Competency Based Learning Material (CBLM)

CNC Machining Centre Operation with CAD CAM
Level-4

Module: Creating and Using G&M Code for Machining

Code: CBLM-OU-LE-CNCCDM-03-L4-V1

National Skills Development Authority
Prime Minister's Office
Government of the People's Republic of Bangladesh

Copyright

National Skills Development Authority
Prime Minister's Office
Level: 10-11, Biniyog Bhaban,
E-6 / B, Agargaon, Sher-E-Bangla Nagar Dhaka-1207, Bangladesh.
Email: ec@nsda.gov.bd
Website: www.nsda.gov.bd.
National Skills Portal: http:\\skillsportal.gov.bd

This Competency Based Learning Materials (CBLM) on "**Creating and Using G&M Code for Machining**" under the CNC Maching Centre Operation with CAD CAM, Level-4 qualification is developed based on the national competency standard approved by National Skills Development Authority (NSDA)

This document is to be used as a key reference point by the competency-based learning materials developers, teachers/trainers/assessors as a base on which to build instructional activities.

This Competency Based Learning Materials is a document for the development of curricula, teaching and learning materials, and assessment tools. It also serves as the document for providing training consistent with the requirements of industry in order to meet the qualification of individuals who graduated through the established standard via competency-based assessment for a relevant job.

This document has been developed by NSDA in association with industry representatives, academia, related specialist, trainer, and related employee.

Public and private institutions may use the information contained in this CBLM for activities benefitting Bangladesh.

Approved by the Authority meeting held on.................

How to use this Competency Based Learning Material (CBLM)

The module, Creating and Using G&M Code for Machining contains training materials and activities for you to complete. These activities may be completed as part of structured classroom activities or you may be required you to work at your own pace. These activities will ask you to complete associated learning and practice activities in order to gain knowledge and skills you need to achieve the learning outcomes.

1. Review the **Learning Activity** page to understand the sequence of learning activities you will undergo. This page will serve as your road map towards the achievement of competence.

2. Read the **Information Sheets.** This will give you an understanding of the jobs or tasks you are going to learn how to do. Once you have finished reading the **Information Sheets** complete the questions in the **Self-Check.**

3. **Self-**Checks are found after each **Information Sheet**. **Self-Checks** are designed to help you know how you are progressing. If you are unable to answer the questions in the **Self-Check** you will need to re-read the relevant **Information Sheet**. Once you have completed all the questions check your answers by reading the relevant **Answer Keys** found at the end of this module.

4. Next move on to the **Job Sheets. Job Sheets** provide detailed information about *how to do the job* you are being trained in. Some **Job Sheets** will also have a series of **Activity Sheets**. These sheets have been designed to introduce you to the job step by step. This is where you will apply the new knowledge you gained by reading the Information Sheets. This is your opportunity to practise the job. You may need to practise the job or activity several times before you become competent.

5. Specification **sheets**, specifying the details of the job to be performed will be provided where appropriate.

6. A review of competency is provided on the last page to help remind if all the required assessment criteria have been met. This record is for your own information and guidance and is not an official record of competency

When working though this Module always be aware of your safety and the safety of others in the training room. Should you require assistance or clarification please consult your trainer or facilitator.

When you have satisfactorily completed all the Jobs and/or Activities outlined in this module, an assessment event will be scheduled to assess if you have achieved competency in the specified learning outcomes. You will then be ready to move onto the next Unit of Competency or Module

Table of Contents

Module Content

Unit of Competency	**Create and Use G&M Code for Machining**
Unit Code	**OU-LE-CNCCDM-03-L4-V1**
Module Title	**Creating and Using G&M Code for Machining**
Module Descriptor	This module covers the skills, knowledge and attitudes required to create and use G&M code for machining. It specifically includes determining machine co-ordinate systems, creating program using G&M code, setting up machine and workpiece and performing 2D machining operation.
Nominal Hours	**60 Hours**
Learning Outcome	After completing the practice of the module, the trainees will be able to perform the following jobs: 1 Determine machine co-ordinate systems 2 Create program using 3 G & M code 4 Set up machine and workpiece 5 5. Perform 2D machining operation

Assessment Criteria

1 Reference position of a machining center is determined
2 Work coordinates are set
3 Appropriate hand tools are selected
4 Preparatory G & M code is created
5 Modal and non-modal codes are identified
6 Absolute and Incremental positioning code is created
7 Circular and linear interpolation are created
8 Cutter radius compensation is selected
9 Tools offset (G43) and work offset (G54) are set
10 Spindle commands and program stop commands are executed
11 Canned cycles are executed
12 Bolt hole codes are executed
13 Pocket milling and contour milling is programmed
14 Sub-program is executed
15 Work holding mechanism is selected
16 Required cutting tools are selected and loaded
17 Datum is defined
18 Entry and exit points are selected
19 Sequence of cutting operations is selected
20 2D operations are performed
21 Canned cycles structures are identified
22 Sub-program structures are identified
23 Pocket milling program structures are identified
24 Canned cycle tricks with subprogram are executed

Learning Outcome 1: Determine Machine Co-Ordinate Systems

Assessment Criteria	1. Reference position of a machining center is determined 2. Work coordinates are set
Conditions and Resources	1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Laptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary tools and equipment 11. Necessary PPE
Contents	1. Reference position of machining center 2. Work coordinates ▪ Machine coordinates ▪ WCS / Work Co-ordinates ▪ Machine reference
Job/Task/Activity	1. Determine the reference position of the machining center 2. Set the work coordinates
Training Method	1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming 9. Role Play
Assessment Method	1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 1: Determine Machine Co-Ordinate Systems

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials '**Create and Use G&M Code for Machining**
2. Read the Information sheet and complete the Self Checks & Check answer sheets on "**Determine machine co-ordinate systems**"	2. Information sheet 1: Determine machine co-ordinate systems 3. Self-check 1: Determine machine co-ordinate systems 4. Check your answer with Answer key 1: Determine machine co-ordinate systems
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet Job Sheet 1.1: Determine and set the work coordinates as per specification Specification Sheet 1.1: Determine and set the work coordinates as per specification

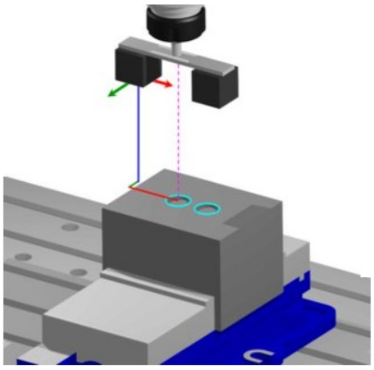
Information Sheet 1: Determine Machine Co-Ordinate Systems

Learning Objective: After completion of this information sheet, the learners will be able to explain, define and interpret the following contents:

1.1 Reference position of machining center
1.2 Work coordinates

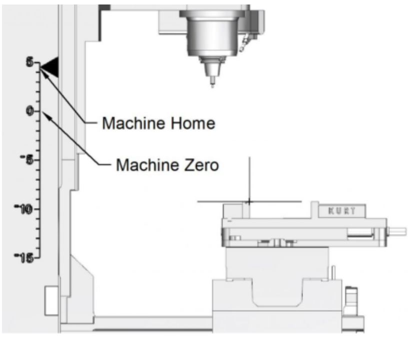
1.1 Reference Position of Machining Center

In CNC (Computer Numerical Control) milling machines, the reference position, often referred to as the machine home position or machine zero, is a crucial point that serves as a starting reference for all machining operations. It's essentially the point where the machine considers itself to be at its origin in terms of its coordinate system.

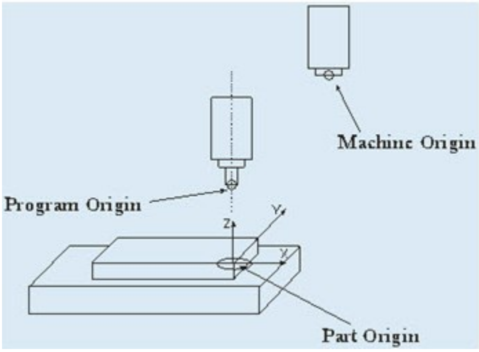


There are typically three main reference positions in CNC milling machines:

a. **Machine zero (absolute zero)**: This is the reference position where all the axes of the machine are at their absolute minimum positions, typically at the extreme end of each axis. It's also the reference point where all program coordinates are based upon. When the machine is powered on or reset, it moves to this position to establish its reference point.



b. **Workpiece zero (part zero)**: This is the reference position on the workpiece where the machining operations are to be performed. It's often set by the operator or programmer based on the design requirements and the desired location for machining operations.

a. 

c. Tool zero: This is the reference position for the cutting tool. It defines the position from which all tool movements are measured. It could be at the tip of the tool, the tool holder, or some other predefined position depending on the machine and the machining process.

d. `Tool offset in CNC milling

In CNC milling, the tool offset refers to the distance between the cutting edge of the tool and the spindle or tool holder. This offset is used to ensure that the tool cuts the workpiece at the correct location, compensating for the tool's geometry and ensuring accuracy in the machining process. Different types of tool offsets, such as length offsets and diameter offsets, are used to account for variations in tool dimensions and achieve precise machining results

To set these reference positions accurately, operators typically use various methods such as manual jogging, probing, or referencing tools. Once set, these reference positions are stored in the machine's memory and can be recalled for subsequent machining operations.

It's important to note that the exact process of setting these reference positions can vary depending on the specific CNC machine, its controller, and the preferences of the operator or programmer. Additionally, some machines may have additional reference positions or variations in terminology, but the concepts generally remain the same.

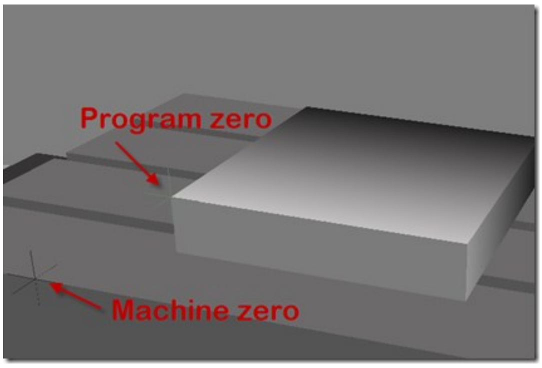
1.2 Work Coordinates

In CNC (Computer Numerical Control) machining, work coordinates and machine coordinates play essential roles in defining the positions and movements of tools and workpieces. Let us break down these terms:

a. Work coordinates system (WCS) or work coordinates (WC)

Work Coordinates refer to the coordinate system defined relative to the workpiece or part being machined. It represents the positions and movements of the cutting tool in relation to the workpiece. The origin (0,0,0) of the work coordinate system is typically set at a specific point on the workpiece, often referred to as the Workpiece Zero or Part Zero.

The work coordinates are used to program the toolpaths and machining operations. They are defined based on the design requirements of the part, allowing precise control over where cuts and features are machined.



b. Machine coordinates:

Machine Coordinates, also known as Cartesian Coordinates or Absolute Coordinates, are the coordinate system defined within the CNC machine itself. The origin (0,0,0) of the machine coordinate system is typically set at a specific reference point within the machine, often referred to as the Machine Zero or Absolute Zero.

Machine coordinates are used by the CNC machine's controller to interpret toolpath instructions and move the cutting tool to the desired positions on the workpiece. These coordinates are independent of any specific workpiece and are used to control the movements of the machine's axes.

c. Machine reference

The Machine Reference is essentially the starting point or reference position of the CNC machine's coordinate system. It's where the machine considers itself to be at its origin. This reference position is typically established during machine setup and calibration.

The Machine Reference can coincide with the Machine Zero point, but this isn't always the case. It's crucial to set this reference accurately to ensure consistent and precise machining operations.

In summary, Work Coordinates (WCS) are specific to the workpiece being machined, defining positions relative to the part, while Machine Coordinates represent the internal coordinate system of the CNC machine. The Machine Reference serves as the starting point for the machine's coordinate system, ensuring accurate and repeatable machining operations. Proper understanding and management of these coordinate systems are crucial for successful CNC machining.

Self-Check Sheet 1: Determine Machine Co-Ordinate Systems

1. What is Reference position of machining center?
   Answer:

2. What is Machine Zero (Absolute Zero) ?
   Answer:

3. What is Workpiece Zero (Part Zero) ?
   Answer:

4. What is Tool Zero?
   Answer:

5. What is the tool offset in CNC milling?
   Answer:

6. What is Work Coordinates System (WCS) or Work Coordinates (WC) ?
   Answer:

7. What is Machine Coordinates?
   Answer:

8. What is Machine Reference?
   Answer:

Answer Sheet 1: Determine Machine Co-Ordinate Systems

1. What is Reference position of machining center?
   Answer: In CNC (Computer Numerical Control) milling machines, the reference position, often referred to as the machine home position or machine zero, is a crucial point that serves as a starting reference for all machining operations.

2. What is Machine Zero (Absolute Zero)
   Answer: This is the reference position where all the axes of the machine are at their absolute minimum positions, typically at the extreme end of each axis.

3. What is Workpiece Zero (Part Zero):
   Answer: This is the reference position on the workpiece where the machining operations are to be performed.

4. What is Tool Zero:
   Answer: This is the reference position for the cutting tool. It defines the position from which all tool movements are measured. It could be at the tip of the tool, the tool holder, or some other predefined position depending on the machine and the machining process

5. What is the tool offset in CNC milling?
   Answer: In CNC milling, the tool offset refers to the distance between the cutting edge of the tool and the spindle or tool holder. This offset is used to ensure that the tool cuts the workpiece at the correct location, compensating for the tool's geometry and ensuring accuracy in the machining process.

6. What is Work Coordinates System (WCS) or Work Coordinates (WCS):
   Answer: Work Coordinates refer to the coordinate system defined relative to the workpiece or part being machined. It represents the positions and movements of the cutting tool in relation to the workpiece. The origin (0,0,0) of the work coordinate system is typically set at a specific point on the workpiece, often referred to as the Workpiece Zero or Part Zero.

7. What is Machine Coordinates:
   Answer: Machine Coordinates, also known as Cartesian Coordinates or Absolute Coordinates, are the coordinate system defined within the CNC machine itself. The origin (0,0,0) of the machine coordinate system is typically set at a specific reference point within the machine, often referred to as the Machine Zero or Absolute Zero.

8. What is Machine Reference:
   Answer: The Machine Reference is essentially the starting point or reference position of the CNC machine's coordinate system. It's where the machine considers itself to be at its origin. This reference position is typically established during machine setup and calibration.

Job Sheet 1.1: Determine and Set the Work Coordinates as Per Specification

Working Procedure

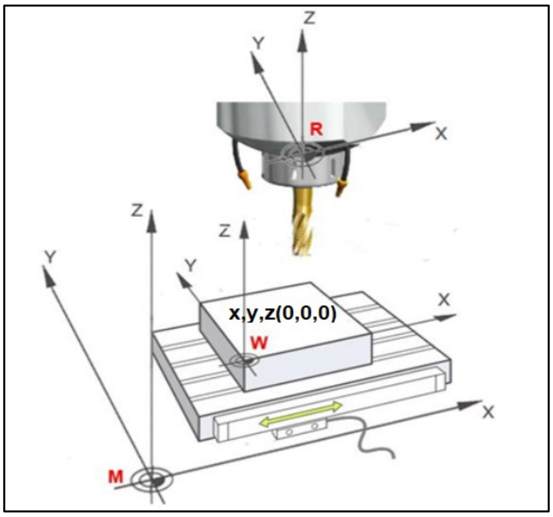
1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. Prepare the Workpiece is properly align
4. Establish a reference position for the machine's axes.
5. Move the machine's spindle to the home position (typically the machine's origin) using the machine's control panel or jog functions.
6. Determine which work coordinate system (G54, G55, etc.) you will use for the machining operations.
7. Use the machine's jog controls to move the spindle to a known reference point on the workpiece or machine table.
8. This reference point will serve as the starting position for setting the work coordinates.
9. Zero the Work Coordinates setting the X, Y, and Z axes to zero at the reference point.
10. Use the appropriate G-code commands (G10, G92) to set the work coordinates to zero.
11. Probe along the X, Y, and Z axes to determine the workpiece's position relative to the machine's reference point.
12. Double-check the work coordinate settings to ensure they accurately reflect the workpiece's position and orientation.
13. Use the machine's jog controls to move the spindle to various locations on the workpiece and verify that the coordinates are correct.
14. Save Work Coordinate Offsets
15. To work coordinates properly set, proceed with programming and executing the machining operations on the CNC milling machine.
16. Ensure that the programmed toolpaths and cutting parameters are aligned with the established work coordinates for accurate machining.
17. Clean tools and equipment.
18. Store all equipment safely.

Specification Sheet 1.1: Determine And Set the Work Coordinates as Per Specification

Name of Job: Set the work coordinates as per specification

Conditions for the job: You must always practice safe operation procedures during the assessment. You will be required to demonstrate OSH competencies to Set the work coordinates as per specification.

Diagram / Design:



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity
1	Hand Gloves	Cotton	pair	01
2	Mask	N 95	Nos	01
3	Safety shoe	As required	pair	01
4	Safety goggles	Standard	Nos	01
5	Apron	Standard	Nos	01

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity
1	CNC Milling Machine		No	01

Required Materials

Sl.	Name Material	Specification	Unit	Quantity
2	Aluminiam Bar	200 x200 x75	No	01

Learning Outcome -2: Create Program Using G & M Code

Assessment Criteria	1. Preparatory G & M code is created 2. Modal and non-modal codes are identified 3. Absolute and Incremental positioning code is created 4. Circular and linear interpolation are created 5. Cutter radius compensation is selected 6. Tools offset (G43) and work offset (G54) are set 7. Spindle commands and program stop commands are executed 8. Canned cycles are executed 9. Bolt hole codes are executed 10. Pocket milling and contour milling is programmed 11. Sub-program is executed
Conditions and Resources	1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Laptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary hand and measuring tools 11. Necessary PPE
Contents	1. Preparatory G & M Code ▪ Repeat ▪ Absolute ▪ Incremental ▪ Linear interpolation ▪ Circular interpolation ▪ Tool change command ▪ Spindle command ▪ Tool radius compensation 2. Modal and non-modal codes 3. Absolute and extended positioning codes ▪ G90 ▪ G91 4. Circular and linear interpolation 5. Compensation of cutter radius 6. Tool offset (G43) and work offset (G54) 7. spindle command and program stop command 8. Canned cycle ▪ Drilling canned cycle ▪ Spot drilling canned cycle

	▪ Boring canned bicycles ▪ Tapping canned cycle ▪ Reverse tapping canned cycle 9. Bolt hole code ▪ G70 (circle) ▪ G71 (Curve) ▪ G72 (straight line) 10. Pocket milling and contour milling codes ▪ G12 (clockwise) ▪ G13 (opposite side of fork) ▪ G47 (Engraved) 11. Sub-program ▪ M97 (internal) ▪ M98 (external) ▪ Looping 12. Job requirement 13. Method of drawing interpretation 14. Continuity of operations in component manufacturing 15. Cutting tool selection process 16. Calculation of cutting speed, feed rate and depth of cut ▪ CNC programming
Job/Task/Activity	1. Generate preparatory G & M code 2. Identify modal and non-modal codes 3. Create absolute and extended positioning codes 4. Create circular and linear interpolation 5. Select the cutter radius compensation 6. Set the tool offset (G43) and work offset (G54). 7. Execute spindle command and program stop command 8. Execute the canned cycle 9. Execute the bolt hole code 10. Perform Programming of pocket milling and contour milling 11. Perform sub program
Training Method	1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming 9. Tole Play
Assessment Method	1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 2: Create Program Using G & M Code

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about about the learning materials	1. Instructor will provide the learning materials ”Create and Use G&M Code for Machining”
2. Read the Information sheet and complete the Self Checks & Check answer sheets on “Create program using G & M code”	2. Information sheet 1: Create program using G & M code 3. Self-check 1: Create program using G & M code 4. Check your answer with Answer key 1: Create program using G & M code
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet ▪ Job Sheet 2.1: Identify modal and non-modal codes as per specification Specification Sheet 2.1: Identify modal and non-modal codes as per specification ▪ Job Sheet 2.2: Create circular and linear interpolation as per specification Specification Sheet 2.2: Create circular and linear interpolation as per specification ▪ Job Sheet 2.3: Select the cutter radius compensation as per specification ▪ Specification Sheet 2.3: Select the cutter radius compensation as per specification Job Sheet 2.4: Set work offset (G54) as per specification ▪ Specification Sheet 2.4: Set work offset (G54) as per specification Job Sheet 2.5: Create Program of pocket milling and contour milling as per specification ▪ Specification Sheet 2.5: Create Program of pocket milling and contour milling as per specification Job Sheet 2.6: Create Program of contour milling as per specification ▪ Specification Sheet 2.6: Create Program of contour milling as per specification

Information Sheet 2: Create Program Using G & M Code

Learning Objective: After completion of this information sheet, the learners will be able to explain, define and interpret the following contents

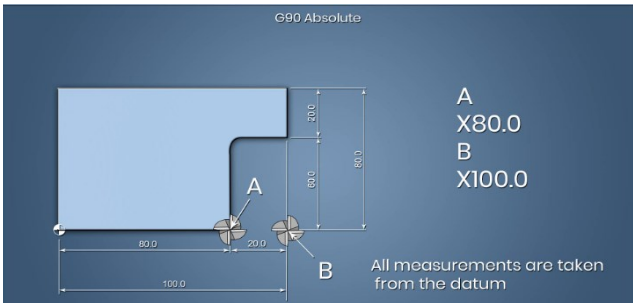
2.1 Preparatory G & M Code
2.2 Modal and non-modal codes
2.3 Absolute and extended positioning codes
2.4 Circular and linear interpolation
2.5 Compensation of cutter radius
2.6 Tool offset (G43) and work offset (G54)
2.7 spindle command and program stop command
2.8 Canned cycle
2.9 Bolt hole code
2.10 Pocket milling and contour milling codes
2.11 Sub-program
2.12 Job requirement
2.13 Method of drawing interpretation
2.14 Continuity of operations in component manufacturing
2.15 Cutting tool selection process
2.16 Calculation of cutting speed, feed rate and depth of cut
2.17 CNC programming

2.1 Preparatory G & M Code

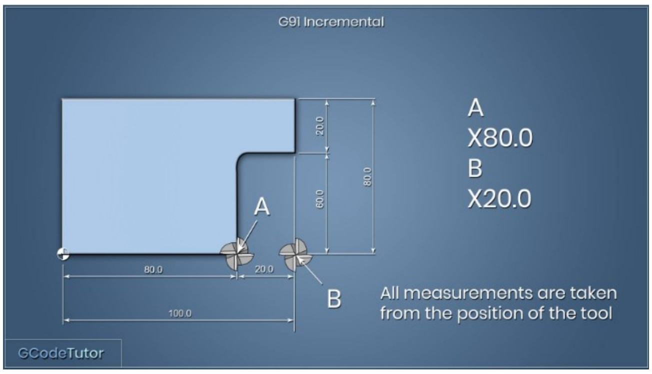
In CNC (Computer Numerical Control) milling, G-codes and M-codes are used to control the movements and functions of the machine. Here's a breakdown of some common preparatory G and M codes and their functions in various modes:

a. **Repeat (G80)**: This command cancels any canned cycles or fixed cycles in effect. It's typically used to ensure that no cycles from previous operations interfere with the current operation.

b. **Absolute (G90)**: In absolute programming mode, coordinates are given from a fixed point of reference. When this command is issued, subsequent X, Y, and Z coordinate values are interpreted as absolute positions from the origin (or machine reference point).

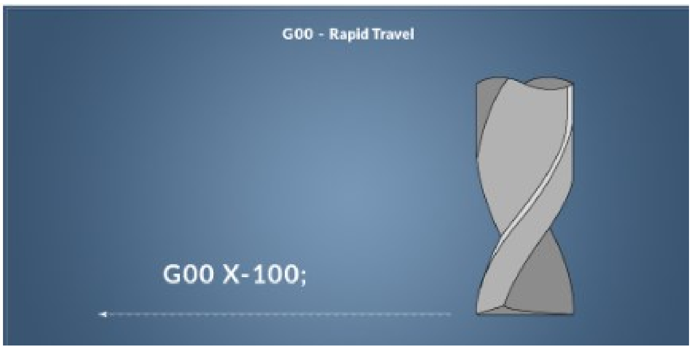


c. **Incremental (G91)**: In incremental programming mode, coordinates are given as incremental distances from the current position. So, each subsequent coordinate value represents a distance to move from the current position rather than an absolute position from a fixed reference point.



d. Rapid tables (G00) : G00 is the rapid travel command in G Code. It is used when the cutter or tool is not removing material so that the time it takes to machine the part is as quick as can be. The top speed is set by the machine parameters and therefore is only controllable by the operator using a rapid override control.

When using a rapid move, be aware of any clamps, vices, and parts that may be in your path. If you are unsure it may be wise to move the X and Y axis first, then on a separate line, move down in Z. It will increase your machining time by a second or two but it will help avoid a 3 Axis crash.



e. Speed (G04): This command is used to specify a dwell or pause time in the program, allowing the machine to dwell for a specified duration before proceeding to the next command.

Sometimes we need to pause the cutter for a brief moment, for that we add a dwell to the code to stop the machine from continuing reading the program for a specified amount of time.

Uses: While drilling with a flat bottom drill and the surface of the bore has a chattered finish, we can stop moving the drill in Z-Axis with it still rotating for half a second to clean up the surface.

G01 Z-20.0 F50; G04 X500; G01 Z5.0;

The few blocks listed above looks like the tool will move 500mm while deep in the part. It won't. The X value in 'G04 X500' forces the tool to dwell for 500 milliseconds before moving onto the next block. The feed rate does not need to be specified again after a

2.1 Tool Change Command

In CNC milling, the tool change process is crucial for replacing cutting tools during machining operations. The tool change command typically involves the use of an M-code, specifically M06. Here's how it works:

a. **M06 command**: The M06 command is used to initiate a tool change. When the CNC controller encounters this command in the program, it stops the machining process and prompts the operator to change the cutting tool.

b. **Tool selection**: Before executing the M06 command, the CNC program specifies which tool should be used for the upcoming machining operations. This information is typically included in the toolpath programming, where each tool is assigned a tool number.

c. 

d. **Tool magazine**: CNC milling machines often have a tool magazine where multiple cutting tools are stored. When the M06 command is executed, the machine moves to a predefined position where the tool change can safely occur. The tool magazine typically has an automatic tool changer (ATC) mechanism that retrieves the specified tool and loads it into the spindle.

e. **Operator intervention**: Once the machine reaches the tool change position, it stops and waits for the operator to perform the tool change. The operator manually removes the current tool from the spindle and inserts the requested tool from the tool magazine.

f. **Tool length offset**: After the new tool is inserted into the spindle, the CNC program may automatically apply the corresponding tool length offset. This offset ensures that the tool's cutting edge is properly aligned with the workpiece surface.

g. **Resume machining**: Once the tool change is complete and the new tool is properly installed, the operator confirms the tool change, and the CNC controller resumes the machining process from where it left off.

The M06 command and the tool change process may vary slightly depending on the specific CNC machine and its control system. It's essential to follow the machine's operating manual and safety guidelines when performing tool changes to ensure efficient and safe machining operations.

2.2 Spindle Command

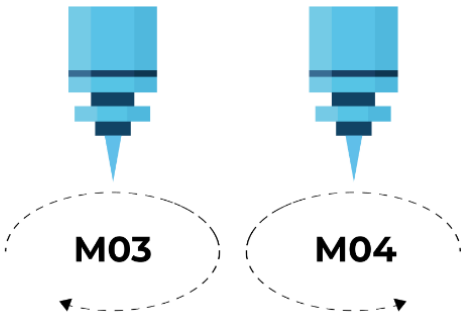
In CNC milling, the term "spindle" typically refers to the rotating cutting tool holder that holds the cutting tool and rotates it at high speeds to perform the cutting operations on the workpiece. The spindle is a crucial component of the milling machine, and controlling its movement and speed is essential for achieving the desired machining results.

The "spindle command" usually refers to the commands issued to control the operation of the spindle during the CNC milling process. These commands are typically part of the CNC program (G-code) and are used to specify parameters such as spindle speed, direction of rotation, and tool engagement.

some common spindle commands used in CNC milling

a. **Spindle speed command (s):** This command is used to set the rotational speed of the spindle. The speed is usually specified in revolutions per minute (RPM). For example, S1000 would command the spindle to rotate at 1000 RPM.
b. **Spindle direction command (m03 and m04):** These commands are used to control the direction of rotation of the spindle.

- M03 command is used to start the spindle in the clockwise direction.
- M04 command is used to start the spindle in the counterclockwise direction.



- Spindle Stop Command (M05): This command is used to stop the rotation of the spindle.

  These are some of the basic spindle commands used in CNC milling. The specific syntax and usage of these commands may vary depending on the CNC machine and controller being used. It's essential to refer to the machine's documentation and programming guidelines for accurate usage of spindle commands.

2.4. Tool Radius Compensation

Two different G-Codes to apply cutter compensation depending on the direction of the cut. G41 is left compensation and G42 is right compensation.

2.5. Modal and Non-Modal Codes

G- and M-codes are modal — they cause the machining system to change from one mode to another. The mode stays active until another command changes it implicitly or explicitly.

EXAMPLE If coolant is turned on (**M07** or **M08**), it stays on until it is explicitly turned off in the program (**M09**).

A few G-codes and M-codes are non-modal. These codes have effect only on the lines on which they occur.

EXAMPLE Dwell (**G04**) is non-modal.

Modal commands are arranged in sets, called modal groups. Only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time.

G4 is a dwell if this were modal it would cause many problems

G0 X50. Y50. (Rapid move)

G4 X5. (Five second dwell)

X0 (This would be a rapid command not another dwell because it remembers the **G0**)

It remembers the **G0** but conveniently forgets the **G4.**

EXAMPLE Inch units (**G20**) vs. millimeter units (**G21**).

A machining system may be in many modes at the same time, with one mode from each modal group being in effect.

For all G-code modal groups, when a machining system is ready to accept commands, one member of the modal group must be in effect. There are default settings for these modal groups. When the machining system is turned on or re-initialized, default values are automatically in effect.

a. Modal Groups for G-Codes

Group 1

{**G00**,**G01**,**G02**,**G03**,**G33**,**G38**, **G73**, **G76**, **G80**,**G81**, **G82**, **G84**,**G85**, **G86**, **G87**, **G88**,**G89**}

motion – This is a group of G-codes for motion called current motion mode — one is always in effect.

Group 2 {**G17**, **G18**, **G19**, **G17.1**, **G17.2**, **G17.3**} plane selection

Group 3 {**G90**, **G91**} distance mode

Group 4 {**G90.1**, **G91.1**} arc IJK distance mode

Group 5 {**G93**, **G94**} feed rate mode

Group 6 {**G20**, **G21**} units

Group 7 {**G40**, **G41**, **G42**, **G41.1**, **G42.1**} cutter radius compensation

Group 8 {**G43**, **G43.1**, **G49**} tool length offset

Group 10 {**G98**, **G99**} return mode in canned cycles

Group 12 {**G54**, **G55**, **G56**, **G57**, **G58**, **G59**, **G59.1**, **G59.2**, **G59.3**} coordinate system selection

Group 13 {**G61**, **G61.1**, **G64**} path control mode

Group 14 {**G96**, **G97**} spindle speed mode

Group 15 {**G07**, **G08**} lathe diameter mode

Modal groups for M-codes are as follows:

Group 4 {**M00**, **M01**, **M02**, **M30**, **M60**} stopping

Group 7 {**M03**, **M04**, **M05**} spindle turning

Group 8 {**M07**, **M08**, **M09**} coolant (special case: **M07** and **M08** may be active at the same time)

Group 9 {**M48**, **M49**} enable/disable feed and speed override controls

Group 10 {operator defined **M100** to **M199**}

b. Non-Modal G-Codes

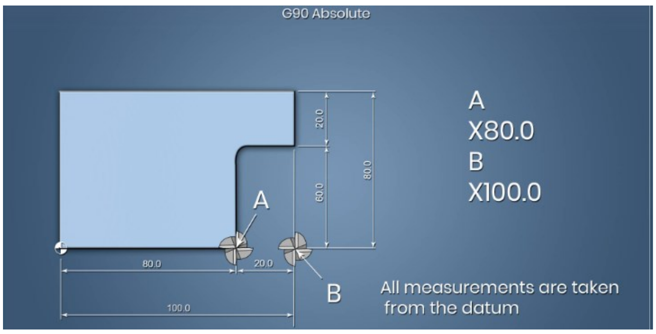
Group 0 {**G04**, **G10**, **G28**, **G30**, **G53**, **G92**, **G92.1**, **G92.2**, **G92.3**}

2.6. Absolute And Extended Positioning Codes

In CNC milling, the G90 and G91 codes are used to specify the positioning mode: absolute and incremental, respectively. These positioning modes determine how the machine interprets coordinate values.

a. G90 - absolute positioning

The G Code G90 is used to define the absolute positioning system. When G90 is active the machine will read all dimensions and movements from the working datum position.



If we were to issue a movement command such as G00 X100.0 Y100.0; then the machine would move 100mm in the plus direction from the datum in both the X and Y axis.

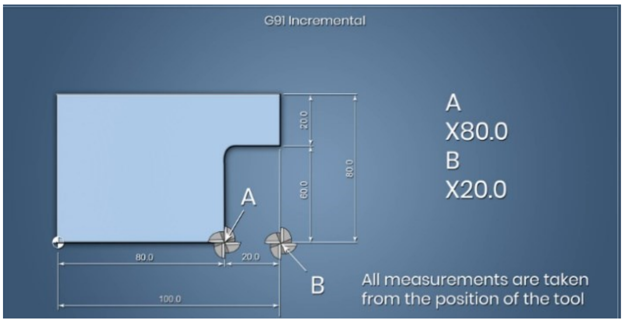
If we were to enter G00 X0.0 Y0.0; we would move the spindle/tool to the datum position.

Each movement command we make will move the tool in relation to the datum position that we have set previously.

Regarding the above drawing. To move the cutter to position A from the datum in the lower left-hand corner of the part, we would give an X dimension of +80. To continue to move to position B we would give an X dimension of +100.

b. G91 - incremental positioning

When working with G91 incremental positioning, We command the tool to move from its current position and not the datum position.

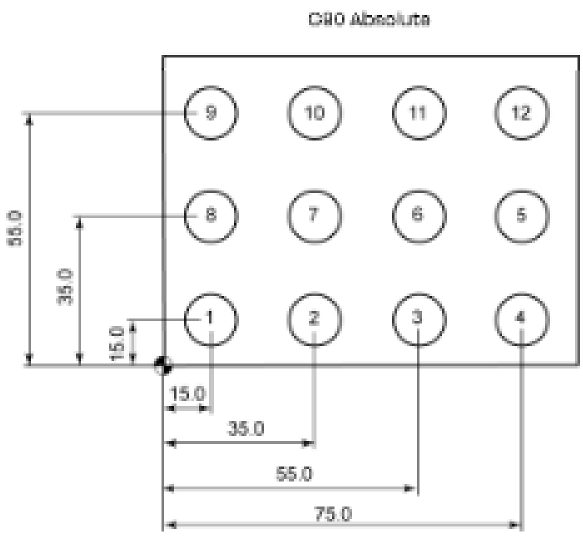


The above drawing shows that to move to position A from the origin we would give a distance of 80mm, the same as absolute. To move from position A to position B we would need to command the X-axis to move 20mm in the plus direction.

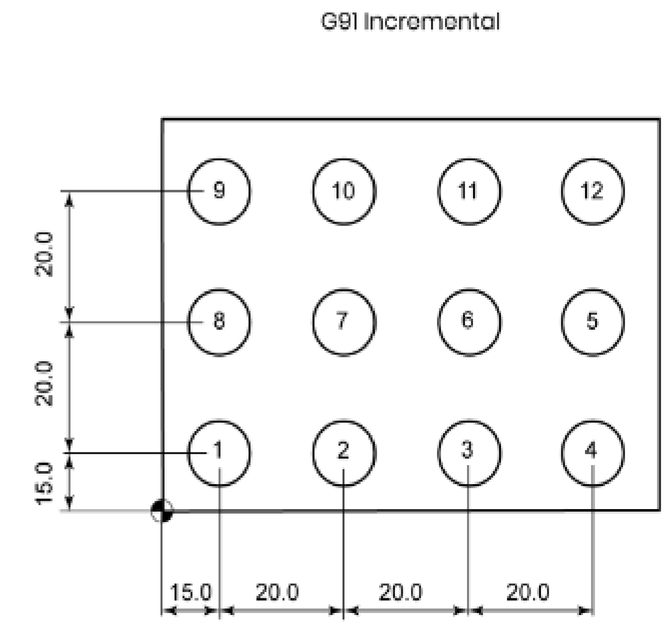
This is because we are giving the distance from the tool position and not the datum position.

We can think of it as the origin or datum is shifting to the centre of the tool after each movement.

Example: G90 - Absolute Positioning

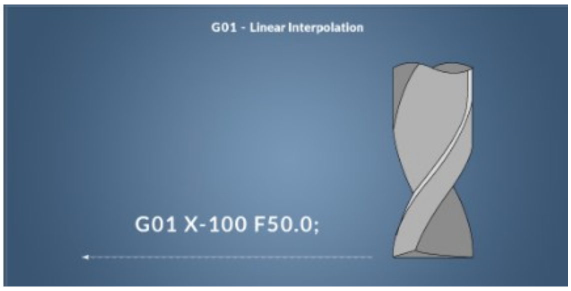
:0002 (G90 EXAMPLE); N2 T0202 (15MM DRILL); G90 G21; S600 M03; G00 X15.0 Y15.0 (POSITION 1); X35.0 (POSITION 2); X55.0 (POSITION 3); X75.0 (POSITION 4); Y35.0 (POSITION 5); X55.0 (POSITION 6); X35.0 (POSITION 7); X15.0 (POSITION 8); Y55.0 (POSITION 9); X35.0 (POSITION 10); X55.0 (POSITION 11); X75.0 (POSITION 12); G00 Z50.0; G28 X0.0 Y0.0 M05; M00;	

Example: G91 - Incremental Positioning:

0003 (G91 EXAMPLE); N2 T0202 (15MM DRILL); G90 G21 (MOVING TO POSITION 1 IN ABSOLUTE); S600 M03; G00 X15.0 Y15.0 (POSITION 1); G91 (INCREMENTAL MODE); X20.0 (POSITION 2); X20.0 (POSITION 3); X20.0 (POSITION 4); Y20.0 (POSITION 5); X-20.0 (POSITION 6); X-20.0 (POSITION 7); X-20.0 (POSITION 8); Y20.0 (POSITION 9); X20.0 (POSITION 10); X20.0 (POSITION 11); X20.0 (POSITION 12); G90 (ABSOLUTE MODE); G00 Z50.0; G28 X0.0 Y0.0 M05; M00;	

2.7. Circular And Linear Interpolation

a. **Linear interpolation (g01)**: This command enables linear interpolation, where the tool moves in a straight line from one point to another at a specified feed rate. It's commonly used for straight cuts or contouring.



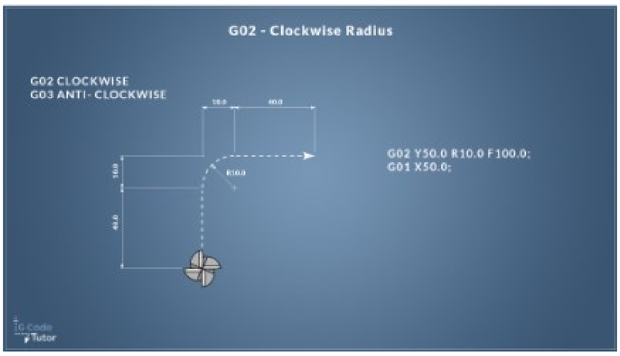
G01 command is used when you are cutting in a straight line. This command has to have a feed rate (F) active before it will run. A typical block would look like G01 X-100 F50.0; You don't need to add this command on every line, as long as there is no other movement G-Code active, for example, you only need to add it after a G02, G03 or a G00 command. In much the same way, a feed rate (F50) is only needed once until you want to change the speed in which you are cutting.

```
G00 X101 Z1.0;
G01 X100 Z-20 F100;
X110 Y-40;
G00 Z300;
```

This command can be used with all axis cutting at the same time, or just one. It is not common to cut with the Z axis as well as X and Y but it is possible if needed.

When programming a profile, it is easier to use cutter compensation G41 and G42 then you do not need to allow for the radius of the cutter when plotting your tool paths, you can simply use the dimensions on the drawing and the machine will offset the cutter to achieve the correct dimensions.

b. Circular interpolation (g02/g03): These commands enable circular interpolation. G02 is for clockwise circular interpolation, while G03 is for counterclockwise. Circular interpolation allows the tool to move along a circular path with specified parameters like center point, radius, and direction.



G02 is used to machine an arc or radius in a clockwise direction and G03 is for anti-clockwise.
Using an ‘R’ value to define the radius, the G02/G03 command moves the cutter in a straight line with the defined radius at the end of this move.

If a feed rate is already active there is no need to add it to this line unless you wish to change the feed rate for the machining of the arc.
G03 Y50.0 R10.0 F100.0;
G01 X50.0;

The centre point of the arc can also be defined using I, J, and K values. I (X) J(Y) and K(Z) defines the auxiliary axis.
using the drawing above, programming an arc using I and J values would look like this:

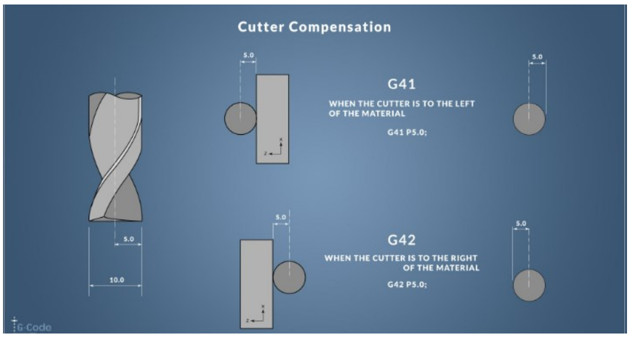
G03 Y50.0 I10.0 J40.0 F100.0;
G01 X50.0;

The K is seldom used but is available if an arc using the Z axis is needed.
When using G02 with G01 and G03 (Counter clockwise arc) any shape can be machined. These three G Codes are the foundation of G Code programming and are the three you will use when cutting material.

2.8. Compensation Of Cutter Radio us

On a CNC machine, it is usually recommended to use climb milling, this needs to be taken into consideration when writing our programs and deciding which direction to apply cutter compensation.

When we wish to cancel the compensation, G40 is used to turn it off. It is a good working practice is to end each section of the program with G40 so if we need to jump back to re-run a tool, we will not have G41 or G42 applied unless it is called upon after the tool change.



Different ways to apply cutter comp

There are three different ways that cutter compensation can be applied

G42 P5.0;

By using a 'P' value we can tell the machine that we wish to offset the cutter by a defined amount, in this case, 5mms.

This technique is usually used on desktop routers and small CNC machines without a built-in tool table

G42 X5.0;

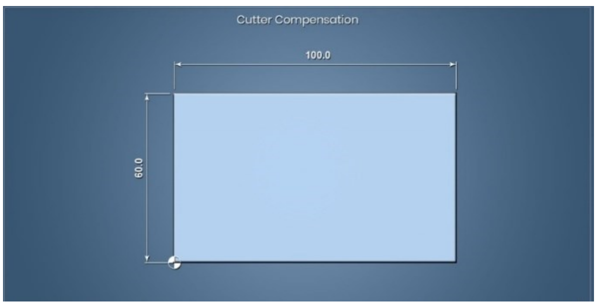
Another way we can write this is by using an 'X' value, it means the same thing.

G42;

On machines with a built-in tool table within their control system, we normally see cutter compensation just applied using a simple G42 (or G41) command

Larger industrial machinery normally uses this style of defining the radius of the tool as the tooling information is entered into the tool table during the set up of the machine. During the tool call line of our program, the machine pulls all the tooling information into its memory and G42 is used to let the machine know that we wish to use cutter compensation. No other information is needed.

Example :



T0101 (10MM ENDMILL);
M06;
G21 G90;
S800 M03;
G41 P5.0;
G00 X0.0 Y0.0;
Z-10;
G01 Y60.0 F100.0;
X100.0;
Y0.0;
X0.0;
G00 Z50.0;
X200.0 Y200.0;

We start by calling the tool, in this case, a 10mm endmill. Then we perform a tool change with the M06 command and set turn the spindle on (M03) at 800RPM. We also set the machine to absolute and metric systems using G21 and G90.

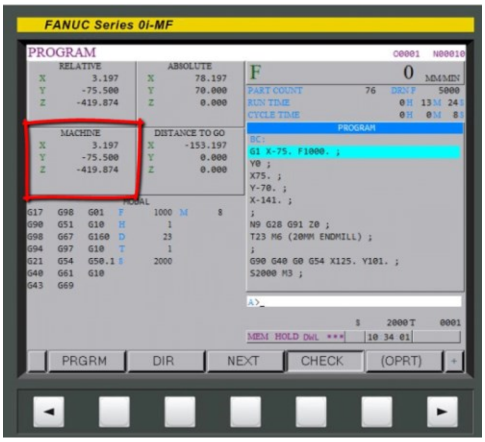
Before we move the cutter, we apply the G41 cutter compensation and tell the machine that we are offsetting it my 5mm with the P value. (5mm being the radius of the tool) We are machining this part by starting at the datum position at the lower left-hand corner and travelling up to Y60.0 position. So, the endmill will need to be offset to the left of the material, hence using G41 and not G42.

2.9. Work offset

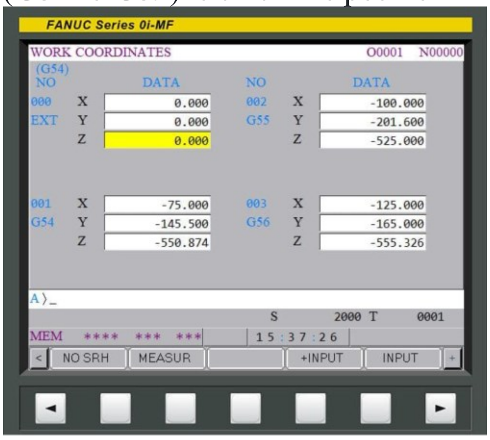
When we write a CNC program we work from a datum on the drawing. All the X and Y figures will be measured from this datum. Once we put the component on the machine it needs to know where the part is. This is called the work offset.
G code usually G54 although as standard you have six of these.
G54 G55 G56 G57 G58 G59
Once you set one of these all you need do is use the G code (54 to 59) and the machine will use that offset.
Each one of these G Codes represents a datum position on the machine.
G0 G54 X0 Y0 (Rapid to X0 Y0 using G54)
G0 G55 X0 Y0 (Rapid to X0 Y0 using G55)
Because the work offset is modal once you state it you don't need to repeat yourself.
G0 G54 X0 Y0 (Rapid to X0 Y0 using G54)X50. Y50. (Still rapid still G54)Z10.

a. Zero Return

When you first turn on your CNC Machine you would normally reference or Zero Return all the axis. The machine then knows where it is.
All machines will have a position display. This position display will have one set of figures normally called "MACHINE". This is the machines position from zero return. So when the machine is at zero return this will read.
X 0.000Y 0.000Z 0.000



The “MACHINE” position tells us how far we are from the machine zero. We don’t use this once we have set ourdatums. This is the position we need to write into the work offset page to tell the control where each datum is (**G54 to G59**) What we do when we are setting (**G54 to G59**) is enter this position in the work offset page.

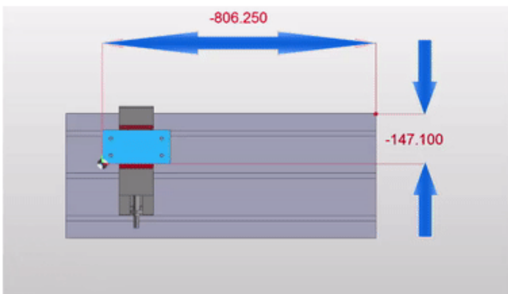


When we subsequently call this G code the machine will use this position as it’s datum.

On the screen above if you programmed **G0 G54 X0 Y0** the machine would move **-75**. in **X** and –**145.5** in **Y**. Thisis it’s new zero position. Every subsequent command will work from this datum.

b. Set the Work Offsets

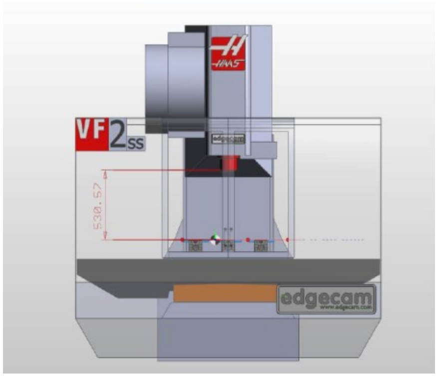
What we do when we are setting the machines datums or Work Offsets is we tell the machine where our datum isfrom Zero Return.



G54 X- 806.250
Y-147.100

In the above case the datum is **806.25** away from **X** Zero Return and **147.1** away from **Y** Zero Return. These will both be minus figures

The Z will be the distance from Zero return to the top of the work-piece.



So in the above case the distance from the spindle nose to the top of the work-piece is **530.570**.

G54 X -806.200
Y-147.100
Z-530.570

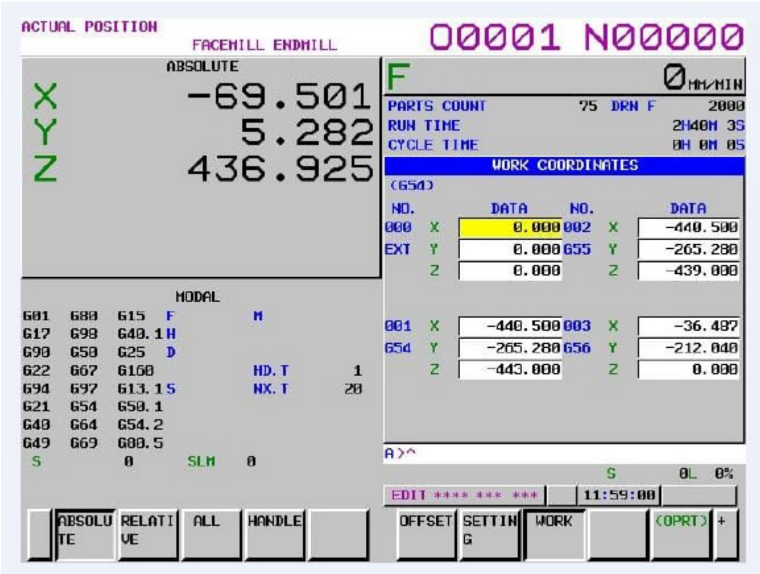
So there you have it your work offset in **X Y** and **Z**.
This is how it looks in the offset file on a Haas machine.
This is an imperial (inch) machine so this datum is **12.568 inches** away from the **X** zero and **8.489** from the **Y** zero.

<< WORK PROBE — WORK ZERO OFFSET — WORK PROBE >>

G CODE	X AXIS	Y AXIS	Z AXIS		
G52	0.	0.	0.		
G54	-12.5680	-8.4890	-23.1480		
G55	0.	0.	0.		
G56	0.	0.	0.		
G57	0.	0.	0.		
G58	0.	0.	0.		
G59	0.	0.	0.		
G154 P1	0.	0.	0.		
G154 P2	0.	0.	0.		
G154 P3	0.	0.	0.		

ENTER A VALUE

Set in G54 G55 and G56. you could use any of these offsets.



2.10. Program Stop Command

Program stop commands are used to halt the execution of the CNC program.
They are typically used for pausing the program, allowing for manual intervention, tool changes, or inspection.
Common program stop commands include:
M00: Program stop (optional stop).
M01: Program stop (conditional stop).
M02: End of program (optional stop).
M30: End of program (reset program).
Example G-code for program stop commands:

2.11. Canned Cycle

Canned cycles are predefined sequences of G-codes that automate common machining operations on CNC milling machines, simplifying programming and reducing the amount of code needed to perform repetitive tasks. Here are some canned cycles commonly used for drilling, spot drilling, boring, tapping, and reverse tapping on CNC milling machines:

a Drilling canned cycle (g81)

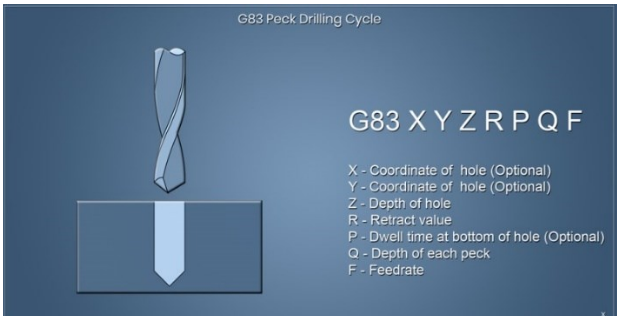
G81 X Y Z R F;
X = Coordinate of hole (Optional)
Y = Coordinate of hole (Optional)
Z = depth of hole
R = Retract value
F = Feedrate

```
Example:
G81 Z-12.0 R1.0 F50.0;
X50.0;
X50.0;
Y40.0;
G80;
```

This cycle is used when drilling a series of holes when a dwell or peck drilling is not needed. This is the most basic drilling cycle used on a CNC milling machine.

The X and Y dimensions are optional if they are omitted on this line the machine expects the spindle to be in the position of the first hole.

b G83 - peck drilling cycle



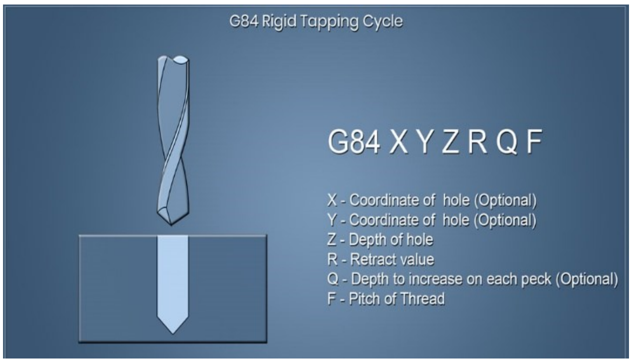
G83 X Y Z R P Q F
X = Coordinate of hole (Optional)
Y = Coordinate of hole (Optional)
Z = Depth of hole
R = Retract value
P = Dwell time at bottom of hole
Q = Depth of each peck
F = Feed rate

Example:
G83 Z-12.0 R1.0 P1000 Q3.0 F50.0;
X50.0;
X50.0;
Y40.0;
G80;

The G83 roughing cycle allows for the production of holes using a peck drilling sequence. A hole is drilling a set amount defined by the Q value then retracts to the position of the R value before continuing to drill the hole until the full depth is reached that is set using the Z value.
This is used when the depth of the hole is deep enough to cause swarf build up, as this cycle chips the swarf.

The R-value is the distance from the datum in Z to the tip of the drill. When the datum is set to the surface of the part, this dimension would be the distance from the surface of the part to the tip of the drill.

c G84 - rigid tapping cycle



G84 X Y Z R Q F;
X = Coordinate of hole (Optional)
Y = Coordinate of hole (Optional)
Z = Depth of hole

Example:
G84 Z-12.0 R5.0 F0.8;
X50.0;
X50.0;
Y40.0
G80;

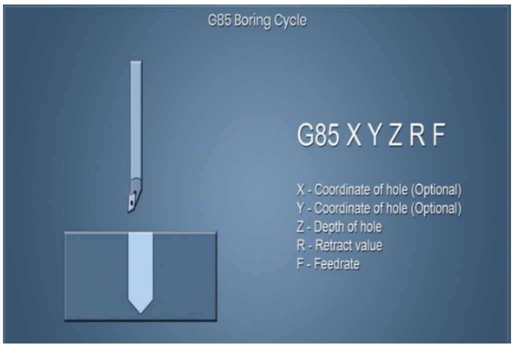
R = Retract value

Q = Depth to increase on each peck (Optional)

F = Pitch of Thread

The G84 rigid tapping cycle used to produce threads in holes without the use of a tapping head.
By adding a Q value, we can turn this tapping cycle into a peck tapping cycle. The machine will tap the amount specified by the Q value before reversing out of the hole to the dimension defined by R. Then continue tapping until the full depth (Z) has been achieved.
For more information check out the G84 - Rigid Tapping Cycle article.

d G85 - boring cycle



G85 X Y Z R F;

X = Coordinate of hole (Optional)

Y = Coordinate of hole (Optional)

Z = Depth of hole

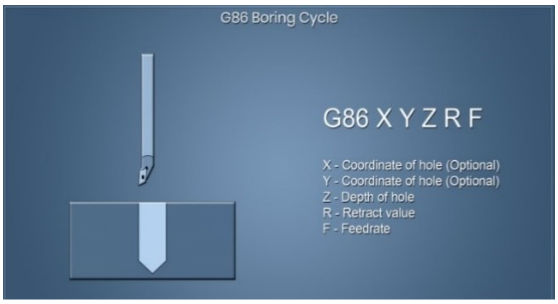
R = Retract value

F = Feed rate

Example:
G85 Z-8.0 R1.0 F30.0;
X50.0;
X50.0 Z-5.0;
Y40.0 Z-6.0;
G80;

The G85 boring cycle bores the hole then feeds out giving a spring cut. The above example demonstrates boring four holes at three different depths.

e G86- boring cycle



G86 X Y Z R F;

X = Coordinate of hole (Optional)

Y = Coordinate of hole (Optional)

Z = Depth of hole

R = Retract value

F = Feed rate

The G86 boring cycle differs from the G85 boring cycle because it bores the hole then stops the spindle before it rapid travels out to the R value, where the G85 cycle bores in then bore out.

2.12. Bolt Hole Code

In CNC milling, G70, G71, and G72 are G-codes used for different types of bolt hole patterns. These codes allow CNC programmers to create bolt hole patterns quickly and efficiently. Here's an explanation of each:

a. G70 - bolt hole circle

G70 is used to create a circular pattern of bolt holes.

Parameters include the center coordinates of the circle, the number of holes, the diameter of the circle, and the angle between holes (optional).

Example:

G70 I... J... L.

I	Radius of the bolt hole circle (– CW / + CCW).
J	Start angle of the first hole.
L	Number of bolt hole

b. G71 - bolt hole curve

G71 is used to create a curve pattern of bolt holes.

Parameters include the start and end points of the curve, the number of holes, and the direction of the curve (clockwise or counterclockwise).

Example:

G71 Bolt Hole Arc

I - Radius
J - Starting angle (degrees CCW from horizontal)
K - Angular spacing of holes (+ or --)
L - Number of holes

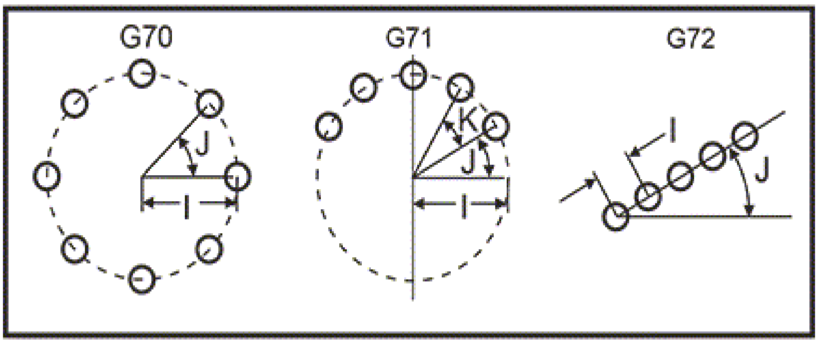
G71 I--, J--,K---,L---

c. G72 - bolt hole straight line

I - Distance between holes
J - Angle of line (degrees CCW from horizontal)
L - Number of holes
indicates optional

This non-modal G code drills L number of holes in a straight line at the specified angle. It operates similarly to G70. For a G72 to work correctly, a canned cycle must be active so that at each position, a drill or tap function is performed.

G70, G71, and G72 Bolt Holes: [I] Radius of bolt circle (G70, G71), or distance between holes (G72), [J] Starting angle from the 3 oclock position, [K] Angular spacing between holes, [L] Number of holes.



d. Rules for bolt pattern canned cycles

- Place the tool at the center of the bolt pattern (for G70 or G71), or at the starting hole location (for G72), before the canned cycle execution.
- The *J* code is the angular starting position and is always 0 to 360 degrees counterclockwise from the three oclock position.
- For G70 and G71 cycles, put an *L0* on the initial canned cycle line to skip drilling at the center of the hole pattern. You can also turn off Setting 28 to prevent a hole from being drilled at the initial X/Y position.

2.13. Pocket Milling and Contour Milling Codes

In CNC milling, pocket milling and contour milling are common machining operations used to remove material from a workpiece to create pockets, slots, and complex shapes. G12, G13, and G47 are G-codes used for pocket milling, contour milling, and engraving operations. Here's an explanation of each:

a. G12 - clockwise circular pocket milling

G12 is used for clockwise circular pocket milling, where the cutting tool moves in a circular motion to remove material from inside a pocket or slot.

Parameters include the start and end points of the arc, the radius of the arc, and the depth of cut.

b. G13 - counterclockwise circular pocket milling

G13 is used for counterclockwise circular pocket milling, where the cutting tool moves in a counterclockwise motion to remove material from inside a pocket or slot.

Parameters are similar to G12.

c. G47 - engraving

G47 is used for engraving operations, where the cutting tool follows a specified path to create engraved text, logos, or other designs on the workpiece.

Parameters include the start point, end point, depth of engraving, and feed rate.

In each of the examples

X and Y specify the coordinates of the milling or engraving operation.

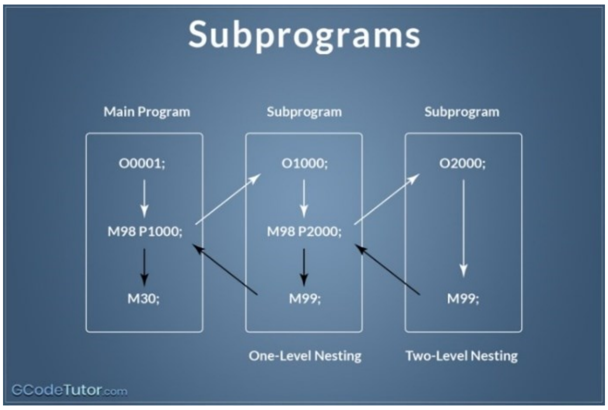
Z specifies the depth of cut or engraving.

I and J specify the incremental distances for circular pocket milling.
F specifies the feed rate for the cutting tool.

These codes (G12, G13, and G47) are essential for performing pocket milling, contour milling, and engraving operations on CNC milling machines, allowing for the creation of intricate and precise features on workpieces.

2.14. Sub-Program

In CNC milling, sub-programs and looping are techniques used to streamline programming, reduce code redundancy, and improve efficiency. M97, M98, and looping commands are commonly used for these purposes. Here's an explanation of each:



Sub-program (M97 and M98):

a. M98 Subprogram Call

The M98 command is used to call a subprogram followed by the program number and the amount of times that we wish to repeat running that subprogram.

A typical M98 block may look like this
M98 P52000;
M98 calls the subprogram
P is the program number (O2000)
The number that precedes the program number is the amount of times it is to be repeated (in this case 5 times)
At the end of the subprogram, the M-Code M99 is used to return to the previous program.
Note: the previous program may be a subprogram.
Up to 4 subprograms can be nested, this means we can call upon a subprogram inside another subprogram up to 4 times

c. M99 Subprogram end

The M-Code M99 will tell the machine that the subprogram that it is currently running has come to an end, if the subprogram has run the number of times specified by the initial program call using M98 P____; It will stop running the subprogram and return from to the previous program and continue to run from the line below the M98 line.

M99 will always return to the program that it came from, even if it is a subprogram. We can not jump back to the main program without returning to the program that called the subprogram.

d. M30 - Program Stop

Once we have used the M-Code M99 to return to the main program we would use M30 to tell the machine that the program has finished and to stop the machine.

An example:

- M98 P52000; (Runs the subprogram O2000 and repeats 5 times)
- M99; (Returns to the program that called O2000 and runs from the block after the M98 command)
- M30;(Machine stop, program finished)

The 'M98 P L' way

Some versions of FANUC approach this subject slightly differently, If your machine does not accept the code above, then try this:

M98 P2000 L5;

M98 - Calls the subprogram

P2000 - Subprogram number (O2000)

L5 - Number of repeats (5 times

2.15. Looping

Looping allows a section of code to be repeated multiple times within a CNC program.

Looping commands include GOTO (branching), IF-THEN-ELSE (conditional branching), and DO (looping).

Looping is often used for repetitive tasks, such as drilling multiple holes or cutting multiple features.

Example (using a simple loop with GOTO):

In each of the examples

P specifies the program number of the sub-program to be called.

N specifies the line number of the CNC program.

GOTO is used to jump to a specified line number, effectively creating a loop.

These techniques (sub-programs and looping) are powerful tools for CNC programmers, allowing for more efficient and organized programming of CNC milling operations. They help reduce code duplication, improve readability, and make it easier to make changes or updates to CNC programs.

2.16. Job Requirement

Job requirements in CNC milling machine operations typically vary depending on the specific role and level of expertise needed. However, some common job requirements for CNC milling machine operators and programmers may include:

Technical skills	▪ Proficiency in reading and interpreting engineering drawings, blueprints, and technical specifications. ▪ Knowledge of CNC machine operations, including setup, tooling, and operation. ▪ Understanding of machining principles, cutting tools, feeds, and speeds. ▪ Ability to use precision measuring instruments such as micrometers, calipers, and gauges to ensure accurate machining processes. ▪ Familiarity with G-code programming and CNC machine controls.
Experience	▪ Experience operating CNC milling machines and other machining equipment. ▪ Knowledge of machining materials such as metals, plastics, and composites. ▪ Experience with CAM (Computer-Aided Manufacturing) software for generating toolpaths and CNC programs.
Problem-solving skills	▪ Ability to troubleshoot and solve machining problems related to tool wear, material defects, and machine malfunctions. ▪ Capability to optimize machining processes for efficiency, quality, and cost-effectiveness.
Attention to detail	▪ Strong attention to detail to ensure precise machining and adherence to engineering specifications. ▪ Ability to perform quality inspections and measurements to verify part accuracy and conformance to specifications.
Safety awareness	▪ Commitment to following safety protocols and procedures to ensure a safe working environment. ▪ Knowledge of machine safety features, personal protective equipment (PPE), and safe machining practices.
Communication skills:	▪ Effective communication skills to collaborate with team members, engineers, and supervisors. ▪ Ability to communicate technical information clearly and accurately.
Adaptability	▪ Flexibility to work with various materials, part sizes, and machining requirements. ▪ Willingness to learn new machining techniques, software, and technologies.
Teamwork	▪ Ability to work effectively as part of a team, collaborating with colleagues to achieve production goals and deadlines. ▪ Willingness to share knowledge and support others in the machining process.

These are general job requirements for CNC milling machine operators and programmers. Specific roles may have additional requirements based on the complexity of the machining operations, industry regulations, and company standards.

2.17. Method Of Drawing Interpretation

Interpretation of 3D Drawings in CNC Machining

- Geometric Dimensioning and Tolerancing (GD&T) Geometric Dimensioning and Tolerancing (GD&T) is a symbolic language used on engineering drawings and computer-generated models.
- Surface Finish Symbols.
- Interpreting Views and Perspectives.
- Material Specifications.

2.18. Continuity of Operations in Component Manufacturing

Continuity of operations in component manufacturing on CNC milling machines is essential to ensure efficiency, productivity, and the timely delivery of parts. Here are some key factors and strategies for maintaining continuity of operations:

Machine maintenance	▪ Regular preventive maintenance is crucial to keep CNC milling machines in optimal working condition. ▪ Schedule routine maintenance tasks such as cleaning, lubrication, and inspection of machine components to prevent unexpected breakdowns and prolong machine life. ▪ Address any issues promptly to minimize downtime and maintain continuous operation.
Tool management	▪ Implement an effective tool management system to ensure that cutting tools are available and properly maintained. ▪ Monitor tool wear and tool life to anticipate tool changes and prevent tool failures during machining operations. ▪ Keep an inventory of spare tools to replace worn or damaged tools quickly and minimize disruptions to production.
Material management:	▪ Maintain adequate stock of raw materials to prevent delays due to material shortages. ▪ Implement efficient material handling and storage practices to minimize material handling time and ensure materials are readily available for machining operations.
Programming and setup optimization:	▪ Optimize CNC programs and machining setups to minimize setup and changeover times. ▪ Use standardized setups and tooling whenever possible to streamline operations and reduce variability between setups. ▪ Implement efficient toolpath strategies to minimize machining time and maximize productivity.
Quality control:	▪ Implement rigorous quality control measures to ensure that parts meet the specified tolerances and quality standards.

	▪ Perform in-process inspections and dimensional checks to detect any issues early in the manufacturing process and prevent the production of defective parts. ▪ Address any quality issues promptly to avoid rework or scrap that could disrupt production schedules.
Workforce training and skill development:	▪ Invest in training and skill development programs for CNC machine operators, programmers, and maintenance technicians to enhance their proficiency and problem-solving abilities. ▪ Cross-train employees to perform multiple tasks and roles within the manufacturing process to ensure flexibility and continuity in operations.
Contingency planning:	▪ Develop contingency plans and backup strategies to address unforeseen events or disruptions that could impact production. ▪ Identify potential risks and vulnerabilities in the manufacturing process and develop mitigation plans to minimize their impact on operations. ▪ Maintain open communication channels with suppliers, customers, and other stakeholders to address any issues or changes in a timely manner.

By focusing on these factors and implementing appropriate strategies, manufacturers can maintain continuity of operations on CNC milling machines, minimize downtime, and ensure the uninterrupted production of high-quality components.

2.19. Cutting Tool Selection Process

Selection Criteria for CNC Cutting Tools

- **Material compatibility:** Select tools compatible with the material being machined, whether it’s aluminum, steel, or composites.
- **Tool geometry:** Consider the geometry of the tool for the specific operation, including angles and flute numbers.
- **Coating:** Opt for coated tools when working with hard materials for increased durability and performance.
- **Size and tolerance:** Ensure the tool size aligns with the machining requirements and can achieve the desired tolerances.

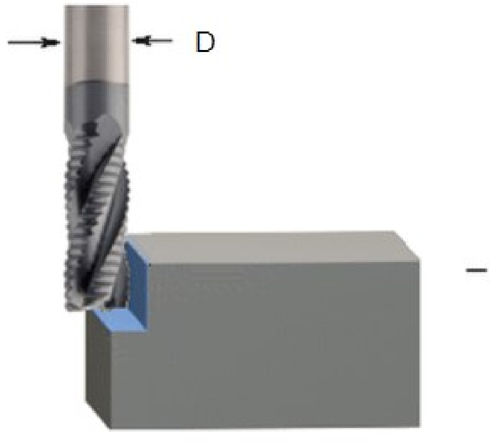
2.20.Calculation of Cutting Speed, Feed Rate and Depth of Cut

Calculating cutting speed, feed rate, and depth of cut is essential for optimizing CNC machining operations and achieving desired machining results. These parameters depend on factors such as the material being machined, the type of cutting tool, and the machining process. Here's how to calculate each parameter:

a. **Cutting Speed (V)**:

G83 - Peck Drilling Cycle

The cutting speed formula is the same for both milling.



Milling

In milling, where the tool is rotation, the cutting speed depends on:
1. Rotation speed of the tool
2. Diameter of the tool.

Metric mode

$$S = \frac{\pi D N}{1000}$$

S in m/min, D in mm, N in RPM.
The 1000 in the denominator is to convert the D in mm to meters.

Inch mode

$$S = \frac{\pi D N}{12}$$

S in ft/min, D in inch, N in RPM.
The 12 in the denominator is to convert the D in inch to feet.

Feed Rate (F)

Feed rate refers to the linear speed at which the cutting tool moves along the workpiece material during machining. It is typically measured in inches per minute (IPM) or millimeters per minute (mm/min).

The formula to calculate feed rate is:

a. Definitions and tips

- Rotation speed n in RPM:
  This is the speed of the spindle in Revolutions Per Minute (RPM)!
- Cutting speed v_c in m/min:
  Speed of a tool (cutter) when cutting normally in m/min. The cutting speed depends on the material. Typical values range from 50 to 500 m/min. It is the speed of a rotating tooth! and we get it from tables. If needed it can be calculated with the diameter of the tool d in mm and the spindle speed n in RPM (Revolutions Per Minute): $v_c = (d \cdot \pi \cdot n)/1000$. With a maximum spindle

speed of 20000 1/min we get: $v_c = d \cdot 63$ m/min. This shows, that we can get a higher speed rate by choosing a bigger diameter of our tool.

- Feed rate (feed speed) v_f in mm/min:
  Tool's distance travelled during one spindle revolution.
- Diameter in mm:
  Diameter of the tool (cutter).
- Number of teeth:
  Number of teeth (cutting edges, flutes) of the used tool (cutter).
- Tooth feed f_z in mm/tooth:
  Also chip load (cpt: chip per tooth) is the amount of material, which should be removed by each tooth of the cutter as it rotates and advances into the work.

The rate of material removal, but also the power requirements and the finish quality of the surface are determined by the feed rate and the cutting speed. They are mostly determined by the material that's being cut. Softer material need lower cutting speeds. Harder cutting tool material need higher cutting speeds.

b. Calculating the feed rate

First, we need to calculate the rotation speed in RPM with the following formula (**n** = rotation speed in RPM, $\mathbf{v_c}$ = cutting speed in m/min, **d** = diameter in mm)

c. Formula to calculate the spindle speed

$$\mathbf{n = (v_c \cdot 1000)/(\pi \cdot d)}$$

The cutting speed for e.g. wood is selected from a table.

Material	**Cutting speed in m/min**
Brass, Bronze, Copper	360
Aluminium (wrought alloy)	300
Cast-Aluminium > 6 % Si	200
Softplastic	150
Hardplastic	100
Steel	90

We get 500 m/min. This gives us a speed for the cutter resp. our spindle of 500000/3.1415 = 159160 RPM (revolutions/min) if we use an 1 mm tool. Ok, here is a problem! The max. speed of my spindle is 20000 RPM! So we stay with 20000 RPM for the next formula to calculate the feed speed

d. Formula to calculate the feed rate

$$\mathbf{v_f = n \cdot z \cdot f_z}$$

In the following table (sorotec.de) we find the tooth feed in mm/tooth for spiral toothed cutter (solid carbide, fishtail, upcut-spiral, 3,175mm (1/8")).

Material	Tooth feed in mm/tooth for d (mm)	0.5	1	1.5	2	2.5	3
Brass, Bronze, Copper		0.003	0.004	0.006	0.008	0.012	0.015

Material	Tooth feed in mm/tooth for d (mm)	0.5	1	1.5	2	2.5	3
Aluminium (wrought alloy)		0.003	0.004	0.006	0.008	0.012	0.016
Cast-Aluminium > 6 % Si		0.002	0.003	0.005	0.007	0.011	0.015
Softplastic		0.008	0.015	0.03	0.04	0.05	0.06
Hardplastic		0.004	0.006	0.010	0.012	0.015	0.018
Steel		0.001	0.003	0.004	0.006	0.008	0.01

The table gives only rough values! to be able to calculate the feed rates. Optimum feed rate will be determined from experience. A good approach is to start off with half of the calculated feed rate and gradually increased to the capacity of the machine and the finish desired.
With our 20000 RPM we get a feed rate of 20000·2·0.01 = 400 mm/min for a 1 mm cutter in wood or 1200 mm/min for a 3 mm cutter.

e. **Depth of Cut (DOC)**

- Depth of cut refers to the distance between the uncut surface of the workpiece and the machined surface resulting from a single pass of the cutting tool. It is typically measured in inches or millimeters.
- The depth of cut is determined based on the desired material removal rate, the rigidity of the machine setup, and the capabilities of the cutting tool.
- The formula to calculate depth of cut is straightforward, but it should be determined based on factors such as tool diameter, material properties, and machining stability.
- The formula to calculate depth of cut is: Depth of cut (t) = D-d/2 mm

It's important to note that these calculations serve as a starting point, and adjustments may be necessary based on machining conditions, tool wear, and desired machining results. Additionally, consulting machining handbooks, tooling catalogs, and empirical data from machining trials can help refine cutting parameters for specific applications.

2.21. CNC Programming

a. CNC Programming

computer numerical control programming, is developing code, or a program of directions, for a computer to operate CNC machines and tools. A CNC programmer studies a paper or digital model of a part and inputs a sequence of instructions on how to make it into a computer.

b. Manual CNC Programming

The manual approach is the most basic form of CNC programming. It involves manually inputting commands into a control console that is typically attached to the CNC machine. Manual CNC programming is a good option for simple parts. However, it can be a time-consuming and error-prone method of communicating instructions to the CNC machine. Because of this, manual CNC programming is not suitable for complex parts. In manual CNC programming, the programmer must have a good understanding of the CNC machine and how it works.

c. Computer-aided manufacturing (CAM)

CAM software is used to create CNC programs for both simple and complex parts. It is the most commonly used method of CNC programming. CAM software is faster and more accurate than manual CNC programming. It provides the programmer with a visual representation of the tool paths which helps them spot any potential issues such as the tool crashing into the workholding fixture. CAM software gives the user more flexibility to translate the program from one type of CNC machine to another without having to rewrite the entire program, as would have to be done when creating manual G-Code.

d. Conversational programming

Conversational programming involves using a CNC machine that has a built-in conversational programming interface. This interface allows the user to input commands in plain English instead of in G-code, as is the case with normal manual programming.

Self-Check Sheet 2: Create Program Using G & M Code

1. What is work offset
   Answer:

2. What is Canned cycle

   Answer:

3. What is the code in canned cycle of Bolt Hole Circle ?
   Answer:

4. Write down G70 - Bolt Hole Circle programming format?
   Answer:

5. What CNC code used in Clockwise Circular Pocket Milling:
   Answer:

6. What CNC code used in Counterclockwise Circular Pocket Milling:
   Answer:

7. What CNC code used for Engraving operation :
   Answer:

8. What is Sub-program
   Answer:

9. What is Looping programming?
   Answer:

10. What is Material Compatibility:
    Answer:

11. What is Tool Geometry:
    Answer:

12. What is Size and Tolerance:
    Answer:

Answer Sheet 2: Create Program Using G & M Code

1. **What is work offset?**
   Answer: When we write a CNC program we work from a datum on the drawing. All the X and Y figures will be measured from this datum. Once we put the component on the machine it needs to know where the part is. This is called the work offset.

2. **What is Canned cycle?**
   Answer: Canned cycles are predefined sequences of G-codes that automate common machining operations on CNC milling machines, simplifying programming and reducing the amount of code needed to perform repetitive tasks.

3. **What is the code in canned cycle of Bolt Hole Circle?**
   Answer: G70 is used to create a circular pattern of bolt holes. Parameters include the center coordinates of the circle, the number of holes, the diameter of the circle, and the angle between holes (optional).

4. **Write down G70 - Bolt Hole Circle** programming format?
   Answer: G70 I... J... L.

I	Radius of the bolt hole circle (– CW / + CCW).
J	Start angle of the first hole.
L	Number of bolt hole

5. **What CNC code used in clockwise Circular Pocket Milling?**
   Answer: G12 is used for clockwise circular pocket milling, where the cutting tool moves in a circular motion to remove material from inside a pocket or slot.

6. **What CNC code used in counter clockwise Circular Pocket Milling?**
   Answer: G13 is used for counter clockwise circular pocket milling, where the cutting tool moves in a counterclockwise motion to remove material from inside a pocket or slot.

7. **What CNC code used for engraving operation?**
   Answer: G47 is used for engraving operations, where the cutting tool follows a specified path to create engraved text, logos, or other designs on the workpiece.

8. **What is sub-program**
   Answer: In CNC milling, sub-programs and looping are techniques used to streamline programming, reduce code redundancy, and improve efficiency. M97, M98, and looping commands are commonly used for these purposes. Here's an explanation of each:

9. **What is looping** programming?
   Answer: Looping allows a section of code to be repeated multiple times within a CNC program. Looping commands include GOTO (branching), IF-THEN-ELSE (conditional branching), and DO (looping). Looping is often used for repetitive tasks, such as drilling multiple holes or cutting multiple features.

10. **What is material compatibility?**
    Answer: Select tools compatible with the material being machined, whether it's aluminum, steel, or composites.

Job Sheet 2.1: Identify Modal and Non-Modal Codes as Per Specification

Working Procedure

1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. Interpret CNC Code
4. Write modal and non-modal codes as per Instruction
5. After completed work Clene work area.

modal G-codes

Group 1 {**G00**,**G01**,**G02**,**G03**,**G33**,**G38**, **G73**, **G76**, **G80**,**G81**, **G82**, , **G88**,**G89**} motion – This is a group of G-codes for motion called current motion mode — one is always in effect.
Group 2 **G17**, **G18**, **G19**, plane selection
Group 3 {**G90**, **G91**} distance mode
Group 5 {**G93**, **G94**} feed rate mode
Group 6 {**G20**, **G21**} units
Group 7 {**G40**, **G41**, **G42**, cutter radius compensation
Group 8 {**G43**, tool length offset
Group 10 {**G98**, **G99**} return mode in canned cycles
Group 12 {**G54**, **G55**, **G56**, **G57**, **G58**, **G59**, coordinate system selection
Group 14 {**G96**, **G97**} spindle speed mode

Non-modal G-codes

Group 0 {**G04**, **G10**, **G28**, **G30**, **G53**, **G92**}

Specification Sheet 2.1: Identify Modal and Non-Modal Codes as Per Specification

Name of Job: Identify modal and non-modal codes as per specification

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to Identify modal and non-modal codes as per specification

Diagram / Design:

G00, **G01**,**G02**,**G03**, **G73**, **G76**, **G80**,**G81**, **G82**, **G88**,**G89 G17**, **G18**,
G19, **G90**, **G91**, **G20**, **G21 G40**, **G41**, **G42**, **G43**, , **G98**, **G99**,
G54, **G55**, **G56**, **G57**, **G58**, **G59**, **G04**, **G10**, **G28**, **G30**, **G53**, **G92**,

Required Personal protective equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	pair	01	
2	Mask	N 95	Nos	01	
3	Safety shoe	As required	pair	01	
4	Safety goggles	Standard	Nos	01	
5	Apron	Standard	Nos	01	

Required Materials

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	Paper	A4	Pcs	1	
2	Pen	Ball point	Pcs	1	

Job Sheet 2.2: Create Circular and Linear Interpolation as Per Specification

Working Procedure

1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. Ensure the CNC machine is properly set up and initialized.
4. Set the cutting tool at the starting point of the circular path and linear path.
5. Choose between clockwise (G02) or counterclockwise (G03) interpolation based on the desired direction of the circular path.
6. Select Linear Interpolation Mode
7. Express the center point of the circle using the X and Y coordinates.
8. Express the coordinates of the endpoint of the linear path (X, Y, and optionally Z coordinates).
9. Specify the radius (or diameter) of the circle.
10. Execute the G-code command for circular interpolation (G02 or G03) along with the parameters set in the previous step.
11. Ensure to end the program appropriately when the operation is complete,
12. Clean tools and equipment.
13. Store all equipment safely.

Program

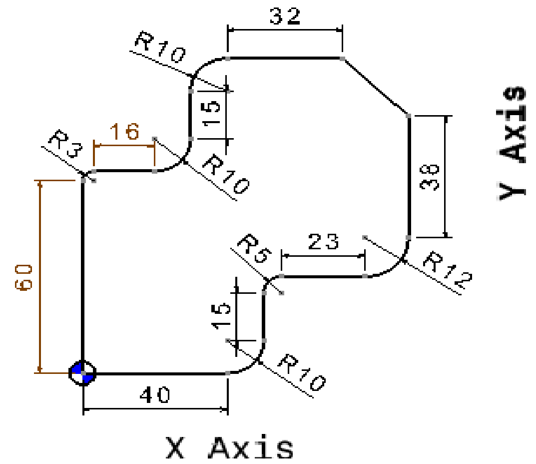
Sequence No	G Code	(X,Y) Value	Feed rate	EoB
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		
		(X,Y) =		

Specification Sheet 2.2: Create Circular and Linear Interpolation as Per Specification

Name of Job: Create circular and linear interpolation as per specification

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to create circular and linear interpolation as per specification

Diagram / Design:



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	pair	01	
2	Mask	N 95	Nos	01	
3	Safety shoe	As required	pair	01	
4	Safety goggles	Standard	Nos	01	
5	Apron	Standard	Nos	01	

Job Sheet 2.3: Select The Cutter Radius Compensation as Per Specification

Working Procedure

A. Collect and wear the PPE.
B. Read job sheet and specification sheet.
C. Ensure that the CNC machine is powered on and properly initialized.
D. Load the CNC program into the machine control unit (MCU).
E. Move the cutting tool to the starting position relative to the workpiece.
F. Activate cutter radius compensation mode most commonly used G-codes for CRC are G41 (left compensation) and G42 (right compensation).
G. Determine the tool offset value based on the radius of the cutting tool. This value represents the distance from the centerline of the tool to the cutting edge.
H. Set the tool offset using the appropriate G-code.
I. Ensure that the tool path is specified relative to the workpiece coordinates and takes into account the cutter radius compensation.
J. Run the CNC program on the machine.
K. Inspect the machined part to ensure that the dimensions match the intended design when machining is complete
L. Measure critical dimensions using precision tools to confirm accuracy.
M. Deactivate cutter radius compensation mode by issuing the appropriate G-code when complete the machining operation
N. Power off finish the machine operation following proper shutdown procedures.
O. Clean tools and equipment.
P. Store all equipment safely.

Program

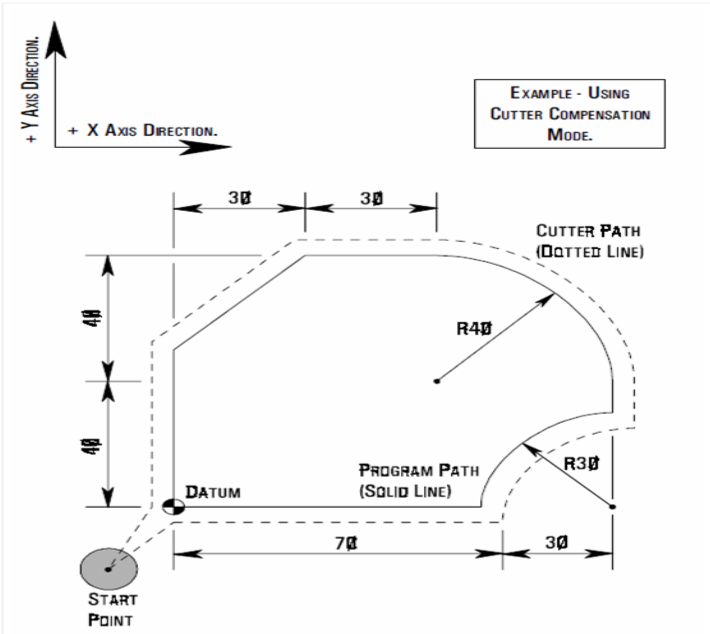
```
O 1234
N171 G00 X-15 Y-15 Z50;
N172 G01 Z0;
N172 G41 X0 Y0 F100; (Start-Up Move)
N173 Y40;
N174 X30 Y80;
N175 X60;
N176 G02 X100 Y40 R40;
N177 G01 Y30;
N178 G03 X70 Y0 R30;
N179 G01X0;
N180 G40 X-15 Y-15; (Cancellation Move)
N190 G00 Z50
```

Specification Sheet 2.3: Select The Cutter Radius Compensation as Per Specification

Name of Job: Select the cutter radius compensation as per specification

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to select the cutter radius compensation as per specification

Diagram / Design:



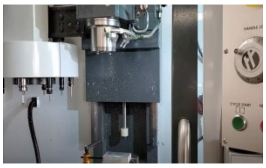
Required Personal protective equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1.	Hand Gloves	Cotton	pair	01	
2.	Mask	N 95	Nos	01	
3.	Safety shoe	As required	pair	01	
4.	Safety goggles	Standard	Nos	01	
5.	Apron	Standard	Nos	01	

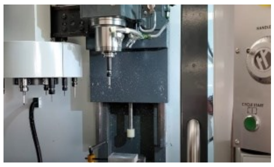
Job Sheet 2.4: Set Work Offset (G54) As Per Specification

Working Procedure

1. Follow OSH
2. Collect and wear the PPE.
3. Read job sheet and specification sheet
4. Prepare the Workpiece to the milling machine table using appropriate clamps or vises.
5. Clamp Edge Finder with tool holder in CNC Mill Chuck
6. Ensure that it's properly aligned and positioned according to your machining requirements.
7. Move the Machine to Reference Point on the workpiece


4


5


6


7

8. Use the machine's control panel or software interface to home or zero all axes (X, Y, and Z).
9. Select G54 Work Offset: In machine's control software,
10. select the appropriate work offset. G54 is commonly used as the default work offset for the first operation.
11. Bring the milling cutter to the reference point on the workpiece.

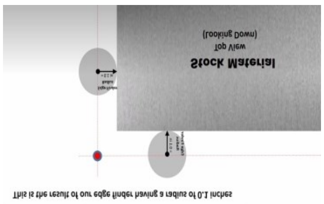

8


9


9


10

12. Use the jog controls to move the tool precisely to the desired location.
13. Set X, Y, and Z Coordinates: Once the tool is positioned at the reference point, set the X, Y, and Z coordinates to zero for the selected work offset (G54).
14. Verify Offset Settings, double-check that the X, Y, and Z coordinates are correctly set to zero for the G54 work offset.
15. Use the machine's display or software interface to confirm the offset values.


13

14


15

16. Save the Work Offset to the CNC machine's memory.
17. perform a test run to ensure that the tool paths and offsets are correctly set up.
18. Perform Home position
19. Clean work area and store tools and equipment

Specification Sheet 2.4: Set Work Offset (G54) as Per Specification

Name of Job: Set work offset (G54) as per specification

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to set the tool offset (G43) and work offset (G54) as per specification.

Diagram / Design:

Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1.	Hand Gloves	Cotton	pair	01	
2.	Mask	N 95	Nos	01	
3.	Safety shoe	As required	pair	01	
4.	Safety goggles	Standard	Nos	01	
5.	Apron	Standard	Nos	01	

Required Materials

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	CNC Machine		No	One	
2	Edge Finder		No	One	

Job Sheet 2.5: Create Program of Pocket Milling as Per Specification

Working Procedure

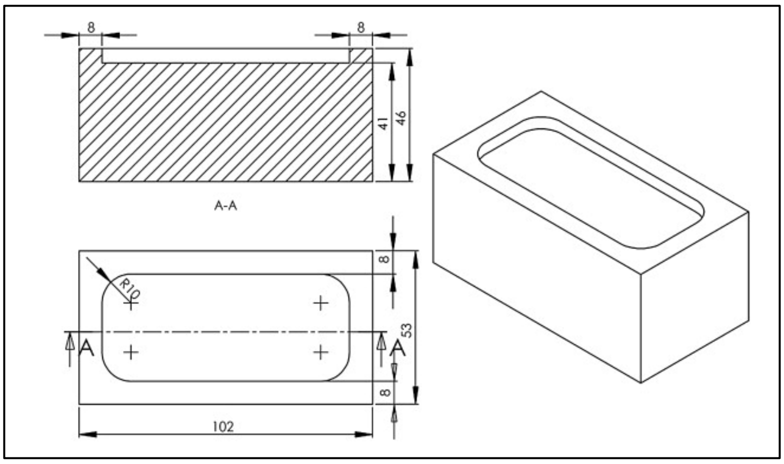
1. Follow OSH
2. Collect and wear the PPE.
3. Read job sheet and specification sheet
4. Clamp the workpiece to the milling machine table and ensure its properly aligned.
5. Choose an appropriate end mill cutter for pocket milling based on the material and desired cutting parameters.
6. Set work offset (g54) accurately.
7. Determine the dimensions and location of the pocket you want to mill on the workpiece.
8. Use cam (computer-aided manufacturing) software to generate the toolpaths for pocket milling.
9. Defining the pocket geometry, tool diameter, cutting depths, stepovers, and other parameters.
10. Generate g-code the toolpaths are calculated
11. Export the g-code file from the cam software.
12. Transfer the g-code file to the CNC machine's control system, either via usb, network, or other methods.
13. Input the appropriate cutting parameters such as spindle speed, feed rate, and cutting depth into the CNC machine's control panel.
14. Start the CNC machine and execute the g-code program for pocket milling.
15. Move the cutter along the programmed toolpaths, removing material to create the pocket.
16. Keep an eye on the milling process to ensure it proceeds smoothly, without any issues like tool breakage or excessive vibration.
17. Perform roughing out the pocket, you may choose to perform a finishing pass to achieve the desired surface finish.
18. Pocket milling is complete to inspect the workpiece to ensure it meets the required specifications.
19. Perform Machine Home position
20. Clean work area
21. Store tools and Equipment

Specification Sheet 2.5: Create Program of Pocket Milling as Per Specification

Name of Job: Program pocket milling as per specification

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to program pocket milling as per specification.

Diagram / Design:



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1.	Hand Gloves	Cotton	pair	01	
2.	Mask	N 95	Nos	01	
3.	Safety shoe	As required	pair	01	
4.	Safety goggles	Standard	Nos	01	
5.	Apron	Standard	Nos	01	

Required Materials

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	CNC Machine		No	01	
2	End Mill Cutter	10 mm	No	01	

Job Sheet 2.6: Create Program of Contour Milling as per Specification

Name of Job: Create Program of contour milling as per specification

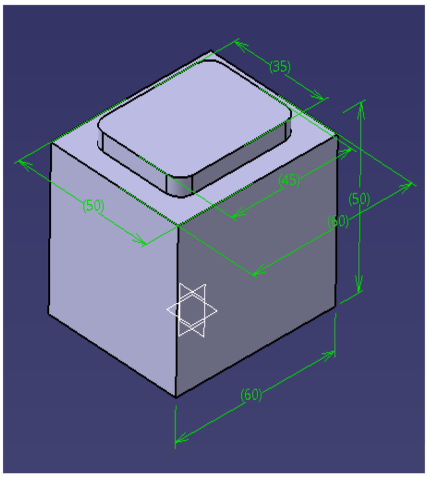
Working Procedure

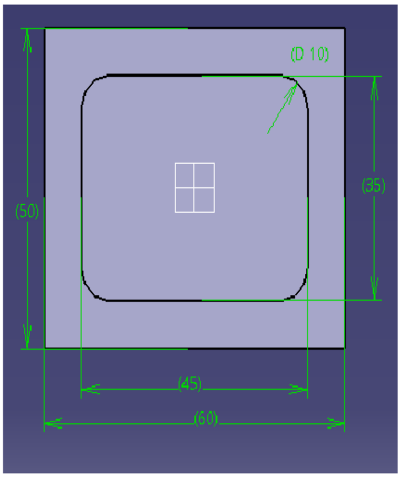
1. Follow OSH
2. Collect and wear the PPE.
3. Read job sheet and specification sheet
4. Prepare the Workpiece to the milling machine table using appropriate clamps or vises.
5. Clamp the workpiece to the milling machine table and ensure proper alignment.
6. Choose an appropriate end mill cutter for contour milling based on the geometry and material of the workpiece.
7. Set work offset (g54) accurately.
8. Determine the contours or profiles you want to mill on the workpiece.
9. Use cam software to generate the toolpaths for contour milling.
10. Define the contour geometry, tool diameter, cutting depths, stepovers, and other parameters.
11. Generate g-code and export the g-code file from the cam software based on the calculated toolpaths.
12. Load g-code or transfer the g-code file to the CNC machine's control system.
13. Set cutting parameters into the CNC machine's control panel.
14. Start the CNC machine and execute the g-code program for contour milling.
15. Move the cutter along the programmed toolpaths to mill the contours.
16. Monitor the milling process to ensure smooth operation and adjust parameters if necessary.
17. Perform a finishing pass to achieve the desired surface finish on the contoured features.
18. Perform contour milling is complete, inspect the workpiece to ensure it meets the required specifications.
19. Perform Machine Home position
20. Clean work area and store tools and equipment

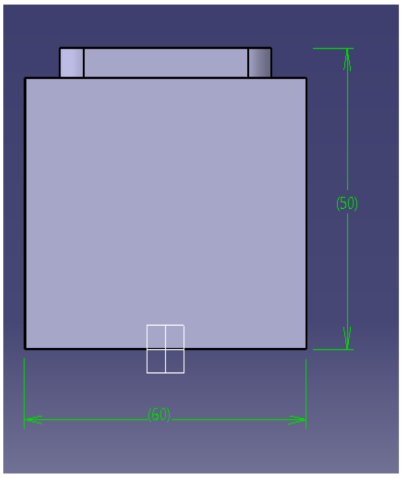
Specification Sheet 2.6: Create Program of Contour Milling as Per Specification

Name of Job: Create Program of contour milling as per specification

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to program contour milling as per specification.







Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1.	Hand Gloves	Cotton	pair	01	
2.	Mask	N 95	Nos	01	
3.	Safety shoe	As required	pair	01	
4.	Safety goggles	Standard	Nos	01	
5.	Apron	Standard	Nos	01	

Required Materials

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	CNC Machine		No	01	
2	End Mill Cutter	10 mm	No	01	

Learning Outcome -3: Set up Machine and Workpiece

Assessment Criteria	1. Work holding mechanism is selected 2. Required cutting tools are selected and loaded 3. **Datum** is defined 4. Entry and exit points are selected
Conditions and Resources	1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Laptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary tools and accessories 11. Necessary Materials 12. Necessary PPE
Contents	1. Work holding mechanism 2. Necessary cutting tools 3. Datum ▪ Main point ▪ Zero point 4. Entry and exit points 5. Program simulation, editing, downloading and saving methods 6. Basic file maintenance procedures
Job/Task/Activity	1. Select the work holding mechanism 2. Select and load the required cutting tool 3. Select entry and exit points
Training Method	1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming
Assessment Method	1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 3: Set up Machine and Workpiece

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials '**Create and Use G&M Code for Machining**
2. Read the Information sheet and complete the Self Checks & Check answer sheets on "**Set up machine and workpiece**"	2. Information sheet 3: Set up machine and workpiece 3. Self-check 3: Set up machine and workpiece 4. Check your answer with Answer key 3: Set up machine and workpiece
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet ▪ Job Sheet 3.1: Identify and select the work holding mechanism Specification Sheet 3.1 Identify and Select the work holding mechanism ▪ Job Sheet 3.2: Select and load the required cutting tool as per specification Specification Sheet 3.2: Select and load the required cutting tool as per specification

Information Sheet 3: Set Up Machine and Workpiece

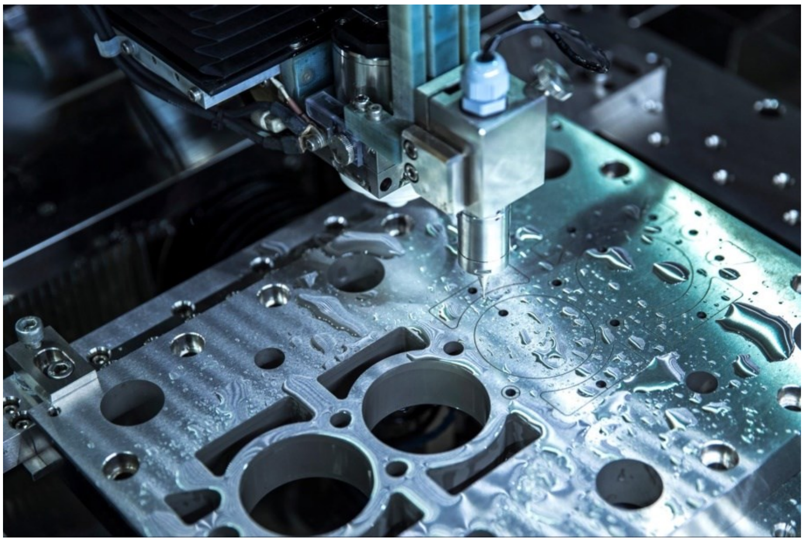
Learning Objective

After completion of this information sheet, the learners will be able to explain, define and interpret the following contents:

3.1 Work holding mechanism
3.2 Necessary cutting tools
3.3 Datum
3.4 Entry and exit points
3.5 Program simulation, editing, downloading and saving methods
3.6 Basic file maintenance procedures

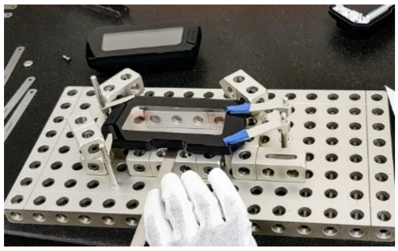
3.1 Work Holding Mechanism

In CNC milling machines, work holding mechanisms are essential components that securely hold the workpiece in place during machining operations. These mechanisms ensure that the workpiece remains stable and accurately positioned to achieve precise cuts and machining processes. There are several common types of work holding mechanisms used in CNC milling machines:



Another important consideration is where your fixturing method is going to hold on to your part. Is it a standard block with parallel edges that can be placed in a vice? Do you need to have multiple sides reachable by the tool in order to avoid an extra operation? Is your part a funky shape that requires a creative method to fixture it in your machine? These are all good questions to ask yourself when choosing a CNC work holding device.

Finally, you want to consider how many parts you will need to make with your fixture. If it is going to be a high-volume operation, it may be worth it to invest some more time and effort to create a custom fixture that can hold many parts at once. If you're only making a few parts, you may want to stick with one of the other fixturing options we'll get into below.

Vise: A vise is one of the most common works holding devices used in CNC milling. It typically consists of two jaws that can be tightened against the workpiece to hold it securely. Vises come in various sizes and configurations to accommodate different workpiece shapes and sizes. They are versatile and suitable for a wide range of milling operations.	
Fixture plate: Fixture plates are flat plates with a grid of holes or slots that allow for the attachment of clamps, bolts, or other work holding accessories. Workpieces are secured to the fixture plate using clamps or bolts, providing a stable mounting surface for machining.	
Clamps and bolts: Clamps and bolts are commonly used to secure workpieces directly to the milling machine's table or fixture plate. They provide a flexible and customizable means of holding workpieces of various shapes and sizes. Clamps can be manual or hydraulic and are tightened to securely hold the workpiece in place during machining.	
Vacuum chuck: Vacuum chucks use suction to hold the workpiece in place on the machine table. They are particularly useful for machining thin or irregularly shaped workpieces that may be difficult to clamp with traditional methods. Vacuum chucks provide uniform holding pressure across the entire surface of the workpiece, minimizing distortion and deformation during machining.	
Magnetic chuck: Magnetic chucks use magnets to hold ferrous workpieces securely in place on the machine table. They provide strong, uniform holding force and are suitable for machining ferrous materials. Magnetic chucks are often used in applications where clamping or bolting may not be practical or when rapid workpiece changeover is required.	
Collet chuck: Collet chucks use collets to grip and hold cylindrical workpieces, such as round stock or drill bits. Collet chucks provide concentric clamping, ensuring accurate positioning of the workpiece relative to the cutting tool. They are commonly used in CNC milling machines for machining small, cylindrical parts with high precision.	

These are just a few examples of work holding mechanisms used in CNC milling machines. The choice of work holding device depends on factors such as the size and shape of the workpiece, the material being machined, and the specific requirements of the machining operation. Proper selection and setup of work holding mechanisms are essential for achieving accurate and efficient CNC milling processes.

3.2 Necessary Cutting Tools

In CNC milling machines, a variety of cutting tools are used to perform different types of machining operations, each serving a specific purpose. Here are some of the most common types of cutting tools used in CNC milling:

End mills: End mills are versatile cutting tools with a cutting edge at the tip and on the sides. They are used for various milling operations such as facing, profiling, slotting, and contouring. End mills come in different types, including square end mills, ball end mills, corner radius end mills, and tapered end mills, each suited for specific applications.	
Face mills: Face mills have cutting edges on the periphery and the face of the cutter. They are primarily used for facing large flat surfaces and producing square shoulders. Face mills are available in different sizes and configurations to accommodate various machining requirements.	
Drills: Drills are used for creating holes in the workpiece. CNC milling machines can use different types of drills, including twist drills, center drills, and spot drills, depending on the specific hole geometry and machining requirements.	
Reamers: Reamers are precision cutting tools used to improve the accuracy and finish of existing holes. They are typically used after drilling operations to achieve tight tolerances and smooth surfaces.	
Milling cutters: Milling cutters are specialized tools used for specific milling operations. They include slab mills, side and face cutters, T-slot cutters, and dovetail cutters, among others. Each type of milling cutter is designed for a particular machining task, such as cutting slots, grooves, or keyways.	
Indexable inserts: Indexable inserts are replaceable cutting inserts that are mounted on milling cutters, drills, and other cutting tools. They are designed for high-speed machining and offer cost-effective tooling solutions. Indexable inserts come in various shapes, sizes, and materials to suit different machining applications.	
Milling inserts: Similar to indexable inserts, milling inserts are replaceable cutting inserts used in milling operations. They are typically mounted on face mills, end mills, and other milling cutters to perform specific cutting tasks such as facing, profiling, and slotting.	

Specialty tools: In addition to standard cutting tools, CNC milling machines may use specialty tools for specific machining tasks. These include chamfer mills, thread mills, countersinks, and engraving tools, among others. Specialty tools are used to create complex features, threads, and surface finishes on the workpiece.	

These are just some examples of the cutting tools commonly used in CNC milling machines. The selection of cutting tools depends on factors such as the material being machined, the desired surface finish, tolerances, and the specific machining operation being performed. Proper tool selection and toolpath programming are essential for achieving efficient and accurate CNC milling processes.

3.3 Datum

In the context of CNC milling machines, a "datum" typically refers to a reference point or surface from which all measurements and machining operations are based. It serves as a fundamental point of origin or orientation for the machining process. Let's break down the terms:

a. Main point

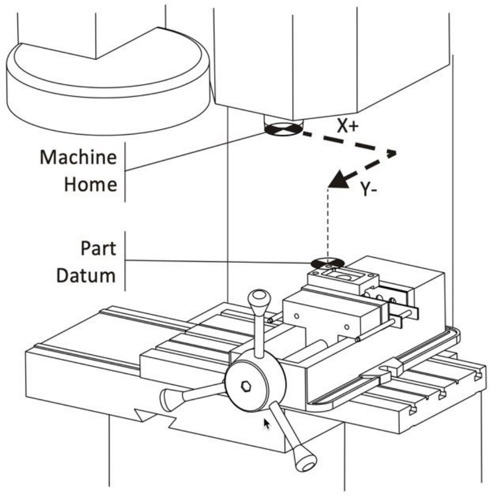
The main point or datum in a CNC milling machine is the primary reference point used to establish the coordinate system for machining operations. It's often referred to as the Workpiece Zero or Part Zero. This point is typically defined by the designer, programmer, or machinist based on the specific requirements of the workpiece being machined. The main point could be located at a specific feature or surface of the workpiece, such as a corner, edge, or center.

Once the main point is established, the CNC machine's controller uses it as the reference for positioning the cutting tool and executing the programmed toolpaths relative to the workpiece. All dimensions and movements are measured from this main point, allowing for precise and accurate machining operations.

b. Zero point in CNC milling machine

The zero point in a CNC milling machine, often referred to as the Machine Zero or Absolute Zero, is the reference point within the machine's coordinate system. It's typically located at a predefined position within the machine's workspace, such as the extreme ends of each axis or a specific corner of the machine table.

When the CNC machine is powered on or initialized, it moves to the zero point to establish its reference position. The zero point serves as the starting reference for all machine movements and tool positioning. It's essential for ensuring consistent and repeatable machining operations across different workpieces and setups.



The zero point in the CNC machine's coordinate system may coincide with the main point or datum of the workpiece, but this isn't always the case. Proper setup and alignment are necessary to ensure that the main point of the workpiece aligns with the zero point of the machine, allowing for accurate machining relative to the workpiece's features and dimensions.

In summary, the main point or datum in CNC milling machines is the primary reference point on the workpiece, while the zero point is the reference point within the machine's coordinate system. Both are crucial for establishing the coordinate system, positioning the cutting tool, and executing precise machining operations.

3.4 Entry and Exit Points

In CNC milling operations, entry and exit points refer to the locations where the cutting tool begins and ends its engagement with the workpiece. Proper selection and positioning of entry and exit points are essential for achieving desired machining results, minimizing tool wear, and preventing surface defects. Here's a breakdown of entry and exit points in CNC milling:

a. Entry point:

- The entry point is the location where the cutting tool first makes contact with the workpiece to begin the machining operation.
- It's strategically chosen to minimize tool wear, reduce cutting forces, and avoid damage to the workpiece surface.
- The selection of the entry point depends on factors such as the geometry of the workpiece, the cutting tool's characteristics, and the desired surface finish.
- Common entry points include corners, edges, or other features of the workpiece where tool engagement can be initiated smoothly and efficiently.
- Entry points are typically programmed into the CNC milling machine's toolpath using CAM (Computer-Aided Manufacturing) software, ensuring precise tool positioning and motion control.

b. Exit point:

- The exit point is the location where the cutting tool disengages from the workpiece at the end of the machining operation.

- It's chosen to minimize tool retraction distance, reduce cycle time, and prevent surface defects such as burrs or marks.
- Similar to entry points, exit points are strategically determined based on the workpiece geometry and machining requirements.
- Exit points are programmed into the toolpath to ensure smooth tool disengagement and minimize any potential damage to the workpiece surface.
- Common exit points include areas where tool disengagement can occur without leaving visible marks or causing abrupt changes in surface finish.

Proper selection of entry and exit points is crucial for optimizing machining efficiency and achieving high-quality results in CNC milling operations. Machinists and programmers often consider factors such as toolpath optimization, tool engagement strategies, and workpiece geometry to determine the most suitable entry and exit points for each machining operation. Additionally, CAM software provides tools and functionalities to facilitate the efficient programming of entry and exit points within the toolpath.

3.5 Program Simulation, Editing, Downloading and Saving Methods

Simulating, editing, downloading, and saving CNC milling machine programs are essential tasks in CNC machining operations. Here's a breakdown of methods commonly used for each of these tasks:

A. Program simulation

Program simulation is the process of digitally simulating the execution of CNC milling machine programs to verify toolpaths, detect errors, and optimize machining operations. There are various software tools available for program simulation, including:

a. CAM Software: Many Computer-Aided Manufacturing (CAM) software packages include built-in simulation capabilities. These tools allow users to import CNC program files, visualize toolpaths, simulate material removal, and analyze machining operations in a virtual environment.
b. Dedicated Simulation Software: There are standalone simulation software programs specifically designed for simulating CNC machining operations. These programs often provide advanced features such as collision detection, machine kinematics simulation, and detailed analysis of cutting forces and tool wear.
c. Machine Tool Simulation: Some CNC milling machines are equipped with built-in simulation features that allow users to simulate toolpaths directly on the machine's control panel. This provides real-time feedback and verification of machining operations before actual cutting begins

B. Editing programs

Editing CNC milling machine programs may involve modifying toolpaths, adjusting cutting parameters, or making other changes to optimize machining operations. Common methods for editing CNC programs include:

a. Text-Based Editors: CNC programs are typically stored as text files containing G-code or other machine-specific programming languages. Text-based editors, such as Notepad or specialized CNC code editors, can be used to manually edit program files.
b. CAM Software: CAM software packages often include editing capabilities that allow users to modify toolpaths, change cutting parameters, add or remove machining operations, and make other adjustments to CNC programs.
c. Machine Control Panel: Some CNC milling machines provide basic editing functionality directly on the machine's control panel. Users can access and modify program files stored in the machine's memory using built-in editing features.

C. Downloading programs

Downloading CNC milling machine programs involves transferring program files from a computer or storage device to the CNC machine's control system. Common methods for downloading programs include:

a. USB Transfer: Many modern CNC milling machines support USB connectivity, allowing users to transfer program files directly from a USB flash drive to the machine's control system.
b. Ethernet or Network Transfer: CNC machines equipped with Ethernet or network connectivity can transfer program files over a local network from a computer or network storage device.
c. RS-232 Serial Transfer: Older CNC machines may use RS-232 serial communication to transfer program files from a computer to the machine's control system using a serial cable connection.

D. Saving programs

Saving CNC milling machine programs involves storing program files for future use or archiving. Common methods for saving programs include:

a. Local Storage: CNC program files can be saved to the internal memory or storage device of the CNC milling machine for easy access and retrieval.
b. External Storage: Program files can also be saved to external storage devices such as USB flash drives, SD cards, or network storage devices for backup or transfer to other machines.
c. Cloud Storage: Some CNC machining operations may choose to save program files to cloud-based storage services for remote access, collaboration, and backup purposes.

These methods for simulating, editing, downloading, and saving CNC milling machine programs provide users with flexibility, efficiency, and control over machining operations. The specific methods used may vary depending on the CNC machine's capabilities, software tools available, and user preferences.

3.6 Basic File Maintenance Procedures

Basic file maintenance procedures in CNC milling machines involve managing program files, tool libraries, and machine settings to ensure smooth operation and organization. Here are some essential file maintenance procedures:

A. Organizing program files

a. Create a systematic directory structure for storing CNC program files. Organize files by project, part number, or job type to facilitate easy retrieval and organization.
b. Use descriptive filenames that indicate the content, revision, and purpose of each program file. Avoid generic filenames or ambiguous naming conventions.
c. Regularly review and clean up obsolete or redundant program files to avoid clutter and confusion.

B. Backing up program files

a. Implement a regular backup schedule to safeguard critical program files against loss or corruption. Back up program files to external storage devices, network drives, or cloud storage services.
b. Store backup copies of program files in secure locations to prevent data loss due to hardware failure, software errors, or other unforeseen circumstances.

C. Version control

a. Establish a version control system to track revisions and changes to CNC program files. Use version numbers or revision dates to differentiate between different iterations of the same program.
b. Maintain a log or documentation that records the history of changes, including modifications made, reasons for revisions, and individuals responsible for updates.

D. Managing tool libraries

a. Maintain an up-to-date tool library that contains information about cutting tools, toolholders, and related parameters used in CNC milling operations.
b. Regularly review and update tool library entries to reflect changes in tool inventory, tool wear, or machining requirements.
c. Ensure consistency and accuracy in tool library entries to prevent errors and discrepancies during tool selection and toolpath generation.

E. Calibration and machine settings:

a. Periodically calibrate CNC milling machine axes, spindle speeds, tool offsets, and other machine settings to ensure accuracy and repeatability in machining operations.
b. Keep detailed records of machine calibration procedures, including calibration dates, measurements, adjustments made, and calibration results.
c. Perform preventive maintenance tasks and routine checks to identify and address any issues or abnormalities in machine performance promptly.

F. Security measures

a. Implement access controls and user permissions to restrict unauthorized access to CNC milling machine settings, program files, and sensitive data.
b. Train personnel on proper file management procedures, security protocols, and data protection practices to prevent accidental or intentional data loss or misuse.

By following these basic file maintenance procedures, CNC milling machine operators and programmers can ensure efficient operation, data integrity, and compliance with quality standards.

Self-Check sheet 3: Set up Machine and Workpiece

1. What is Work holding mechanism
   Answer:

2. Why use Fixture Plate?
   Answer:

3. Why use Clamps and Bolts:
   Answer:

4. Why use Vacuum Chuck:
   Answer:

5. Why use Magnetic Chuck:
   Answer:

6. Why use Collet Chuck:
   Answer:

7. What is Zero Point in CNC Milling Machine:
   Answer:

8. What is Program Simulation:
   Answer:

9. What types of method to use transfer Programs in CNC machine
   Answer:

Answer Sheet 3: Set up Machine and Workpiece

1. **What is Work holding mechanism**
   Answer: In CNC milling machines, work holding mechanisms are essential components that securely hold the workpiece in place during machining operations. These mechanisms ensure that the workpiece remains stable and accurately positioned to achieve precise cuts and machining processes.
2. **Why use Fixture Plate?**
   Answer: Fixture plates are flat plates with a grid of holes or slots that allow for the attachment of clamps, bolts, or other work holding accessories. Workpieces are secured to the fixture plate using clamps or bolts, providing a stable mounting surface for machining.
3. **Why use Clamps and Bolts?**
   Answer: Clamps and bolts are commonly used to secure workpieces directly to the milling machine's table or fixture plate. They provide a flexible and customizable means of holding workpieces of various shapes and sizes.
4. **Why use Vacuum Chuck?**
   Answer: Vacuum chucks use suction to hold the workpiece in place on the machine table. They are particularly useful for machining thin or irregularly shaped workpieces that may be difficult to clamp with traditional methods.
5. **Why use Magnetic Chuck?**
   Answer: Magnetic chucks use magnets to hold ferrous workpieces securely in place on the machine table. They provide strong, uniform holding force and are suitable for machining ferrous materials.
6. **Why use Collet Chuck?**
   Answer: Collet chucks use collets to grip and hold cylindrical workpieces, such as round stock or drill bits. Collet chucks provide concentric clamping, ensuring accurate positioning of the workpiece relative to the cutting tool.
7. **What is Zero Point in CNC Milling Machine?**
   Answer: The zero point in a CNC milling machine, often referred to as the Machine Zero or Absolute Zero, is the reference point within the machine's coordinate system. It's typically located at a predefined position within the machine's workspace, such as the extreme ends of each axis or a specific corner of the machine table.
8. **What is Program Simulation?**
   Answer: Program simulation is the process of digitally simulating the execution of CNC milling machine programs to verify toolpaths, detect errors, and optimize machining operations.
9. **What types of method to use transfer Programs in CNC machine?**
   Answer: Common methods for downloading programs include
   USB Transfer: Many modern CNC milling machines support USB connectivity, allowing users to transfer program files directly from a USB flash drive to the machine's control system. **Ethernet or Network Transfer**: CNC machines equipped with Ethernet or network connectivity can transfer program files over a local network from a computer or network storage device. **RS-232 Serial Transfer:** Older CNC machines may use RS-232 serial

Job Sheet 3.1: Identify and Select the Work Holding Mechanism

Working Procedure:

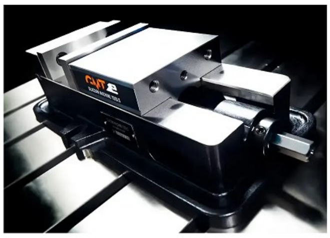
1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. Understand the dimensions, material, and geometry of the workpiece.
4. Determine the maximum workpiece size and weight the machine can accommodate.
5. Identify Suitable Work Holding Options: Based on the workpiece specifications and machine capabilities, identify potential work holding mechanisms that are suitable for the job.
6. Choose the most appropriate work holding mechanism based on the operation.
7. Ensure the setup required for the selected work holding mechanism.
8. Check for proper fitment and any necessary adapters or accessories required.
9. Verify that the workpiece is securely held and properly aligned for machining, and make any necessary adjustments.
10. Select work holding mechanism, fixture positions, and any special considerations.
11. Clean tools and equipment.
12. Store all equipment safely.

Identification No.	Holding device Name
1.	Vise
2.	Step Clamps
3.	Angle Plate Figure
4.	Magnetic Table
5.	Dovetail Fixture Mostly used on %-Axis part machining
6.	Dovetail Fixture Mostly used on %-Axis part machining
7.	Portable vacuum table for holding plastic parts in our CNC machine.
8.	Using a Square chuck is an easy way to hold round parts on a VMC
9.	Custom work holder(jig) for machining a brake caliper part.

Specification Sheet 3.1 Identify and Select the Work Holding Mechanism

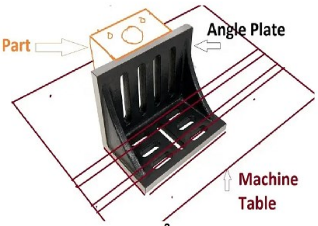
Name of Job: Identify and Select the work holding mechanism

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to **Identify and Select the work holding mechanism Diagram / Design:**



1

2



3

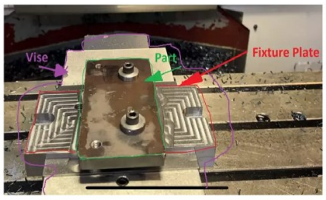
4

5

6

7

8



9

Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	pair	01	
2	Mask	N 95	Nos	01	
3	Safety shoe	As required	pair	01	
4	Safety goggles	Standard	Nos	01	
5	Apron	Standard	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1.	CNC Milling Machine	As required	No	01	
2.	Vise	As required	No	01	
3.	Step Clamps	As required	No	01	
4.	Angle Plate Figure	As required	No	01	
5.	Magnetic Table	As required	No	01	
6.	Dovetail Fixture	As required	No	01	
7.	Portable vacuum	As required	No	01	
8.	Square chuck	As required	No	01	
9.	work holder(jig)	As required	No	01	

Job Sheet 3.2: Select and Load The Required Cutting Tool as Per Specification

Job Name: Select and load the required cutting tool as per specification

Working Procedure

1. Collect and wear the PPE.
2. Understand the specifications of the machining job including the material to be machined, required surface finish, tolerances, and machining operations to be performed.
3. Determine the type and specifications of the cutting tool needed for the job based on the machining requirements. Consider factors such as: Tool material, Tool geometry, Tool size, and Tool holder type.
4. Select the appropriate cutting tool from the tool crib or storage area based on the identified requirements.
5. Ensure that the tool is in good condition, free from damage or excessive wear.
6. Check the cutting edges for sharpness and integrity.
7. Loosen the tool holder's clamping mechanism.
8. Insert the cutting tool into the holder, ensuring proper alignment.
9. Tighten the clamping mechanism securely to hold the tool in place. Use a torque wrench if specified by the manufacturer to ensure proper tightening torque.
10. Insert the assembled tool holder with the cutting tool into the CNC milling machine's spindle.
11. Ensure that the tool holder is seated properly and securely in the spindle taper.
12. Use the spindle lock or drawbar mechanism (if applicable) to secure the tool holder in the spindle.
13. Measure the tool length using a tool presenter or height gauge to establish the tool's length offset for accurate machining.
14. Record details of the selected cutting tool, including its specifications, tool number, and any length offsets used.
15. Clean tools and equipment.
16. Store all equipment safely.

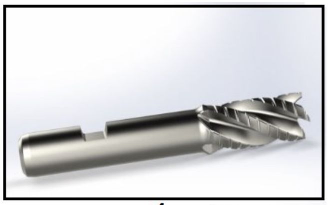
Cutter No	Cutter Name
1.	Roughing End Mill
2.	End Mill
3.	Drill Bits
4.	Face Mill
5.	Face Mill
6.	Reamers
7.	Thread Mill
8.	Slab Mill
9.	Gear Cutters

Specification Sheet 3.2: Select and Load The Required Cutting Tool as Per Specification

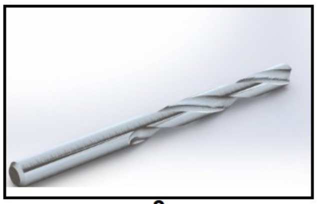
Name of Job: Select and load the required cutting tool as per specification

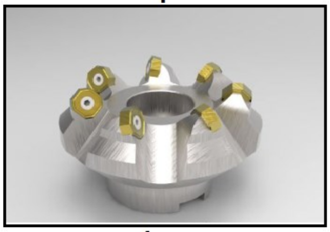
Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to select and load the required cutting tool as per specification.

Diagram / Design:


1


2

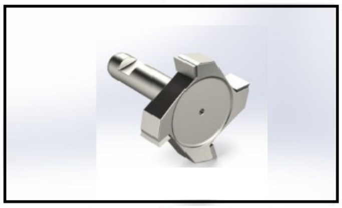

3

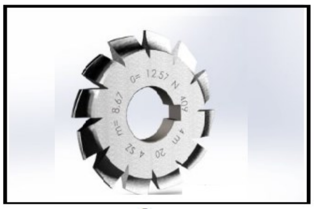

4


5


6


7


8


9

Required Personal Protective Equipment (PPE): Pre/Person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	pair	01	
2	Mask	N 95	Nos	01	
3	Safety shoe	As required	pair	01	
4	Safety goggles	Standard	Nos	01	
5	Apron	Standard	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	CNC Milling Machine		No	01	

Learning Outcome -4: Perform 2D Machining Operation

Assessment Criteria	1. Sequence of cutting operations is selected 2. 2D operations are performed 3. Canned cycles structures are identified 4. Sub-program structures are identified 5. Pocket milling program structures are identified 6. Canned cycle tricks with subprogram are executed
Conditions and Resources	1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Laptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary Tools and Equipment 11. Necessary Materials 12. Necessary PPE
Contents	1. Continuity of cutting operations 2. 2D operation ▪ Facing ▪ Contour ▪ Pocket ▪ Drilling ▪ To carve 3. Canned cycle structure 4. Subprogram structure 5. Pocket milling program structure 6. Canned cycle techniques with subprograms 7. CNC milling machine cleaning and maintenance procedures 8. Procedures for managing data protection.
Job/Task/Activity	1. Select the continuation of the cutting operation 2. Complete the 2D operation 3. Identify the canned cycle structure 4. Identify the sub-program structure 5. Identify the pocket milling program structure 6. Execute canned cycle techniques with subprograms

Training Method	1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming
Assessment Method	1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 4: Perform 2D Machining Operation

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials 'Create and Use G&M Code for Machining"
2. Read the Information sheet and complete the Self Checks & Check answer sheets on "**Perform 2D machining operation**"	2. Information sheet 4: Perform 2D machining operation 3. Self-check 4: Perform 2D machining operation 4. Check your answer with Answer key 4: Perform 2D machining operation
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet Job Sheet 4.1: Perform 2D Drilling operation as per specification. Specification sheet 4.1: Perform 2D Drilling operation as per specification. Job sheet 4 .2 Input and write a part program of Drilling cycle (Canned Cycle) using G81 as per specification. Specification sheet 4 .2 Input and write a part program drilling cycle (Canned Cycle) using G81 as per specification. Job Sheet 4.3: Perform sub-programing as per specification Specification sheet 4.3: Perform sub-programing as per specification Job sheet 4.4 Execute a Canned cycle part program as per specification Specification sheet4.4 Execute a Canned cycle part program as per specification

Information Sheet 4: Perform 2D Machining Operation

Learning Objective

After completion of this information sheet, the learners will be able to explain, define and interpret the following contents:

4.1. Continuity of cutting operations
4.2. 2D operation
4.3. Canned cycle structure
4.4. Subprogram structure
4.5. Pocket milling program structure
4.6. Canned cycle techniques with subprograms
4.7. CNC milling machine cleaning and maintenance procedures
4.8. Procedures for managing data protection

4.1. Continuity Of Cutting Operations

Continuity of cutting operations on a CNC milling machine refers to the smooth and uninterrupted execution of machining processes to achieve the desired outcomes efficiently and accurately. Here are several key aspects that contribute to maintaining continuity in cutting operations:

a. Optimized toolpaths

Utilize Computer-Aided Manufacturing (CAM) software to generate optimized toolpaths that minimize tool retractions, reduce air cutting, and avoid unnecessary rapid movements. Efficient toolpath generation ensures smoother transitions between machining operations, minimizing cycle times and improving productivity.

b. Proper workpiece fixturing

Securely fixture the workpiece to the machine table to prevent movement or vibration during cutting operations. Proper workpiece fixturing helps maintain dimensional accuracy and surface finish while reducing the risk of tool chatter or workpiece deformation.

c. Tool management

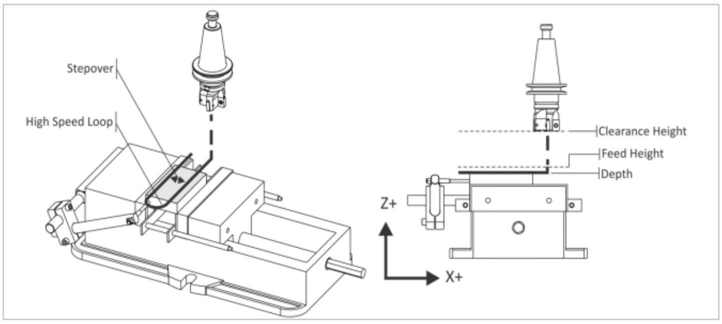
Implement effective tool management practices to ensure the availability of sharp and properly calibrated cutting tools throughout the machining process. Regular tool inspection, tool life monitoring, and tool replacement minimize tool wear and prevent premature tool failure, ensuring continuity in cutting operations.

d. Machine maintenance

Conduct regular preventive maintenance on the CNC milling machine to keep it in optimal condition. This includes lubrication of moving parts, calibration of machine axes, inspection of spindle and toolholder runout, and cleaning of coolant systems. Well-maintained machines operate more reliably, reducing the risk of unplanned downtime and ensuring continuous cutting operations.

e. Chip evacuation

Ensure effective chip evacuation to prevent chip buildup on cutting tools, workpiece surfaces, and within the machining area. Proper chip evacuation mechanisms, such as through-spindle coolant, chip conveyors, or air blast systems, help maintain cutting performance and surface quality by removing chips promptly and preventing chip recutting.



f. Monitoring and feedback

Implement real-time monitoring systems to track cutting parameters, tool condition, and machine performance during machining operations. Monitoring systems provide feedback on tool wear, spindle load, surface finish, and other key indicators, allowing operators to make adjustments as needed to maintain continuity and quality in cutting operations.

g. Operator training and skill

Ensure that machine operators are adequately trained and skilled in CNC machining techniques, tool setup, program loading, and troubleshooting. Well-trained operators can identify issues quickly, respond to alarms or warnings, and take corrective actions to keep cutting operations running smoothly.

By addressing these factors, CNC milling machine operators and programmers can maintain continuity in cutting operations, maximize productivity, and achieve high-quality machining outcomes. Continual improvement in process optimization, tool management, and machine maintenance contributes to sustained performance and efficiency in CNC milling operations.

4.2. 2D Operation

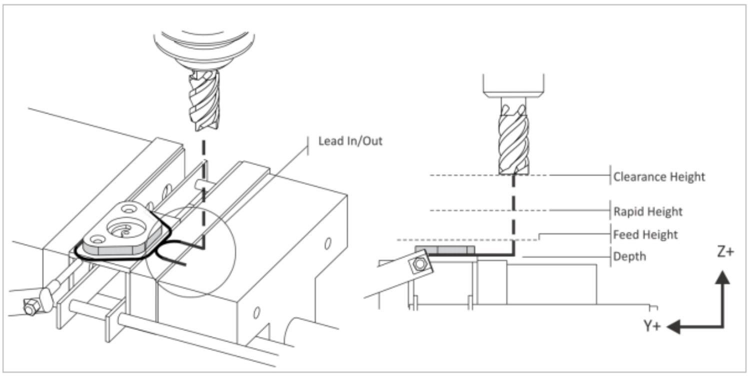
Here's a brief explanation of each of the 2D machining operations you mentioned in the context of CNC milling machines:

a. Facing

Facing is a machining operation where the CNC milling machine removes material from the surface of a workpiece to create a flat and even surface. It involves cutting across the entire surface of the workpiece using a face mill or end mill, typically to remove any excess material, achieve a specific thickness, or prepare the surface for subsequent machining operations.

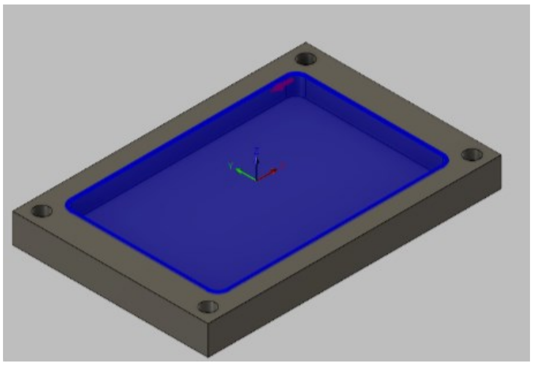
b. Contouring

Contouring involves cutting along the outline or perimeter of a 2D profile on the workpiece. It follows a specified path or contour defined by the CAD/CAM software, allowing the CNC machine to accurately shape the workpiece according to the desired design or geometry. Contouring operations are commonly used for profiling, engraving, or creating complex shapes with precise dimensions.



c. **Pocketing**

Pocketing is a machining operation where the CNC milling machine removes material from within a defined area or pocket on the workpiece. It involves cutting out the interior region of the workpiece to a specified depth, creating a cavity or recess with flat or contoured walls. Pocketing operations are used for creating slots, holes, or recesses in the workpiece, such as pockets for fasteners or clearance areas for other components.



d. **Drilling**

Drilling is a machining operation where the CNC milling machine creates holes in the workpiece using a drill bit or specialized drilling tool. It involves rotating the drill bit and applying axial force to penetrate the workpiece material and form a hole of the desired diameter and depth. Drilling operations are essential for creating holes for fasteners, dowel pins, or other components in the workpiece.

These 2D machining operations are fundamental to CNC milling and are widely used across various industries for manufacturing a wide range of parts, components, and products with precision and efficiency. The specific parameters, tooling, and strategies employed for each operation may vary depending on the material, geometry, and requirements of the workpiece.

4.3. Canned Cycle Structure

Canned cycles in CNC milling refer to pre-programmed sequences of operations that automate common machining tasks. They are designed to simplify programming and improve efficiency by reducing the amount of code needed to perform certain operations. The structure of a canned cycle typically consists of several components:

a. **G-Code command**: Canned cycles are initiated using specific G-codes (predefined G-codes). For example:
    - G81: Drilling cycle
    - G82: Drilling cycle with dwell
    - G83: Peck drilling cycle
    - G84: Tapping cycle
    - G85: Boring cycle
    - G86: Boring cycle with dwell
    - G87: Back boring cycle
    - G88: Reaming cycle
    - G89: Reaming cycle with dwell
    - G90: Absolute positioning mode
    - G91: Incremental positioning mode

b. **Parameters**: Each canned cycle requires specific parameters to be set based on the machining operation being performed. These parameters include:
    - Feed rate
    - Spindle speed
    - Depth of cut
    - Tool position (initial and final)
    - Dwell time (if applicable)
    - Retract height (for clearing chips or avoiding collisions)
    - Peck depth (for peck drilling)

c. **Block structure**: Canned cycles are typically programmed within a block structure, where the G-code command is followed by the necessary parameters. For example:
    - G81 X__ Y__ Z__ R__ F__ (for a drilling cycle)
    - G82 X__ Y__ Z__ R__ F__ P__ (for a drilling cycle with dwell)

d. **Safety considerations**: It's essential to consider safety aspects when using canned cycles, such as ensuring proper tool clearances, avoiding collisions, and verifying program correctness.

e. **Integration with tool change**: Canned cycles can be integrated into larger CNC programs, including tool change operations, to automate entire machining processes efficiently.

f. **Post-processing**: After the canned cycle is executed, it's essential to program appropriate movements to position the tool for subsequent operations or to move the tool away from the workpiece safely.

Overall, the structure of canned cycles aims to simplify CNC programming by providing standardized commands for common machining tasks while ensuring safety and efficiency in the machining process.

4.4. Subprogram Structure

Subprograms in CNC milling, also known as subroutines or macros, are reusable sections of code that perform specific tasks within a CNC program. They are used to simplify programming, reduce redundancy, and improve code readability. The structure of a subprogram in CNC milling typically consists of several components:

a. **Definition**: Subprograms are defined using a unique name followed by parameters (if any). The definition typically occurs at the beginning of the CNC program or in a separate program file. For example:

b. **Call**: To execute a subprogram, the main CNC program calls the subprogram by its name. The call is typically made wherever the functionality of the subprogram is required. For example:

c. **Parameters**: Subprograms can accept parameters to customize their behavior for specific tasks. Parameters are passed from the main program to the subprogram during the call. For example:

d. **Return**: After executing the subprogram, control returns to the main program. This is usually achieved using the M99 (End of subprogram) command. For example:

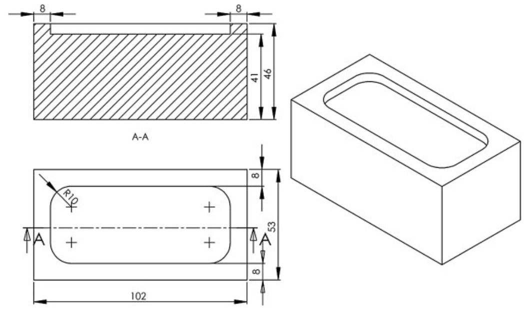
e. **Safety considerations**: Ensure that subprograms are properly defined and called within the CNC program to avoid errors or unexpected behavior. Verify tool clearances and program correctness to prevent collisions or damage to the machine or workpiece.

f. **Organization**: Subprograms can be organized into separate program files or sections within the main CNC program to improve code organization and maintainability.

Overall, the structure of subprograms in CNC milling machines allows for modular and reusable code, facilitating efficient programming and maintenance of CNC machining processes.

4.5. Pocket Milling Program Structure

Pocket milling is a common machining operation in CNC milling, used to remove material from within a pocket or cavity on a workpiece. The program structure for pocket milling in CNC typically involves the following components:



a. **Tool setup**: Begin by setting up the cutting tool and securing it in the spindle. Ensure the tool parameters (such as tool diameter, length, and geometry offsets) are correctly configured in the CNC machine's controller.

b. **Workpiece setup**: Secure the workpiece on the CNC machine's table or fixture, ensuring it is properly aligned and securely clamped to prevent movement during machining.

c. **Tool path generation**: Generate the toolpath for pocket milling using CAM (Computer-Aided Manufacturing) software. The toolpath defines the cutting motions that the CNC machine will execute to remove material from the pocket efficiently. The toolpath should consider factors such as cutting direction, cutter engagement, stepovers, and entry/exit strategies to optimize machining efficiency and surface finish.

d. **Program initialization**: Begin the CNC program by initializing necessary parameters such as the spindle speed (S), feed rate (F), coolant settings, tool change (if applicable), and coordinate system selection (G54, G55, etc.).

e. **Pocket boundary definition**: Specify the boundary of the pocket to be machined using appropriate G-code commands. This typically involves moving the tool to the starting position at the edge of the pocket.

f. **Cutting depth and stepovers**: Determine the cutting depth and stepover values based on the desired material removal rate, tool capabilities, and surface finish requirements. Use G-code commands such as G73 (High-speed peck drilling cycle) or G71 (Rough turning cycle) for efficient material removal.

g. **Machining Koop**: Execute the machining loop to remove material from the pocket. This loop involves moving the tool along the defined toolpath while ensuring proper cutting conditions (feed rate, spindle speed, tool engagement) to achieve the desired machining results.

h. **Retract and ramp movements**: Include retract and ramp movements between machining passes to ensure smooth tool entry and exit from the material, minimizing tool wear and potential cutting issues.

i. **Finish passes**: After roughing out the pocket, include finishing passes to achieve the final dimensions and surface quality. Adjust cutting parameters (feed rate, stepover) for finishing operations to achieve the desired surface finish.

j. **Program end**: Once the pocket milling operation is complete, end the CNC program by retracting the tool to a safe position, turning off the spindle, and releasing any clamps or fixtures holding the workpiece.

k. **Safety considerations**: Throughout the programming process, ensure that proper safety measures are followed, including tool and workpiece inspection, toolpath simulation, and consideration of machine limits to prevent collisions and ensure operator safety.

By following this structured approach, CNC programmers can effectively create pocket milling programs that efficiently remove material from workpieces while achieving the desired dimensional accuracy and surface finish.

4.6. Canned Cycle Techniques with Subprograms

Combining canned cycles with subprograms in CNC milling can significantly streamline programming and enhance the efficiency of machining operations. Here's how you can integrate these techniques:

a **Define subprograms**: Create subprograms for specific machining tasks, such as drilling, tapping, or pocketing. Each subprogram encapsulates the G-code commands and parameters required to perform the task efficiently.
Example Subprogram for Drilling:

b **Implement canned cycles**: Utilize canned cycles within the subprograms to automate common machining operations. This allows you to simplify the main program by calling the appropriate subprogram for each machining task.
Example Main Program:

c **Parameter passing**: Pass parameters to subprograms as needed to customize the machining operations. Parameters can include coordinates, feed rates, depths, tool numbers, or any other relevant values.
Example Parameter Passing:

d **Error handling**: Implement error handling within subprograms to handle exceptions or unexpected conditions gracefully. This can include checking for tool/tool holder collisions, workpiece clamping issues, or out-of-bounds movements.
Example Error Handling:

e **Modularity and Reusability**: Design subprograms to be modular and reusable across multiple CNC programs. This reduces duplication of code and simplifies maintenance.

f **Optimization**: Continuously optimize subprograms and canned cycles for improved machining efficiency, such as by adjusting feed rates, cutting depths, or toolpath strategies.

g **Testing and Validation**: Thoroughly test and validate subprograms in a simulated environment before deploying them on the CNC machine to ensure they perform as expected and meet quality standards.

By combining canned cycle techniques with subprograms in CNC milling, programmers can achieve greater automation, efficiency, and flexibility in machining operations, leading to reduced programming time and improved productivity.

4.7. CNC Milling Machine Cleaning and Maintenance Procedures

Cleaning and maintenance are essential for keeping CNC milling machines in optimal condition, ensuring longevity, accuracy, and safety. Here are some procedures for cleaning and maintaining CNC milling machines:

a. Daily cleaning

- Wipe down the machine surfaces with a clean, dry cloth to remove dust, debris, and coolant residue.
- Use compressed air or a vacuum cleaner with a soft brush attachment to remove chips and swarf from the machine's enclosure, tool changer, and chip conveyor.
- Inspect the way covers, bellows, and seals for any damage or contamination. Clean and lubricate them if necessary.
- Check the coolant levels and top up as needed. Clean the coolant tank and filters regularly to prevent contamination.

b. Weekly cleaning

- Perform a more thorough cleaning of the machine's internal components, such as the spindle, tool holder, and tool changer.
- Inspect the way surfaces, ball screws, and linear guides for any signs of wear or damage. Clean and lubricate them according to the manufacturer's recommendations.
- Check the coolant system for leaks, clogs, or other issues. Clean and flush the coolant lines if necessary.

c. Monthly maintenance

- Inspect and clean the electrical components, including the control panel, circuit boards, and wiring connections. Check for loose connections, corrosion, or damage.
- Lubricate the machine's lubrication points using the appropriate lubricants recommended by the manufacturer.
- Perform a thorough inspection of the spindle, bearings, and tool holder. Replace any worn or damaged components as needed.

- Check the accuracy of the machine's positioning and repeatability using calibration tools. Make any necessary adjustments to ensure accuracy.

d. Quarterly and yearly maintenance

- Schedule professional maintenance and calibration services from qualified technicians to ensure the machine's optimal performance.
- Replace worn-out or damaged parts, such as belts, bearings, seals, and filters, according to the manufacturer's recommended maintenance schedule.
- Perform a comprehensive inspection of the machine's structural integrity, alignment, and safety features. Make any necessary repairs or adjustments.

e. Operator training and safety

- Provide training to machine operators on proper cleaning and maintenance procedures.
- Emphasize the importance of safety precautions, such as wearing appropriate personal protective equipment (PPE) and following lockout/tagout procedures when performing maintenance tasks.

f. Documentation and record-keeping

- Keep detailed records of cleaning and maintenance activities, including dates, tasks performed, and any issues encountered.
- Refer to the machine's maintenance manual and documentation provided by the manufacturer for specific guidelines and recommendations.

Regular cleaning and maintenance of CNC milling machines are essential for preserving their performance, reliability, and safety. By following these procedures diligently, you can extend the lifespan of your CNC machine and minimize downtime due to unexpected failures or malfunctions.

4.8. Procedures for Managing Data Protection

Managing data protection in CNC milling involves safeguarding sensitive information related to machining processes, toolpaths, designs, and intellectual property. Here are some procedures to ensure data protection in CNC milling:

a. Access control

- Limit access to CNC machines and associated computers to authorized personnel only.

- Implement user authentication mechanisms such as passwords, PINs, or biometric verification to control access to CNC programming software and machine controls.

b. Data encryption

- Encrypt sensitive data, including CAD/CAM files, CNC programs (G-code), and design blueprints, to prevent unauthorized access or interception.
- Utilize industry-standard encryption algorithms and protocols to secure data both at rest and in transit.

c. Network security

- Secure CNC machines and computer systems connected to the network with firewalls, antivirus software, and intrusion detection/prevention systems.
- Implement secure Wi-Fi networks and VPNs (Virtual Private Networks) to protect data transmission between CNC machines and external devices.

d. Physical security

- Secure CNC machine workshops and facilities with access controls, surveillance cameras, and alarm systems to prevent unauthorized physical access or theft of equipment and data storage devices.
- Store physical media containing sensitive data (such as USB drives or memory cards) in locked cabinets or safes when not in use.

e. Data backup and disaster recovery

- Regularly back up CNC programs, design files, and other critical data to secure storage devices or cloud-based platforms.
- Develop and test comprehensive disaster recovery plans to ensure rapid restoration of data and operations in the event of system failures, data breaches, or disasters.

f. Employee training and awareness

- Provide training to CNC operators, programmers, and other personnel on data protection policies, procedures, and best practices.
- Raise awareness about the importance of data security, confidentiality, and compliance with regulatory requirements.

g. Vendor and supplier management

- Evaluate the security practices of CNC machine vendors and software suppliers to ensure they meet industry standards and regulatory requirements for data protection.
- Establish contractual agreements that include provisions for data security, confidentiality, and compliance with applicable laws and regulations.

h. Regulatory compliance

- Stay informed about data protection regulations relevant to CNC milling operations, such as GDPR (General Data Protection Regulation) in the European Union or CCPA (California Consumer Privacy Act) in the United States.
- Ensure compliance with data protection laws and regulations by implementing appropriate security measures, conducting regular audits, and maintaining documentation of compliance efforts.

Self-Check Sheet 4: Perform 2D Machining Operation Tools

1. **What is Facing operation?**
   Answer:

2. **What is Contouring**?
   Answer:

3. **What is Pocketing**?
   Answer:

4. **What is Drilling**?
   Answer:

5. **Why need maintenance in CNC milling?**
   Answer:

Answer Sheet 4: Perform 2d Machining Operation Tools

1. **What is Facing operation?**
   Answer: Facing is a machining operation where the CNC milling machine removes material from the surface of a workpiece to create a flat and even surface.

2. **What is Contouring?**
   Answer: Contouring involves cutting along the outline or perimeter of a 2D profile on the workpiece. It follows a specified path or contour defined by the CAD/CAM software, allowing the CNC machine to accurately shape the workpiece according to the desired design or geometry. Contouring operations are commonly used for profiling, engraving, or creating complex shapes with precise dimensions.

3. **What is Pocketing?**
   Answer: Pocketing is a machining operation where the CNC milling machine removes material from within a defined area or pocket on the workpiece. It involves cutting out the interior region of the workpiece to a specified depth, creating a cavity or recess with flat or contoured walls.

4. **What is Drilling?**
   Answer: Drilling is a machining operation where the CNC milling machine creates holes in the workpiece using a drill bit or specialized drilling tool. It involves rotating the drill bit and applying axial force to penetrate the workpiece material and form a hole of the desired diameter and depth.

5. **Why need maintenance in CNC milling?**
   Answer: Cleaning and maintenance are essential for keeping CNC milling machines in optimal condition, ensuring longevity, accuracy, and safety. Here are some procedures for cleaning and maintaining CNC milling machines:

Job Sheet 4.1: Perform 2D Drilling Operation as Per Specification.

Job Name: Perform 2D Drilling operation as per specification.

Procedure:

1. Wear PPE.
2. Read the job sheet and specification sheet.
3. Interpret the technical diagram to determine materials required for the job.
4. Take note of measurements according to specification if required.
5. Setup the workpiece (raw material) onto the milling machine's worktable.
6. Secure the workpiece firmly using clamps or a vice to prevent movement during driilling.
7. Choose the appropriate drill based on the required operation, material type, and desired hole.
8. Install the selected tool holder securely into the spindle of the CNC milling machine.
9. Set the origin or zero point on the CNC machine's coordinate system based on the measured dimensions of the workpiece.
10. Measure the tool length and diameter accurately using a tool presenter or a similar measuring device.
11. Input the tool offset values into the CNC machine's control panel to ensure precise tool positioning during machining.
12. Write or generate a CNC program using CAM (Computer-Aided Manufacturing) software.
13. Specify the toolpaths, cutting parameters, feed rates, and spindle speeds required for the drilling operation.
14. Move the milling machine's spindle to the starting position specified in the CNC program.
15. Start machine and Emergency switch on .
16. Unlock program lock key
17. Press edit mood and press program directory
18. Input Program Name and write program as per specification.
19. Execute the machine .(The program Below)

```
%
O001:
N 05 G90 G20;
N 10 G28 X0 Y0 Z0;
N 15 G0 Z100 G54 T1D1;
N 20 M03 S1000 ;
N 25 G81 G99 X0 Y55 Z2 I-10 F150;
N 30 G91 X17 N3;
N 40 Y-15;
N 45 X-17 N2;
N 50 G90 G0 G80 Z100;
N 55 M30;
```

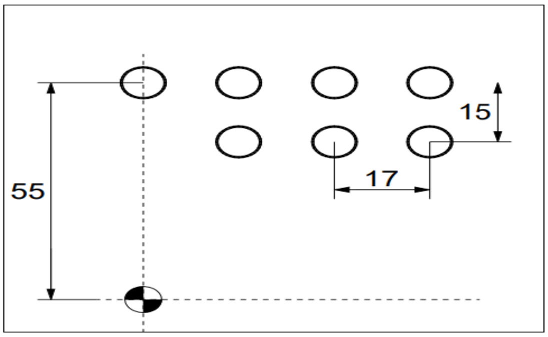
20. After complete Inputting program check program again.
21. Shutdown machine properly.

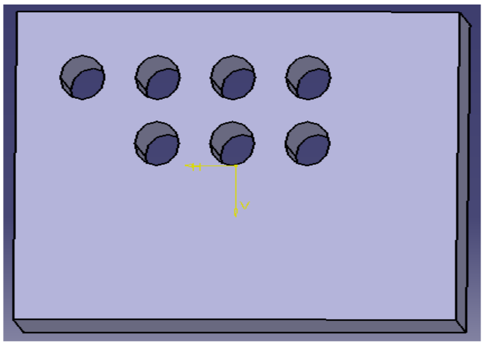
Specification Sheet 4.1: Perform 2D Drilling Operation as Per Specification

Job Name : Perform 2D Drilling operation as per specification.

Conditions for the job: You must practice safe operation procedures at all times during the performance this job. You will be required to demonstrate OSH competencies **Perform 2D Drilling operation as per specification.**

Diagram / Design





Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	Pair	01	
2	Mask	N 95	Nos	01	
3	Safety shoe	As required	Pair	01	
4	Safety goggles	Stranded	Nos	01	
5	Apron	Stranded	Nos	01	

Required Tools and equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	Vernier caliper	150 mm	Pcs	01	
2	Steal Ruler	150 mm	Pcs	01	
3	Spanner set	10-22 mm	Set	01	
4	Soft hammer	2 pound	Pcs	01	
5	Drill bit	10 mm	Pcs	01	

Required Materials

Sl.	Name of materials	Specification	Unit	Quantity	Remarks
1	Paper	As required	Pcs	01	
2	Pen	As required	Pcs	01	
3	MS Bar	100 X 70 X 25 mm	Pcs	01	

Job Sheet 4 .2 Input and Write a Part Program of Drilling Cycle (Canned Cycle) Using G81 as Per Specification

Procedure

1. Wear PPE.
2. Read the job sheet and specification sheet.
3. Interpret the technical diagram to determine materials required for the job.
4. Take note of measurements according to specification if required.
5. Setup the workpiece (raw material) onto the milling machine's worktable.
6. Secure the workpiece firmly using clamps or a vice to prevent movement during drilling.
7. Choose the appropriate drill based on the required operation, material type, and desired hole.
8. Install the selected tool holder securely into the spindle of the CNC milling machine.
9. Set the origin or zero point on the CNC machine's coordinate system based on the measured dimensions of the workpiece.
10. Measure the tool length and diameter accurately using a tool presenter or a similar measuring device.
11. Input the tool offset values into the CNC machine's control panel to ensure precise tool positioning during machining.
12. Write CNC program Drilling cycle(Canned Cycle) Program Use G81
13. Specify the toolpaths, cutting parameters, feed rates, and spindle speeds required for the drilling operation.
14. Move the milling machine's spindle to the starting position specified in the CNC program.
15. Start machine and Emergency switch on .
16. Unlock program lock key
17. Press edit mood and press program directory
18. Input Program Name and write program as per specification.
19. Execute the machine .(The program Below)

```
% O002:
N 05     G28 X0 Y0 Z0;
N 10     G90 G43 G21
N 15     G0Z25 M8 M41;
N 20     G0 G54 T1 D1;
N 25     M03 S1000 ;
N 30     G81 G98 X15 Y15 Z2 I-20 F200
N 35     X85
N 40     Y85
N 45     X15
N 50     X50 Y75
N 55     G93 I50 J50
N 60     G91 Q-45 N3
N 65     G80 G0 G90 G44 Z30
N 70     M30
```

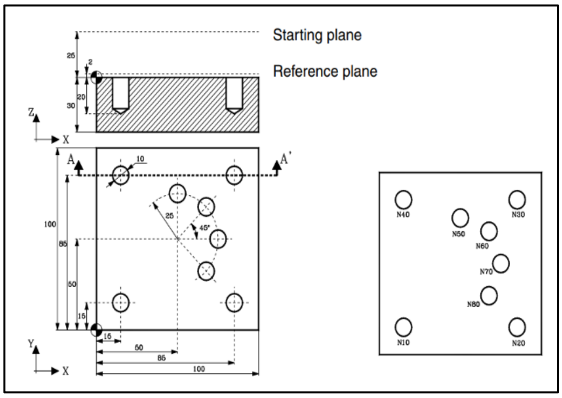
20. After completing the Inputting program check the program again.

21. Shutdown machine properly.

Specification Sheet 4.2: Input And Write a Part Program of Drilling Cycle (Canned Cycle) Using G81 as Per Specification

Conditions for the job: You must practice safe operation procedures at all times during the performance this job. You will be required to demonstrate OSH competencies Input and write a part program Drilling cycle(Canned Cycle) Program Use G81 as per specification. .

Diagram / Design



- Definition of the drilling points in: Absolute Cartesian coordinates.
- Incremental Polar coordinates with repetition.
- Tool: Ø10 mm helical drill bit.
- Cutting conditions: S=1000 rpm. F=200 mm/min.

Required Personal Protective Equipment (PPE): Pre/person

SL.	Name of The PPE	Specification	Unit	Quantity	Remarks
1	Hand gloves	Cotton	PAIR	01	
2	Mask	N 95	NOS	01	
3	Safety shoe	As required	PAIR	01	
4	Safety goggles	Stranded	NOS	01	
5	Apron	Stranded	NOS	01	

Required Tools and equipment

Sl.	Name of tools and Equipment	Specification	Unit	Quantity	Remarks
1	Vernier caliper	150 mm	Pcs	01	
2	Steel Ruler	150 mm	Pcs	01	
3	Spanner set	10-22 mm	Set	01	
4	Soft hammer	2 pounds	Pcs	01	

Required Materials

Sl.	Name of materials	Specification	Unit	Quantity	Remarks
1	Paper	As required	Pcs	01	
2	Pen	As required	Pcs	01	
3	MS Bar	100 X 70 X 25 mm	Pcs	01	

Job Sheet 4.3: Perform Sub-Programing as Per Specification

Working Procedure:

1. Collect and Ware PPE.
2. Read job sheet and specification sheet
3. Set up the workpiece and tooling as required for the sub-program operation.
4. Ensure proper workpiece alignment and tool offsetting.
5. Implement a tool change if the sub-program requires a different cutting tool than the main program.
6. Ensure that the correct sub-program number and any required parameters are specified.
7. Execute the specific machining operations or sequence of commands defined within the sub-program.
8. After completing the sub-program operations, return control to the main program.
9. Ensure that the sub-program is designed for reuse in multiple parts of the main program or in different programs altogether.
10. Test the sub-program independently to ensure that it performs the intended operations correctly.

Main Program	Sub Program
O1000 ;	O1001 ;
N1 T1 M6 ;	N101 Y30 ;
N2 G0 G90 G40 G21 G17 G94 G80 ;	N102 Y50 ;
N3 G54 X10 Y10 S? M3 ;	N103 Y70 ;
N4 G43 Z100 H1 ;	N104 X30 ;
N5 Z5 ;	N105 X50 ;
N6 G81 R3 Z-20 F? M8 ;	N106 X70 ;
N7 M98 P1001; (Call Sub program O1001)	N107 X90 ;
N8 G0 G90 Z100	N108 Y50 ;
N9 T2 M6 ;	N109 Y30 ;
N10 G0 G90 G40 G21 G17 G94 G80 ;	N110 Y10 ;
N11 G54 X10 Y10 S? M3 ;	N111 X70 ;
N12 G43 Z100 H1 ;	N112 X50 ;
N13 Z5 ;	N113 X30 ;
N14 G84 G99 G95 R3 Z-20 F1.25 M8 ;	N114 G80 ;
N15 M98 P1001;(Call Sub program O1001)	N115 M99 (Go to Main Program)
N16 G0 G90 Z100 ;	
N17 T0 M6 ;	
N18 M30 ;	

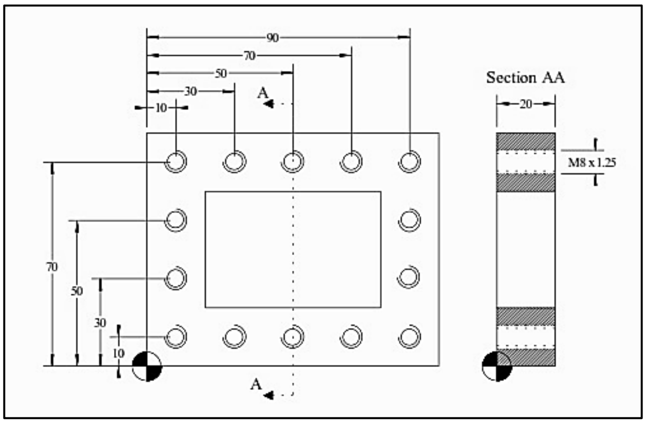
11. After complete work
12. Clean tools and equipment
13. Store all equipment safely.

Specification Sheet 4.3: Perform Sub-Program as Per Specification

Name of Job: Perform sub-program in as per specification

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to **Perform sub-program in as per specification**

Diagram



Required Personal protective equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	pair	01	
2	Breathing mask	N 95	Nos	01	
3	Ear plugs	As required	Nos	01	
4	Safety shoe	As required	pair	01	
5	Safety goggles	Stranded	Nos	01	
6	Apron	Stranded	Nos	01	

Required Tools and equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	CNC Milling Machine		No	01	
2	Drill	8.5 mm	No	1	
3	Tap	10 mm	No	1	

Job Sheet 4.4: Execute a Canned Cycle Part Program as per Specification

Procedure

1. Wear PPE.
2. Read the job sheet and specification sheet.
3. Interpret the technical diagram to determine materials required for the job.
4. Take note of measurements according to specification if required.
5. Determine tools and equipment to hold a work piece.
6. Start machine and Emergency switch on .
7. Unlock program lock key
8. Press edit mood and press program directory to input program as per specification. (The program Below)

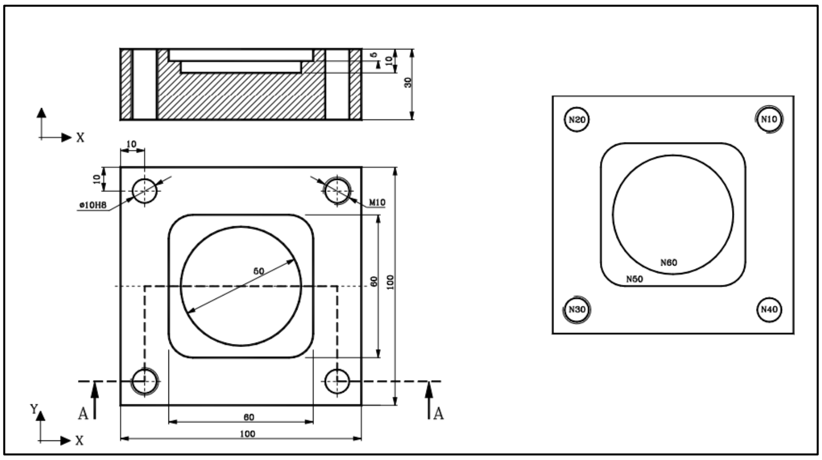
```
% O003:
N  05      G28 X0 Y0 Z0;
N  10      G90 G43 G21
N  15      G0Z25 M8 M42;
N  20      G0 G54 M03 S 1800 T6 D 6;   (Center punching drill bit.)
N  25      G0  G43 Z20;
N  30      G82 G99 X40 Y40 Z2 I-5 K300
N  35      X-40
N  40      Y-40
N  45      X40
N  50      G80
N  55      G0 G44 Z100
N  60      M06 T9 D9                    (Drill bit)
N  65      G0 G90 G43 Z20 F200 S1050 M4 M42
N  70      G81 G99 X40 Y40 Z2 I-35
N  75      X-40 Y-40
N  80      G0 G44 Z100
N  85      M06 T8 D8                    (Reamer)
N  90      G0 G90 G43 Z20 F100 S500 M4 M41
N  95      G85 G99 X-40 Y40 Z2 I-30 K200
N  100     X40 Y-40
N  105     G80
N  110     G0 G44 Z100
N  115     M06 T12 D12 ; Tap.
N  120     G0 G90 G43 Z20 F450 S300 M4 M41
N  125     G84 G99 X40 Y40 Z2 I-35 K200
N  130     X-40 Y-40
N  135     G80
N  140     G0 G44 Z100
N  145     M06 T2 D2                    (  End mill for pockets.)
N  150     G0 G90 G43 Z20 F250 S1600 M4 M42
N  160     N50 G87 G98 X0 Y0 Z2 I-5 J-30 K30 B5 D2 H200 L-1
N  165     N60 G88 G98 X0 Y0 Z-3 I-10 J-25 B5 D2 H200 L1
N  170     G80
N  175     G0 G44 Z100
N  180     M30
```

22. After completing the Inputting program check the program again.
23. Shutdown machine properly.

Specification Sheet 4.4: Execute A Canned Cycle Part Program as Per Specification

Conditions for the job: You must practice safe operation procedures at all times during the performance this job. You will be required to demonstrate OSH competencies to **Execute a Canned cycle part program as per specification**

Diagram / Design



Definition of a rectangular pocket and a circular pocket.

- Center drill 8 mm
- Drill 8.4 mm
- Ramer: 8.5 mm
- Tap : 10 mm
- Tool: End mill with 4 teeth and Ø10 mm.
- Cutting conditions: S=1600 rpm.
- Roughing feed rate: 300 mm/min.
- Finishing feed rate: 200 mm/min

Required Personal protective equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	Pair	01	
2	Mask	N 95	Nos	01	
3	Safety shoe	As required	Pair	01	
4	Safety goggles	Stranded	Nos	01	
5	Apron	Stranded	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	Vernier caliper	150 mm	Pcs	01	
2	Steal Ruler	150 mm	Pcs	01	
3	Spanner set	10-22 mm	Set	01	
4	Soft hammer	2 pound	Pcs	01	
5	End Mill Cutter	4 teeth Ø10 mm.	Pcs	01	
6	Center drill bit	8mm	Pcs	01	
7	Drill bit	8.4 mm	Pcs	01	
8	Reamer	8.5 mm	Pcs	01	
9	Tap set	10 mm	Pcs	03	

Required Materials

Sl.	Name of materials	Specification	Unit	Quantity	Remarks
1	Paper	As required	Pcs	01	
2	Pen	As required	Pcs	01	
3	MS Bar	100 X 100 X 30 mm	Pcs	08	

Review of Competency

Below is yourself assessment rating for module "**Creating and Using G&M Code for Machining**"

Assessment of performance Criteria	Yes	No
1. Reference position of a machining center is determined	☐	☐
2. Work coordinates are set	☐	☐
3. Appropriate hand tools are selected	☐	☐
4. Preparatory G & M code is created	☐	☐
5. Modal and non-modal codes are identified	☐	☐
6. Absolute and Incremental positioning code is created	☐	☐
7. Circular and linear interpolation are created	☐	☐
8. Cutter radius compensation is selected	☐	☐
9. Tools offset (G43) and work offset (G54) are set	☐	☐
10. Spindle commands and program stop commands are executed	☐	☐
11. Canned cycles are executed	☐	☐
12. Bolt hole codes are executed	☐	☐
13. Pocket milling and contour milling is programmed	☐	☐
14. Sub-program is executed	☐	☐
15. Work holding mechanism is selected	☐	☐
16. Required cutting tools are selected and loaded	☐	☐
17. Datum is defined	☐	☐
18. Entry and exit points are selected	☐	☐
19. Sequence of cutting operations is selected	☐	☐
20. 2D operations are performed	☐	☐
21. Canned cycles structures are identified	☐	☐
22. Sub-program structures are identified	☐	☐
23. Pocket milling program structures are identified	☐	☐
24. Canned cycle tricks with subprogram are executed	☐	☐

I now feel ready to undertake my formal competency assessment.

Signed:

Date:

Development of CBLM

The Competency based Learning Material (CBLM) of '**Creating and Use G&M Code for Machining' (Occupation: CNC Maching Centre Operation with CAD CAM, Level-4)** for National Skills Certificate is developed by NSDA with the assistance of SIMEC System Ltd., ECF Consultancy & SIMEC Institute of Technology JV (Joint Venture Firm) in the month of June, 2024 under the contract number of package SD-9B dated 15th January 2024.

SL No.	Name & Address	Designation	Contact Number
1	Rofiqun Nabi	Writer	01841-604582
2	Uttam Kumar das	Editor	01716-220932
3	Engr. Md. Zuwel Parves	Co-Ordinator	01737-278906
4	Engr. Md. Nazrul Islam	Reviewer	01711-273708

Reference:

1. **https://machmotion.com/downloads/GCode/Mach4-G-and-M-Code-Reference-Manual.pdf**,
2. **https://acssgt01.tripod.com/documents/G-M_Codes.pdf**, ,
3. **https://tecnoedu.com/Download/g_and_m_programming_for_cnc_lathes.pdf**,
4. https://www.cnccookbook.com/g-code-m-code-reference-list-cnc-mills,
5. **https://assets.new.siemens.com/siemens/assets/api/uuid:a27afac9-4273-43a7-bd63-ab4867e1805c/2-C101-G-codes-M-codes.pdf**,