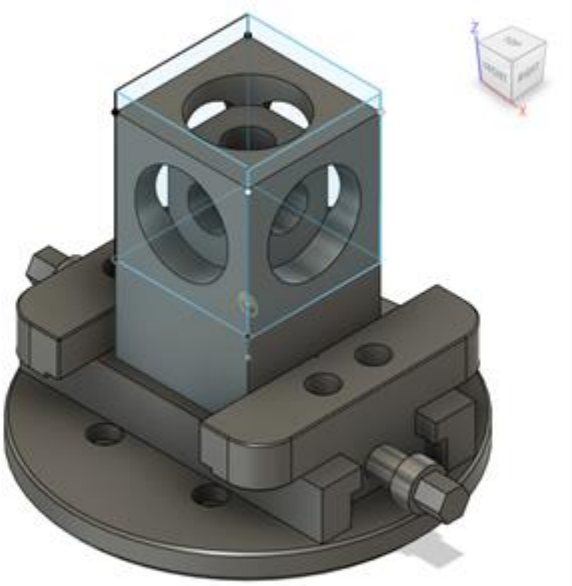


Figure 4.21 Drawing Created to Assist Toolpath Creation

Upon this, the 2D face operation is set exactly like the previous 2D face operations, and both 2D pocket operation destined to create the cylindrical geometries are programmed like the ones created for the top face. The 25 mm clearance height must be set before, can also be applied here since now the movement necessary is on the A-axis.

The cutting strategies for the next three faces are precisely equal to the one just explained and because of this reason, they won´t be repeated. Nonetheless, the rotation of the 4^{th} and 5^{th} axis are extremely important and will be presented next.

At this point, the machine is set on the G-55 workoffset and an A-axis rotation motion is necessary. This motion will lead to the G-56 workoffset and once again, the A-axis will need to rotate again to G-58 and once more ending on G-57, meaning the machine needs to rotate the A-axis 90 degrees in the negative direction for each face. To do this it is simply necessary too create a manual pass through with the line: "A-90 B0" at the end of each setup, with exception being on G-57, since it is the last, the line should be: “A-90 B90”, which will make the machine move back to the G-54 workoffset allowing for a more secure part removal.

The program tree must match the one presented on Figure 4.22.

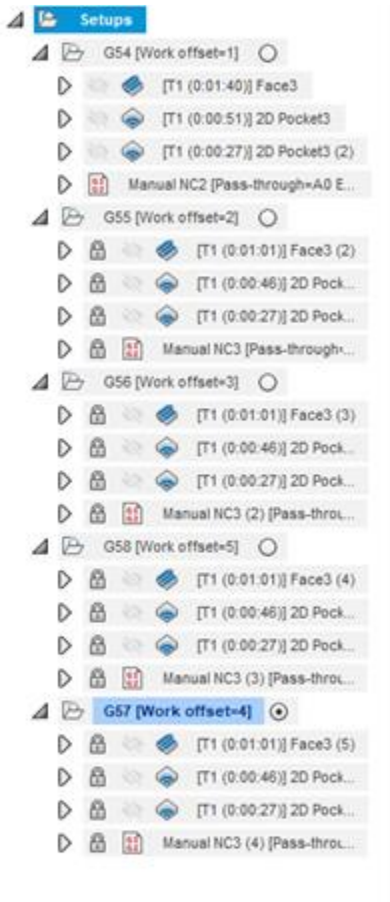


Figure 4.22 Operations Tree of Part 2

Concluding all this, the G-code file can now be generated and introduced on Mach3Mill to run the program and manufacture the part. Nevertheless, and although it wasn´t stated before, when working with 3+2 motions, before introducing the stock onto the vise, the operator must always make sure both A and B-axis are on position zero withing the G-54 workoffset. If such matter is not true, the operator must go on Mach3Mill activate Fixture 1 (G54) and, on the MDI panel, type in A0 B0 and press enter.

This step is crucial and if skipped may lead to the machine work on a different offset from the one programmed.

The final result is represented next on Figure 2.12.

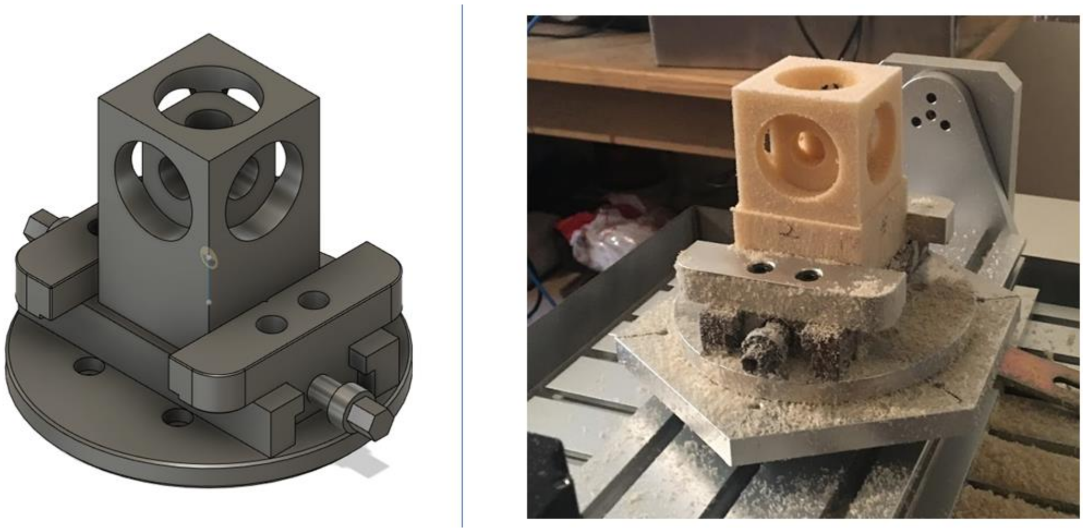


Figure 4.23 Part 3 Final Result comparation between the model on Fusion360 and the real model produced

4.3 5-Axis Continuous Procedures

The 5-axis continuous toolpath creation is most of the times perceived as a complex endeavour, however, Fusion360 interface allows for the user to create these types of toolpaths without much effort.

To attest/demonstrate this assertion, two study cases were created, presenting two very different types of 5-axis continuous toolpaths.

The first one, Part 4 has its technical drawing located on Attachment IX and a 3D model of the same is presented on Figure 4.24.

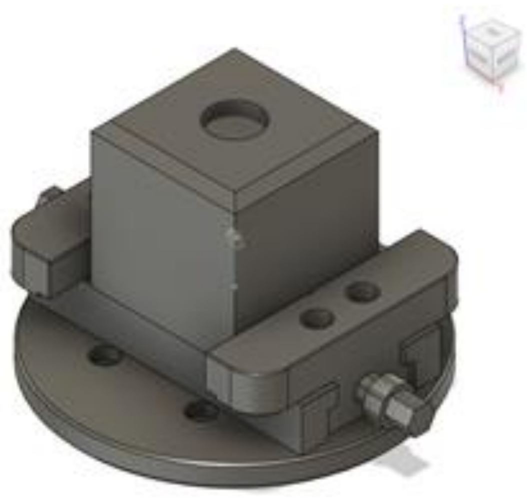


Figure 4.24 Part 4 with a block dimension of 60x60x60 mm to be machined using 5-axis continuous toolpaths

It is very important to clarify that, the manufacture processes of this part contains four different strategies: 2D face, 2D contour, 2D pocket and a swarf; only the last one will be addressed since it is the only relevant one to matter in question.

The swarf strategy is a 5-axis continuous approach used to machine with the side of the tool, being able to cut both contours and faces. For part 4, this strategy will be used to machine the chamfers on the top face.

Starting by defining the feeds and speeds for the operation, the spindle speed is set to 3000 rpm and all feedrates to 1500 mm/min.

Moving to the geometry tab, since the objects to select are chamfers, drive mode should be changed to contours and selection mode set to contour pairs, needing only to click the bottom part of the chamfer followed by the top part of the chamfer. The model on Fusion360 will appear as in Figure 4.25.

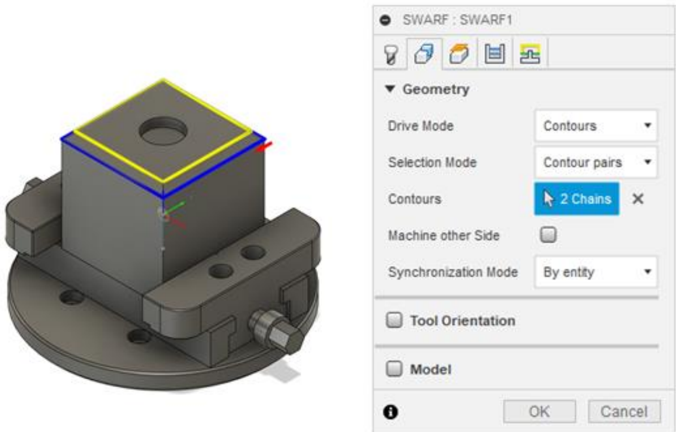


Figure 4.25 Swarf Heights Tab Parameters

It is important to respect this selection order since the tool will follow the part selected first (blue path represented on the image above), the red arrow indicated the direction the tool will move.

On the heights tab, the most important parameter is the bottom height. This represents the lower point the tool tip will reach, and being this a swarf operation the cut will be done with the side of the tool, defining this height to be below the chamfer´s base will allow to establish (in the passes tab) an additional tool offset. Considering this, bottom height was set to 5 mm below the chamfer´s base, the remaining heights were left with the default values given by the software.

Now, on the passes tab, tool offset can be set to 5 mm, corresponding the heights set prior, tangential fragment distance offset is set to 10 mm which will allow the tool change its direction of cut without colliding with the stock (the tool being used has a 8 mm diameter).

The remnant parameters are left with the default configuration, nonetheless, there is a parameter called sideways tilt that is set to 0 degrees. This constraint allows to specify the interval of angles that the machine will never overtake, but, in this case, the geometry intended to be machined surrounds the entire part, and so there can be no limits applied to the rotary motors or the toolpaths will not generate.

To finalize the operation program, linking factors can be left as default, since the machining area of the part is the external part of the top face.

Directing the attentions now to the physical machining process, even though the CAM program was correctly programmed, and every procedure was followed, the machine failed manufacture the part.

On Figure 4.26 is presented an illustration of Fusion360´s simulation and what the HY-6040 was delivering while running the part program.

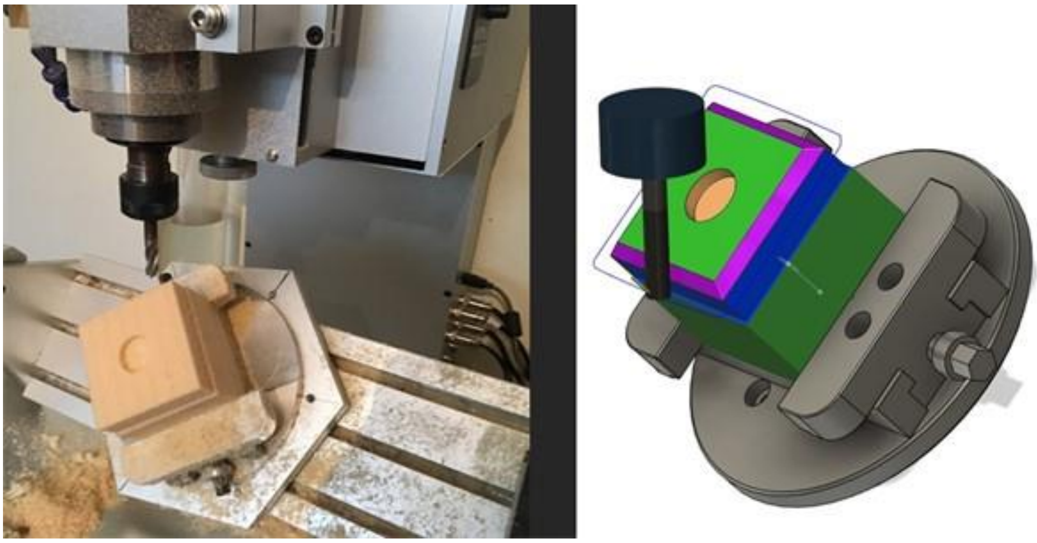


Figure 4.26 Swarf Strategy Reality vs Simulation

Although the simulation shows the programmed toolpath, cutting th chamfer with the tool respecting the 5 mm offset, in reality, the tool never touched the stock material, and the program was stopped when collision was imminent.

Upon this, a series of tests were made with additional 5-axis continuous programs in conjunction with different programming approaches all with negative results, which lead to conclude that, even with the 5.axis construction, having a controller with special capabilities (TCP, DWO, RTCP) is a crucial factor.

The subsequent and last part programmed (Part 5), was created as one of the tests mentioned before and will also be subjected to these circumstances, nonetheless the programming process will be explained showing the final result.

The part was created using the Create Form option on Fusion 360, the geometry is arbitrary with the only condition being the creation of a surface to be manufactured using 5-axis continuous movements and it is represented on Figure 4.27 (being this an arbitrary geometry with random forms and dimensions, the technical drawing of the same was not created).

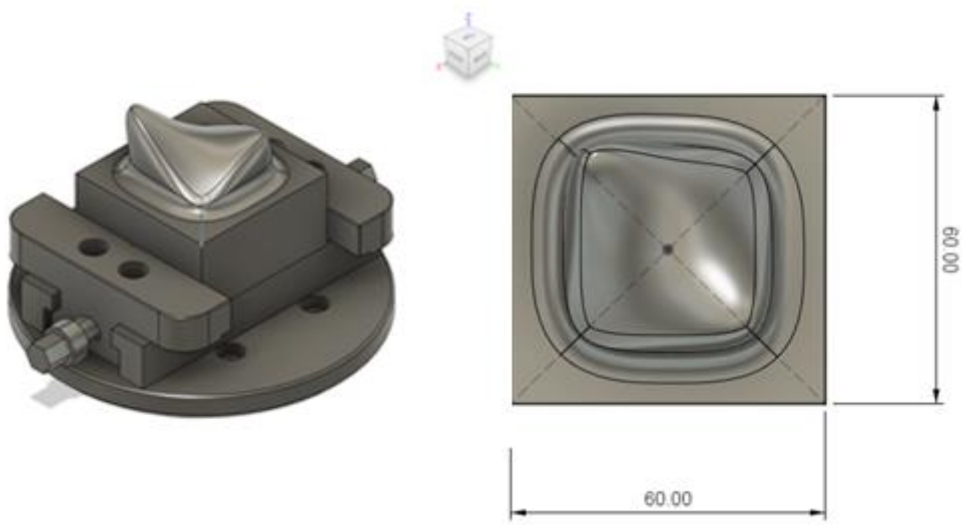


Figure 4.27 Part 5 to be machined with 5-axis continuous toolpath and containing a tool change

The program is initiated, like the precious ones with a 2D face and a 2D contour operation and using the 8 mm flat mill. Upon this, a 3D adaptive is created, Fusion 360 will automatically recognize the part´s geometry and the only aspect that differs from the other part programs presented till now is that, on the passes tab maximum roughing stepdown and fine stepdown must be equal and set to 2.5 mm, this means that, for each pass made by the tool, 2.5 mm of material will be removed in depth. If the values did not match, and the fine stepdown was set to, for example 1 mm, the toolpath would have additional passes removing only 1 mm in depth, leading to higher machining time and higher tool waste. Stock to leave should also be activated and set to 0.5 mm since this is a roughing operation.

Reached this stage, it is necessary to execute a tool change. This is done using a manual force tool change code (Figure 4.28).

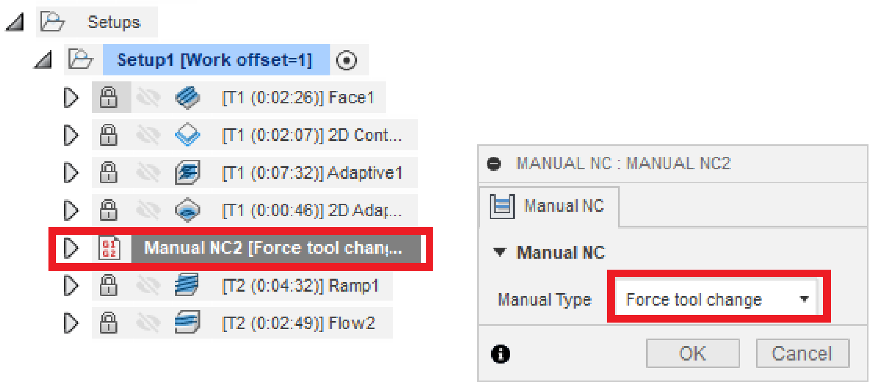


Figure 4.28 Manual Tool Change

This will lift the spindle to the highest position possible and stop it. The operator needs now only to change the tool and press RUN once more.

The next stage is to create a 3D ramp strategy which will create a semi-finishing toolpath. Since this is a 3D strategy the software is able to create the optima parameters for the operations in question.

Finishing the program with a 3D flow toolpath, this is the 5-axis continuous strategy that, like the previous one, will be created with all optimal parameters. Usually, in these type of operations, the main concern should be with the range of rotation given to the 4th and 5th axis, however, since nether the stock or the workholding solutions for this case are a concern the values can be left as default.

This last step should always be confirmed using the simulation tool on Fusion360, otherwise is almost impossible to know the safe working limits of the rotary motors. For this part, the toolpath was created without any problems and the Figure 4.29 represents a point on the strategy where the tool angle is visible.

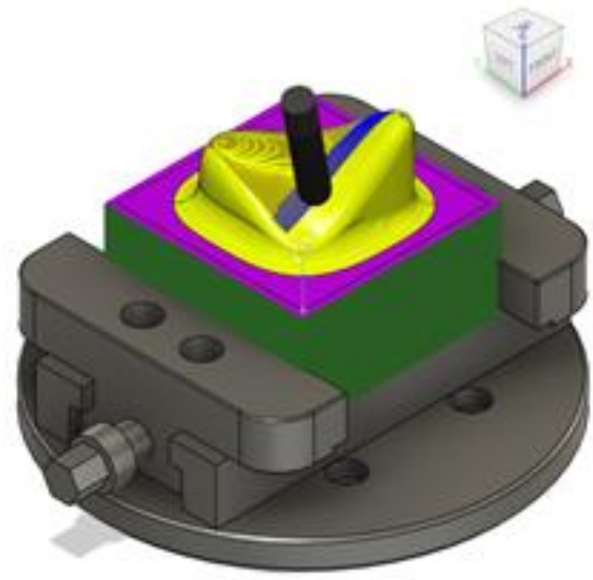


Figure 4.29 5-Axis continuous 3D flow toolpath represented by the blue lines and being executed by a ball mill

It must be reminded that Fusion360 does not have machine simulation so, the tool angle visible on Figure 4.29 is translated, in reality to workpiece movement. Since this operation is being made with a ball mill, the same rotation ensures that the tool is cutting the material with its side.

Once more, even with the program correctly executed, all the strategies were well performed apart from the 5-axis continuous toolpath. Since there was no collision the program was left to run until the end, however, the tool movements were incorrect (Figure 4.30).

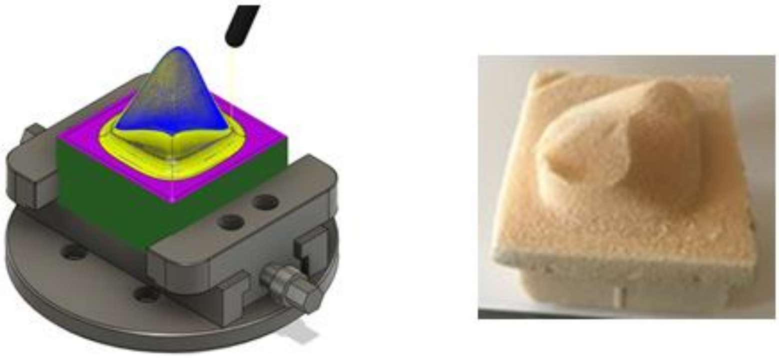


Figure 4.30 Part 5 Simulation vs Reality

It is concluded that: the machine can properly work with 3 and 3+2 strategies, and that all procedures established support not only an easier workflow but also a safer one.

The use of the model vise with the MRZP is extremely important when working with multi-axis and if any doubts endure the CAD block model with all the workoffsets configurations and the attachments created must be consulted.

It was also visible that Fusion360 is a very capable CAM system, allowing the user to use CAD elements to create the intended toolpaths.

When it comes to 5-axis continuous machining, the toolpath movements are nothing like the ones programmed which can be due to the simplicity of the post-processor (not possessing more powerful capabilities to have full knowledge of the tool´s position and orientation) or due to the fact that the way Mach3Mill interprets the G-code created on Fusion360 is not the most appropriated.

5 CONCLUSIONS AND FUTURE WORK

Reaching the epilogue of the document and with all working procedures established, some conclusions can be drawn, leading to a future work on improving the equipment.

5.1 Conclusions

Upon the realisation of the work described in the previous chapters, come conclusions were taken and will be presented next. These final assumptions tend not only to subjects related with the HY-6040 machine, but also with all the software used throughout the dissertation.

When dealing with CNC machines it is extremely important to first study/analyse the machine fully, and only after that working with it. This was the objective of chapter 3, to understand how the machine was built, the implications that would come from the table/table configuration and how this would reflect in the workflow.

The post-processor was another key factor, upon understanding how Fusion360 worked, the edit/configuration of the post-processor file was completely directed to a workflow that would make both CAM programming and Mach3Mill preparation simple and secure for both operator and machine.

With the inclusion of the MRZP, and the construction of the virtual fixture system, machine accuracy was ensured and a big amount of preparation work was reduced, since the operator does not need to set the Fixture offsets on Mach3Mill (defining the WCS on Fusion360 is enough).

One other important aspect relies on the nowadays offer when it comes to supplementary software (extensions). Even though Fusion360 is a very complete and powerful tool, doing some research and installing some available extensions or additional software can prove to be a big help, and it can lead to a more personalized CAM workflow.

To extend on this matter, when programming some parts, Fusion360 allows to create toolpath patterns when managing multiple faces with equal operations. This feature was used for part 3 and the simulation resultant exhibited a valid toolpath , however, using the program Visual Studio Code with the extensions added it was possible to conclude that the post-processor could not convert the patterns into proper G-code without having to test it on the machine. This saved material, tool-wear and the possibility of any error that could lead to damages of the part was machined.

The fact that a system like Fusion360 also allows the constant transition between CAD and CAM, making it possible to create elements (dots, lines, shapes) to delimit toolpath is also a noteworthy factor that results on more fluid/free workflows for different parts.

Upon defining and understanding all this perceptions and methods, 3-axis and 3+2-axis machining become simple, and respecting the procedures established the main concerns to have in mind is whenever A or B-axis rotation is needed. The pass through message and the WCS defined need to be the correct ones.

Directing the attention now to the 5-axis continuous machining, the main conclusion drawn from the work is that both post-processor and controller of 5-axis CNC machines are one of the most important factors for the machine work.

Having a post-processor with advanced properties (like DWO or TCP) that can output reliable G-code files to a controller that holds good communication with the machine is essential.

For multi-axis toolpath to be correctly executed it is required that machine gets the precise information correspondent to both tool position and orientation.

This means, although the machine possess 5-axis, and both 3 and 3+2 work is running perfectly, to be able to create 5-axis continuous toolpath additional work is required. This work is presented next.

5.2 Future Work

As stated before, in order to get the machine used on this dissertation to fully work with 5-axis continuous movements some additional work is needed.

Here, the improvements needed as well as some additional ones, relating to an upgrade in terms of structure and machining power and CAM works, are listed next:

- Further development of the post-processor;
- Study the communication between post-processor and controller;
- Adapt and install stronger motors;
- Attain more workholding systems;
- Developed new CAM workflows.

BIBLIOGRAPHIC REFERENCES

[1] “ISO - ISO 513:2012 - Classification and application of hard cutting materials for metal removal with defined cutting edges — Designation of the main groups and groups of application.” https://www.iso.org/standard/59932.html (accessed May 08, 2020).

[2] “BLOG DEL DEPARTAMENTO DE INSTALACIONES: Materiales según la norma ISO.” https://departamentofrigorificohesperides.blogspot.com/2016/05/materiales-segun-la-norma-iso.html (accessed May 08, 2020).

[3] Sandvik, “Cutting tool materials.” https://www.sandvik.coromant.com/en-gb/knowledge/materials/pages/cutting-tool-materials.aspx (accessed Apr. 15, 2020).

[4] B. Maschinen, A. Investition, G. Beschaffungen, B. Ersatzbeschaffungen, and S. Mittelherkunft, *Advanced Machining Processes of Metallic Materials Theory Modelling and Applications*. .

[5] “Basic End Mills Technical Information | MSC Industrial Supply Co.” https://www.mscdirect.com/basicsof/end-mills (accessed May 08, 2020).

[6] “MachiningCloud App for Better Tooling Decisions.” https://www.machiningcloud.com/ (accessed May 08, 2020).

[7] I. 60812, “International Standard International Standard,” *61010-1 © Iec2001*, vol. 2006, p. 13, 2006.

[8] “Cnc Machine Axis Diagram. diagram of the omag blade5 nc900 5 axis cnc machine with. fusion360 cam post processor for 5 axis table table ac. cnc milling. cnc cad cam. cnc lathe axis youtube. the what why and how of 5 axis cnc machining. cnc teknolojisi ve eksen mant youtube. what is a cnc machine and what is its working principle. ns cnc ultra precision machining system. 5 axis cnc machining shop. [A.2002-acura-tl-radio.info].” http://a.2002-acura-tl-radio.info/page-b/cnc-machine-axis-diagram-73790.html# (accessed May 11, 2020).

[9] K. Apro, *Secrets of 5-axis Machining*. 2008.

[10] “Total Guide to CNC Jigs, Fixtures, and Workholding Solutions for Mills.” https://www.cnccookbook.com/cnc-jigs-fixtures-workholding-solutions-milling/ (accessed Mar. 23, 2020).

[11] I. NexGenCAM, *Fundamentals of CNC*. 2012.

[12] “10020 - Step Clamps | Wixroyd.” https://www.wixroyd.com/en/catalog/work-holding/manual-clamping-height-setting/standard-manual-clamping/standard-over-clamps/10020/g+m+c+s+a (accessed Apr. 28, 2020).

[13] N. P. Maniar, A. B. Andhare, and C. M. Patel, “Design of 28 Operations , 4 Axis – 360 o Indexing Milling Fixture for CNC,” vol. 1, no. 5, pp. 129–131, 2009.

[14] “Modular workholding saves time and money - Practical Machinist : Practical Machinist.” https://www.practicalmachinist.com/cutting-tool-engineering/mod-squad/ (accessed Mar. 16, 2020).

[15] J. Roberts, “Welding Begins on Orion Pathfinder,” 2015.

[16] “five-axis vertical machining center | NMV Series | 5 Axis CNC Machine | DMG MORI | Hobby shops near me, Hobby desk, Hobbies that make money.” https://www.pinterest.com/pin/81909286950711705/ (accessed Mar. 30, 2020).

[17] “573813499568242688 @ twitter.com.” [Online]. Available: https://twitter.com/matsuura_uk/status/573813499568242688.

[18] “Cath-mounted heart pumps for short-term use | Future Medicine India.” https://www.futuremedicineindia.com/cath-mounted-heart-pumps-for-short-term-use/ (accessed Mar. 30, 2020).

[19] “2e01e3665eda92965760e9a05dffe8d3403df374 @ cardiacvascularnews.com.” [Online]. Available: http://cardiacvascularnews.com/fda-approves-abiomeds-first-in-human-trial-of-impella-ecp-worlds-smallest-heart-pump/.

[20] “In a heartbeat: Tiny propeller keeps blood flowing -- ScienceDaily.” https://www.sciencedaily.com/releases/2016/09/160921100129.htm (accessed May 06, 2020).

[21] “The bizarre reactor that might save nuclear fusion | Science | AAAS.” https://www.sciencemag.org/news/2015/10/bizarre-reactor-might-save-nuclear-fusion# (accessed Mar. 30, 2020).

[22] “1037300619 – Puntland Post.” https://puntlandpost.net/2019/03/20/hayadda-amnesty-international-oo-warbixin-ka-diyaarisay-duqeymaha-maraykanka-ee-soomaaliya/attachment/1037300619/ (accessed Mar. 30, 2020).

[23] “Cloud Powered 3D CAD/CAM Software for Product Design | Fusion 360.” https://www.autodesk.com/products/fusion-360/overview (accessed Apr. 29, 2020).

[24] “CAM Workflow.” https://www.werkzeugtechnik-schmitz.de/en/cam-workflow.html (accessed Apr. 07, 2020).

[25] “3D CNC Milling & Machining: CAM Toolpaths.” https://www.cnccookbook.com/3d-cnc-milling-machining-cam-toolpaths/ (accessed Apr. 13, 2020).

[26] “SprutCAM – CAD/CAM system for 5 axis milling processing.” https://sprutcam.com/sprutcam/5-axis-milling-programming/ (accessed Apr. 13, 2020).

[27] “ModuleWorks release 2017.08 CAD/CAM components | ModuleWorks.” https://www.moduleworks.com/moduleworks-release-2017-08-cadcam-components/ (accessed Apr. 13, 2020).

[28] “What CNC programming style best suits your operations?” https://www.mazakusa.com/news-events/blog/what-cnc-programming-style-best-suits-your-operations/ (accessed Apr. 14, 2020).

[29] “Blum-Novotest GmbH | Measuring and Testing Technology | Touch Probes | Laser Measuring Systems | Measuring Machines | Test Stands.” https://www.blum-novotest.com/en.html (accessed May 07, 2020).

[30] “Partners in THINC | Okuma | Blum | Automation | Workpiece Probing | Tool Setting.” https://www.okuma.com/bridging-the-skills-gap-—-part-7-workpiece-

probing-and-tool-setting (accessed Apr. 14, 2020).

[31] “Sistemas de medição a laser para centros de usinagem CNC da BLUM.” https://www.blum-novotest.com/pt/produkte/messkomponenten/lasercontrol.html (accessed May 26, 2020).

[32] “Cutter Compensation.” https://machmotion.com/blog/what-is-cutter-compensation (accessed Apr. 14, 2020).

[33] “crc.” https://archiv.hobbycnc.hu/CNC/Programming/crc.htm (accessed Apr. 14, 2020).

[34] “Cutter Compensation for Beginners - Fusion 360 Blog.” https://www.autodesk.com/products/fusion-360/blog/cutter-compensation/ (accessed Apr. 14, 2020).

[35] “Understanding 5-Axis Machining Concepts | Autodesk University.” https://www.autodesk.com/autodesk-university/class/Understanding-5-Axis-Machining-Concepts-2018 (accessed Jun. 10, 2020).

[36] “G254 - Dynamic Work Offset (DWO) (Group 23) | Customer Resource Center.” https://staging-diy.haascnc.com/g254-dynamic-work-offset-dwo-group-23-1 (accessed Jun. 10, 2020).

[37] “Visual Studio Code - Code Editing. Redefined.” https://code.visualstudio.com/ (accessed Aug. 05, 2020).

ATTACHMENT I – COMMON ALPHANUMERIC ADDRESS CODE

Code	Meaning
A	Rotations about X-axis
B	Rotations about Y-axis
C	Rotations about Z-axis
D	Cutter diameter compensation (CDC) offset address
F	Feed rate
G	G-code (preparatory code)
H	Tool length offset (TLO)
I	Arc center X-vector, also used in drill cycles
J	Arc center Y-vector, also used in drill cycles
K	Arc center Z-vector, also used in drill cycles
M	M-Code (miscellaneous code)
N	Block Number
O	Program Number
P	Dwell time
Q	Used in drill cycles
R	Arc radius, also used in drill cycles
S	Spindle speed in RPM
T	Tool number
X	X-coordinate
Y	Y-coordinate
Z	Z-coordinate

ATTACHMENT II – MACH3 MILLING CODES

G00	**Rapid Positioning Motion (X, Y, Z, A, B)**
G01	Linear Interpolation Motion (X, Y, Z, A, B)
G02	Circular Interpolation Motion CW (X,Y,Z,A,I,J,K,R,F)
G03	Circular Interpolation Motion CCW (X,Y,Z,A,I,J,K,R,F)
G04	Dwell (P) (P=Seconds)
G09	Exact Stop, Non-Modal
G17	Circular Motion XY Plane Selection (G02 or G03)
G18	Circular Motion ZX Plane Selection (G02 or G03)
G19	Circular Motion YZ Plane Selection (G02 or G03)
G20	Inch Coordinate Positioning
G21	Metric Coordinate Positioning
G28	Machine Zero Return Thru Ref. Point (X,Y,Z,A,B)
G29	Move to Location Through G28 Ref. Point (X,Y,Z,A,B)
G40	Cutter Comp Cancel
G41	2D Cutter Compensation, Left (X,Y,D)
G42	2D Cutter Compensation, Right (X,Y,D)
G43	Tool Length Compensation + (H,Z)
G49	Tool Length Compensation Cancel G43/G44/G43
G52	Work Offset Positioning Coordinate
G53	Machine Positioning Coordinate, Non-Modal (X,Y,Z,A,B)
G54	Work Offset Positioning Coordinate #1
G55	Work Offset Positioning Coordinate #2
G56	Work Offset Positioning Coordinate #3
G57	Work Offset Positioning Coordinate #4
G58	Work Offset Positioning Coordinate #5
G59	Work Offset Positioning Coordinate #6
G73	HS Peck Drilling Canned Cycle (X,Y,A,B,Z,I,J,K,Q,P,R,L,F)
G74	Reverse Tapping Canned Cycle (X,Y,A,B,Z,J,R,L,F)

G76	Fine Boring Canned Cycle (X,Y,A,B,Z,I,J,P,Q,R,L,F)
G77	Black Bore Canned Cycle (X,Y,A,B,Z,I,J,Q,R,L,F)
G80	Cancel Canned Cycle
G81	Drill Canned Cycle (X,Y,A,B,Z,R,L,F)
G82	Spot Drill / Counterbore Canned Cycle (X,Y,A,B,Z,P,R,L,F)
G83	Peck Drill Deep Hole Canned Cycle (X,Y,A,B,Z,I,J,K,Q,P,R,L,F)
G84	Tapping Canned Cycle (X,Y,A,B,Z,J,R,L,F)
G85	Bore In - Bore Out Canned Cycle (X,Y,A,B,Z,R,L,F)
G86	Bore In - Stop - Rapid Out Canned Cycle (X,Y,A,B,Z,R,L,F)
G87	Bore In - Manual Retract Canned Cycle (X,Y,A,B,Z,R,L,F)
G88	Bore In - Dwell - Manual Retract Canned Cycle (X,Y,A,B,Z,P,R,L,F)
G89	Bore In - Dwell - Bore Out Canned Cycle (X,Y,A,B,Z,P,R,L,F)
G90	Absolute Positioning Command
G91	Incremental Positioning Command
G92	Global Work Coordinate System
G93	Inverse Time Feed Mode ON
G94	Inverse Time Feed OFF / Feed Per Minute ON
G98	Canned Cycle Initial Point Return
G99	Canned Cycle R Plane Return

ATTACHMENT III – MACH3 MILLING M-CODES

M00	Program Stop
M01	Optional Program Stop
M02	Program End
M03	Spindle ON Clockwise (S)
M04	Spindle ON Counterclockwise (S)
M05	Spindle Stop
M06	Tool Change (T)
M08	Coolant ON
M09	Coolant OFF
M30	Program End and Reset
M31	Chip Auger Forward
M33	Chip Auger Stop
M34	Coolant Spigot Position Down, Increment
M35	Coolant Spigot Position Up, Decrement
M36	Pallet Part Ready
M41	Spindle Low Gear Override
M42	Spindle High Gear Override
M50	Execute Pallet Change
M83	Auto Air Jet ON
M84	Auto Air Jet OFF
M88	Coolant Through Spindle ON
M97	Local Sub-Program Call (P,L)
M98	Sub-Program Call (P,L)
M99	Sub-Program / Routine Return of Loop (P)

ATTACHMENT IV – FUSION360 MANUFACTURE DESCRIPTION

Introduction

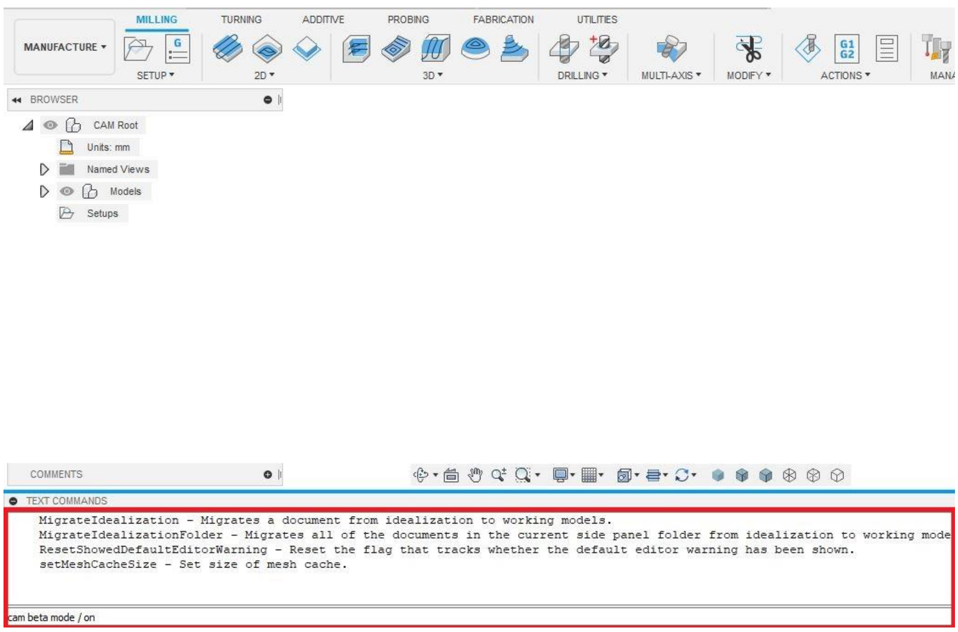
The following document was built with the intent of providing additional information to the help files from the software Autodesk Fusion 360 when it comes to the manufacture of components.

There are some initial insights that should be considered when working with this program. While creating working with CAM on Fusion 360, the user can always switch from the manufacture mode to the design mode and create additional CAD elements to help on the CAM program (drawings for toolpath limits, toolpath directions, etc.), which gives an enormous versatility when it comes to programming process.

Other aspect is that Fusion 360 possesses an App store and a scripts/add-ins features that allows the user to use additional elements when working.

The last consideration is that this document was develop based on the educational student version of Fusion 360, which contains all existent capabilities.

This way, when using the software for manufacture work, the first thing to always do is open the text commands panel and type in: "cam beta mode / on" and press enter to activate additional functionalities to the manufacture mode, and be able to make full use of Fusion360´s functionalities (Figure below).



Setup

When working with Fusion 360 to create CAM parts, this is the first tab to use in order to define the type of work intended. It also contains features that allow the user to arrange/personalize the CAM program and create different types of workflows for the manufacture of different parts.

This tab contains 6 options that will be described separately next.

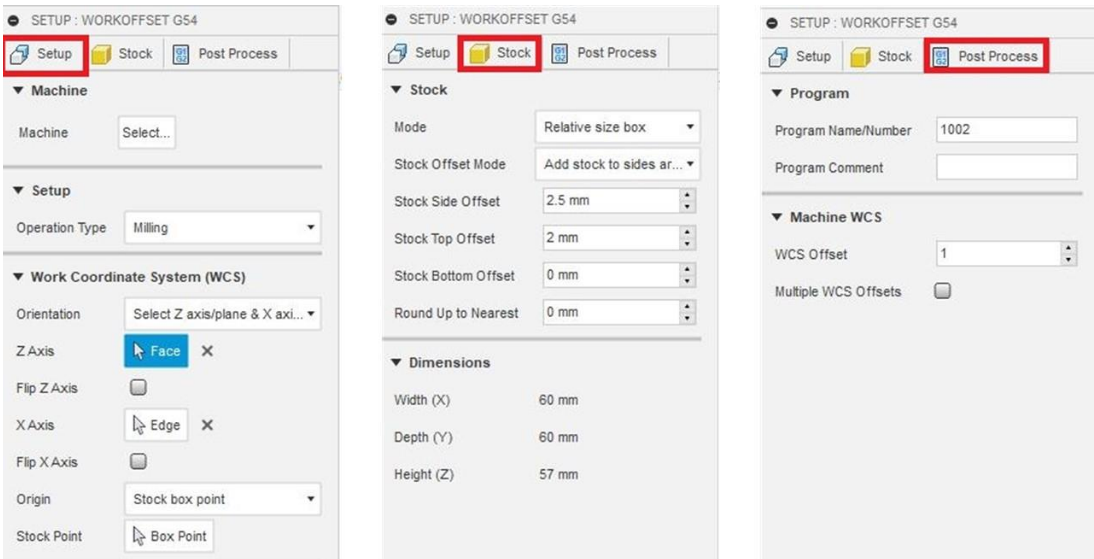
New Setup

This is usually the first option to be used in every CAM program, and it is always embodied with three different parts.

The first part is the Setup (Figure (a)), in which is allowed to establish the machine intended for the manufacture process, the type of operation (milling, turning, etc.), the establish of the zero part point, the definition of the model (of the part) and fixture system.

The second (Figure(b)), allows the definition of a stock. There are a different number of stock options permitting the user to program machining processes from stock boxes or simply modifying existing parts increasing the diversity of Fusion´s workflow.

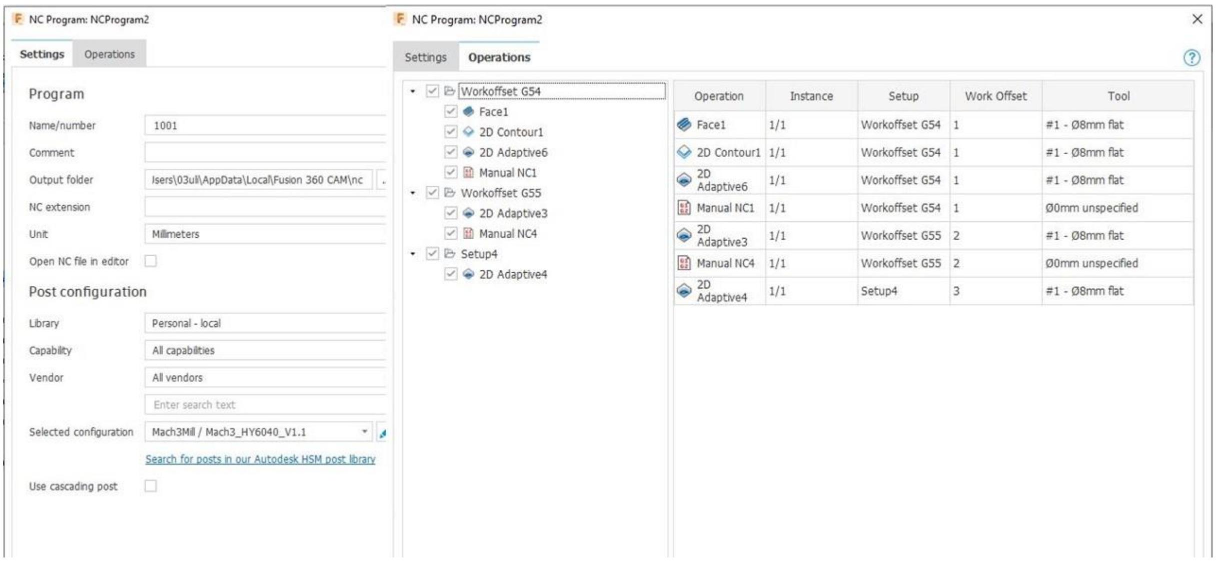
The last options (Figure (c)) refers to the post process of the setup, defining the name of the program and the number of work coordinate system (WCS) offsets to use.



a) b) c)

Create NC Program

This feature is normally use when the CAM program is complete, and the only step left to do is generating the G-code file. The user can define or change the name intended for the G-code file and which ones of the programmed CAM strategies are to be exported. It is here that the selection of the post processor is done (Figure below).

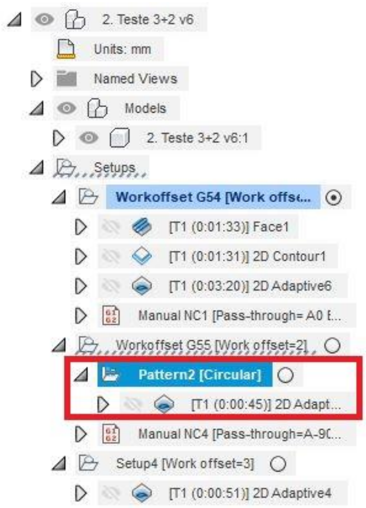


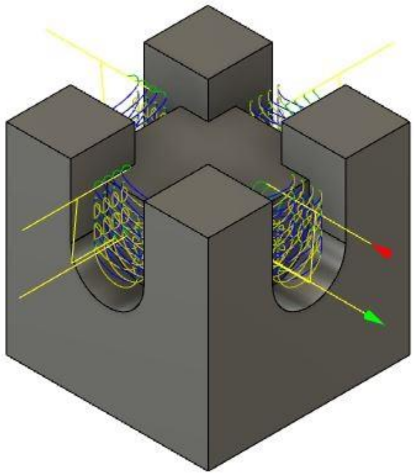
New Folder

The only purpose of this feature is to allow the user to personalize/organize the program is anyway intended. It creates a new folder on the operations tree allowing for arrange the operations, for instance, in type or tool being used.

New Pattern

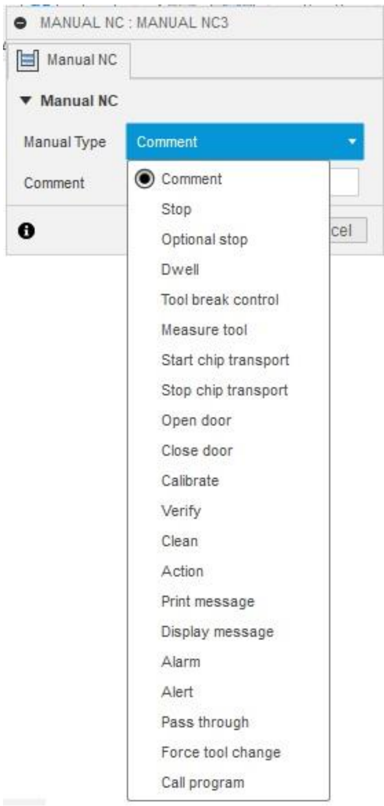
Just like when working on CAD projects, this option allows the user to save time programming, and, when the part requires a few equal strategies on a giving pattern, using this option will automatically create all operations with all the same parameters. Although, one important factor to consider when using this option is if the post processor of the machine that it will be used can understand this information or not.





Manual NC

This feature, just like the name indicates, allows the user to add manual G-code lines to the program. This can include a huge variety of functions, from tool change to a comment or even a stop. It also allows to add G-code lines to command the machine to travel to a certain intended position.



Probe WCS

This last setup options allows to generate probing strategies with the purpose of defining the WCS point on the real-world part (in other words defining the zero-part point on the real machine), or establish workspace dimensions depending on what is needed.

Cutting Strategies

The cutting strategies on Fusion 360 are arranged in 4 big groups. The major groups are denominated 2D and 3D, retaining the most used operations. The 2D group is basically dedicated to simpler toolpaths using 2.5 and sometimes 3 axis motion and the 3D group is for more complex 3 axis motion and even with some multi axis tilting options. However, the biggest different between these two is that, when using 2D strategies the user has to give the software all the information regarding the area to machine, while if using the 3D strategies, the software recognizes what to machine, and the user only needs to make some few adjustments.

There are also a third group for drilling operations and a fourth group dedicated to multi axis toolpaths (4 and 5 axis).

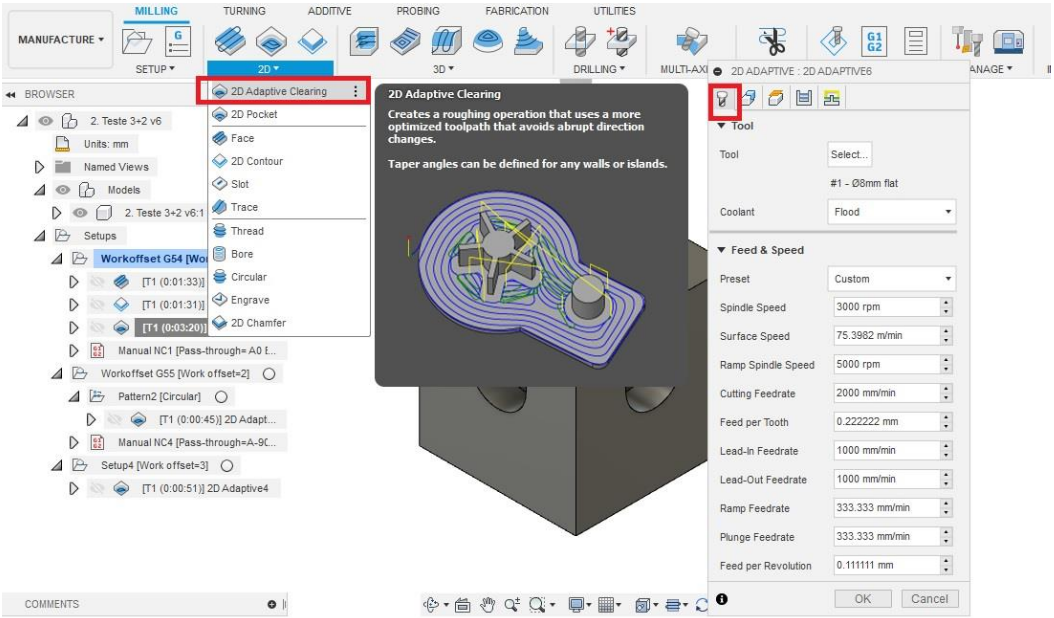
2D Strategies

There are a great number of 2D strategies, but before studying them is important to understand that the workflow of the software, regarding this type of strategies, is always the same. There are always 5 tabs with the same functions to define: tool, geometry heights, passes and linking. And for the most part of the strategies the tabs are filled with the same values, this way, all the strategies are going to be presented next, but in order to not repeat every step, the tabs will be defined only in the initial strategies.

2D Adaptive Clearing

This strategy creates a roughing operation that uses a more optimized toolpath that avoids abrupt direction changes.

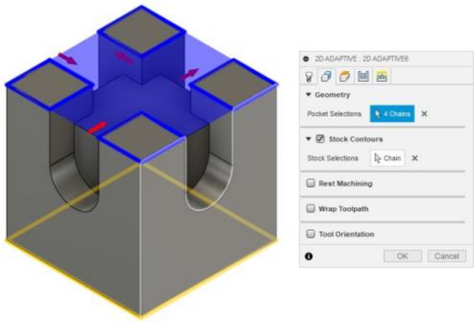
When selecting this strategy, the first tab (tool) is used to define the tool, the coolant type and the feeds and speeds of the operation (not only the ones for cutting but also the trajectory feeds and speeds) and presented in the figure below.



The geometry tab is where the user defines where the tool needs to actually cut the material, using pocket selection the limits of the toolpath should be selected and the area where the toolpath is created is shown in a blue colour (Figure below), the red arrows can be selected to change the blue area. It is also important to activate the stock contours so that there are no collisions, and the cutting movements don't start only in the model.

The wrap toolpath exists so that, if the user desires, the toolpath will be wrapped around a cylinder so it can be output using axis substitution around a rotary axis, or as full multi-axis moves.

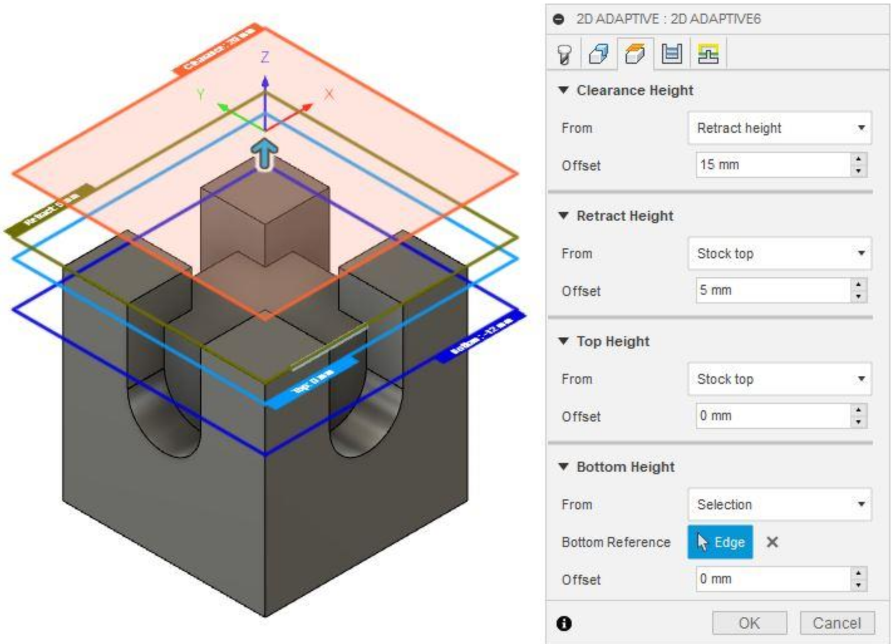
The tool orientation option, when enable allows the user to give the tool an orientation different to the one selected on the setup.



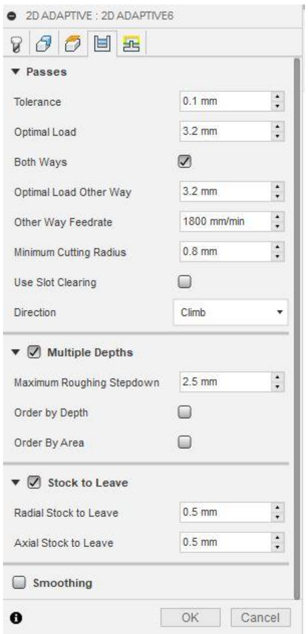
Moving to the heights tab, this is where the user defines the height and depth of cut, and also the height to which the tool will retract (everything related with tool height movements is defined here).

As represented on the next figure, the clearance plane defines the height from which the tool will enter and leave this operation, the retract plane defines the height that the tool moves up to before the next cutting pass. The top plane defines the height from which metal removing will start and the bottom is where this metal removing stops.

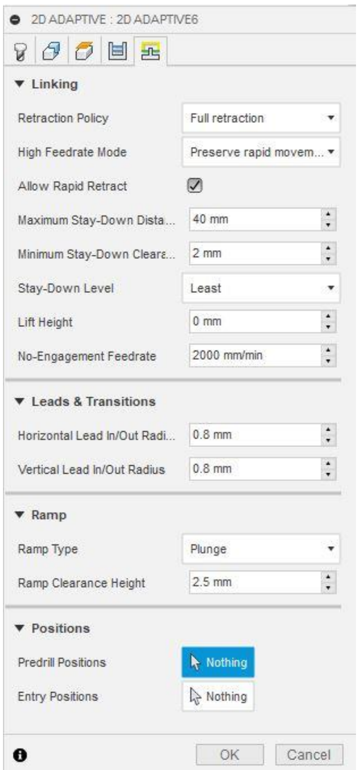
All heights can be added offsets and can be determined using different references like stop top/bottom, model top/bottom or a defined selection.



The passes tab defines the amount of material to be removed from each pass, how many passes, the direction to be used, and it also allows to add stock to leave for some following finishing passes.



The last tab is referent to the linking movements and it is used to defined how the leads and transitions happen, how the tool will start the cut (helix, plunge or predrill) and how the tool behaves during both cutting and the transitioning movements.



The programming of every strategy in the software involves setting up all these five tabs, and since they are always the same, they will only be mentioned again if any modification is presented.

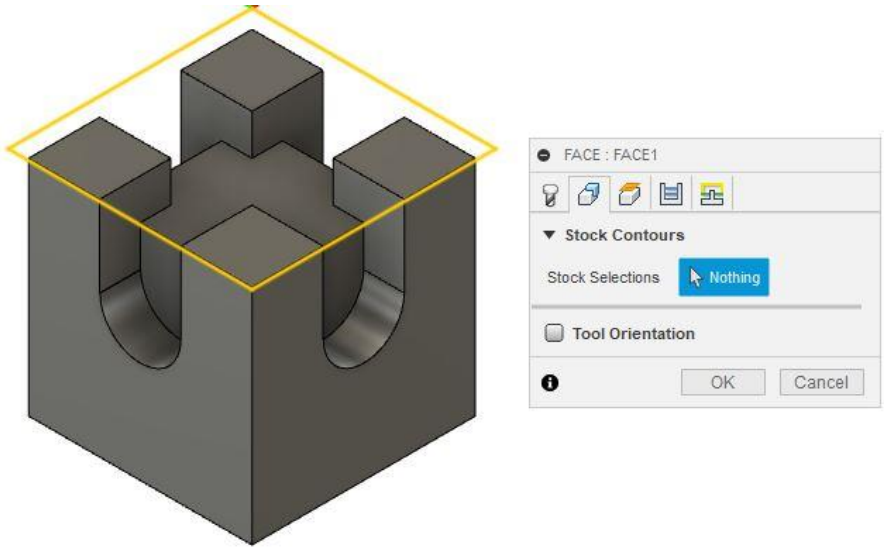
2D Adaptive Pocket

The only difference between a 2D adaptive clearing and a 2D adaptive pocket is that this last strategy should be use when the area to be machined is surrounded by material from all sides, which requires additional attention when it comes to the beginning of the cut.

Although this difference exists, the way to configure these two cutting strategies is exactly the same.

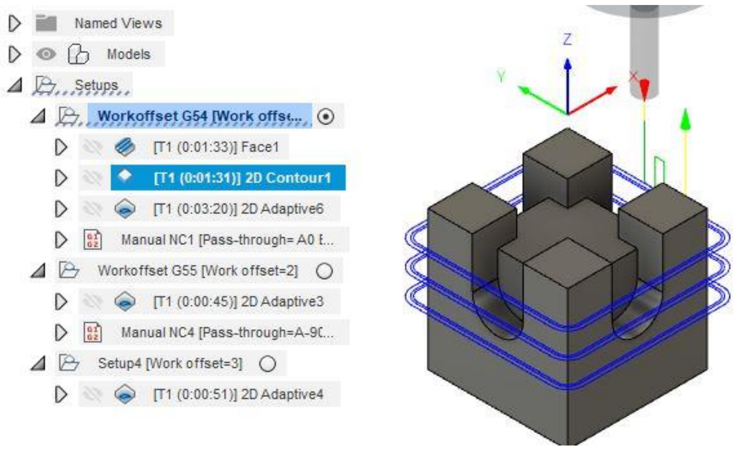
Face

This feature, just like the name indicates, is used for facing operations. The workflow of this operation differs from the previous operations only in one aspect, the geometry tab, where the only thing needed to select is the stock contour represented on the figure below (for a personalized facing operation the user can draw the contours intended for the operation and select it).



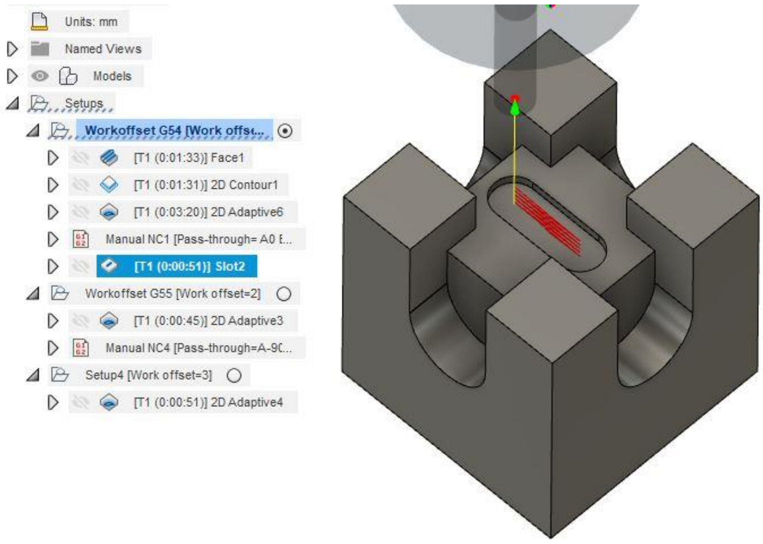
2D Contour

This feature allows the user to create open or close 2D contours on different Z levels. It is possible to also create multiple passes, and the way to program this operation it is just like the previous one.



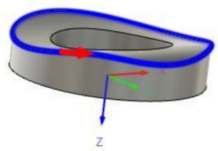
Slot

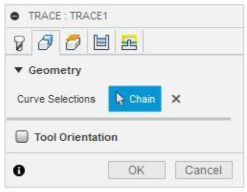
This is another function that it is programmed exactly like a 2D adaptive clearing or a 2D pocket, being dedicated to the creation of slots. The only concern regarding this operations is that, the select geometry needs to have been creating using the slot geometry on the CAD options, otherwise the software will not recognize it and the only way to machine it will be creating one different cutting strategy.



Trace

This is a 3-axis strategy that machines contours with varying Z values and with, or without, left and right-side compensation. The programming process of this operation is equal to the 2D contour, the only concern is that the selected curve needs to be the one with the z variation.





Thread

Once again, like the name indicates, this operation allows to create threads. The important points for the correct creation of this cutting strategy is to select the hole intended to be threaded and the correct tool.

Bore / Circular

Although these are two different options for cutting strategies, both are used for the same purpose and create equal toolpaths. They are both used for milling cylindrical pockets and islands, and the software allows to select multiple geometries while using these strategies.

Engraving

This strategy machines along contours with V-shaped chamfered walls, so the main point when programming this operation is to select the proper geometries and the correct tool for the cutting process.

2D Chamfer

This is the last of the 2D cutting strategies and allows to machine along contours creating a chamfered surface. Just like the previous operation is important to select both the right geometries and the right tool.

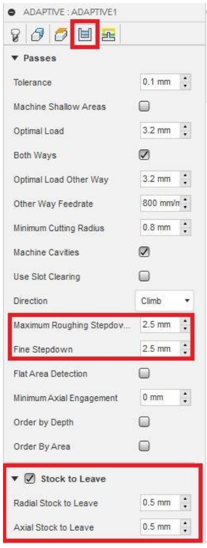
3D Cutting Strategies

These are more complex strategies than the previous ones, working with 2.5, 3 and more axis. While on 3D strategies the user does not need to provide every information to restrict the toolpath, the software is designed to handle those aspects.

Adaptive Clearing

This a roughing strategy available for clearing large quantities of material effectively. It is unique in that it guarantees a maximum tool load at all stages of the machining process and makes it possible to cut deep and with the flank of the tool without risk of breakage.

The only selection needed to program this is the stock or stock area intended and the software automatically recognizes the model. An important aspect is that, being this a roughing strategy and in order to optimize the toolpath, on the passes tab, the maximum roughing step down value should be equal to the fine stepdown value, and there should always be give some stock to leave like shown on the figure below. This also means these operations should always be followed by some finishing operations.



Pocket Clearing

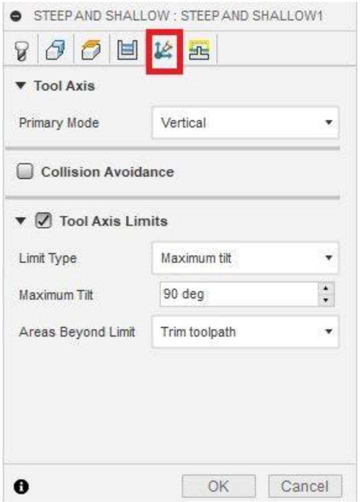
Just like the in the 2D strategies, the only difference between pocket clearing and the adaptive clearing seen before is that pocket clearing should be used when the area to machine is surrounded by material that doesn't needs cutting by all sides.

The programming of this operation is equal to previous and the same considerations must be respected.

Steep and Shallow

It is a finishing strategy that machines steep areas using contour passes and shallow areas using parallel or scallop passes. This strategy is used on parts that consist of steep areas and shallow areas in their geometry, for example, pats with 3D freeform surfaces. This feature is part of an extension, which means it is not available in all Fusion licenses but allows to save time by generating a toolpath for both steep and shallow areas and also incorporates a range of controls to allow machining of both steep and shallow regions efficiently.

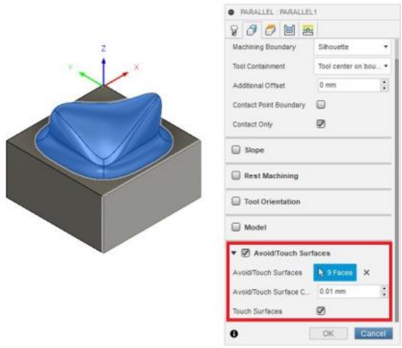
Steep and Shallow cutting strategy has an additional programming tab where the user is able to activate multi axis function and define maximum axis tilt angle.



Parallel

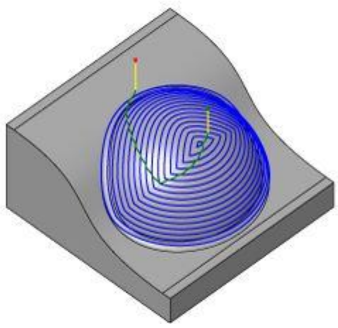
This is a finishing strategy where the passes are parallel in the XY plane and follow the surface in the Z direction. Parallel passes are best suited for shallow areas and down milling. To automatically detect shallow areas, the machining can be limited to a maximum angle between the tool tip and the surface. By selecting the down milling option, tool deflection can be minimized when machining complex surfaces.

An additional parameter on the geometry tab when programming strategies like this one is the possibility to demand the tool to touch or not the selected surfaces which allows for more control of what is machined.



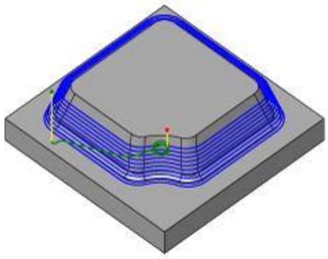
Scallop

This strategy creates passes that are at a constant distance from one another by offsetting inward along the surface. The passes follow sloping and vertical walls to maintain the stepover. Although scallop finishing can be used to finish and entire part, it is most used for rest machining, following a combination of contour and parallel passes.



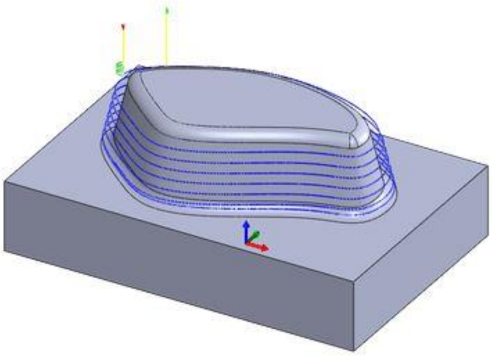
Contour

This is the best strategy for finishing steep walls but can also be used for semi-finish and finish machining on the more vertical areas of a part. If a slope angle is specified, for example 35 to 90 degrees, the steeper areas are machined, leaving the shallower areas up to 35 degrees for more appropriate strategies.



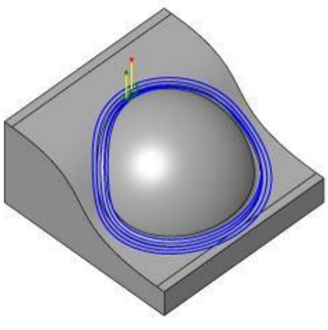
Ramp

This strategy creates a finishing operation intended for steep areas similar to the contour strategy, however this strategy ramps down walls rather than machining with a constant Z, as is the case for contour, ensuring that the tool is engaged at all times which can be important for certain materials such as ceramics.



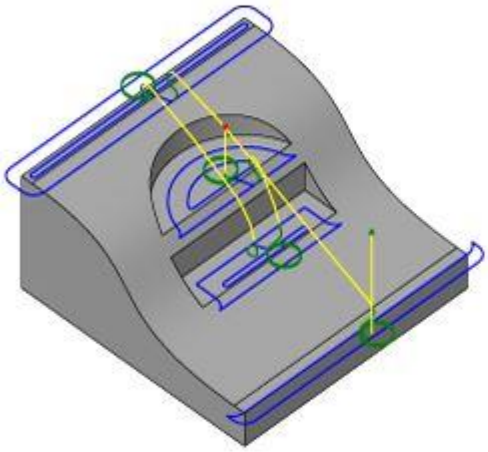
Pencil

Used to create toolpaths along internal corners and fillets with small radii, removing material that no other tool can reach. Whether using single or multiple passes, the pencil strategy is ideally suited for cleaning up after other finishing strategies.



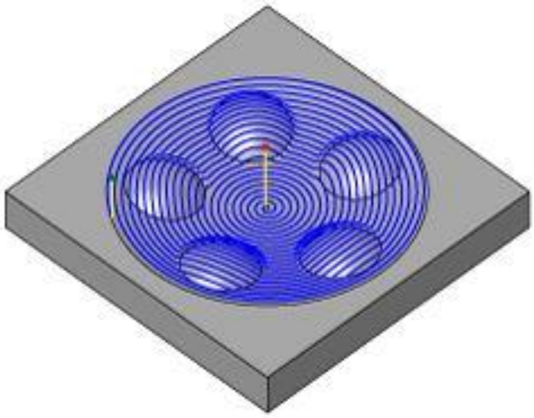
Horizontal

This is a clearing strategy that automatically detects all the flat areas of the part and clear them with an offsetting path. When the flat area is shelved above the surrounding areas, the cutter moves beyond the flat areas to clean the edges. Using the optional maximum stepdown, horizontal faces can be machined in stages, making the horizontal clearing suitable for both semi-finishing and finishing.



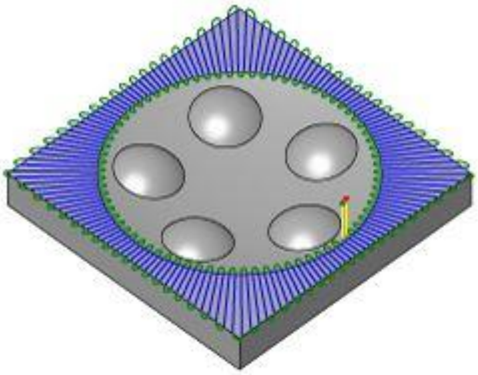
Spiral

Creates a spiral toolpath from a given center point, generating a constant contact as it machines within a given boundary. It is ideally suited for use on round shallow parts using tool contact angles up to 40 degrees, in conjunctions with contour passes for the more vertical faces. The center point of the detail to be machined is located automatically or can be user-specified. This strategy also supports tool contact angles.



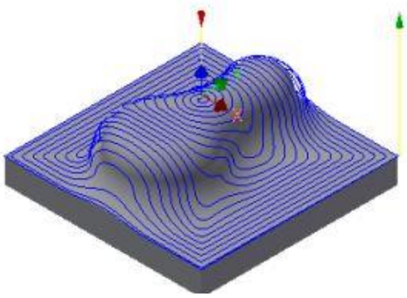
Radial

Like spiral machining, radial machining also starts from a center point, providing with the user with the ability to machine radial parts. It also provides the option to stop short of the center of the radial passes, where they become very dense. The center point of the detail to be machined is located automatically or can be user-specified. This routine can also be used with tool contact angles.



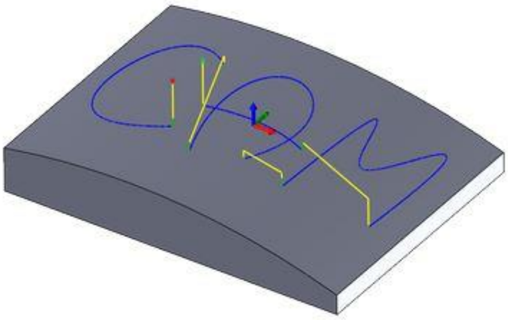
Morphed Spiral

This is a very similar strategy to the spiral, however, a morphed spiral operation generates the spiral from the selected boundary as opposed to a spiral operation which trims the generated passes to the machining boundary. This means that it can be used for additional surfaces for which spiral is not appropriate. It can also be very useful when machining free-form/organic surfaces. Although the scallop strategy is often used for these types of surfaces, both the sharp corners and the linking transitions between the generated passes can result in visible marks. The morphed spiral strategy generally provides a much smoother toolpath by avoiding these issues.



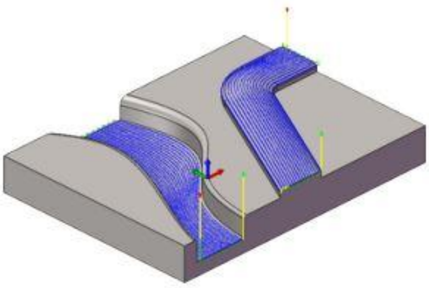
Project

Another finishing strategy that allows to machine along contours with the center of the tool. The provided contours are always projected onto the surface and so do not have to be on the actual surface. Project is commonly used for engraving text or symbols on a surface. The tool is moved into the surface by either entering an axial offset or a negative stock to leave.



Morph

It is a finishing strategy for machining shallow areas between selected contours with a consistent cutting direction.

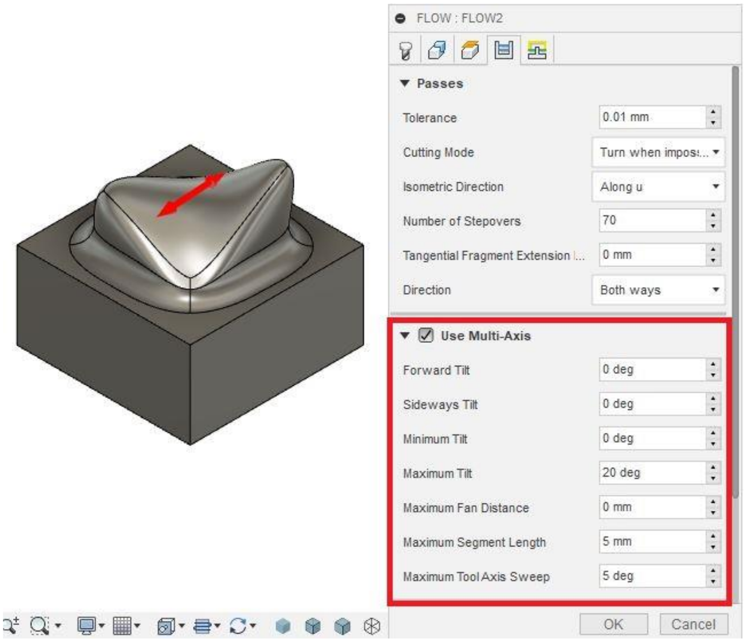


Flow

This is a finishing strategy, which follows the isocurves of a surface, to machine parts with curved surfaces. The machining passes and points of the toolpath are evenly spaced to provide consistent cutting motion over free-form shapes.

To get continuous toolpath from one surface to another, the user needs to ensure that the underlying isocurves of the surfaces align with each other and set the passes to run in the same isometric direction.

Although flow is a 3-axis strategy by default, multi axis can be enable and axis tilting can be limited and the isometric cutting direction can also be changed.



Drilling

This option allows to create drilling cycles with the additional option for hole recognition, which automatically finds hole features in any plane and creates drilling operations. The drilling operations are the easiest to program, the main concern is selecting the right tool and geometry.

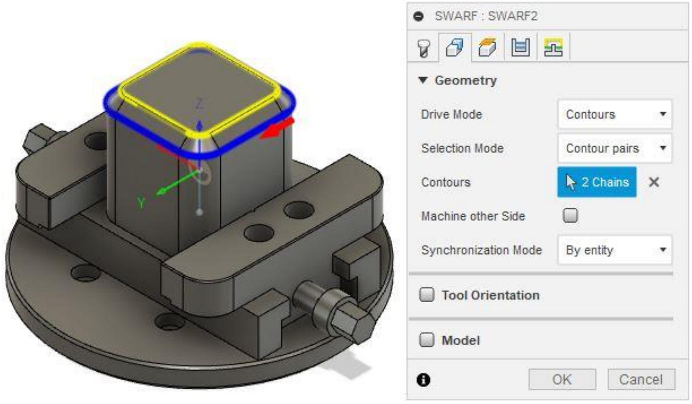
Multi-Axis

These strategies are dedicated to 4 or 5 axis continuous machining and are the most complex ones.

Swarf

Used for machining with the side of the tool. This strategy is best suited for chamfered, bevelled edges, or tapered walls.

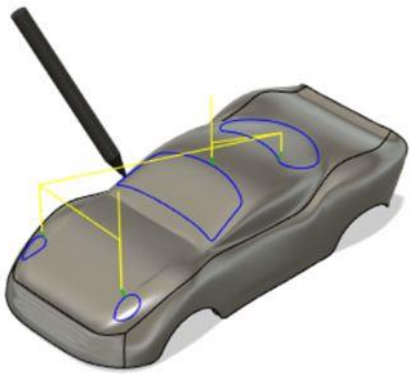
The selection mode allows to choose both contours or faces. For contour selection, first it must be selected the lower chain followed by the top chain, while for face there only needs to be selected the faces that delimit the path the tool must follow.



This strategy also allows for the user to establish single or multiple cut passes and offset the tool.

Multi-axis contour

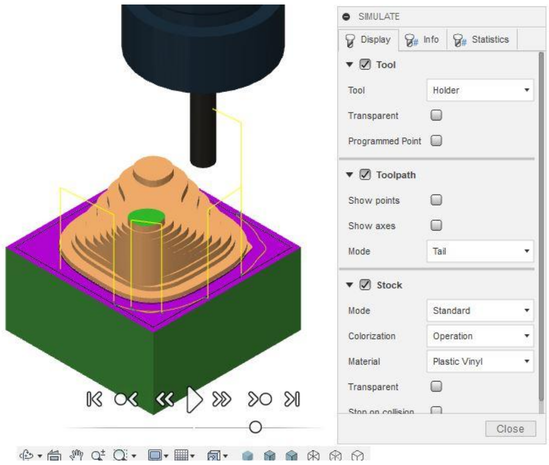
This strategy is used for machining with the tip of the tool along a given contact curve. By default, the tool is normal to the surface. Lead/lag and sideways tilt can be applied, when desired, to control the contact point on the tool. The strategy allows center, left and right compensation.



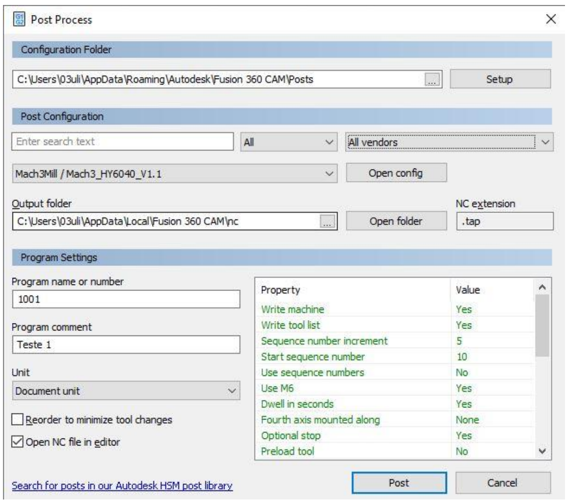
Actions

This is the interface regarding simulation, the post process and the creation of a setup sheet for the CAM programs.

Basically, the simulation shows all the toolpath created and allows the visualisation of the cutting process. The user can activate stock tool and holder, which leads for collision warning.



The post process feature is where the G-code files are generated, and there are a lot of elements that allow the personalization of the files. Fusion 360 comes with a vast list of post processors, and it has two additional post processors libraries, one called “Personal Post Library” which is a folder located on the Fusion 360´s files located on the user computer where modified post processor versions can be placed for use. The other called “Cloud Posts” is another file for the user´s post processors but working on cloud, this means that, if the user decides to use a different computer, all that needs to be done is login in the account and the post processors on this folder can be used.



ATTACHMENT V – VISE AND SETUP DEFINITION PROCEDURE

Fusion 360 Setup Definition

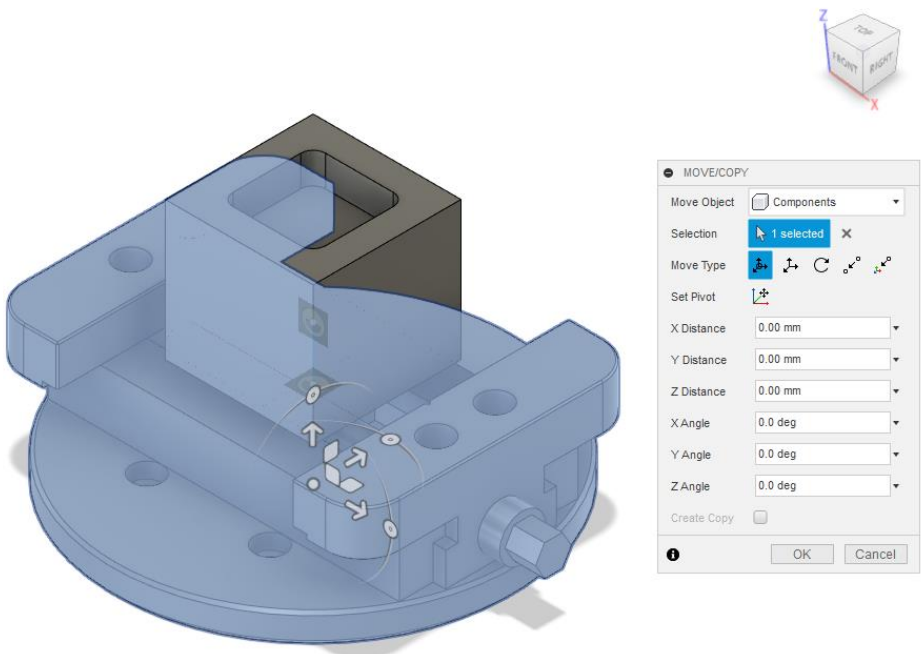
The procedure here presented explains how to import the workholding solution used in the HY-6040 machine, how insert the part onto it and how to define the different setups according with the fixtures defined on Mach3Mill.

Vise Insertion

The correct procedure is introducing the vise on the part model environment after this last one is created. To be able to do this, the vise model file needs to be saved in the same folder has the part model file.

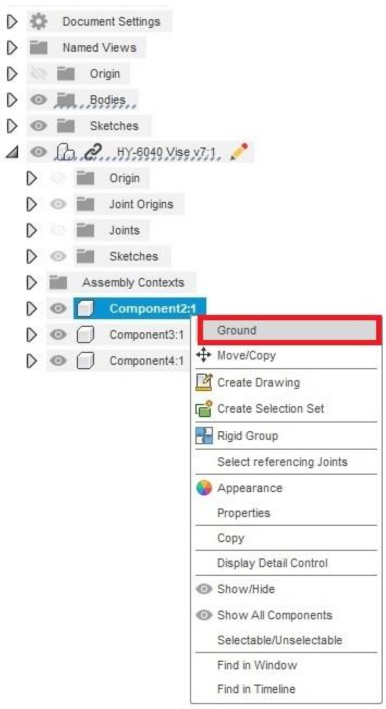
With this done, simply open the data panel, right click on the vise and select: “Insert into Current Design”.

The vise will appear on the workspace as showed in the image below.

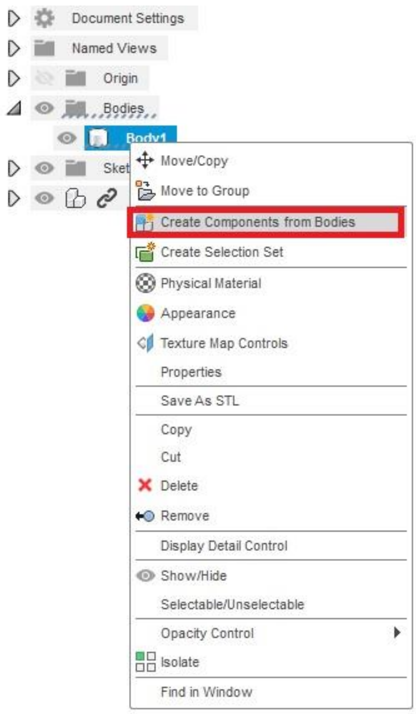


It is important to press OK and not move the vise from the (0, 0, 0) position.

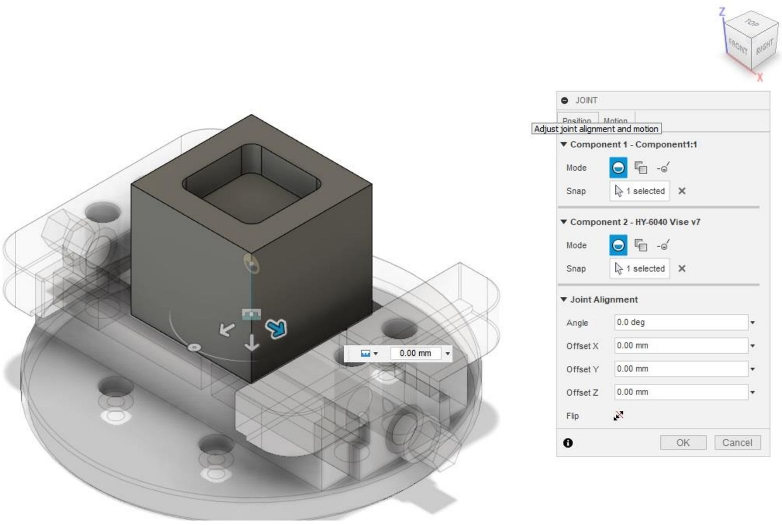
Open the vise tree to inspect it´s elements and ground component 2:1.



Now, remaining on the tree, find the part model, right click it and set Create Components from Bodies. This enables the option to create joints between the part component and the vise.



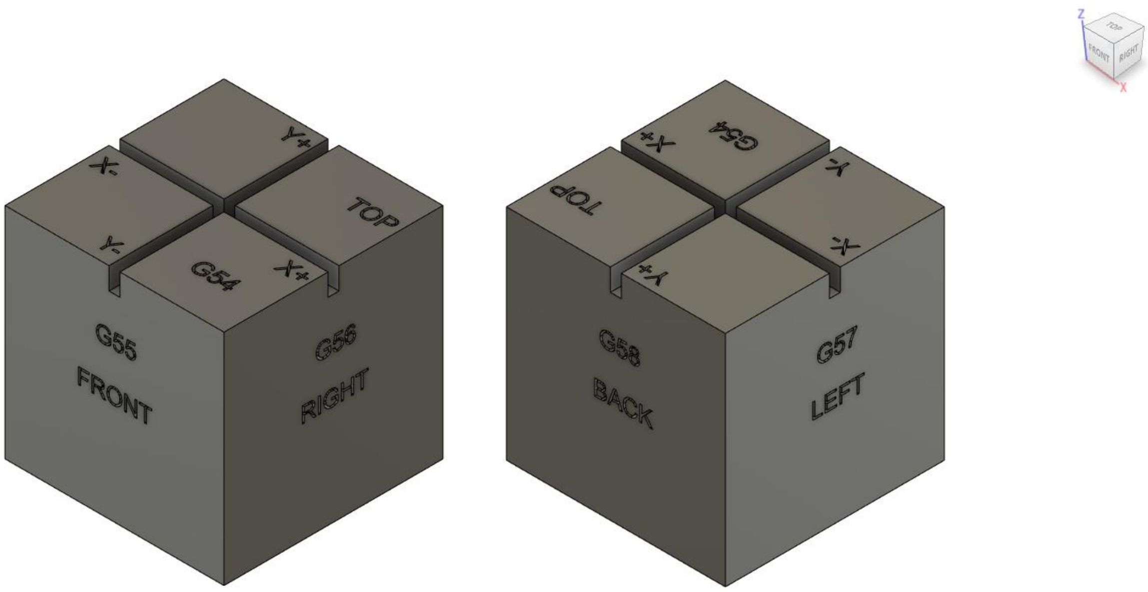
Continuing, a rigid joint must be set between the middle point of the bottom face of the part and the point located on the vise surface. If necessary, the model can be moved but never the vise.



The vise jaws can now be moved by simply dragging them with the mouse and the vise installations is completed. The user may now go to the manufacture environment.

Setup and Workoffset Definition

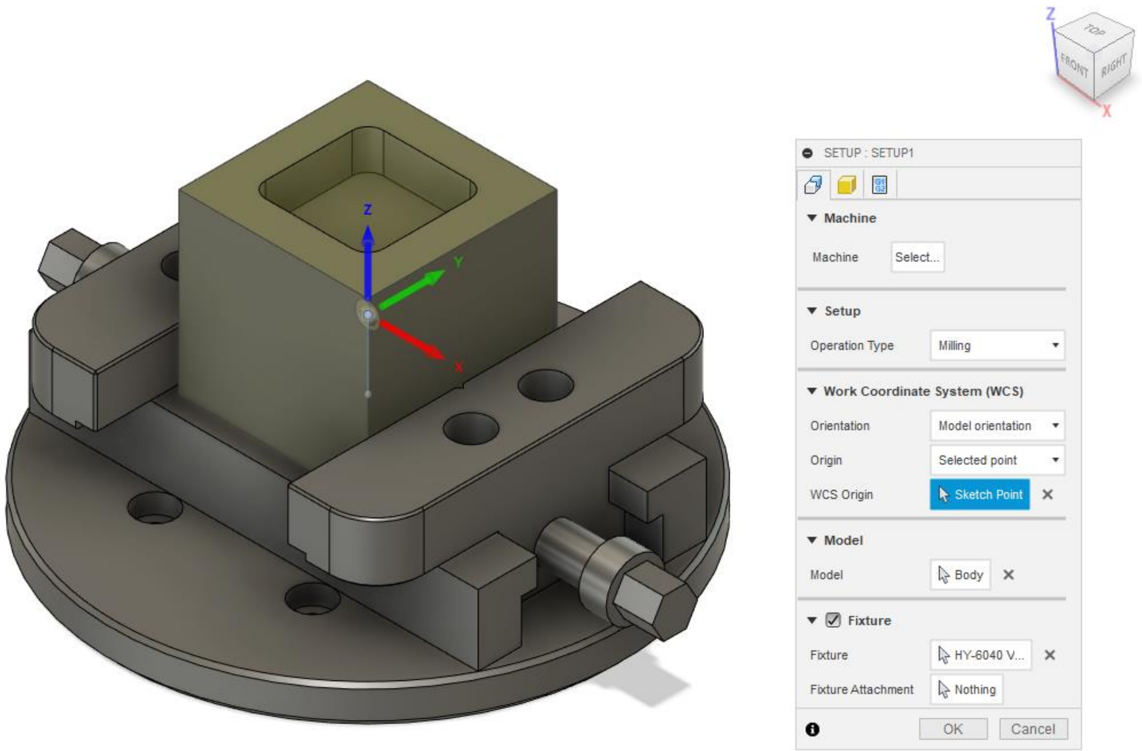
The setup and workoffset creation depend on the face to be machined. There are five workoffsets available: G54, G55, G56, G57 and G58 and they are defined on Mach3Mill according to the file “3+2 Model for Axis Orientaion” showed in the following image.



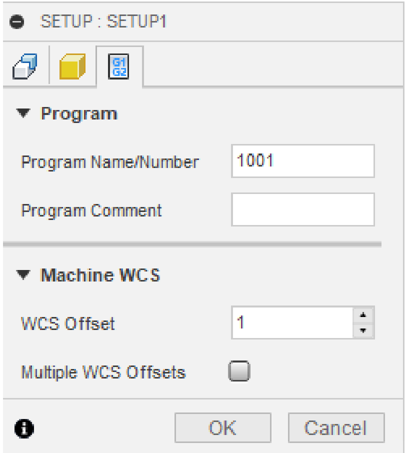
Adding on this, it is imperative to attribute the MRZP point contained on the vise model as the WCS origin.

Using the part from the previous example, if it was intended to machine the top face, the setup created would have to respect the G54 workspace.

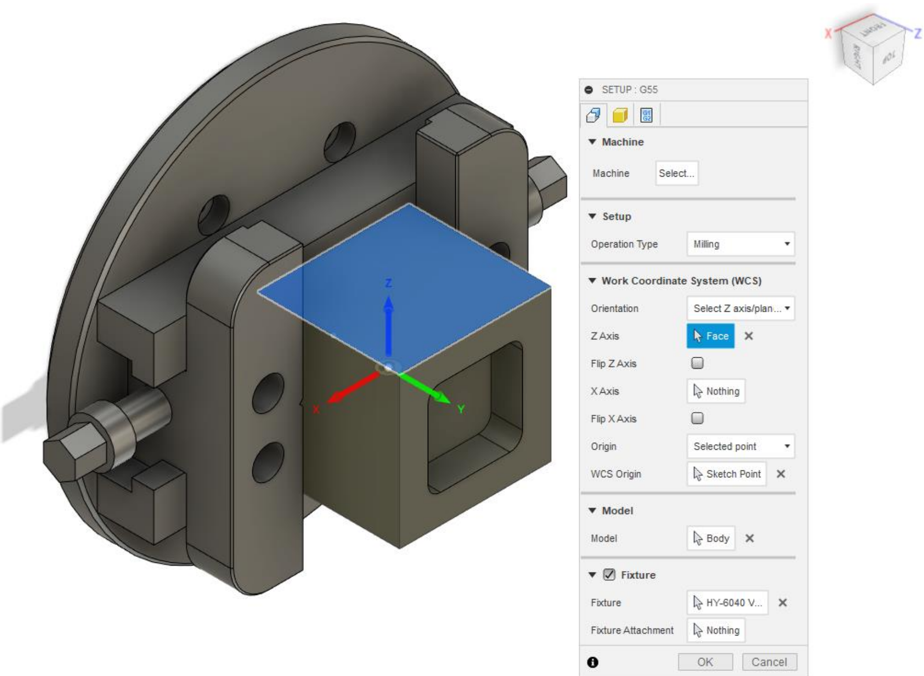
To do this, click New Setup, define the vise as fixture and the part as model, give the axis the correct orientation and on WCS Origin use the selection model and select the MRZP. This configuration should match the one showed on the subsequent image.

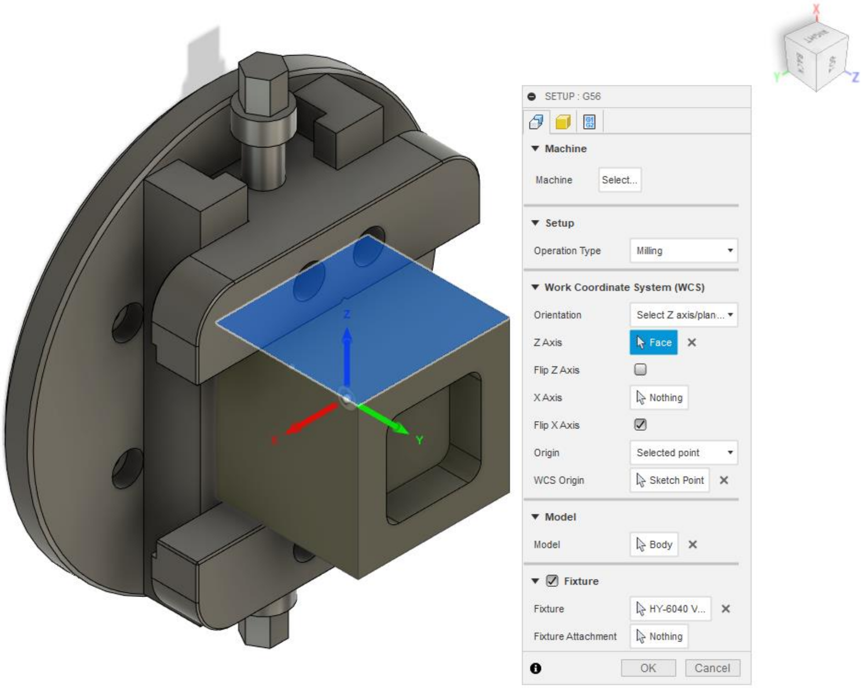


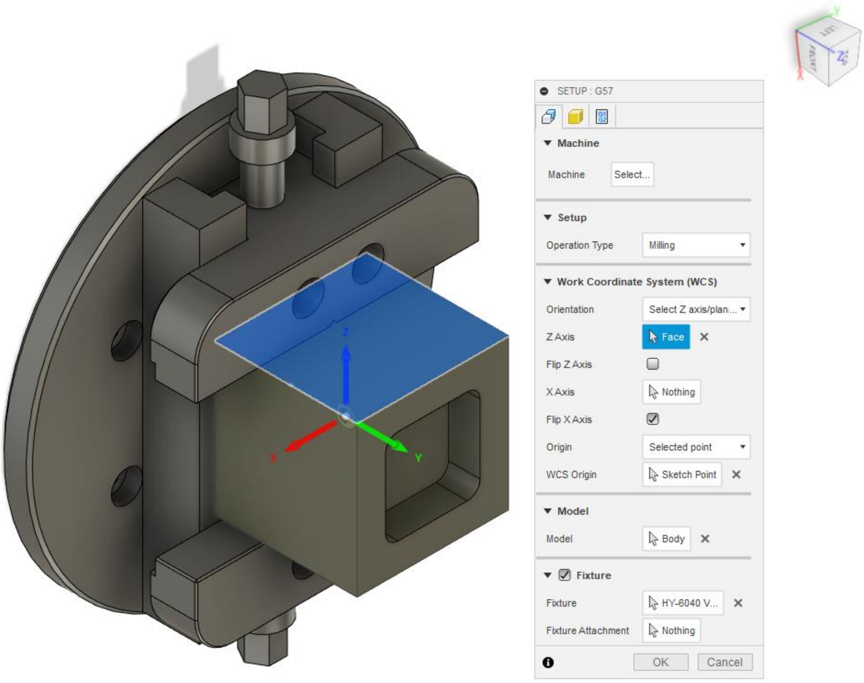
The only other matter to define is in the post processor tab, the WCS offset should be set to 1 (2 for G55; 3 for G56; 4 for G57; 5 for G58).

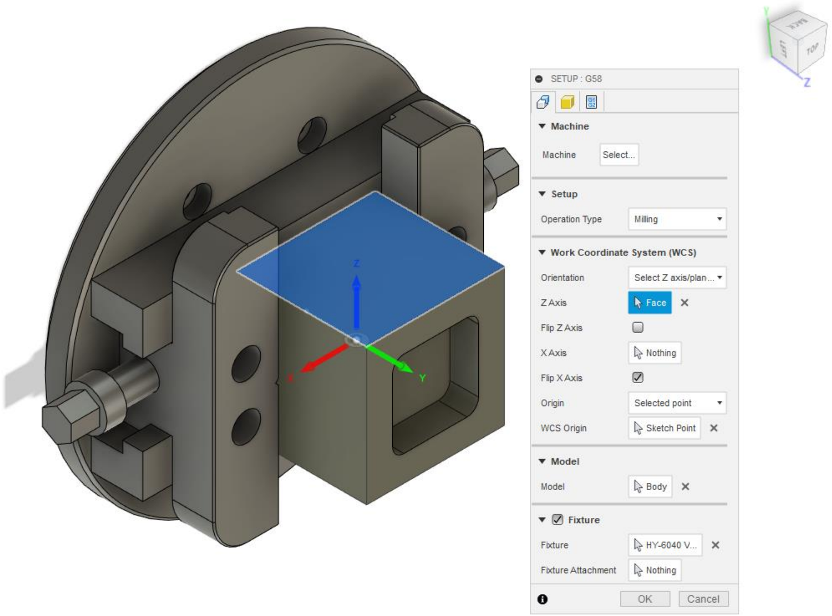


The following sequence of imagens shows the axis orientation for the other workoffsets with the purpose of solving every uncertainty that could remain.




SETUP : G56
Machine
Machine
Select...
Setup
Operation Type
Milling
Work Coordinate System (WCS)
Orientation
Select Z axis/plan...
Z Axis
Face
Flip Z Axis
X Axis
Nothing
Flip X Axis
Origin
Selected point
WCS Origin
Sketch Point
Model
Model
Body
Fixture
Fixture
HY-6040 V...
Fixture Attachment
Nothing
OK
Cancel

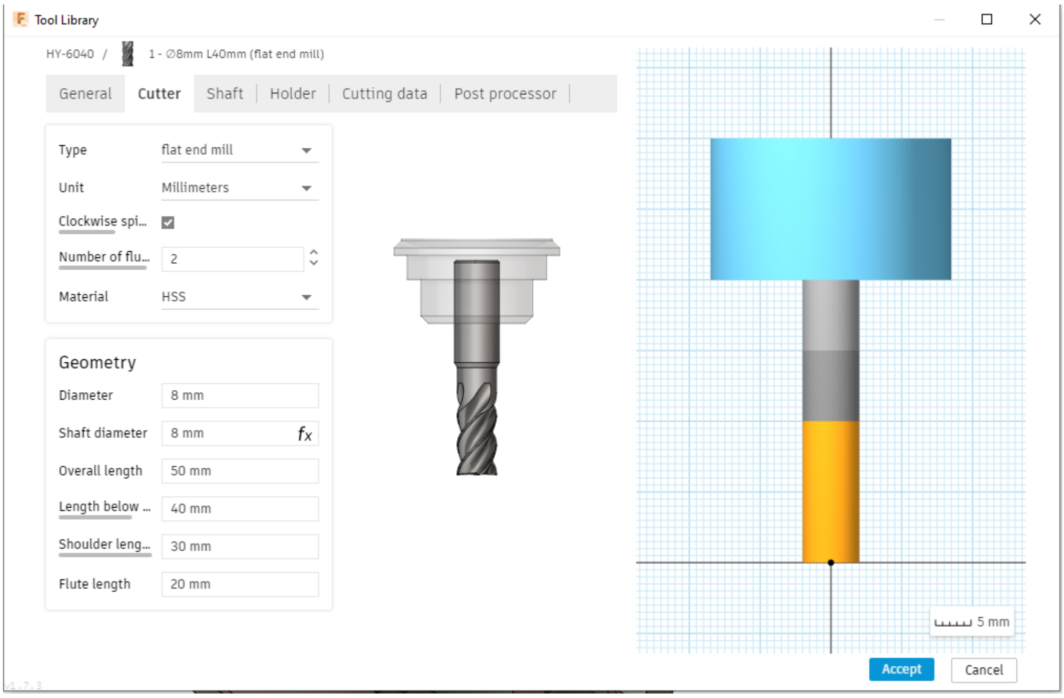

SETUP : G57
Machine
Machine
Select...
Setup
Operation Type
Milling
Work Coordinate System (WCS)
Orientation
Select Z axis/plan...
Z Axis
Face
Flip Z Axis
X Axis
Nothing
Flip X Axis
Origin
Selected point
WCS Origin
Sketch Point
Model
Model
Body
Fixture
Fixture
HY-6040 V...
Fixture Attachment
Nothing
OK
Cancel



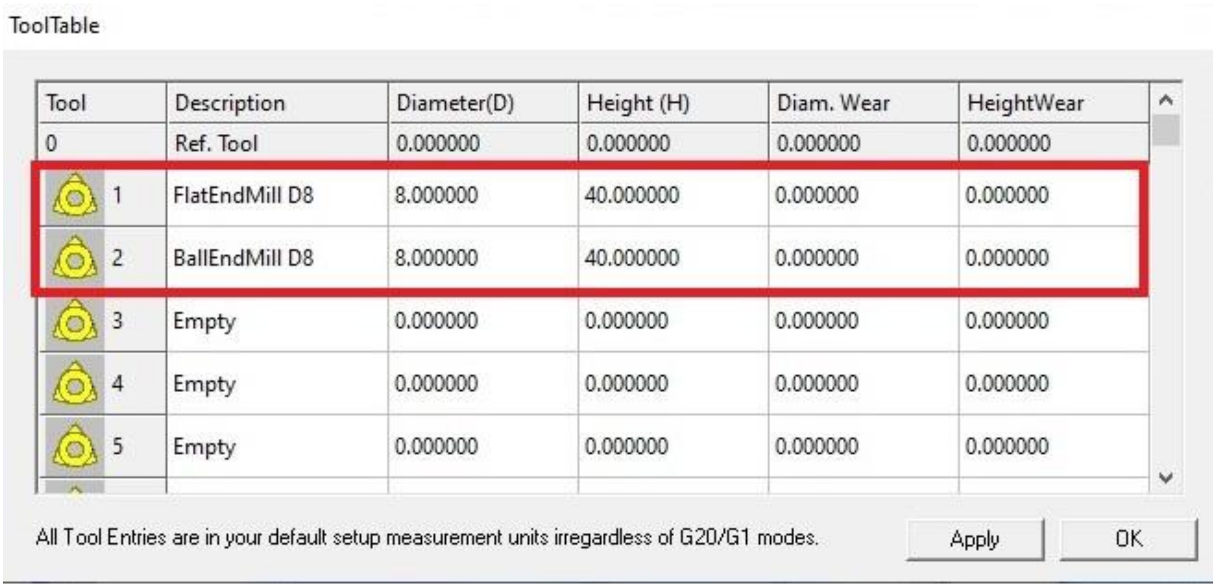
Tool Programming

Regarding the tools, they must be programmed both on Fusion 360 and on Mach3Mill. Before explaining this process is important to clarify some aspects: being that this machine does not have an automatic tool change, it is defined that the length below the holder should always be set to 40 mm. This way, when working with multiple tools the change for error is reduced since the operator will not have to introduce different values for tool offset.

Upon this, on Fusion 360 simply open the tool library, click on create tool and set the tool parameters like the example on the image below.



The same way on Mach3Mill:


ToolTable

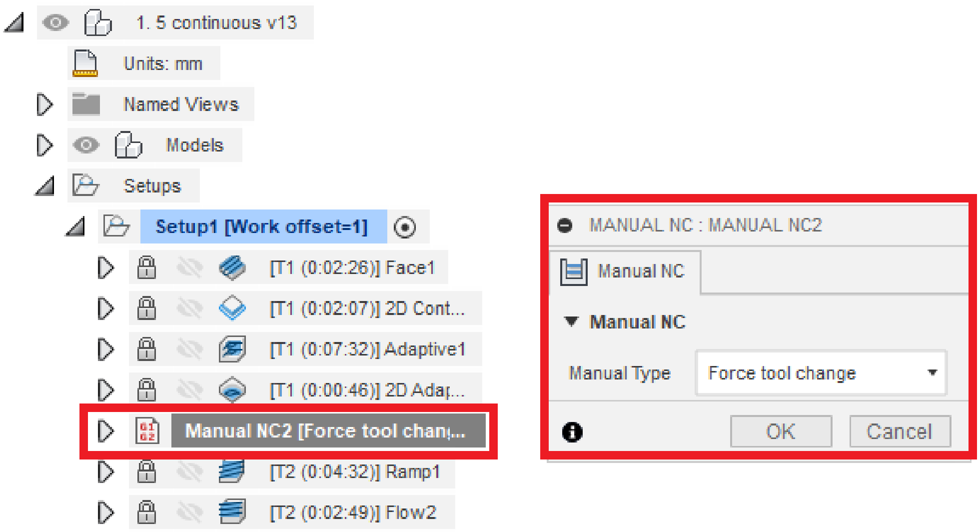
Tool	Description	Diameter(D)	Height (H)	Diam. Wear	HeightWear
0	Ref. Tool	0.000000	0.000000	0.000000	0.000000
1	FlatEndMill D8	8.000000	40.000000	0.000000	0.000000
2	BallEndMill D8	8.000000	40.000000	0.000000	0.000000
3	Empty	0.000000	0.000000	0.000000	0.000000
4	Empty	0.000000	0.000000	0.000000	0.000000
5	Empty	0.000000	0.000000	0.000000	0.000000

All Tool Entries are in your default setup measurement units irregardless of G20/G1 modes. Apply OK

While the diameter compensates for when the tool is machining on the left or right-side, the height (tool length offset) compensates for the vertical differences.

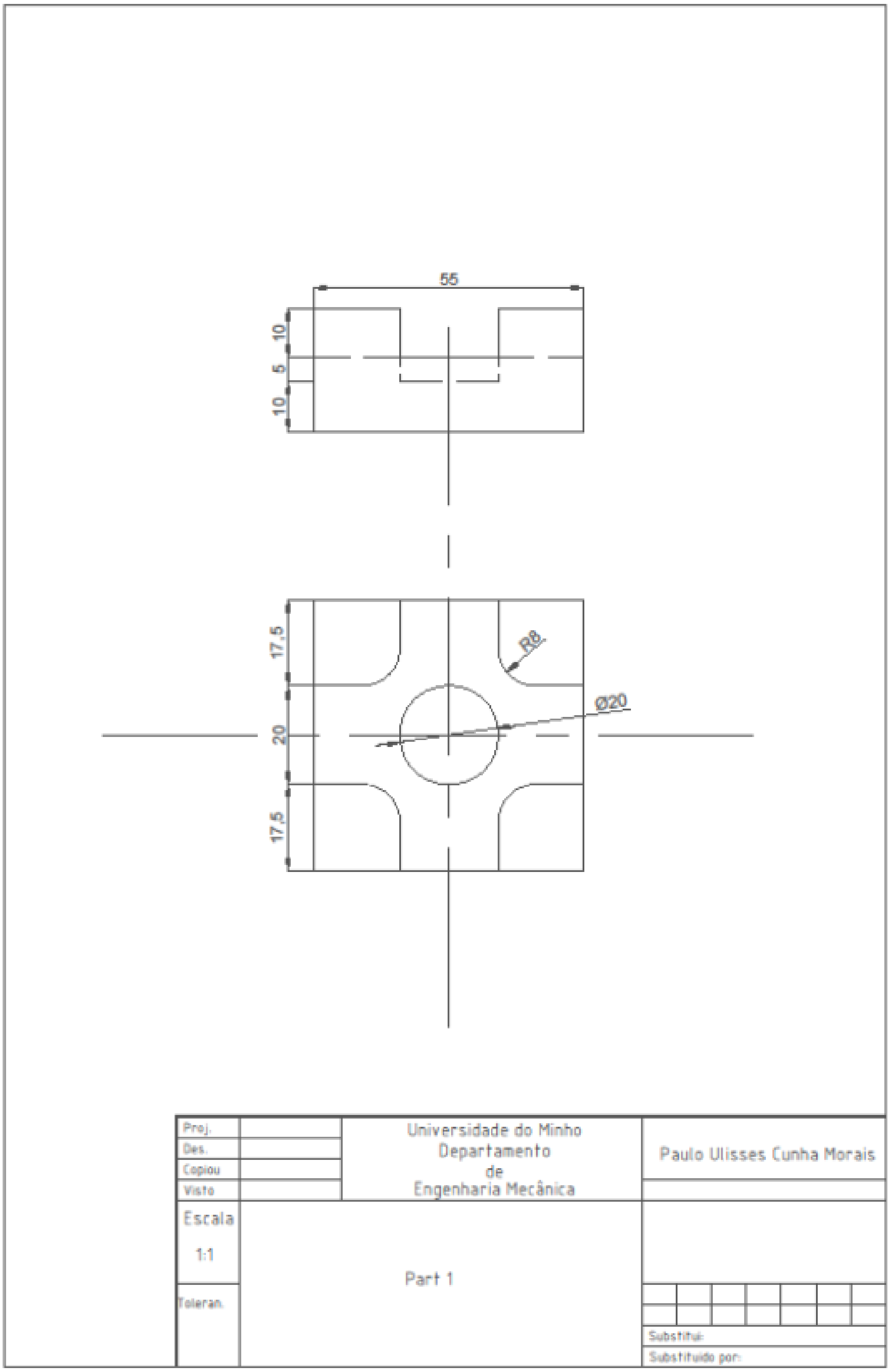
It is very important that the number of the tools match in both systems, because the G-code files will use the number programmed on Fusion 360 while Mach3Mill, upon reading the line, will associate the same number with its own tool library.

To command the tool change in Fusion a manual G-code line must be created where the tool change is necessary (see figure below).

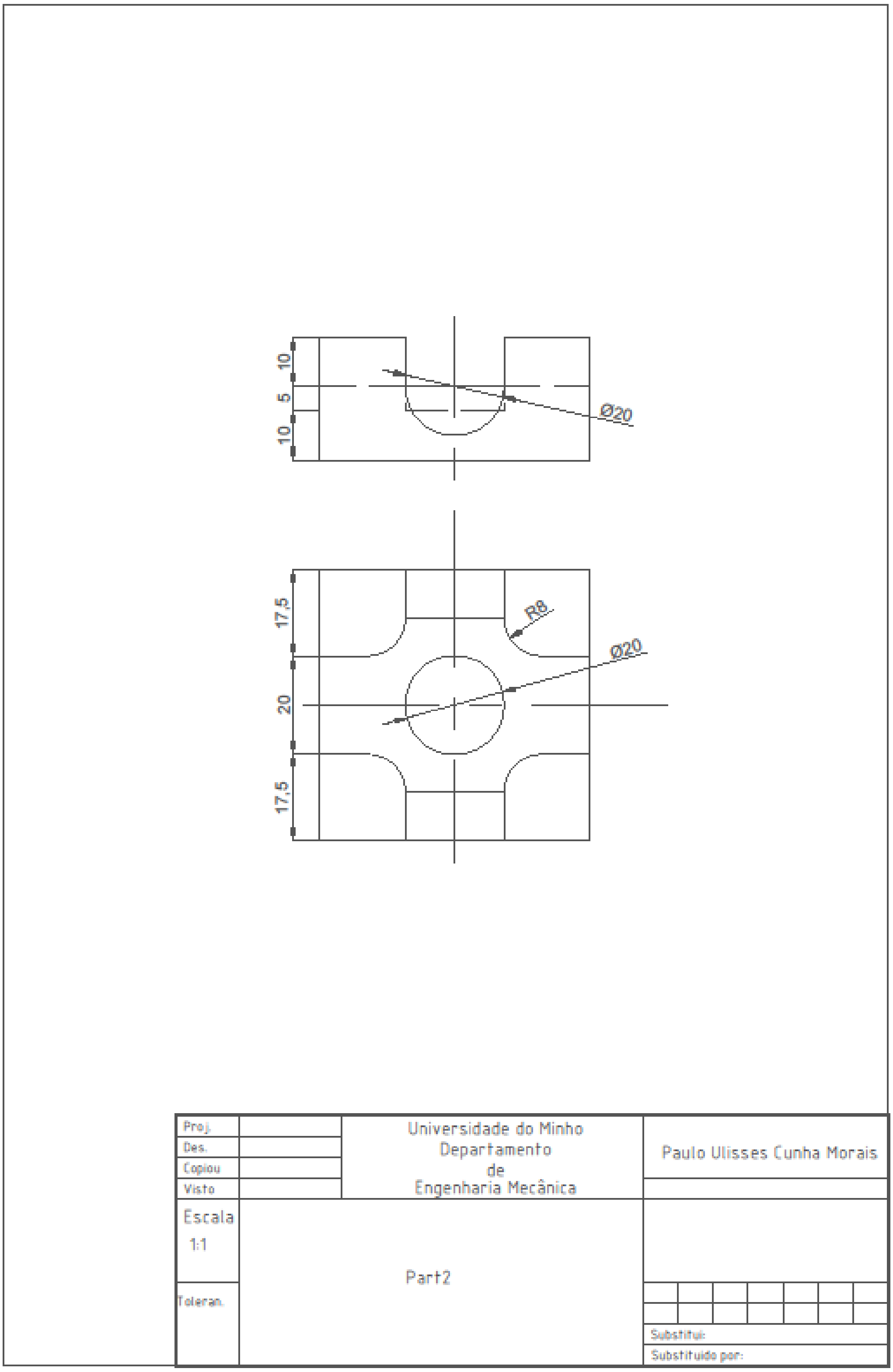


When in the machining process, Mach3Mill will read the G-code lines referent to the tool change, and will command the machine to lift and stop the spindle. The operator can now securely remove the tool, and insert the new tool with the 40 mm of tool length and press RUN.

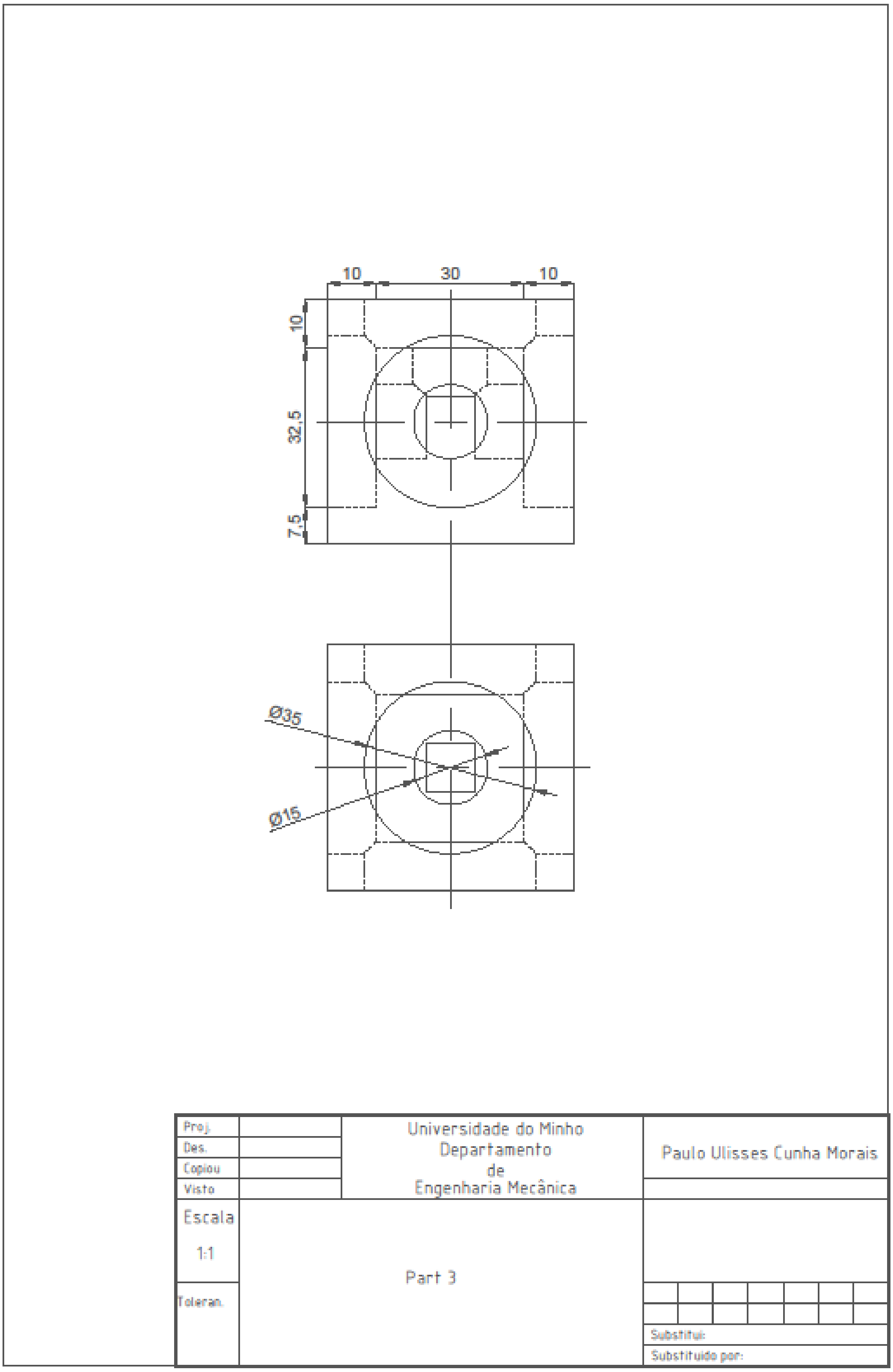
ATTACHMENT VI – TECHNICAL DRAWING PART 1



ATTACHMENT VII – TECHNICAL DRAWING PART 2



ATTACHMENT VIII – TECHNICAL DRAWING PART 3



ATTACHMENT IX – TECHNICAL DRAWING PART 4