



The present work was submitted to the Faculty of Engineering

Development of selected routines (procedures) in the CAM post-processing for CNC Vertical Machining Centers

Bachelor Thesis

by

Tuguldur Tulga

Supervisor 1 / Examiner 1:
Faculty

Prof. Dr.-Ing. Gunther C. Stehr, GMIT Engineering

Supervisor 2 / Examiner 2:
Ulaanbaatar

Batbayar Erdenee, CEO Darkhangeomash,

Advisor

Ulaanbaatar/Nalaikh,

8/16/18

Statutory Declaration

Tulga Tuguldur

Last Name, First Name

Student ID Number

I hereby affirm in lieu of an oath that I provided the submitted bachelor thesis independently and without undue external help. I did not use any sources other than those stated. In case that the work is additionally submitted on a data medium, I declare that the written and the electronic form are completely identical. The work was not submitted in the same or similar form to any examination authority.

Place, Date

Signature

Acknowledgments

This bachelor work is a result of some phenomenal help and support extended to me by many individuals at GMIT and Darkhangeomash: I would like to express my sincere gratitude to my supervisors Prof. Dr.-Ing. Gunther C. Stehr and Batbayar Erdenee, CEO Darkhangeomash, for their continued help and guidance throughout my thesis work.

A special thanks go to Lkham Erdenekhuu, safety engineer at Darkhangeomash, for his encouragement and valuable lessons. Also, I would like to thank Chinggis, head engineer at Darkhangeomash, and Ganzorig, CNC operator at Darkhangeomash for their help in manufacturing.

Abstract

CAD/CAM software are used for modelling and simulating. However, they are not enough for the CNC machines to manufacture a product. Here, the post-processing plays an important role customizing the CAD/CAM codes and making them understandable for the machine. It doesn't matter whether the CNC machine is modern or expensive, each machines need post-processing to ensure the codes are optimum for the machines to operate at its finest.

The goal of this thesis was to develop a modified post-builder specifically for "Victor Taichung Vcenter-102e" CNC machine which is in operation at Darkhangeomash company. The task included among other things such as creating 3D-model and operations on "Siemens PLM NX 10" software and making a real-time simulation on "Nanjing SwanSoft" CNC simulator for verifying the G-code. The G-codes were generated by "Siemens PLM NX 10" and post-processed by using NX post-builder software considering. The post processed G-code is then used on Vc-102e machine to manufacture a piece which is assigned by Batbayar Erdenee, CEO Darkhangeomash.

Table of Contents

Statutory Declaration	i
Acknowledgments	ii
Abstract	iii
1 Introduction.....	1
1.1 Background	1
1.2 Objectives.....	1
2 Machining	1
2.1 Metal removal processes.....	1
2.2 Metal cutting tools.....	2
2.2.1 End Mills	3
2.2.2 Center-Cutting end mills	3
2.2.3 Face Mill.....	4
2.2.4 Hole-making tools	5
3 CNC machines	5
3.1 Operation procedures.....	6
3.2 Coordinate systems.....	7
4 CAM programs (Siemens PLM NX).....	7
4.1 Introduction to NX.....	8
4.2 3D-modelling	8
4.3 Operation generating.....	9
4.4 Post-processing.....	11
5 Vertical Machining Center (Vc-102e, Victor-Taichung).....	12
5.1 Safety procedures	12
5.2 Machine codes for Vc-102e.....	13
5.2.1 Tool function	13
5.2.2 (Miscellaneous) M functions.....	13
5.3 (Preparatory) G functions	16

6	Post-processor building for 3-axis Vertical Machining Center	16
6.1	Settings of G and M codes	16
6.2	Programming.....	17
6.2.1	Program start sequence.....	17
6.2.2	Operation start sequence.....	18
6.2.3	Toolpath settings.....	19
6.2.4	Operation End Sequence.....	20
6.2.5	Program End Sequence.....	21
7	Results	22
7.1	Operations on NX 10.....	22
7.2	An output of G-codes from NX post builder	22
7.3	Simulation on “Swansoft” CNC.....	24
7.4	Machined piece	24
8	Discussion	25
9	Recommendation for future work	26
	Appendices.....	29
	G-code list for Fanuc controller.....	29
	M-code list for Fanuc controller	32
	Post-processed G-code by NX 10, using 3axis	34
	Meaning of characters in machine code	36

1 Introduction

1.1 Background

In the metalworking industry, the application of CNC machining centers plays an important role in the efficient and fast production of high precision parts. To implement the CAM (Computer Aided Manufacturing) approach in the industrial production process, the usage of the 3D-CAD data set from the design procedure of the metal parts should be used for the machining programming in the CNC center. However, a direct import of the data file is not possible, because the CNC center programming takes place based on machine language. Therefore, a specified CAM post-processing procedure has to be developed as a transformation tool between the CAD data set and the machine code for the CNC center.

1.2 Objectives

Objectives of this thesis were to develop a modified NX post-builder for „Victor Taichung Vcenter-102e” CNC machine and that were to be used to manufacture a piece. The work involved modelling the piece on NX 10 CAD/CAM software and making a simulation on “Nanjing Swansoft CNC simulator” to verify the G-code before sending them to the CNC machine.

2 Machining

Machining is any of various processes in which a piece of raw material is cut into a desired final shape and size by a controlled material-removal process. Machining is a part of the manufacture of many metal products, but it can also be used on materials such as wood, plastic, ceramic, and composites. Much of modern-day machining is carried out by computer numerical control (CNC), in which computers are used to control the movement and operation of the mills, lathes, and other cutting machines.(1)

2.1 Metal removal processes

There are 3 main types of metal removal processes - conventional machining, abrasive processes, nontraditional processes shown in Fig 1. In conventional machining, a sharp cutting tool is used to mechanically cut the material to achieve the desired geometry. The three principal machining processes are turning, drilling, and milling. Another group of material

removal process is the abrasive processes, which mechanically remove material by the action of hard, abrasive particles. Finally, there are the nontraditional processes, which use various energy forms other than a sharp cutting tool or abrasive particles to remove material.(2)

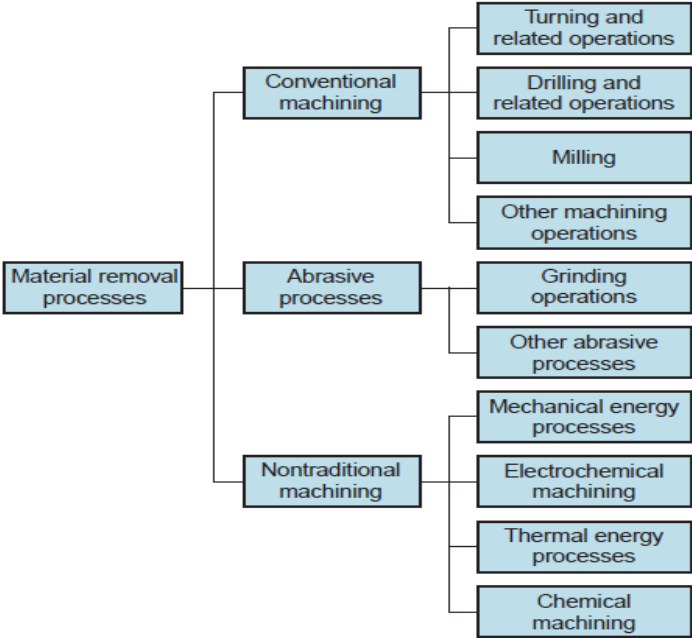


Figure 1 – Metal removal processes (2)

2.2 Metal cutting tools

Various types of tools are available designed for different CNC operations. Therefore, only the tools used in this operation are discussed, such as end mills, face mills, and drills. In Table 1, the available tools offered by the company is shown.

No. of tool	Name of tool	Diameter, mm	Length, mm	Number of teeth
1	EM_D	6	198.40	2
2	EM_D	3	125.60	3
3	EM_D	20	185.60	4
4	EM_D	20	189.00	4
5	FMI_D	40	153.30	4
6	FMI_D	50	109.20	4
7	Drill_D	10	187.60	-
8	Drill_D	8.5	176.25	-
9	Drill_D	16	199.00	-

10	Drill_D	17	240.00	-
11	Drill_D	20	182.50	-
12	Setting_tool	16		-

Table 1 – Availabe tools at the company

2.2.1 End Mills

End milling tools include flat, ball, bull nose and chamfer as shown in Fig2.

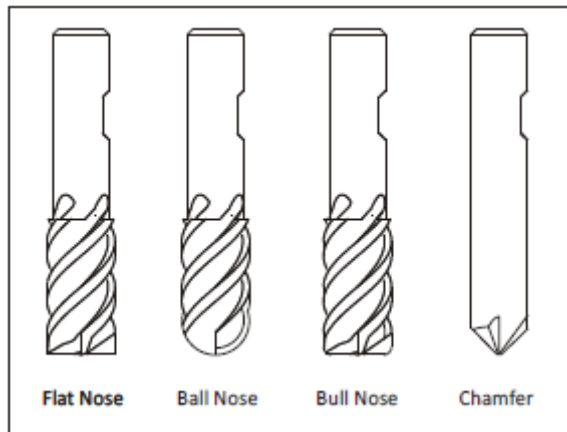


Figure 2 - Types of end mills (3)

Flat nose mills are used for milling 2D contours and pockets. Ball nose mills are used for 3D milling. Bull-nose end mills have a radius corner. They are used to create a fillet on the bottom of a wall. Because they are sturdier than an end mill they are also sometimes used for roughing operations. Chamfer mills have an angled nose used to create a chamfer or to de-burr parts. The number of Flutes Milling tools usually have either two or four cutting flutes. Two flute cutters provide more chip clearance when in close areas. Four flute mills are more rigid, can be fed faster, and are preferred when greater chip clearance is not required, such as when milling an outside contour.(3)

Flat nose mills are used for cavity milling operation as it has the most suitable shape for the cavity's geometry.

2.2.2 Center-Cutting end mills

Milling tools are either center cutting or non-center cutting. Center cutting mills can plunge straight down into material, while non-center cutting tools cannot. Non-center cutting end mills require a pilot hole, ramping, or helical motion to plunge into a material.

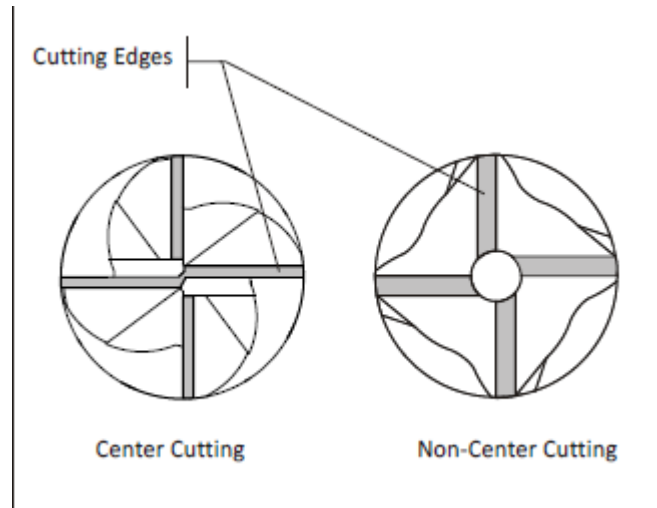


Figure 3 - Cutting edges of center and non-center cutting(3)

Milling tools that are available in the company are all non-center. Thus, a helical motion is applied to plunge into a material which is illustrated as the yellow line in Fig.4

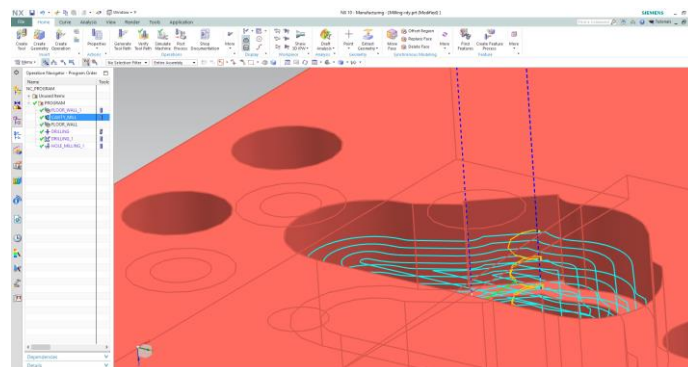


Figure 4 - Cavity milling in NX

2.2.3 Face Mill

A face mill has cutting inserts that are replaced when worn. They are rigid, may have up to eight or more cutting edges, and can remove material quickly. They are used for the first machining operation to quickly create a flat finished face on the part shown in Fig 6.

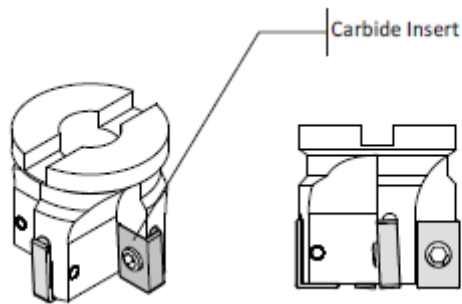


Figure 5 - Face mill(3)

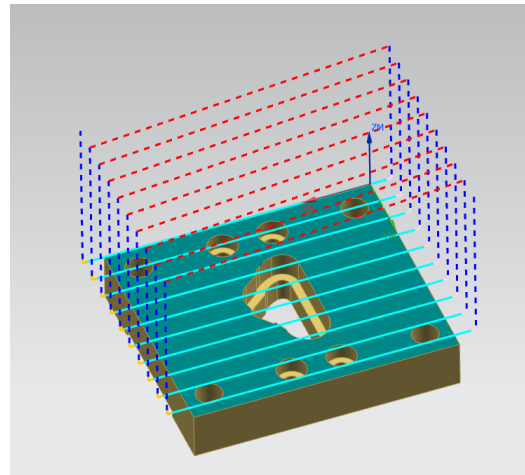


Figure 6 – NX surface roughing operation

2.2.4 Hole-making tools

Center-spot drills

Center (spotting) drills are short and very rigid drills used to create a conic on the face of the part. Because they do a sharp point and resist bending, they locate the hole precisely. The conic helps prevent the subsequent drill from wobbling and ensure the drill is located precisely and drill straight down.

Countersink drills are used to create the conical face for a machine screw. Combined spotting-countersinks are used to create a screw clearance hole and countersink in one operation.

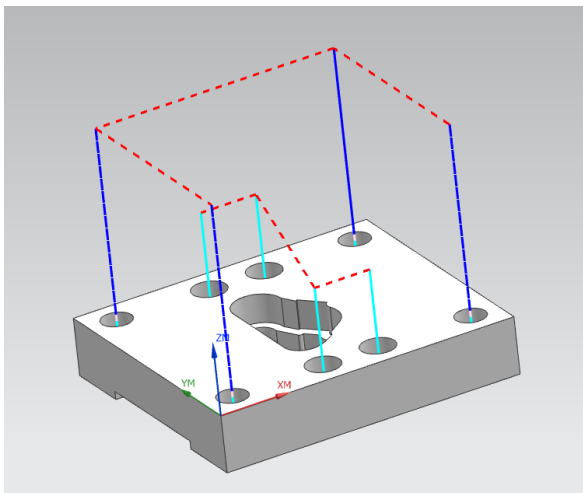


Figure 7 – Drilling operations

3 CNC machines

Numerical control (NC) refers to the automation of machine tools that are operated by abstractly programmed commands encoded on a storage medium, as opposed to manually

controlled via handwheels or levers, or mechanically automated via cams alone. The NC machines based on existing tools that were modified with motors that moved the controls to follow points fed into the system on paper tape. These early servomechanisms were rapidly augmented with analog and digital computers, creating the modern computer numerical controlled (CNC) machine tools that have revolutionized the design process.(4)

CNC Machines fall into many categories. There are milling, cutting, drilling, bending machines and many more. They differ in principles and functionality but share the aim of refining materials. As opposed to manually operated machines, CNC machines are controlled by control systems. Control system is a special software whose task is to control individual parts of the machine-like motors or valves in order to achieve the desired result. The system itself is commanded either manually by an operator or through a specialized code, mostly the G-code or rather called machine code. G-code is a semi-standardized language for CNC Machines whose instructions describe motions and actions of the machine. (3)

3.1 Operation procedures

In Fig 8, the operation procedures of the CNC machine are shown. First, the engineering drawing according to the design of the workpiece is prepared. CAD software can be used in this procedure. Then based on the information in the engineering drawing, the computer numerical controlled machinery program (CNC machinery program) will be composed. The CNC machinery program includes all the geometrical and technical information. The geometrical information decides the target position of the movement of the tools, cutting direction and movement priority, etc. Technical information includes the choice of tools, the rotational speed of the main axis, the rotating direction, cutting speed, etc. The CNC machinery program will input the geometrical and technical information into the digital controllers, while the input devices include paper tape, magnetic tape, external keyboard, etc.

After the digital controller has processed all the information, the moving path of the tools and the suitable procedures will be available. Lastly, the digital controller will control the whole

machine and its tools to process the materials into a required workpiece.

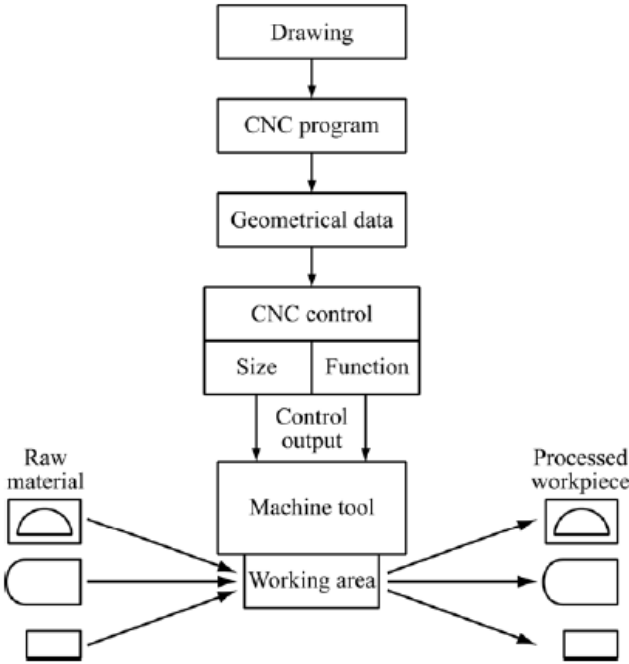


Figure 8. Simplified operation procedures of CNC machine

3.2 Coordinate systems

The coordinate system plays an important role in CNC machining. They describe the positions and the movements of the machine in CNC code. There are two types of coordinate systems, absolute and incremental. In an absolute system, the position of the points is relative to the origin. But in an incremental system, the position of the points is relative to the current position.

4 CAM programs (Siemens PLM NX)

The production method that requires a computer to control the machines is called a computer-aided manufacturing, simply called CAM. CAM is closely related to the computer-aided design (CAD) because the output information about the products from the CAD can assist the composing of the production program. (2)

A CNC machinery program is composed of simple alphabets and numerical values. It is hard to understand and will take a long time to calculate and compose. Thus, it is easy to make mistake when programming. To eliminate these potentials of making mistakes, some computer-aided programming software is used, such as NX, SPLIT, COMPACT, MASTERCAM, etc. Draft of the working steps and data according to the engineering drawing

are made, and then the application software is used to analyze the data automatically to produce the CNC machinery program. Application software has a number of advantages:

- Usually, some easy symbolic language is used to represent geometric and technical data.
- Can reduce the time required to describe the workpiece and processing procedures.
- Minimize the works like data entry, calculation, etc, so it can reduce the mistakes of the CNC machinery program.
- Application software can usually be applied to different machines, so the programmer can ignore the different coding of different machines.
- Some application software can even have a simulation program, so as to help the programmer to check the mistakes and to modify the program. This can reduce the time and money wasted when something goes wrong. (2)

4.1 Introduction to NX

NX is an advanced CAM/CAD/CAE software developed by Siemens “PLM Software”.

- CAD (Computer-aided): this software makes product design more efficient by automating processes that were once manual, such as traditional drafting. You can also do solid modeling (creating 3d representations of your designs)
- CAM (Computer-aided manufacturing): it is used to control manufacturing machine tools like lathe, milling or electrical discharge machine.
- CAE (Computer-aided engineering): is the software related with the engineering analysis simulations. (6)

4.2 3D-modelling

Using NX 10 software assigned part shown in Fig 9, is modelled, shown in Fig 10.

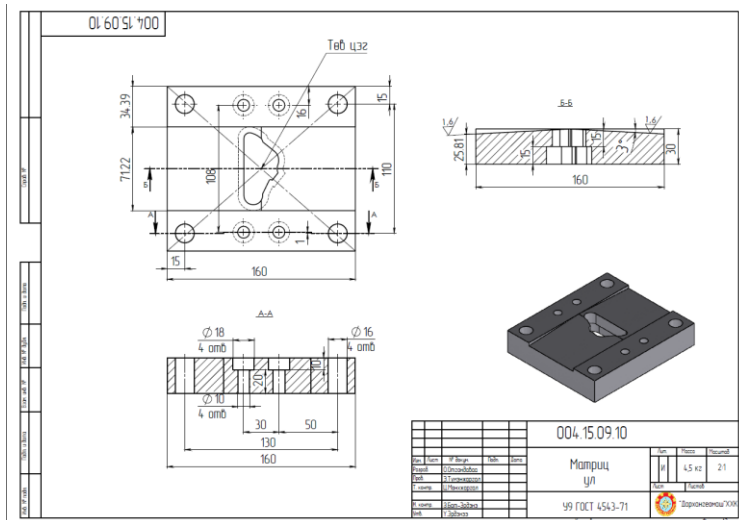


Figure 9 – Drawing of assigned part

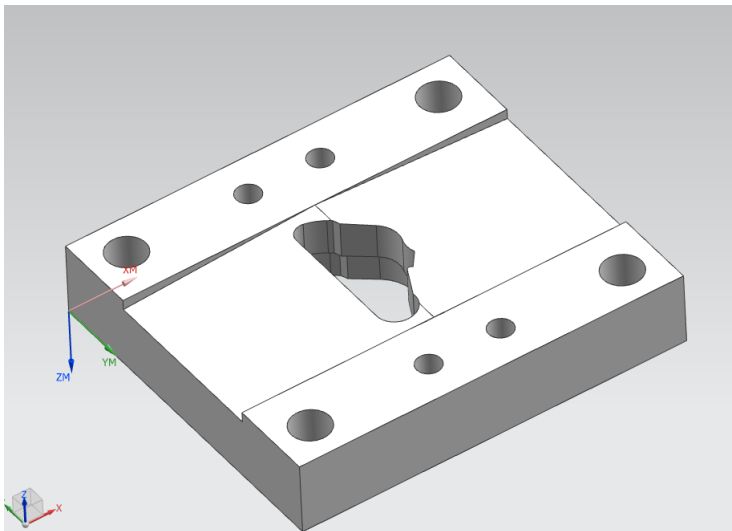


Figure 10 - Modelled part on NX 10

4.3 Operation generating

Using operation generating, various types of machining methods such as millings, drilling, hole making, turning, wire-EDM can be applied to a working part. Each method has its own subtypes widening its range of applications, shown in Fig 11.

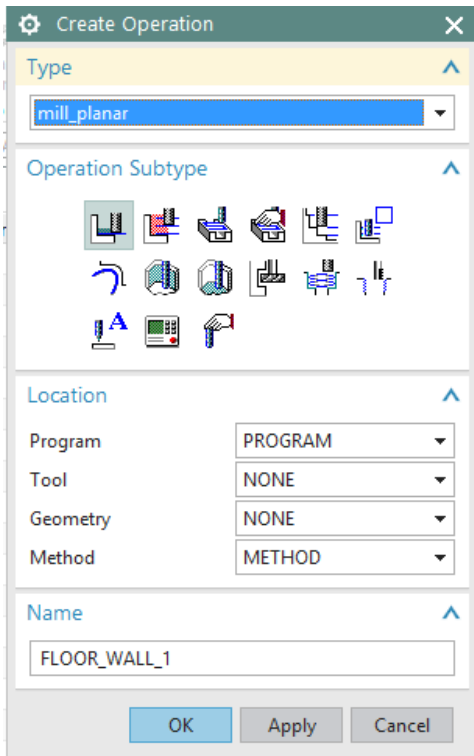


Figure 11 - A layout of NX machining methods

To generate operation, some essential parameters are needed. First geometries of the working part are to be specified including the actual part, check body (optional), working geometries, and blank. They are shown in Fig 12.

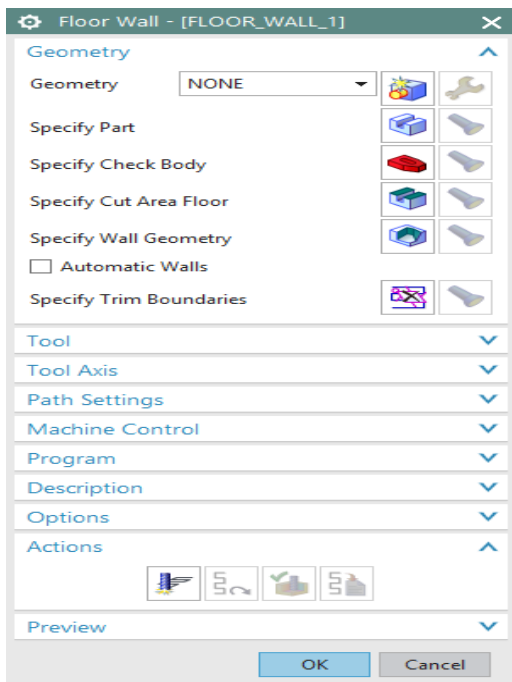


Figure 12 – Operation parameters on NX

There are several ways of assigning a blank. Using a solid geometry as the blank or allowing the software to assign a prismatic block with desired offsets in the X, Y, and Z directions.

As we have already created a block we can use that as the blank geometry. If a sort of clamping is included, then it can be specified by the check body to prevent a collision between machine tool path and part holders or clamps. Although, an operation can be generated without specifying. In some cases, the collision can be avoided by increasing travel distance of a tool to a working face. But, it will increase the total time of operation. Another important decision to make is to select the right shape and size of the tool. It is important to consider geometries, material, size of a working material and machining specification. In most cases, one tool is not enough to complete a whole operation. Thus, it is necessary to attach multiple tools to a tool turret.

In this certain case, however, tool selections are limited since they are provided by the company. Available tools are shown in Table.

4.4 Post-processing

The primary use of the Manufacturing Application is to generate tool paths for manufacturing parts. Generally, we cannot just send an unmodified tool path file to a machine and start cutting because there are many different types of machines. Each type of machine has unique hardware capabilities, requirements and control systems. For instance, the machine may have a vertical or a horizontal spindle; it can cut while moving several axes simultaneously, etc. The controller accepts a tool path file and directs tool motion and other machine activity (such as turning the coolant or air on and off).

Naturally, as each type of machine has unique hardware characteristics; controllers also differ in software characteristics. For instance, most controllers require that the instruction for turning the coolant on be given in a particular code. Some controllers also restrict the number of M codes that are allowed in one line of output. This information is not in the initial NX tool path. Therefore, the tool path must be modified to suit the unique parameters of each different machine/controller combination. The modification is called Post Processing. (5)

5 Vertical Machining Center (Vc-102e, Victor-Taichung)

Vertical machining centers are designed mainly for machining as the XY motions at a given depth. This is called 2-1/2 axis machining. Most of these machines also allow simultaneous XYZ motions. This feature is called 3-axis machining. Because of their design, cutting on sides is limited, unless special attachments are used. (6)



Figure 13 Vcenter-102e CNC machine at Darkhangeomach

5.1 Safety procedures

To operate the machine, safety regulations of the company are followed. For the prevention of an accident, whether man-made or an occasional disaster to both the machine and the people around the machine, some precautions have to be taken. Even though there are devices and warning signs on the machine some accidents happen due to a neglect of the machine and the machine being incorrectly operated. In this particular case, loads of metal chips are produced increasing the hazard around the workplace. Thus, the following steps are followed during a production of a workpiece:

- Wearing proper work clothes. Avoid long sleeves or overhanging parts
- Safety glasses, safety shoes, and a hard hat are some safety features used:
 - Safety glasses: preventing the eyes from getting injured by chips
 - Safety shoes: for the prevention of slipping, and for the protection of the feet.
 - Hard hat: for the prevention of hair being pulled into the machine and for protecting the head.

Some important notations operating CNC machine:

- The door must be closed before starting the machine
- Do not touch the chips or the tip of the tools by hand
- Do not clean off the chips or adjust the position of the coolant during the operation of the machine
- The operation of the machine must be stopped when the work is taken off the chuck by the foot switch.
- The way for stopping the machine: Use the following mentioned buttons or switches to stop the machines in an emergency situation. The function and position of these buttons or switches should be kept in mind.

5.2 Machine codes for Vc-102e

Few machine codes, such as M-code, G-code, Tool function, etc. play an important role in programming the machine. Thus, to make it easier to read and understand, some commands and processes are explained.

5.2.1 Tool function

Tool functions are used to select tools and when it is inputted, the magazine automatically turns taking the shorter route to position the specified tool. It uses an address T and 4-digit number.

5.2.2 (Miscellaneous) M functions

The miscellaneous function command consists of the address M followed by two digits. Miscellaneous functions are similar to preparatory functions in that they are encoded instructions which cause machine features to be activated, such as turning coolant on or causing a tool change to occur. The main difference between preparatory and miscellaneous functions is that the event or purpose for which the preparatory functions are activated occurs before the start of the movements in the block in which they are programmed, whereas the event for which the miscellaneous functions are activated has to occur at the end of the operation (block) in which they are programmed. (8)

M00 (Temporary program stop) – When it is read by machine's control unit, it sets the spindle to stop rotating, a coolant-supply, M03, M04, and M08 commands to be canceled. However, G, F and S codes are not affected by this. Also, there must be no other commands set in one line. It's a single block command.

M01 (Optional stop) - It has a similar function to M00. But, depending on the operator it can be selected whether the machine is stopped or not. Also, it is a single block command.

M02 (Program End) – It resets the control unit causing a program stop. Commanded by a single block.

M30 (Reset) – It is used at the end of the program stopping it completely. When the operation is stopped, it calls the beginning of the memory allowing it to operate using endless tape.

M03, M04, M05 (Spindle start/stop) – M03 and M04 starts spindle to rotate in normal and reverse direction respectively. They can be canceled by M00, M01, M02, M03/M04, M05, M06, M19, M30 commands.

M06 (Tool change) – Before executing an M06 command, T code must be written in order to call the tool. They can be written at the same command block. However, other commands are not allowed with the same block with M06. There are some conditions to be satisfied before executing this command.

1. Tool clamping system should be under tool clamping situation.
2. Y and Z axes must be located at the second reference position. The mode is to be MDI and then execute G91 G30 Y0. Z0.
3. Tool number called must be different with the tool number on the spindle.

Flowchart of a tool changing process is shown in Fig14.

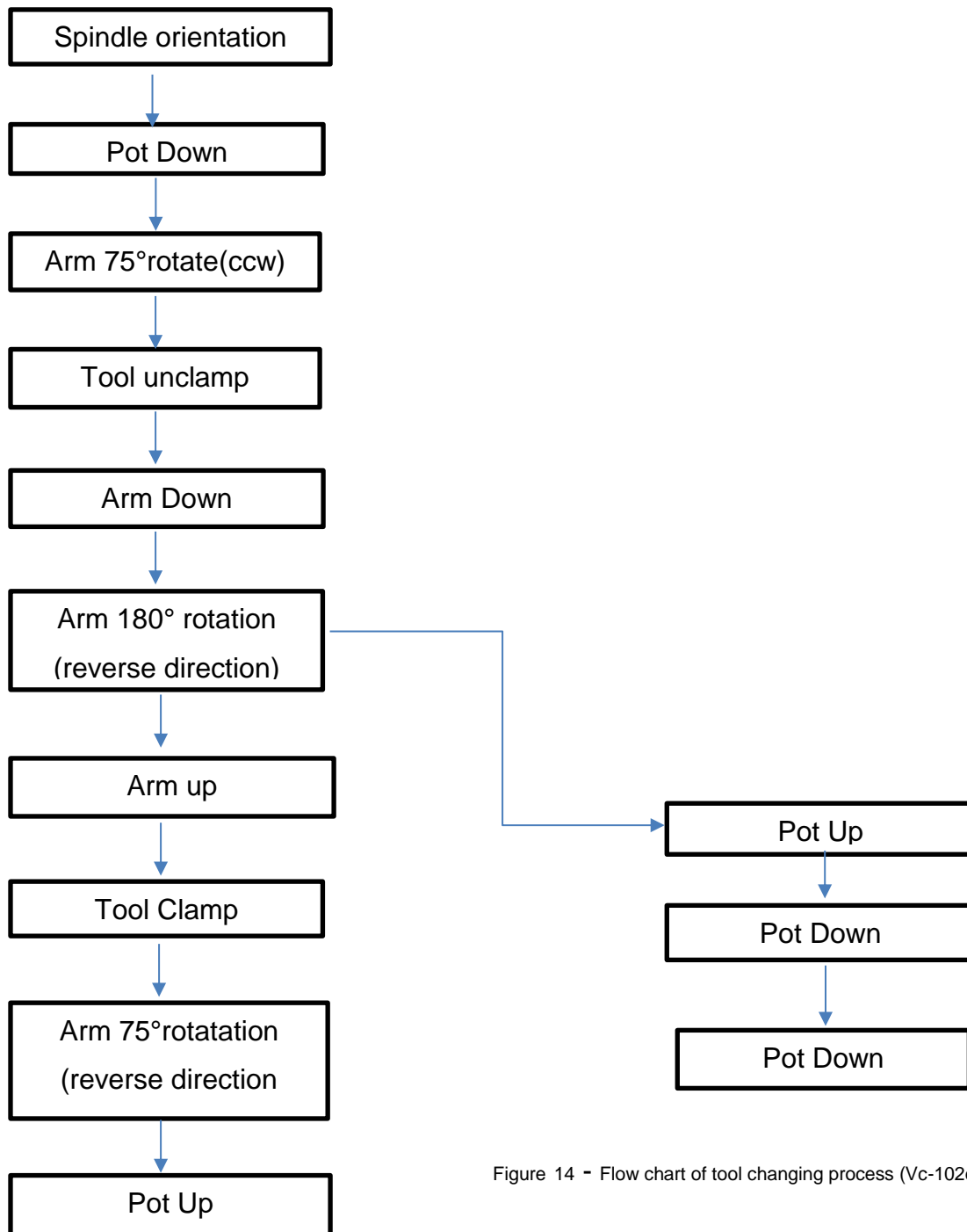


Figure 14 - Flow chart of tool changing process (Vc-102e)

M08, M09 (Coolant discharge stop) – M08 command discharges coolant from ports positioned around the spindle. M09 command stops coolant. If coolant off button is pressed on the control panel during an operation, coolant will not be discharged when the machine reads M08 command. Thus, coolant off button should be checked before operating.

5.3 (Preparatory) G functions

G-codes are the preparatory functions that identify the type of activities the machine will execute. A program block may contain one or more G-codes. The letter address G and specific numerical codes allow communication between the controller and the machine tool. This combination of letters and numerical values is commonly called G-Code. In order to perform a specific machining operation, a G-Code must be used. There are two types of G-Codes: modal and non-modal. Modal commands remain in effect, in multiple blocks, until they are changed by another command from the same group. Non-modal commands are in effect only for the block in which they are stated.(10)

G00 (Positioning) – This command moves X/Y/Z axes in rapid rate to the specified position following absolute or incremental command.

G01 (Linear Positioning) – It is used for straight cutting. The tool moves in a straight line from the current position to the command position at the rate specified by the feed function.

G02, G03 (Circular Interpolation) – The tool moves along a circular passage according to the commanded plane, cutting direction, coordinate system, circular arc size, and rapid rate.

6 Post-processor building for 3-axis Vertical Machining Center

6.1 Settings of G and M codes

Settings of G&M codes are in sub-tabs of programming tab. Depending on a model of CNC machine, some parameters in these tabs need some customizations in order to operate properly. Without an instruction manual written for the specific machine, it would be much more difficult to understand its programming. The manual includes G&M code lists, which are shown in the appendix, designed for the machine. By understanding its meaning and replacing it with a same/similar code given by post builder, a customized code can be developed.

6.2 Programming

A CNC program can be written in so many different ways that make it impossible to find any degree of consistency between individual part programmers, even within the same company. Operators can greatly influence any programming style, based on their experience of using many programs to machine various parts. Operators can also communicate their preferences to the CNC programmer, who should use such information in the best way to improve program structure.(8)

The structure of the programming needs to be considered first. A well written program will increase the quality of the product, save time, and be easier to read for the operator. In general, the code has 5 main steps show below:

- Program start sequence
- Operation start sequence
- Toolpath
- Operation end sequence
- Program end sequence

The most programming modification occurs in Program and Tool path tab. Modifications are mainly done either adding/removing/replacing command blocks and custom command, and changing its word sequencing.

6.2.1 Program start sequence

Program start sequence or rather called initial blocks, is a set of preparatory commands (G-codes) are the one that starts the code blocks, show in in Fig15. These G-codes are usually included to cancel the commands that may be in effect or to initiate the program mode. Program start and end sequences are used once at the beginning and the end.

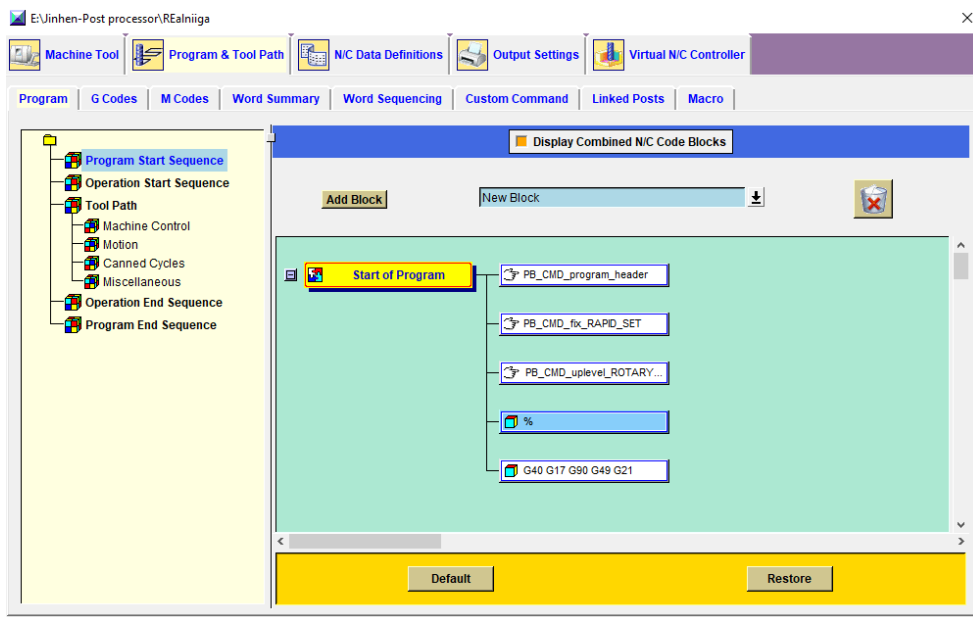


Figure 15 – Program start sequence on NX post builder.

G40 G17 G90 G49 G21 block is shown in Fig15. This block is a **program start sequence** or safety blocks. G40 and G49 are called cancellation commands. G49 cancels tool compensation that might be left from the previous operation. G17 selects an X-Y plane. G21 sets a unit of parameters in mm.

6.2.2 Operation start sequence

Operation start sequence, tool path, and operation end sequence are repeated for each operation cycle. It starts with a custom command to introduce the name of the operation and it is placed between MOM_set_seq_on and MOM_set_seq_off commands to place a line number only at the operation name. It eliminates all other line numbers making the code a bit easier to read. G30 is used instead of default G28 to prevent collision between tool changing system and a powertrain.

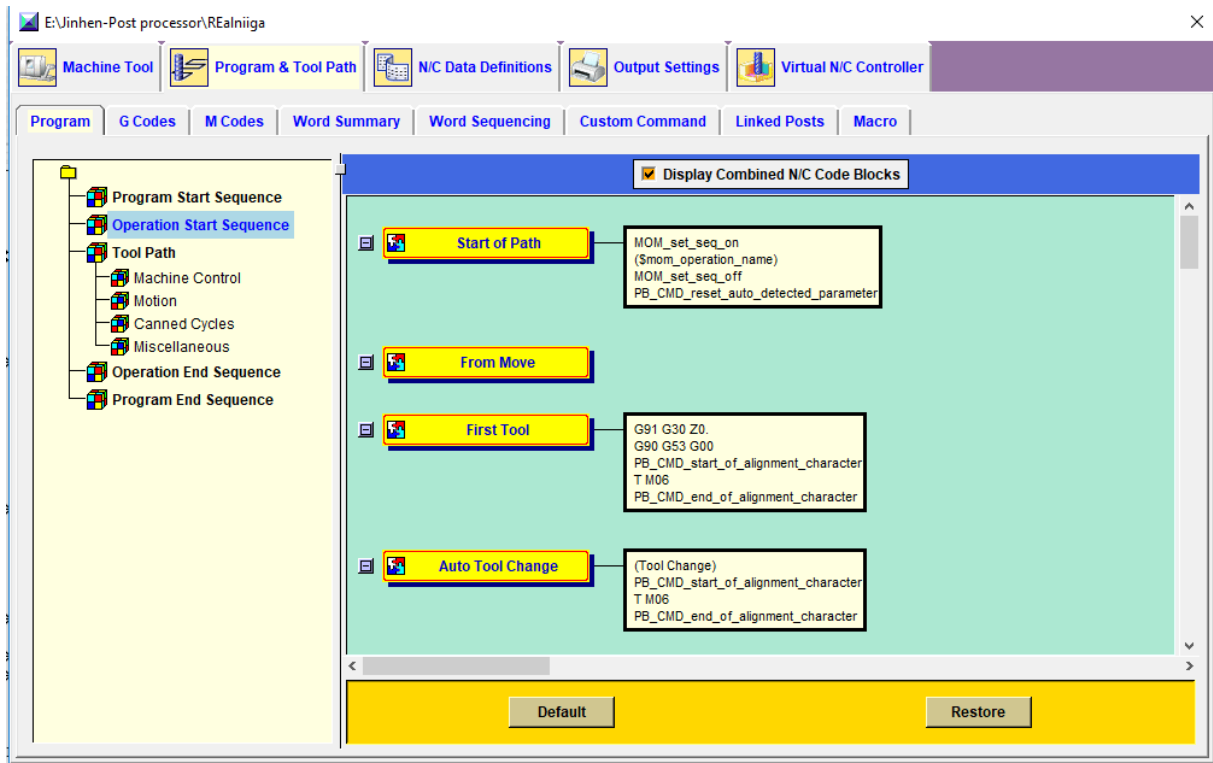


Figure 16 – Operation start sequence on NX post builder

6.2.3 Toolpath settings

The least changes are made in this part. Only the circular record is changed to quadrant shown in Fig 17.

XY plane is the primary work plane for machining on a vertical machining center. Quadrant is one fourth of a 2-dimensional grid in a plane of the rectangular coordinate system for measurement. It also represents an arc of 90° that is one fourth of a circle. Quadrant 1 is located in the upper right corner and quadrants 2 through 4 proceed in the counterclockwise direction.(2) (10)

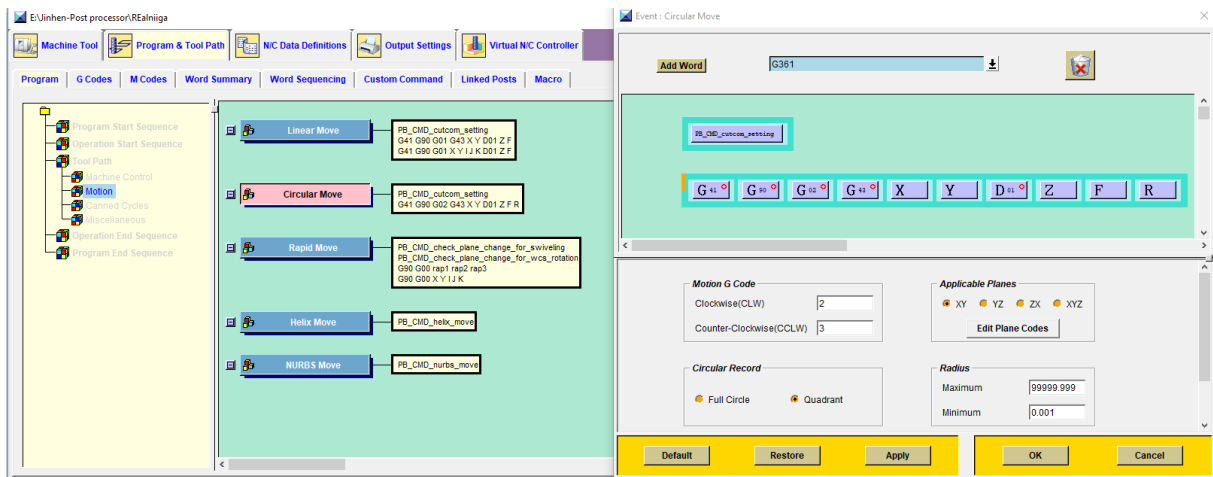


Figure 17 – Tool path on NX post builder

6.2.4 Operation End Sequence

This sequence consists of 4 blocks. M09 to stop the coolant, G91 G30 Z0.- moving to the reference point for safe tool changing, M05 and M01 to stop spindle and option. It is repeated after each operation start sequence.

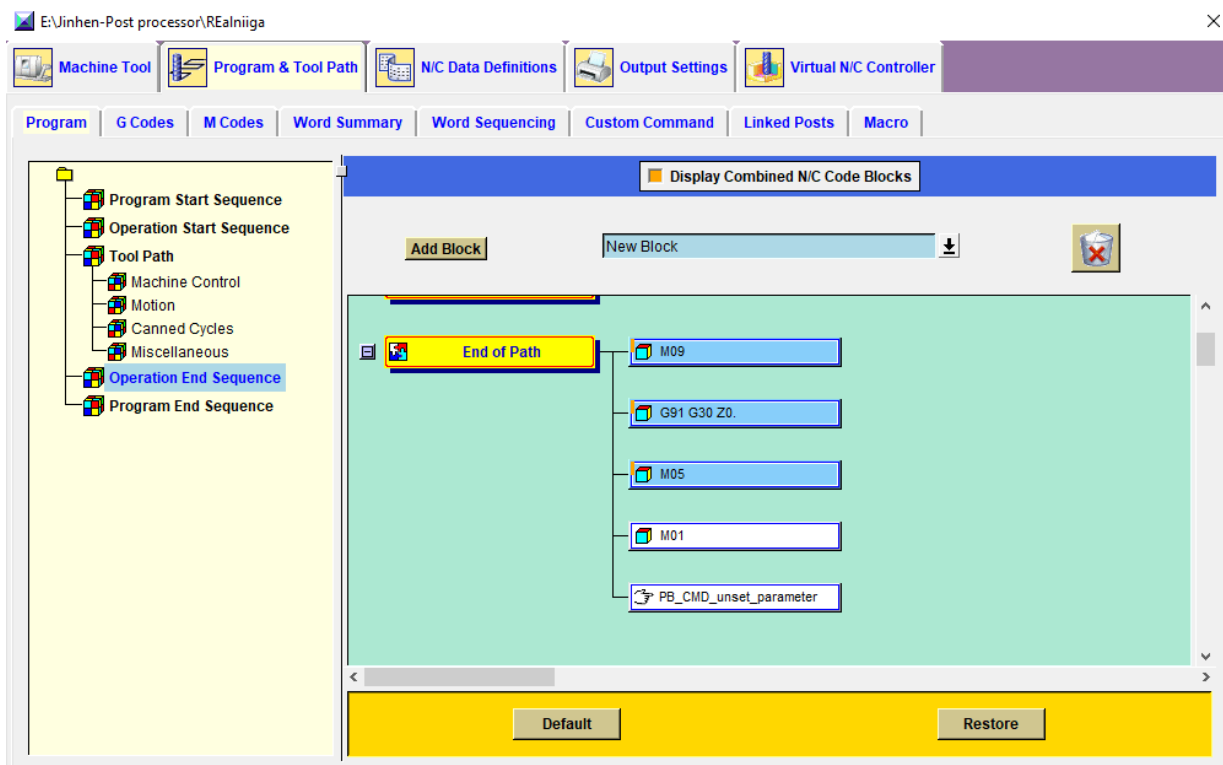


Figure 18 – Operation end sequence on NX post builder

6.2.5 Program End Sequence

Here the machine moves to the initial position and with M30 block it ends the program.

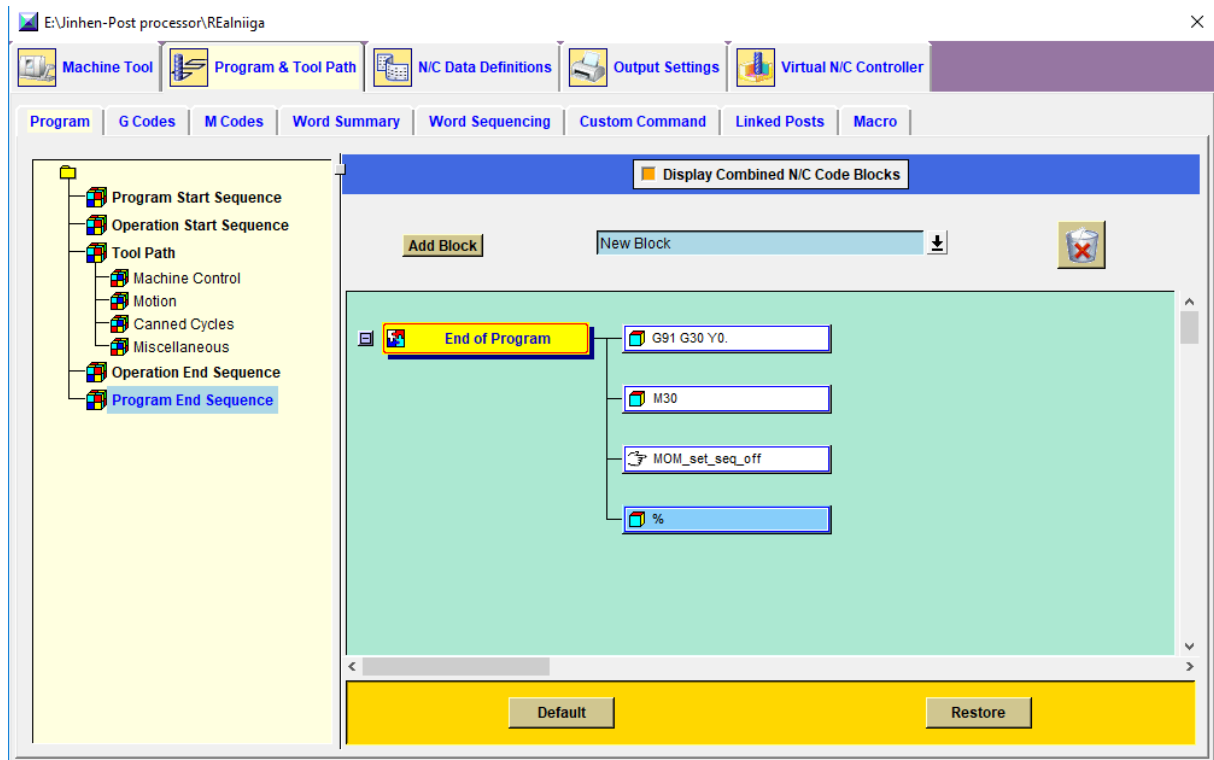


Figure 19 – Program end sequence on NX post builder

7 Results

7.1 Operations on NX 10

Different methods are used for removing various geometries such as cavity milling for complex geometry in the middle of the part, drilling for through holes, hole milling for blinded holes, and floor-wall to remove top blank. In Fig 20, the whole operation is shown.

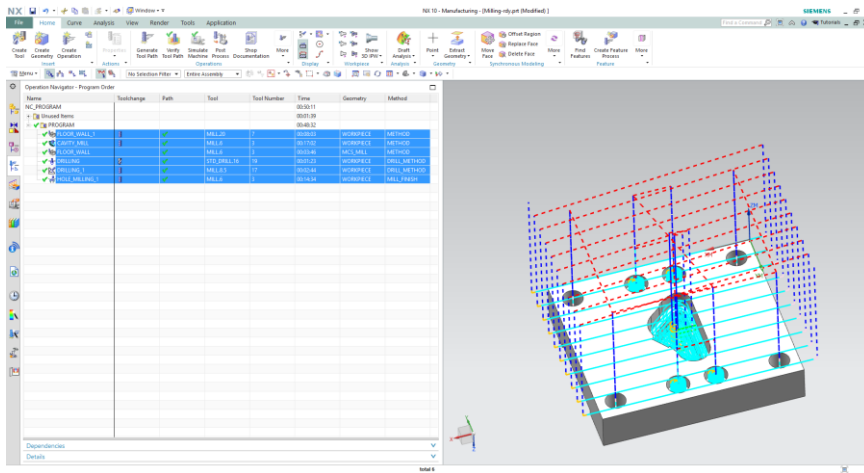


Figure 20 – Operations made on the workpiece.

7.2 An output of G-codes from NX post builder

```
O7777
%
G40 G17 G90 G49 G21
N1 (FLOOR_WALL_1)
G91 G30 Z0.0
G91 G30 Y0.0
G90 G53 G00
T7 M06.
G94 G90 G54
G43 H07 Z103.0
S796 M03
M08
-----> Tool path code #1
M09
G91 G30 Z0.0
M05
M01
N2 (CAVITY_MILL)
(Tool Change)
T3 M06
G90 G54
G00 G43 H03 Z103.0
S1989 M03
```

```

M08
-----> Tool path code #2

M09
G91 G30 Z0.0
M05
M01
N3 (FLOOR_WALL)
G90 G54
M08
-----> Tool path code #3

M09
G91 G30 Z0.0
M05
M01
N4 (DRILLING)
  (Tool Change)
T19 M06
G90 G54
G00 G43 H19 Z100.0
S995 M03
M08
-----> Tool path code #4

M09
G91 G30 Z0.0
M05
M01
N5 (DRILLING_1)
  (Tool Change)
T17 M06
G90 G54
G00 G43 H17 Z100.
S1872 M03
M08
-----> Tool path code #5

M09
G91 G30 Z0.0
M05
M01
N6 (HOLE_MILLING_1)
  (Tool Change)
T3 M06
G90 G54
G00 G43 H03 Z100.0
S1592 M03
M08
-----> Tool path code #6

M09
G91 G30 Z0.0
M05
M01
G91 G30 Y0.0
M30
%
```




Figure 22 – half finished part on Vc-102e

8 Discussion

During the work process, several difficulties are faced mainly due to lack of pre-considerations. Before starting the actual work, limitations of the machine and availabilities of blank and tools should be considered. Two aligned surfaces of the part shown in Fig 10 are not possible to be operated by the CNC machine because of its axis limitation. This limitation caused me to re-consider the operation plan. Also, because of limited tool selections, 4 holes with 8 diameters are drilled with 8.5 diameter tool.

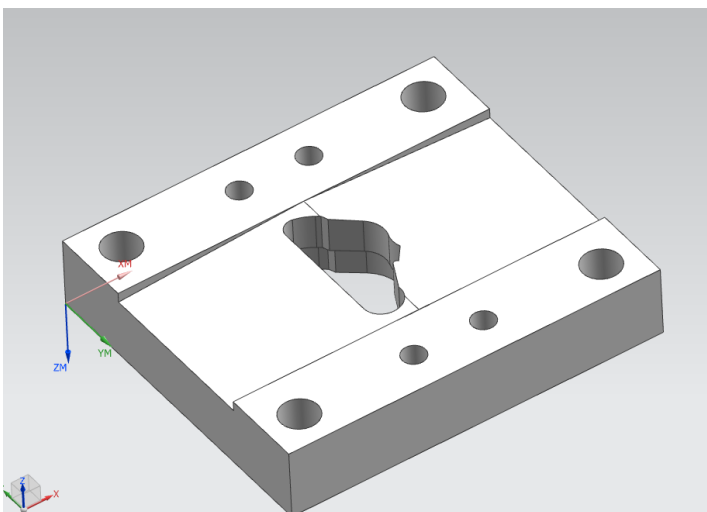


Figure 10 – Modelled part on NX 10

Furthermore, to test the G-code it would be better to use foam, plastic, or other soft blanks that will cause less damage to the machine when the machine code is not developed properly. During the test of first post-processed machine code, it had a custom command included a G49 command block which cancels the tool compensation at the operation end. And when tool returned to the reference point, it collided with the workpiece causing a damage to the tool. On the other hand, proof-reading every operation line is a must.

9 Recommendation for future work

As a part of the thesis, modified machine code/G-code is developed for the Vc-102e machine with Fanuc Series 0i-MF controller using NX post builder software. Developing a post-builder that the is able to read was the aim of this thesis. Thus, some important aspects of economically manufacturing were not considered such as cutting parameters, feed and speed rates, an optimum type for each operations resulting a reduction in production time. Also, more custom commands can be included making the G-code easier to read by the operator.

References

1. MAS.863/4.140J-P7 [Internet]. [cited 2018 Aug 16]. Available from: <http://fab.cba.mit.edu/classes/863.12/people/laia.mogassoldevila/projects/p7.html>
2. Fundamentals of Modern Manufacturing Materials Processes and Systems Fourth Edition Mikell P Groover.pdf.
3. Fundamentals_of_CNC_Machining.by.Autodeskpdf.
4. Joy A. Design of a Computer Numeric Control System with Open Software Tools. Int J Adv Res Electr Electron Instrum Energy. 1970 Jan 1;3(2):7439–44.
5. Introduction to Computer Aided Manufacturing.pdf.
6. Altes_Francesc.pdf [Internet]. [cited 2018 Jul 31]. Available from: https://www.theseus.fi/bitstream/handle/10024/131919/Altes_Francesc.pdf?sequence=1&isAllowed=y

7. nx10um1.pdf [Internet]. [cited 2018 Aug 9]. Available from: <https://www.star-m.jp/eng/service/usermanual/nx10um1.pdf>
8. Smid P. CNC Control Setup for Milling and Turning: :313.
9. Computer Numerical Control of Machine Tools Second edition By G. E. Thyer.pdf.
10. Evans K. Programming of CNC machines. Fourth edition. South Norwalk, Connecticut: Industrial Press, Inc; 2016. 473 p.
11. Instruction Manual(ENFKWA1100E) - Vc-102E by Victor Taichung Machinery Works Co. Ltd.
12. VCENTER-102E Vertical Machining Center (VMC) from Victor Taichung CNC Machine Tool Builder [Internet]. [cited 2018 Aug 16]. Available from: <http://www.victortaichung.com/machine-tools/en/Vcenter-102E.html#parentHorizontalTab2>
13. Fundamentals of Modern Manufacturing Materials Processes and Systems Fourth Edition Mikell P Groover.pdf.
14. Manufacturing Engineering And Technology Seventh Edition By Steven R. Schmid and Serope Kalpakjian.pdf.
15. nx10.user.manual.pdf.
16. Khurmi RS, Gupta JK. Theory of machines: [a textbook for the students of B.E. / B. Tech., U.P.S.C. (Engg. Services) ; Section "B" of A.M.I.E. (I). Ram Nagar, New Delhi: Eurasia Pub. House; 2008.
17. Ghazanfari A. NX 10 for Engineering Design. :207.
18. CNC Programming Handbook 2nd Edition by Peter Smid.pdf.
19. Youssef HA, El-Hofy H. Machining technology: machine tools and operations. Boca Raton: CRC Press; 2008. 633 p.
20. G Codes and M Codes for 3D printing.pdf.
21. SINUMERIK 840D sl G code programming.pdf.
22. Computer Numerical Control of Machine Tools Second edition By G. E. Thyer.pdf.

23. SINUMERIK and SINAMICS Equipment for Machine Tools.pdf.
24. Smid P. CNC Control Setup for Milling and Turning: :313.
25. Sinha SK. CNC programming using Fanuc custom B macro. New York: McGraw-Hill; 2010. 274 p.
26. Control System for CNC Cutting Machine by Bc. Filip Mešťánek.
27. Rothbart HA, editor. Cam design handbook. New York: McGraw-Hill; 2004. 606 p.
28. Limited D. G and M Programming for CNC Milling Machines. :98.
29. Smid P. FANUC CNC custom Macros. N Y. 2005;336.
30. Adivarekar MS. Development of a postprocessor for a multi-axis CNC milling center. :58.
31. Rabe H. The Design of a CNC Mill for Product Prototyping. :135.
32. Evans K. Programming of CNC machines. Fourth edition. South Norwalk, Connecticut: Industrial Press, Inc; 2016. 473 p.
33. Igunnuoda_Olorunmuyiwa.pdf.
34. Dallali M. Concept of CNC Machine Centers Bachelor thesis. :45.

Appendices

G-code list for Fanuc controller

G-code	Group	Function
G00 ☆	01	Positioning (Rapid traverse)
G01 ☆		Linear interpolation (Cutting feed)
G02		Circular interpolation / Helical cutting CW
G03		Circular interpolation / Helical cutting CCW
G04	00	Dwell, exact stop
G09		Exact stop

G10		Data setting
G10.6		Tool retract and recover
G11		Data setting mode cancel
G15☆	17	Polar coordinates command canceled
G16		Polar coordinates command
G17☆	02	XY plane selection
G18☆		ZX plane selection
G19☆		YZ plane selection
G20	06	Input in inch
G21		Input in mm
G22☆	04	Stored stroke check function on
G23		Stored stroke check function off
G27	00	Reference point return check
G28		Reference to reference point
G29		Return from the reference point
G30		2 nd , 3 rd , 4 th reference point return
G30.1		Floating reference point return
G31		Skip function
G33	01	Thread cutting
G37	00	Automatic tool length measurement
G39		Corner offset circular interpolation
G40☆	07	Cutter compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
G40.1 (G150) ☆	19	Normal direction control cancel mode
G41.1 (G151)		Normal direction control left side on
G42.1 (G152)		Normal direction control right side on
G43	08	Tool length compensation + direction
G44		Tool length compensation-direction
G45	00	Tool offset increase
G46		Tool offset decrease
G47		Tool offset a double increase

G48		Tool offset a double decrease
G49	08	Tool length compensation cancel
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancels
G51.1		Programmable mirror image
G52	00	Local coordinate system setting
G53		Machine coordinate system selection
G54	14	Work coordinate system 1 selection
G54.1		Additional workpiece coordinate sys. Select
G55		Work coordinate system 2 selection
G56		Work coordinate system 3 selection
G57		Work coordinate system 4 selection
G58		Work coordinate system 5 selection
G59		Work coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	0	Macro command, Macro call
G66	12	Custom macro modal call
G67		Custom macro modal call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74		Counter tapping cycle
G76		Fine boring
G80		Canned cycle cancel
G81		Drilling cycle, spot boring
G82		Drilling cycle, counterboring
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle

G86		Boring cycle
G87	09	Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute command
G91		Incremental command
G92	00	Programming of absolute zero point
G92.1		Workpiece coordinate system preset
G94	05	Feed per minute
G95		Feed per rotation
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Canned cycle-аар анхны цэг дээрээ очих
G99		Canned cycle-аар R цэг дээр буцаж очих

Table 2

M-code list for Fanuc controller

M code	Functions	M code	Functions
M00	Program stop	M50	
M01	Option stop	M51	
M02	Program end	M53	
M03	Spindle forward	M54	
M04	Spindle reverse	M56	
M05	Spindle stop	M57	

M06	Auto tool change (Call macro O9029)	M58	
M07	Oil mist ON	M59	Air blow OFF
M08	Coolant ON	M60	
M09	Coolant OFF / Oil mist OFF	M61	
M10	4 th axis clamp	M62	
M11	4 th axis unclamp	M70	DNC ON
M13	Spindle CW and Coolant ON	M71	DNC OFF
M14	Spindle CCW and Coolant ON	M73	Y-axis mirror image OFF
M15	Pot up	M74	Y-axis mirror image ON
M16	Pot down	M75	X-axis mirror image OFF
M19	Spindle orientation ON	M76	X-axis mirror image ON
M20	Spindle orientation OFF	M80	Tool NO. Initial prepare
M21	Extra M code or the fixture clamping	M81	Tool Initial start
M22	Extra M code or the fixture unclamping	M82	
M23	Extra M code or the fixture clamping and tilting up	M83	Workpiece counter ON
M24	Extra M code or the fixture clamping and tilting down	M85	
M25		M89	
M26		M90	
M27	Workpiece measurement select (Spindle Protect OFF)	M91	
M28	Tool length measurement select (Spindle protect ON)	M92	
M29	Rigid Tapping	M93	
M30	Program end	M98	Cal subprogram
M31	Extra M code	M99	End subprogram
M32	Extra M code		
M33			
M34			
M40	Screw Conveyor ON		
M41	Screw Conveyor OFF		
M42	Chips conveyor ON		

M43	Chips conveyor OFF		
M45			
M48	Override cancel ON		
M49	Override cancel OFF		

Post-processed G-code by NX 10, using 3axis

```

%
N0010 G40 G17 G90 G70
N0020 G91 G28 Z0.0
N0030 T07 M06
N0040 T03
N0050 G00 G90 X6.811 Y.0001 S796 M03
N0060 G43 Z4.0551 H07
N0070 Z.1181
N0080 G01 Z0.0 F9.8 M08
-----
N0860 G91 G28 Z0.0
N0870 T03 M06
N0880 T19

```

```

N0890 G00 G90 X3.1874 Y2.5026 S1989 M03
N0900 G43 Z4.0551 H03
-----
N4190 G00 X3.0381 Y2.5347 S2653 M03
-----
N5820 G91 G28 Z0.0
N5830 T19 M06
N5840 T17
N5850 G00 G90 X.5906 Y.5906 S995 M03
N5860 G43 Z3.937 H19
N5870 G81 X.5906 Y.5906 Z-1.4688 R.0984 F9.8
N5880 Y4.9213
N5890 X5.7087
N5900 Y.5906
N5910 G80
N5920 G91 G28 Z0.0
N5930 T17 M06
N5940 T03
N5950 G00 G90 X2.5591 Y4.8819 S1872 M03
N5960 G43 Z1.5748 H17
-----
N6060 G91 G28 Z0.0
N6070 T03 M06
N6080 T07
N6090 G00 G90 X2.5591 Y.6299 S1592 M03
N6100 G43 Z3.937 H03
-----
N9740 G00 Z3.937
N9750 M02
%
```

Meaning of characters in machine code

ADDRESS CHARACTERS	
CHARACTER	MEANING
A	Additional rotary axis parallel and around the X-axis
B	Additional rotary axis parallel and around the Y-axis
C	Additional rotary axis parallel and around the Z-axis
D	Tool radius offset number; (Turning) Depth of cut for multiple repetitive cycles
E	User Macro Character
F	Feed rate; (Turning) Precise designation of thread lead
G	Preparatory Function
H	Tool Length Offset number
I	Incremental X coordinate of circle center; (Turning) parameter of fixed cycle
J	Incremental Y coordinate of circle center
K	Incremental Z coordinate of circle center; (Turning) parameter of fixed cycle
L	Number of repetitions (subprogram, hole pattern); Fixed offset group number
M	Miscellaneous Function
N	Sequence or block number
O	Program number
P	Dwell time, program number, and sequence number designation in subprogram; (Turning) Sequence number start for multiple repetitive cycles
Q	Depth of cut, shift of canned cycles; (Turning) Sequence number end for multiple repetitive cycles
R	Point R for canned cycles, as a reference return value; Radius designation of a circle arc; Angular displacement value for coordinate system rotation
S	Spindle-Speed Function
T	Tool Function
U	Additional linear axis parallel to X-axis
V	Additional linear axis parallel to Y-axis
W	Additional linear axis parallel to Z-axis
X	X coordinate
Y	Y coordinate
Z	Z coordinate

Figure 21 – Meaning of all characters used in machind code (11)

Vcenter-102E Machine Specifications



Item / Model	Units	Vcenter-102E
Travel		
X axis travel	mm	1020
Y axis travel	mm	600
Z axis travel	mm	500
Distance		
Spindle center to column	mm	600
Spindle nose to table surface	mm	150 ~ 710
Table		
Table work area	mm	1100 x 510
Dimension of T-slot	mm	5 x 18 x 100
Max. table load	kg	700
Spindle		
Spindle taper		B7-60
Spindle motor - cont./30min	KW	11/15
Spindle speed	rpm	6000
Feed rate		
Rapid feed rate - X/Y/Z	m/min	24 / 24 / 20
Axis feed motor - X/Y/Z	KW	2.5 / 2.5 / 2.5
Cutting feedrate by table	m/min	7.5
X/Y ball screw (dia. x pitch)	mm	40 x P12
Z ball screw	m/min	40 x P10
Tools		
Max. tool length	mm	250
Max. tool weight	kg	7
Magazine capacity		24 (Round)
Max. tool diameter (without adjacent tools)	mm	80
Tool exchange time	sec.	3.5
Full stud angle	deg.	60 (opt. 45)
Tool selection method		Random
Machine		
Power requirement	kVA	30
Min/Max. air pressure	kg/cm ²	5.5 ~ 8.5
Coolant tank capacity	L	200
Std. NC controller		FANUC DiMate-MD
Floor space requirement	mm	2750 x 2485
Max. machine height	mm	2640
Net weight	kg	6100

*Machine and controller specifications are subject to change without notice.

Standard Accessories

- Fully enclosed splash guard
- T nuts for table slot
- Hand tools and toolbox
- Rigid tapping
- High pressure coolant (5 bars/50Hz, 7 bars/60Hz)
- Remote MPG
- 3-step warning light
- Leveling blocks
- Fanuc manuals by CD-ROM

Optional Accessories

- Dynamic graphic display (including Manual guide D)
- Spindle oil cooler
- Air conditioner for electric cabinet
- Chip conveyor with cart
- Air gun
- Coolant gun
- Oil skimmer
- Oil mist collector
- Fanuc manuals
- CE marked components

Machine Layout

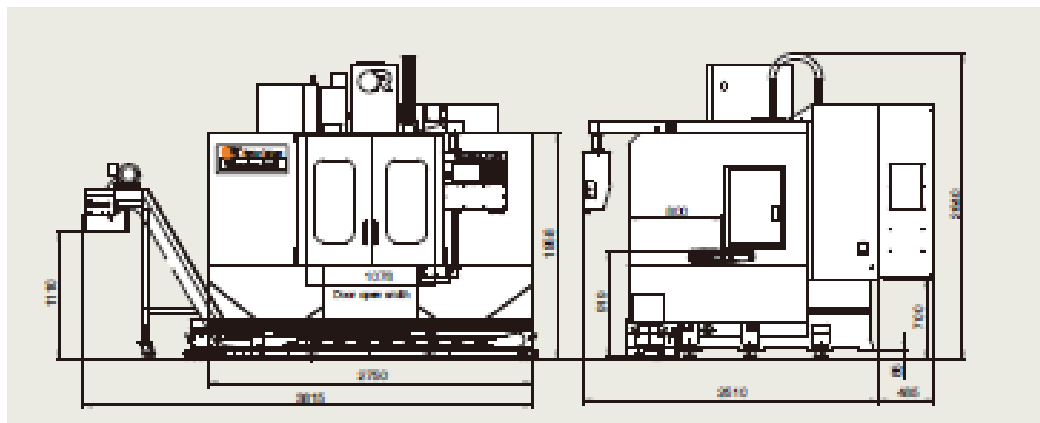


Figure 22 – Specifications of Vcenter-102E machine (12)

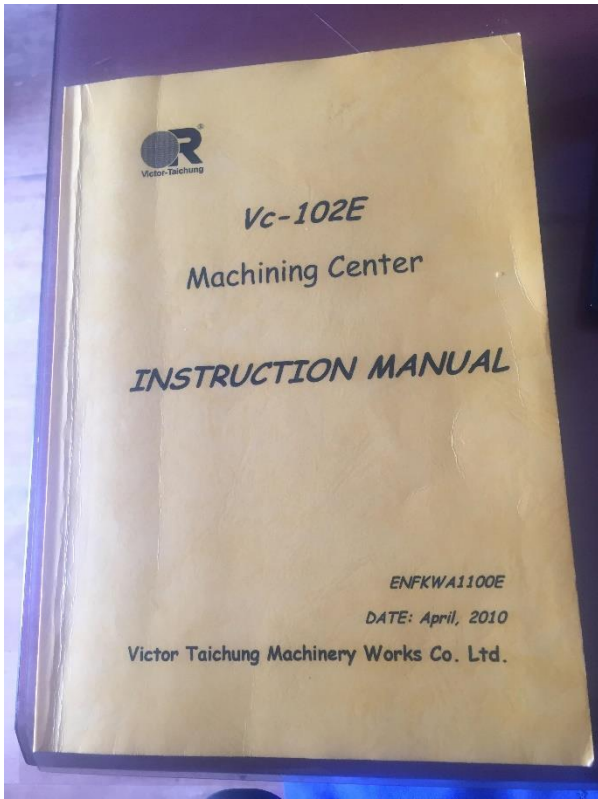


Figure 23 – Instruction manual for Vc-102e machining center



Figure 24 – Blank



Figure 2523 – Corner drilling operation



Figure 26 – Control panel of Vc-120e machine