



# **Competency Based Learning Material (CBLM)**

**Perform CNC Maching Centre Operation with CAD CAM  
Level-4**

**Module: Performing CNC Machining Center Operation with CAD  
CAM**

**Code: CBLM-OU-LE-CNCCDM-02-L4-V1**



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



## 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](mailto:ec@nsda.gov.bd)  
Website: [www.nstda.gov.bd](http://www.nstda.gov.bd).  
National Skills Portal: <http://skillsportal.gov.bd>

This Competency Based Learning Materials (CBLM) on “Performing CNC Machining Center Operation with CAD CAM” 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.

National Skills Development Authority (NSDA) is the owner of this document. Other interested parties must obtain written permission from NSDA for reproduction of information in any manner, in whole or in part, of this Competency Standard, in English or other language.

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, Performing CNC Machining Center Operation with CAD CAM 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 practice the job. You may need to practice 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 through 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

Copyright .....	ii
How to use this Competency Based Learning Material (CBLM).....	vi
Module Content .....	1
<b>Learning Outcome 1: Prepare CNC Machining Centre.....</b>	<b>2</b>
Learning Experience 1: Prepare CNC Machining Centre.....	4
Information Sheet 1: Prepare CNC Machining Centre.....	5
Self Check sheet 1: Prepare CNC Machining Centre .....	14
Answer sheet 1: Prepare CNC Machining Centre .....	15
Job Sheet 1.1: Identify The Main Components of CNC Machining Center as Per Specification.....	18
Specification Sheet 1.1 Identify the Main Components of CNC Machining Center as Per Specification .....	19
Job Sheet 1.2: Check Coolant Level in CNC Machine as Per Specification .....	20
Specification Sheet 1.2: Check Coolant Level in CNC Machine as Per Specification.....	21
Job Sheet 1.3: Turn the CNC Machining Center Power on And Off as Per Specification .....	22
Specification Sheet 1.3: Turn the CNC Machining Center Power on And Off as Per Specification .....	23
Job Sheet 1.4: Perform ATC Function .....	24
Specification sheet 1.4: Perform ATC function.....	25
Job Sheet 1.5: Clamp The Workpiece with Clamping Kit.....	26
Specification Sheet 1.5: Clamp The Workpiece with Clamping Kit as Per Specification .....	27
<b>Learning Outcome -2: Select Cutting Tools .....</b>	<b>28</b>
Learning Experience 2: Select Cutting Tools.....	30
Information sheet 2: Select Cutting Tools.....	31
Self Check sheet 2: Select cutting tools .....	40
Answer sheet 2: Select Cutting Tools .....	41
Job sheet 2.1: Identify Different Types of Cutting Tools as Per Specification .....	44
Specification Sheet 2.1 Identify Different Types of Cutting Tools as Per Specification .....	45
Job Sheet 2.2: Select Cutting Tools According to the Appropriate Sequence for an Operation .....	46
Specification Sheet 2.2: Select Cutting Tools According to the Appropriate Sequence for an Operation .....	47
<b>Learning Outcome -3: Operate CNC Machining Center.....</b>	<b>50</b>
Learning Experience 3: Operate CNC Machining Center.....	50
Information Sheet 3: Operate CNC Machining Center .....	51
Self Check Sheet 3: Operate CNC Machining Center.....	66
Answer Sheet 3: Operate CNC Machining Center .....	67
Job Sheet 3.1: Accomplish CNC Machine Axis Positioning According to Job Requirements .....	72
Specification Sheet 3.1: Accomplish CNC Machine Axis Positioning According to Job Requirements.....	73
Job Sheet 3.2: Perform Coping, Edit, Load and File Creation as Per Specification .....	74
Specification Sheet 3.2: Complete Copy, Edit, Load and File Creation as Per Specification .....	75
Job Sheet 3.3: Perform Work Offset and Tool Offset in CNC Milling Machine.....	76
Specification Sheet 3.3 Perform Work Offset and Tool Offset in CNC Milling Machine .....	77
Job Sheet 3.4: Perform Program Start, Program Stop, and Complete a Single Block, Dry Run, Feed Hold as Per Specification.....	80

Specification Sheet 3.4: Perform Program Start, Program Stop and Complete a Single Block, Dry Run, Feed Hold as Per Specification .....	81
Job 3.5: Perform Dry Run in CNC Machine as Per Specification .....	80
Specification Sheet 3.5 Perform Dry Run in CNC Machine as Per Specification .....	81
Job Sheet 3 .6 Input a Part Program as Per Specification. ....	83
Specification Sheet 3.6 Input a Part Program as Per Specification. ....	85
<b>Review of Competency .....</b>	<b>87</b>

## Module Content

<b>Unit of Competency</b>	<b>Perform CNC Machining Center Operation</b>
<b>Unit Code</b>	<b>OU-LE-CNCCDM-02-L4-V1</b>
<b>Module Title</b>	<b>Performing CNC Machining Center Operation</b>
<b>Module Descriptor</b>	This module covers the skills, knowledge and attitudes required to perform CNC machining center operation. It specifically includes preparing CNC machining center, selecting cutting tools and operating CNC machining center.
<b>Nominal Hours</b>	<b>55 Hours</b>
<b>Learning Outcome</b>	After completing the practice of the module, the trainees will be able to perform the following jobs: <ol style="list-style-type: none"> <li>1. Prepare CNC Machining Centre</li> <li>2. Select cutting tools</li> <li>3. Operate CNC Machining center</li> </ol>

### Assessment Criteria

1. Major components of CNC machining center in identified
2. Air pressure, coolant level and lubrication oil are checked
3. CNC Machining center power on & off is performed
4. Machine reference is performed
5. Override keys are adjusted as required
6. Hand Jog rate is performed as required
7. ATC function is performed
8. Workpiece is clamped with clamping kits
9. Various types of cutting tools are identified
10. The appropriate sequence of cutting tools for an operation is selected
11. Appropriate cutting tools are selected as per job materials
12. Appropriate cutting parameters are followed as per manufacturer chart
13. Different types of cutting tools inserts are selected
14. CNC machine axes positioning is performed as per job requirement
15. Copy, edit, load, and create files are executed
16. Program start, program stop, and a single block, dry run, feed hold is performed
17. Manual data input (MDI) in CNC machining center is performed
18. Auxiliary function keys are performed
19. Tool length compensation is performed
20. Work offset and tool offset are performed
21. Program is transferred from storage devices
22. Dry and actual run is executed

## Learning Outcome 1: Prepare CNC Machining Centre

Assessment Criteria	<ol style="list-style-type: none"> <li>1. Major components of CNC machining center in identified</li> <li>2. Air pressure, coolant level and lubrication oil are checked</li> <li>3. CNC Machining center power on &amp; off is performed</li> <li>4. Machine reference is performed</li> <li>5. Override keys are adjusted as required</li> <li>6. Hand Jog rate is performed as required</li> <li>7. ATC function is performed</li> <li>8. Workpiece is clamped with clamping kits</li> </ol>
Conditions and Resources	<ol style="list-style-type: none"> <li>1. Workplace or Simulated Workplace</li> <li>2. CBLM</li> <li>3. Handout</li> <li>4. Laptop</li> <li>5. Multimedia Projector</li> <li>6. Paper, Pen, Pencil,</li> <li>7. Internet Facilities</li> <li>8. White Board and</li> <li>9. Audio Video Devices</li> <li>10. Necessary tools and equipment</li> <li>11. Necessary PPE</li> </ol>
Contents	<ol style="list-style-type: none"> <li>1. Main components of CNC machining center <ul style="list-style-type: none"> <li>▪ Machine on-off switch</li> <li>▪ Emergency switch</li> <li>▪ Machine reference switch</li> <li>▪ Override controller</li> <li>▪ Hand jog rate</li> <li>▪ Work holding device</li> <li>▪ Tool holder</li> <li>▪ Clamping kit</li> </ul> </li> </ol>

	<ul style="list-style-type: none"> <li>➤ the vice</li> <li>➤ Clamping set</li> <li>➤ parallel bars</li> </ul> <ol style="list-style-type: none"> <li>2. Need to check air pressure, coolant level and lubrication oil.</li> <li>3. How to turn CNC machining center</li> <li>4. Machine reference</li> <li>5. Requirement to adjust override key</li> <li>6. Hand jog rate</li> <li>7. ATC function</li> <li>8. Clamping method with workpiece clamping kit</li> </ol>
Job/Task/Activity	<ol style="list-style-type: none"> <li>1. Identify the main components of CNC machining center</li> <li>2. Check air pressure, coolant level and lubrication oil</li> <li>3. Turn the CNC machining center power on and off</li> <li>4. Complete the ATC function</li> <li>5. Clamp the workpiece with clamping kit</li> </ol>
Training Method	<ol style="list-style-type: none"> <li>1. Discussion</li> <li>2. Presentation</li> <li>3. Demonstration</li> <li>4. Guided Practice</li> <li>5. Individual Practice</li> <li>6. Project Work</li> <li>7. Problem Solving</li> <li>8. Brainstorming</li> <li>9. Role Play</li> </ol>
Assessment Method	<ol style="list-style-type: none"> <li>1. Written Test</li> <li>2. Demonstration</li> <li>3. Oral questioning</li> <li>4. Portfolio</li> </ol>

## Learning Experience 1: Prepare CNC Machining Centre

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 <b>‘Prepare CNC Machining Centre’</b>
2. Read the Information sheet and complete the Self Checks & Check answer sheets on <b>“Prepare CNC Machining Centre”</b>	2. Read Information sheet 1: Prepare CNC Machining Centre 3. Answer Self-check 1: Prepare CNC Machining Centre 4. Check your answer with Answer key 1: Prepare CNC Machining Centre
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet Task Sheet 1.1: Identify the Main Components of CNC Machining Center Specification Sheet 1.1 Identify the Main Components of CNC Machining Center  Job Sheet 1.2: Check Coolant Level in CNC Machine Specification Sheet 1.2: Check Coolant Level in CNC Machine  Job Sheet 1.3: Turn the CNC Machining Center Power on and off Specification Sheet 1.3: Turn the CNC Machining Center Power on and off  Job Sheet 1.4: Align Cutting Tools Using ATC Function Specification sheet 1.4: Align Cutting Tools Using ATC Function  Job Sheet 1.5: Clamp the Workpiece with Clamping Kit Specification Sheet 1.5: Clamp The Workpiece with Clamping Kit



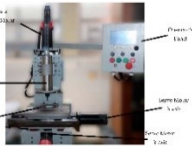



## Information Sheet 1: Prepare CNC Machining Centre






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

- 1.1 Main components of CNC machining center
- 1.2 Need to check air pressure, coolant level and lubrication oil.
- 1.3 Points to consider to turn CNC machining center
- 1.4 Machine reference
- 1.5 Requirement to adjust override key
- 1.6 Hand jog rate
- 1.7 ATC function
- 1.8 Clamping method of workpiece with clamping kit

### 1.1 Main Components of CNC Machining Center

The main components of a CNC machining center include

Machine on-off switch	This switch is used to power the CNC machining center on and off.	
Emergency switch	A safety feature that immediately stops the machine in case of an emergency or when safety is compromised.	
Machine reference switch	Used to establish the reference point for the CNC machine's movements.	
Override controller	Allows operators to manually adjust the feed rate, spindle speed, or other parameters during machining operations.	
Hand jog rate	Control mechanism for manually moving the machine's axes at different speeds for setup or troubleshooting.	
Work holding device	Fixtures or mechanisms used to securely hold the workpiece in place during machining.	

Tool holder	Holds the cutting tools securely in place during machining operations.	
Clamping kit	Contains various clamps, bolts, nuts, and other accessories used to secure workpieces to the work holding device.	
Vice	A type of work holding device commonly used to hold smaller workpieces securely in place during machining.	
Clamping set	Similar to a clamping kit, it includes components necessary for securing workpieces to the machine bed or work holding device.	
Parallel bars	Precision bars used for aligning and leveling workpieces on the machine bed.	

## 1.2 Need to Check Air Pressure, Coolant Level and Lubrication Oil.

It's crucial to regularly check these three elements for optimal performance and longevity of your CNC milling machine. Here's a breakdown of how to inspect each:

### a. Air Pressure

- **Locate the air pressure gauge:** This gauge should be easily identifiable on the machine, often situated near the air preparation unit or control panel.
- **Consult the machine manual:** The manual will specify the recommended air pressure range for the machine.
- **Compare the gauge reading to the recommended range:** If the pressure falls outside the acceptable range, adjust it using the pressure regulator provided in the air preparation unit (refer to the manual for instructions).
- **Inspect for leaks:** Listen for any hissing sounds or visually check for signs of air escaping from hoses or fittings. Leaks need to be addressed promptly to maintain appropriate pressure.

### b. Coolant Level

- **Identify the coolant reservoir:** This is typically a transparent tank located on the machine.

- **Observe the level indicator:** The reservoir should have a level indicator showing the minimum and maximum acceptable coolant levels.
- **Ensure the coolant level is within the recommended range:** If the level is low, top it up with a fresh coolant mixture as per the manufacturer's specifications. Avoid overfilling, as it can overflow during operation.
- **Check the coolant quality:** The coolant should be clean and free of debris or discoloration. If it appears cloudy or discolored, it may need to be replaced. Refer to the manual for recommended coolant change intervals.

#### c. Lubrication Oil

- **Consult the machine manual:** The manual will specify the lubrication points on the machine and the recommended type and frequency of lubrication.
- **Locate the lubrication points:** These are usually grease fittings or oil reservoirs situated at various locations on the machine, such as guideways, spindle bearings, and slideways.
- **Apply lubricant as per the manual:** Use the recommended type and amount of lubricant at each point. Over-lubrication can be detrimental, so follow the instructions carefully.
- **Visually inspect for leaks:** Look for any signs of oil leaking from fittings or seals. Leaks need to be addressed promptly to prevent damage to the machine.

### 1.3 Points to Consider to Turn CNC Machining Center

Here's a general guideline, but remember to consult your machine manual for the most accurate and up-to-date instructions:

#### a. Preparation

- Ensure the work area is clear and free of obstructions. This includes materials, tools, and personnel who are not directly involved in the operation.
- Verify that the workpiece is securely fastened in the work holding device.
- Double-check that the cutting tool is properly installed and tightened in the tool holder.
- Clear any debris or chips from the machine table and surrounding areas.
- Close all safety doors and guards.

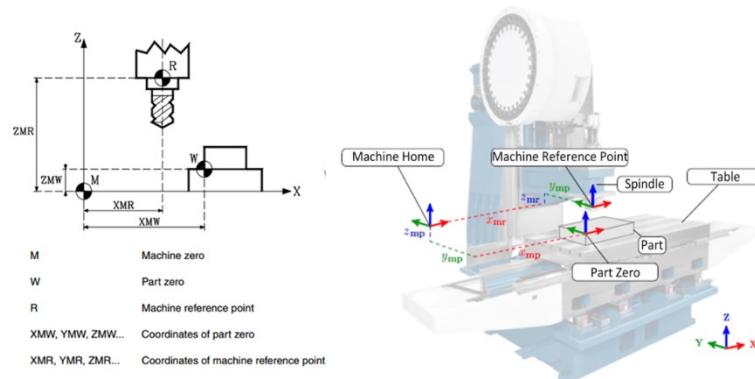
#### b. Turning on the machine

- Locate the main power switch and turn it on. This is typically located on the control panel or near the machine base.
- Release the emergency stop button (if engaged). The emergency stop button is a large red button typically located on the control panel and should only be pressed in case of an emergency.
- Power on the control panel. This might involve a separate switch or button on the panel itself.

- Follow the specific instructions in the manual to boot up the CNC system and load the desired program. This may involve entering codes, selecting options from a menu, or using a separate computer interface.

### c. Machine reference

In CNC (Computer Numerical Control) milling, the reference point, often referred to as the machine zero point or the workpiece zero point, is a crucial concept. It establishes the origin from which all machining operations are defined. Here's a breakdown:



### d. Machine zero point (machine home position)

This is the reference point for the CNC machine itself. It's typically a predefined location that the machine returns to when not in operation or when the operator initiates a homing sequence. The machine zero point serves as the reference for all movements made by the machine during operation. It's important for ensuring consistency and accuracy in machining operations.

### e. Workpiece zero point

This is the reference point for the specific workpiece being machined. It's usually established based on the design requirements of the part and the machining program. The workpiece zero point serves as the origin for all machining coordinates used in the CNC program. It's set relative to the machine zero point, often through the process of workpiece setup and alignment.

### f. Establishing the workpiece zero point

The workpiece zero point can be set manually by the operator using edge finders, dial indicators, or other measuring tools to locate specific features on the workpiece.

Alternatively, it can be set automatically using probing routines integrated into the CNC machine, where the machine locates reference points on the workpiece surface based on programmed instructions.

### g. Coordinate system

Once the workpiece zero point is established, a coordinate system is defined relative to it. Common coordinate systems include Cartesian coordinates (X, Y, Z) for three-axis milling machines and additional axes for multi-axis machines. The CNC program specifies tool paths and machining operations using coordinates within this system.

In summary, the reference point in CNC milling establishes the origin for all machining operations. The machine zero point provides a consistent reference for machine movements, while the workpiece zero point defines the origin specific to the workpiece being machined. Establishing and accurately setting these reference points are essential for precise and repeatable machining operations.

## 1.4 Requirement To Adjust Override Key

Here are some common requirements for adjusting the override key on a CNC machine:

### a. Authorization and training

- Operators typically need proper authorization and training to adjust override settings on CNC machines.
- Training ensures that operators understand the implications of changing override settings and how it can affect the machining process and part quality.

### b. Safety considerations

- Operators should be aware of safety considerations when adjusting override settings.
- Changing parameters such as feed rate or spindle speed can impact the cutting forces and tool wear, potentially affecting operator safety and machine integrity.

### c. Machine mode

- Depending on the CNC machine's configuration and control system, override adjustments may be accessible in different machine modes, such as manual mode, auto mode, or MDI (Manual Data Input) mode.
- Operators should ensure they are in the appropriate mode before attempting to adjust override settings.

### d. Control interface

- CNC machines feature control interfaces where operators can access override functions.
- The override key may be labeled as "Feed Override," "Spindle Override," or something similar, depending on the machine's manufacturer and control system.

### e. Limitations and guidelines

- CNC machines often have limitations on how much operators can adjust override settings.
- Manufacturers typically provide guidelines on the permissible range of override adjustments to prevent excessive strain on the machine and tooling.

### f. Real-time monitoring

- Some CNC machines offer real-time monitoring of machining parameters, allowing operators to observe the effects of override adjustments on the machining process.
- Operators can use this feedback to make informed decisions and optimize machining operations.

### g. Documentation and recordkeeping

- It's essential to maintain documentation of override adjustments and recordkeeping for quality control purposes.
- Keeping track of override settings helps identify trends, troubleshoot issues, and maintain consistency in machining processes.

## 1.5 Hand Jog Rate

The hand jog rate on a CNC milling machine refers to the speed at which the operator can manually move the machine's axes (X, Y, and Z) using the jog controls. This feature allows for:

- Precise positioning of the cutting tool during setup and alignment.
- Inspection of the workpiece and tool during the machining process.
- Manual intervention in case of minor adjustments or troubleshooting.



The hand jog rate is typically adjustable through dedicated controls on the machine's panel or jog handle. These controls can be:

- Physical knobs or dials: These allow the operator to manually adjust the jog speed in a stepless manner.
- Selection buttons: The machine might offer predefined jog speed options through buttons, allowing the operator to choose between different speeds.
- Software interface: Some advanced CNC machines might allow adjusting the jog rate through the software interface.

Here are some key points to remember about hand jog rate:

- Safety: Always exercise caution when using the hand jog function. Ensure the work area is clear and never reach into the machine while it's in motion.
- Start slow: It's recommended to begin with the lowest jog rate when initially positioning the tool or workpiece. Gradually increase the speed as needed while maintaining control and awareness of your surroundings.
- Matching the task: Choose a suitable jog rate based on the task at hand. For delicate positioning, use a slower rate, while for larger movements, a higher speed might be appropriate.
- Manual vs. automatic: Remember that the hand jog function serves for manual manipulation and should not be used as a substitute for programmed automated movements.

## 1.6 ATC Function

The ATC function, or Automatic Tool Changer function, is a crucial feature found in many CNC (Computer Numerical Control) machines, particularly in machining centers. The primary purpose of the Automatic Tool Changer is to enable the CNC machine to change cutting tools automatically during the machining process without manual intervention. This capability enhances efficiency, reduces downtime, and enables the machine to perform complex machining operations seamlessly.



Here are some key aspects of the ATC function:

### a. Tool storage

- CNC machining centers equipped with an ATC typically have a tool magazine or carousel that stores multiple cutting tools.
- The tool storage capacity can vary depending on the machine model and configuration, ranging from a few tools to dozens or even hundreds of tools.

### b. Tool selection

- The CNC control system manages the selection of cutting tools from the tool magazine based on the programmed machining instructions.
- During the machining process, the control system determines which tool is required for the upcoming operation and commands the ATC to retrieve the appropriate tool from the tool magazine.

### c. Tool change process

- When it's time to change tools, the CNC machine pauses the machining operation and moves the spindle to the designated tool change position.
- The ATC then engages with the selected tool in the tool magazine and exchanges it with the currently installed tool in the spindle.
- After the tool change is complete, the CNC machine resumes the machining process with the newly installed tool.

### d. Tool length measurement

- Some CNC machines incorporate tool length measurement probes or sensors to ensure accurate tool positioning and tool length compensation.
- Before installing a new tool, the machine may automatically measure the length of the tool to compensate for any variations and maintain machining accuracy.

### e. Tool change time

- The efficiency of the ATC function is often measured by the tool change time, which refers to the time it takes for the machine to complete a tool change cycle.

- Faster tool change times contribute to higher productivity and throughput in CNC machining operations.

**f. Tool management and monitoring**

- CNC machines with ATC capabilities often include features for tool management and monitoring.
- Operators can use the CNC control interface to view the status of tools in the magazine, track tool life, and receive alerts for tool wear or tool breakage.

Overall, the Automatic Tool Changer function significantly enhances the capabilities and efficiency of CNC machining centers by automating the tool change process. It enables the machine to perform a wide range of machining operations with minimal downtime and operator intervention, making it an indispensable feature in modern manufacturing environments.

## 1.7 Clamping Method of Workpiece with Clamping Kit

A workpiece clamping kit is a set of components and accessories used to securely hold a workpiece in place during machining operations on a CNC (Computer Numerical Control) machine or manual machining equipment. The clamping method employed with a workpiece clamping kit depends on factors such as the workpiece geometry, material, and machining requirements.

Here are some common clamping methods used with workpiece clamping kits:

**a. T-slot clamping**

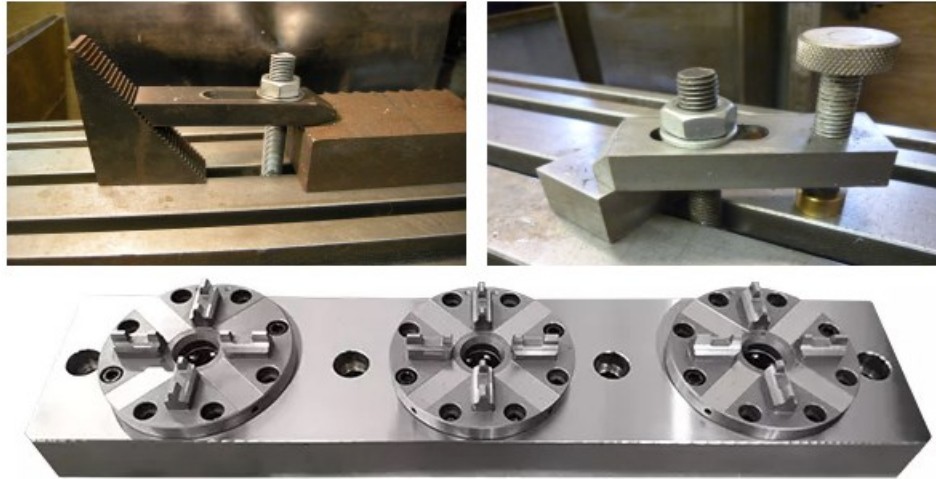
- T-slot clamping involves securing the workpiece to the machine table using T-slot nuts, bolts, and clamping studs.
- T-slot tables have slots or grooves that allow for flexible positioning of clamping components.
- T-slot clamps, step blocks, and clamping studs are used to hold the workpiece securely in place.

**b. Toe clamps**

- Toe clamps are used to secure workpieces to the machine table by applying pressure from the side.
- Toe clamps typically consist of a threaded stud and a clamping foot that presses against the workpiece.
- They are particularly useful for holding irregularly shaped or thin workpieces securely.

**c. Clamping straps and clamps**

- Clamping straps and clamps are used to hold down larger workpieces or fixtures.
- Straps are wrapped around the workpiece and tightened using clamping mechanisms such as cam locks or hand knobs.
- They provide uniform pressure distribution and are often used for securing larger or irregularly shaped workpieces.



**workpiece clamping kit**

**d. Vacuum chucking:**

- Vacuum chucking utilizes suction to hold the workpiece in place on the machine table.
- Vacuum tables or chucks create a seal between the workpiece and the table surface, generating a vacuum that holds the workpiece securely.
- This method is commonly used for holding thin or non-metallic workpieces that may be difficult to clamp using traditional methods.

**e. Magnetic clamping:**

- Magnetic chucks are used to hold ferromagnetic workpieces securely in place.
- The magnetic chuck generates a magnetic field that adheres the workpiece to the machine table.
- This method is often used for quick and easy setup of workpieces with flat and ferrous surfaces.

**f. Modular fixturing systems:**

- Modular fixturing systems consist of a variety of components such as clamps, vise jaws, and locating pins that can be combined and arranged to hold different types of workpieces.
- These systems offer flexibility and repeatability in workpiece setup and are commonly used in high-volume production environments.

The choice of clamping method depends on factors such as the workpiece material, size, shape, and machining requirements. By selecting the appropriate clamping method and utilizing a comprehensive workpiece clamping kit, operators can ensure efficient and accurate machining operations while maintaining workpiece stability and safety.

## Self-Check Sheet 1: Prepare CNC Machining Centre

1. Write the name main components of CNC machining center.

**Answer:**

2. How to need check air pressure?

**Answer:**

3. How to need check coolant level?

**Answer:**

4. How to turn CNC machining center?

**Answer:**

5. What does machine reference depend on?

**Answer:**

6. Write how to Machine Mode and Control Interface to adjust override key?

**Answer:**

7. How to control hand jog rate?

**Answer:**

8. Write key aspects of the ATC function?

**Answer:**

9. Write any two (02) method of clamping method with workpiece clamping kit?

**Answer:**

## Answer Sheet 1: Prepare CNC Machining Centre

### 1. Write the name main components of CNC machining center.

Answer:

- Machine on-off switch
- Emergency switch
- Machine reference switch
- Override controller
- Hand jog rate
- Work holding device
- Tool holder
- Clamping kit
- Vice
- Clamping set
- Parallel bars

### 2. How to need check air pressure?

Answer: Following is the Checking process of air pressure

- **Locate the air pressure gauge:** This gauge should be easily identifiable on the machine, often situated near the air preparation unit or control panel.
- **Consult the machine manual:** The manual will specify the recommended air pressure range for the machine.
- **Compare the gauge reading to the recommended range:** If the pressure falls outside the acceptable range, adjust it using the pressure regulator provided in the air preparation unit (refer to the manual for instructions).
- **Inspect for leaks:** Listen for any hissing sounds or visually check for signs of air escaping from hoses or fittings. Leaks need to be addressed promptly to maintain appropriate pressure.

### 3. What are the needs to check coolant level?

Answer: Following are the needs to check **coolant level**

- **Identify the coolant reservoir:** This is typically a transparent tank located on the machine.
- **Observe the level indicator:** The reservoir should have a level indicator showing the minimum and maximum acceptable coolant levels.
- **Ensure the coolant level is within the recommended range:** If the level is low, top it up with a fresh coolant mixture as per the manufacturer's specifications. Avoid overfilling, as it can overflow during operation.
- **Check the coolant quality:** The coolant should be clean and free of debris or discoloration. If it appears cloudy or discolored, it may need to be replaced. Refer to the manual for recommended coolant change intervals.

### 4. What are the points to consider to turn CNC machining center?

Answer: Here's a general guideline, but remember to consult your machine manual for the most accurate and up-to-date instructions:

**Preparation:**

- Ensure the work area is clear and free of obstructions. This includes materials, tools, and personnel who are not directly involved in the operation.
- Verify that the workpiece is securely fastened in the work holding device.
- Double-check that the cutting tool is properly installed and tightened in the tool holder.
- Clear any debris or chips from the machine table and surrounding areas.
- Close all safety doors and guards.

**Turning on the Machine:**

- Locate the main power switch and turn it on. This is typically located on the control panel or near the machine base.
- Release the emergency stop button (if engaged). The emergency stop button is a large red button typically located on the control panel and should only be pressed in case of an emergency.
- Power on the control panel. This might involve a separate switch or button on the panel itself.
- Follow the specific instructions in the manual to boot up the CNC system and load the desired program. This may involve entering codes, selecting options from a menu, or using a separate computer interface.

**5. What does machine reference depend on?**

**Answer: CNC milling machines:**

- CNC milling machines are used to remove material from a workpiece to create complex shapes and features.
- They utilize rotating cutting tools to perform operations such as drilling, milling, contouring, and pocketing.
- CNC milling machines come in various configurations, including vertical milling machines (VMCs) and horizontal milling machines (HMCs).

**CNC turning machines:**

- CNC turning machines, also known as lathes, are used to produce cylindrical parts by rotating a workpiece against a stationary cutting tool.
- They are commonly used for creating shafts, pins, bushings, and other cylindrical components.
- CNC turning machines can be equipped with single-axis, two-axis, or multi-axis capabilities for performing complex turning operations.

**CNC machining centers:**

- CNC machining centers combine milling, drilling, tapping, and other machining processes into a single machine.
- They offer versatility and efficiency for producing a wide range of parts with high precision.
- CNC machining centers can have three or more axes of motion for multi-axis machining operations.

6. Write how to Machine Mode and Control Interface use to adjust override key?

**Answer: machine mode:**

- Depending on the CNC machine's configuration and control system, override adjustments may be accessible in different machine modes, such as manual mode, auto mode, or MDI (Manual Data Input) mode.
- Operators should ensure they are in the appropriate mode before attempting to adjust override settings.

**Control interface:**

- CNC machines feature control interfaces where operators can access override functions.
- The override key may be labeled as "Feed Override," "Spindle Override," or something similar, depending on the machine's manufacturer and control system.

7. How to control hand jog rate?

**Answer:** The hand jog rate is typically adjustable through dedicated controls on the machine's panel or jog handle. These controls can be:

- Physical knobs or dials: These allow the operator to manually adjust the jog speed in a stepless manner.
- Selection buttons: The machine might offer predefined jog speed options through buttons, allowing the operator to choose between different speeds.
- Software interface: Some advanced CNC machines might allow adjusting the jog rate through the software interface.

8. Write key aspects of the ATC function?

**Answer:**

- Tool Storage
- Tool Selection
- Tool Change Process
- Tool Length Measurement
- Tool Change Time
- Tool Management and Monitoring

9. Write any two (02) method of clamping method with workpiece clamping kit?

**Answer: T-Slot Clamping:**

- T-slot clamping involves securing the workpiece to the machine table using T-slot nuts, bolts, and clamping studs.
- T-slot tables have slots or grooves that allow for flexible positioning of clamping components.
- T-slot clamps, step blocks, and clamping studs are used to hold the workpiece securely in place.

**Magnetic Clamping:**

- Magnetic chucks are used to hold ferromagnetic workpieces securely in place.
- The magnetic chuck generates a magnetic field that adheres the workpiece to the machine table.
- This method is often used for quick and easy setup of workpieces with flat and ferrous surfaces.

## Task Sheet 1.1: Identify the Main Components of CNC Machining Center

### Working Procedure:

1. Follow OSH.
2. Read job sheet and specification sheet.
3. Collect all Tools & Equipment.
4. Wear the PPE.
5. Take Paper and Pen
6. Interpret CNC Milling machine different components.
7. Identify the CNC Milling machine different components.
8. Check the machine parts
9. Note part name
10. Clean working area.
11. Store all equipment safely.

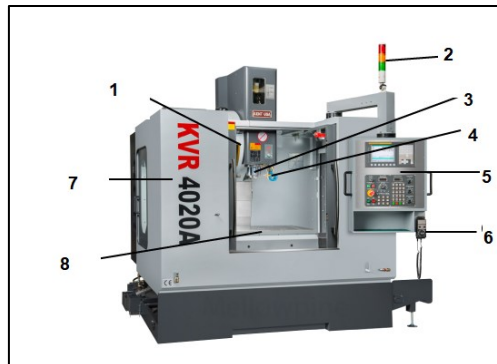
Sl. No.	Machine Part identifying Number	Component Name
1.	1.	Tool Changer
2.	2.	Indicator
3.	3.	Spindle
4.	4.	Coolant system
5.	5.	CNC Controller
6.	6.	CNC Pendant
7.	7.	Table or work Bed
8.	8.	Encloser

## Specification Sheet 1.1 Identify the Main Components of CNC Machining Center

**Name of Job:** Identify the main components of CNC machining center 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 the main components of CNC machining center 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 Tools and Equipment**

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	CNC Milling Machine		Pcs	01	

## **Job Sheet 1.2: Check Coolant Level in CNC Machine**

### **Working Procedure:**

1. Follow OSH
2. Read job sheet and Specification sheet.
3. Collect tools equipment and Checking device
4. Collect PPE as required
5. Ensure that the CNC machine is powered off and that all moving parts have come to a complete stop.
6. Use appropriate personal protective equipment (PPE) such as gloves and safety glasses to protect yourself from any splashing coolant.
7. Located within the CNC machine's enclosure or on the exterior of the machine, depending on the machine's design.
8. Depending on the machine, the coolant reservoir may have a removable cap or lid.
9. Open the reservoir to access the coolant inside.
10. Inspect the Coolant Carefully Level Using a flashlight, if necessary, visually inspect the coolant level inside the reservoir.
11. Marked range on the reservoir or according to the machine manufacturer's specifications.
12. Ensure that the coolant level is above the minimum level to prevent the pump from running dry.
13. Ensure coolant level is below the recommended range, add coolant to the reservoir.
14. Use the appropriate type of coolant recommended by the machine manufacturer.
15. Avoid overfilling the reservoir to prevent spillage or overflow during machine operation.
16. Verified the coolant level and, if necessary, added coolant, securely close the reservoir's cap or lid.
17. Wipe any spills or coolant residue from the reservoir and surrounding areas to maintain a clean working environment.
18. Dispose of any waste coolant properly according to local regulations.
19. Document the coolant level check, noting the date and any actions taken, in the machine's maintenance log or records.
20. Ensuring that the coolant reservoir is properly closed and secured
21. Power on the CNC machine and resume operations as needed.

## Specification Sheet 1.2: Check Coolant Level in CNC Machine

**Name of Job:** Check air pressure, coolant level and lubrication oil 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 check coolant level in CNC machine 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	Coolant /Cutting Oil		Lit	10	
2	Water		Lit	10	

## **Job Sheet 1.3: Turn the CNC Machining Center Power on and Off**

### **Working Procedure:**

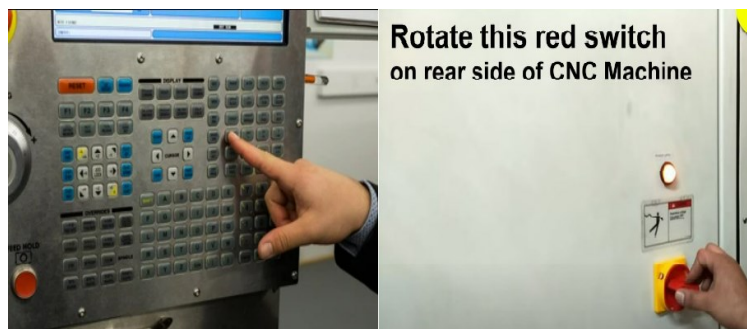
1. Follow OSH
2. Read job sheet and Specification sheet.
3. Collect tools equipment and Checking device
4. Collect PPE as required
5. Ensure that the area around the CNC machine is clear of obstacles and debris.
6. Check that all safety guards and interlocks are in place and functioning properly.
7. Press emergency stop button is accessible and not engaged.
8. Switch on the main power supply to the CNC machine.
9. Perform power switches or controls on for individual components such as the coolant system or spindle.
10. Ensure these are also powered on if required.
11. Boot up the CNC control unit by pressing the power button or following the manufacturer's instructions.
12. Wait for the control unit to initialize and display the control interface.
13. Perform a homing sequence to establish the machine's reference position.
14. This typically involves moving the machine axes to predefined home positions to ensure accurate positioning during operation.
15. Perform Safety Checks:
16. Perform a final visual inspection of the machine setup, including tooling, workpiece, and coolant levels.
17. Ensure all personnel are clear of the machine area before starting operation.
18. Pause the machining program using the control interface.
19. Emergency Stop and End of Program:
20. Retract the cutting tool from the workpiece and move it to a safe position.
21. Remove the machined workpiece from the machine table or fixture.
22. Unload any unused cutting tools from the tool changer or spindle.
23. Follow the manufacturer's instructions or standard operating procedures to properly shut down the CNC machine and control unit.
24. Switch off the main power supply to the machine and any auxiliary components.
25. Clean the machine table, work area, and any spilled coolant or debris.
26. Dispose of any waste materials or coolant properly according to regulations.

## Specification Sheet 1.3: Turn the CNC Machining Center Power on and Off

**Name of Job:** Turn the CNC machining center power on and off 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 Turn the CNC machining center power on and off 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	

## **Job Sheet 1.4: Align Cutting Tools Using ATC Function**

### **Working Procedure:**

1. Follow OSH
2. Read job sheet and Specification sheet.
3. Collect tools equipment and Checking device
4. Collect PPE as required
5. Ensure that the CNC machine is powered on and that all safety precautions are observed.
6. Verify that the ATC system is enabled and functional.
7. Use the CNC control interface to select the desired tool for the upcoming machining operation.
8. Perform inputting the tool number or selecting from a predefined tool library.
9. Perform tool change sequence through the CNC control interface.
10. Command signals the ATC system to begin the tool change process.
11. Perform engaging with a tool, it retracts to a safe position away from the workpiece.
12. Ensures that the spindle and tool holder are clear for tool exchange.
13. Releases the current tool from the spindle or tool holder.
14. Unlocking the tool holder mechanism or activating pneumatic or hydraulic clamping systems.
15. ATC system positions the tool magazine or carousel to access the desired tool.
16. Movement can involve rotation and translation to align the selected tool with the spindle.
17. Selected tool is positioned in front of the spindle, the ATC system engages the tool holder mechanism.
18. Ensures that the cutting tool is properly aligned for accurate machining.
19. After the tool change, the machine may measure the length of the newly inserted tool to ensure accurate machining.
20. After complete Job store all tools and equipment

## Specification Sheet 1.4: Align Cutting Tools Using ATC Function

**Name of Job:** Perform ATC function 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 ATC function 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	

## **Job Sheet 1.5: Clamp the Workpiece with Clamping Kit**

### **Working Procedure:**

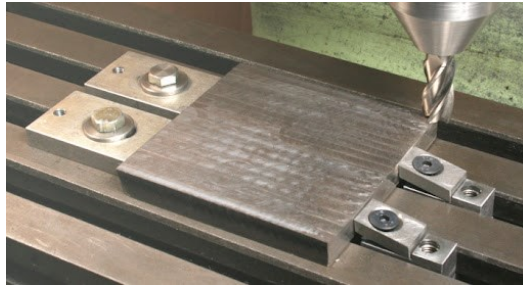
1. Follow OSH
2. Read job sheet and Specification sheet.
3. Collect tools equipment and Checking device
4. Collect PPE as required
5. Choose a clamping kit suitable for the size and shape of your workpiece and compatible with the T-slot size of the machine table.
6. Ensure that the workpiece is clean and free of debris.
7. Verify that all surfaces to be machined are adequately squared and aligned.
8. Position the Workpiece on the Machine Table
9. Place the workpiece on the CNC machine table in the desired machining position.
10. Align the workpiece with the machine's reference points or coordinate system as needed for the machining operation.
11. Determine the optimal locations on the workpiece for clamping.
12. Choose locations that provide secure and stable support for the workpiece during machining.
13. Insert T-nuts from the clamping kit into the T-slots of the machine table at the selected clamping points.
14. Ensure that the orientation of the T-nuts matches the orientation of the clamping components you'll use.
15. Place step blocks or studs from the clamping kit onto the T-nuts in the T-slots.
16. Position them so that they align with the clamping points on the workpiece.
17. Place clamps from the clamping kit onto the step blocks or studs.
18. Align the clamps with the edges or surfaces of the workpiece to securely hold it in place.
19. Use a wrench or torque driver to tighten the clamps securely onto the workpiece.
20. Apply even pressure to ensure that the workpiece is held firmly without distortion.
21. Double-check the tightness of all clamps and ensure that the workpiece is securely held in place.
22. Apply gentle pressure or attempt to move the workpiece to confirm stability.
23. Confirm that the workpiece is securely held in place and ready for machining.
24. After complete the job store all tools and equipment

## Specification Sheet 1.5: Clamp the Workpiece with Clamping Kit

**Name of Job:** Clamp the workpiece with clamping kit 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 Clamp the workpiece with clamping kit 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	

## Learning Outcome -2: Select Cutting Tools

Assessment Criteria	<ol style="list-style-type: none"> <li>1. Various types of cutting tools are identified</li> <li>2. The appropriate sequence of cutting tools for an operation is selected</li> <li>3. Appropriate cutting tools are selected as per job materials</li> <li>4. Appropriate cutting parameters are followed as per manufacturer chart</li> <li>5. Different types of cutting tools inserts are selected</li> </ol>
Conditions and Resources	<ol style="list-style-type: none"> <li>1. Workplace or Simulated Workplace</li> <li>2. CBLM</li> <li>3. Handout</li> <li>4. Laptop</li> <li>5. Multimedia Projector</li> <li>6. Paper, Pen, Pencil,</li> <li>7. Internet Facilities</li> <li>8. White Board and</li> <li>9. Audio Video Devices</li> <li>10. Necessary Tools and Equipment</li> <li>11. Necessary Materials</li> <li>12. Necessary PPE</li> </ol>
Contents	<ol style="list-style-type: none"> <li>1. Various types of cutting tools <ul style="list-style-type: none"> <li>▪ Face mill</li> <li>▪ Drill</li> <li>▪ T-slot cutter</li> <li>▪ Solid and mill</li> <li>▪ Ball nose and mill</li> <li>▪ Bull nose and mill</li> <li>▪ Boring bar</li> <li>▪ Tapping tool</li> <li>▪ Insert</li> <li>▪ Chamfering tools</li> <li>▪ Fillet tools</li> </ul> </li> <li>2. Selection of cutting tools according to appropriate consistency</li> <li>3. Job materials <ul style="list-style-type: none"> <li>▪ Mild steel</li> <li>▪ Aluminum</li> <li>▪ Alloy steel</li> <li>▪ Bronze</li> <li>▪ Copper</li> <li>▪ Stainless steel</li> <li>▪ Tool steel</li> <li>▪ Plastic</li> <li>▪ Taflon</li> </ul> </li> </ol>

	<ul style="list-style-type: none"> <li>▪ Fiber</li> <li>▪ Nylon</li> </ul> <ol style="list-style-type: none"> <li>4. Selection appropriate cutting tool according to the job material</li> <li>5. Manufacturer's chart for selecting cutting tools</li> <li>6. Cutting parameters <ul style="list-style-type: none"> <li>▪ Table feed</li> <li>▪ Spindle speed</li> <li>▪ Cutting depth</li> <li>▪ Cutting speed</li> <li>▪ Stepmover</li> <li>▪ Feed per tooth</li> <li>▪ Plunge rate</li> </ul> </li> <li>7. Cutting tool insert</li> </ol>
Job/Task/Activity	<ol style="list-style-type: none"> <li>1. Identify different types of cutting tools</li> <li>2. Select cutting tools according to the appropriate sequence for an operation</li> <li>3. Select the appropriate cutting tool according to the job material</li> <li>4. Follow the appropriate cutting parameters according to the manufacturer's chart</li> </ol>
Training Method	<ol style="list-style-type: none"> <li>1. Discussion</li> <li>2. Discussion</li> <li>3. Presentation</li> <li>4. Demonstration</li> <li>5. Guided Practice</li> <li>6. Individual Practice</li> <li>7. Project Work</li> <li>8. Problem Solving</li> <li>9. Brainstorming</li> <li>10. Role Play</li> </ol>
Assessment Method	<ol style="list-style-type: none"> <li>1. Written Test</li> <li>2. Demonstration</li> <li>3. Oral questioning</li> <li>4. Portfolio</li> </ol>

## Learning Experience 2: Select Cutting Tools

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 <b>‘Select cutting tools’</b>
2. Read the Information sheet and complete the Self Checks & Check answer sheets on <b>“Select cutting tools”</b>	2. Read Information sheet 1: <b>Select cutting tools</b> Answer Self-check 1: <b>Select cutting tools</b> Check your answer with Answer key 1: <b>Select cutting tools</b>
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	3. Job/Task Sheet and Specification Sheet Task Sheet 2.1: Identify different types of cutting tools Specification Sheet 2.1: Identify different types of cutting tools  Job Sheet 2.2: Select cutting tools according to appropriate sequence of operation Specification Sheet 2.2: Select cutting tools according to appropriate sequence of operation

## Information Sheet 2: Select Cutting Tools









### Learning Objective:



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

- 2.1 Various types of cutting tools
- 2.2 Selection of cutting tools according to appropriate consistency
- 2.3 Job materials
- 2.4 The need to select the appropriate cutting tool according to the job material
- 2.5 Manufacturer's chart
- 2.6 Cutting parameters
- 2.7 Cutting tool insert

### 2.1 Various Types of Cutting Tools

Here are the various types of cutting tools you mentioned:

<p><b>Face mill:</b> A face mill is a cutting tool used for milling flat surfaces, typically using a rotating cutter with multiple cutting edges.</p>	
<p><b>Drill:</b> A drill is a tool used for making holes in various materials by rotating a cutting tool, typically a drill bit, which applies force along the axis of the hole.</p>	
<p><b>T-slot cutter:</b> A T-slot cutter is a type of milling cutter designed to cut T-shaped slots in a workpiece, commonly used in machine tool tables.</p>	
<p><b>Solid end mill:</b> Solid end mills are cutting tools used for milling operations, featuring a solid cutting end. They come in various shapes and sizes for different milling applications.</p>	
<p><b>Ball nose end mill:</b> A ball nose end mill is a type of milling cutter with a rounded tip, suitable for contouring and 3D machining of curved surfaces.</p>	
<p><b>Boring bar:</b> A boring bar is a tool used in machining to enlarge or refine the diameter of a pre-drilled hole. It typically features a single cutting edge.</p>	
<p><b>Tapping tool:</b> A tapping tool is used to create internal threads in a hole, typically by cutting and forming threads into a previously drilled hole.</p>	
<p><b>Insert:</b> Inserts are replaceable cutting tips used in various cutting tools such as milling cutters, turning tools, and drills. They allow for economical tool maintenance and customization.</p>	

<p><b>Chamfering tools:</b> Chamfering tools are used to create chamfers, which are beveled edges at the intersection of two surfaces. Chamfers are commonly used to ease assembly or enhance aesthetics.</p>	
<p><b>Fillet tools:</b> Fillet tools are used to create fillets, which are rounded interior or exterior edges along the contours of a workpiece. Fillets are used to strengthen parts and improve their appearance.</p>	

Each of these cutting tools has its specific application and is used in various machining operations across different industries.

## 2.2 Selection Of Cutting Tools According to Appropriate Consistency

Selecting cutting tools based on appropriate consistency involves matching the characteristics of the cutting tool with the material being machined, as well as considering factors such as tool life, surface finish, and machining efficiency. Here's how you might select cutting tools based on different consistencies of materials:

### a. Soft materials (e.g., Aluminum, Plastics):

- For soft materials, carbide cutting tools are often suitable due to their sharpness and ability to maintain cutting edges.
- High-speed steel (HSS) tools can also be effective for softer materials, especially if cost is a concern.
- Consider using high-rake angle cutting tools for efficient chip evacuation and reduced cutting forces.

### b. Hard materials (e.g., hardened steel, titanium alloys):

- For hard materials, carbide cutting tools with specialized coatings like TiAlN (titanium aluminum nitride) or TiCN (titanium carbonitride) are preferred due to their high wear resistance.
- Consider using inserts or end mills specifically designed for hard milling applications.
- Use cutting parameters such as lower speeds and feeds to reduce heat generation and prolong tool life.

### c. Brittle materials (e.g., ceramics, glass):

- Diamond-coated tools or PCD (polycrystalline diamond) tools are often used for machining brittle materials due to their exceptional hardness and abrasion resistance.
- Carefully control cutting forces and minimize vibrations to prevent chipping or fracturing of the workpiece.

### d. Tough materials (e.g., stainless steel, Inconel):

- Use carbide cutting tools with tough substrates and coatings designed to withstand high temperatures and abrasion.
- Consider using coolant or cutting fluids to reduce heat buildup and improve chip evacuation.
- Use proper tool geometry to manage cutting forces and prevent tool wear.

e. **Abrasive materials (e.g., composites, fiber-reinforced plastics):**

- Specialized carbide or diamond-coated cutting tools are often used for machining abrasive materials to maintain sharp cutting edges and prevent premature wear.
- Consider using proper cutting speeds and feeds to prevent overheating and delamination of the workpiece.
- Use dust extraction systems or cutting fluids to control the spread of abrasive particles and maintain tool performance.

In summary, selecting cutting tools according to appropriate consistency involves understanding the properties of the material being machined and choosing tools with suitable materials, coatings, and geometries to optimize cutting performance and tool life.

## 2.3 Job Materials

Here's a breakdown of suitable cutting tools for the materials you listed, considering their consistency:

### Metals:

- **Mild steel:** This is a relatively soft metal. You can use **High-Speed Steel (HSS)** tools for most applications. For high-production environments or heavy cuts, consider **Tetranized HSS** or **Carbide** tools.
- **Aluminum:** Similar to mild steel, HSS tools work well for aluminum. Carbide tools can be beneficial for high-speed machining or continuous cutting.
- **Alloy steel:** The hardness of alloy steel varies depending on the specific alloy. HSS tools may be sufficient for softer alloy steels, but **Carbide** is generally recommended for most applications.
- **Bronze:** Bronze is softer than steel. HSS tools are suitable, but Carbide can offer advantages in terms of wear resistance and longer tool life.
- **Copper:** Another soft metal, copper can be effectively machined with HSS tools.
- **Stainless steel:** Varies in hardness depending on the grade. HSS can work for softer grades, but **Carbide** is the preferred choice for most stainless-steel machining due to its higher wear resistance.
- **Tool steel:** Extremely hard and wear-resistant. Carbide tools are necessary for machining tool steel.

### Plastics:

- **General plastics:** HSS or **Carbon steel** tools can be used for many plastics. However, specific plastic types may require specialized tool materials to avoid melting or excessive wear.
- **Teflon (PTFE):** Prone to melting at high temperatures. Specialized **coated tools** or **cryogenic cooling** techniques are often used to minimize heat generation during cutting.

- **Fiber:** The type of fiber (e.g., glass fiber, carbon fiber) influences tool selection. HSS or **diamond-coated** tools might be suitable depending on the specific fiber and its reinforcement material.
- **Nylon:** Relatively soft plastic. HSS or even Carbon steel tools can be effective.

## 2.4 Selection Appropriate Cutting Tool According to the Job Material

Selecting the appropriate cutting tool according to the job material is crucial for several reasons:

- a. **Optimized cutting performance:** Different materials have varying properties such as hardness, toughness, abrasiveness, and thermal conductivity. Choosing the right cutting tool ensures that it can effectively withstand the forces and temperatures generated during the machining process, leading to optimized cutting performance.
- b. **Extended tool life:** Using the correct cutting tool for a specific material helps to minimize tool wear and damage. Tools that are designed to handle the particular properties of the material will last longer, reducing the frequency of tool changes and associated downtime.
- c. **Surface finish quality:** The choice of cutting tool influences the surface finish quality of the machined part. Proper tool selection ensures that the cutting edges interact with the material in a way that produces smooth surfaces, minimizing the need for secondary finishing operations.
- d. **Dimensional accuracy:** Different cutting tools may have varying degrees of accuracy when machining different materials. Selecting the appropriate tool helps maintain dimensional accuracy and consistency in the machined parts.
- e. **Prevention of workpiece damage:** Using the wrong cutting tool can result in workpiece damage such as chipping, cracking, or deformation. By selecting the appropriate tool, you reduce the risk of damaging the workpiece and minimize scrap rates.
- f. **Cost efficiency:** Choosing the right cutting tool based on the job material can lead to cost savings in the long run. Although specialized tools may have higher upfront costs, they often provide better performance and longer tool life, resulting in reduced overall machining costs.
- g. **Safety:** Using inappropriate cutting tools for specific materials can pose safety hazards to machine operators and other personnel in the machining environment. Proper tool selection helps ensure safe machining operations.

In summary, selecting the appropriate cutting tool according to the job material is essential for achieving efficient machining processes, maintaining quality standards, and ensuring the safety of personnel and equipment. It involves considering factors such as material properties, cutting tool materials, coatings, geometries, and cutting parameters to achieve the desired machining outcomes.

## 2.5 Manufacturer's Chart for Selecting Cutting Tools

Important of manufacturer's charts when selecting cutting tools:

- a. **Optimized Performance:** Manufacturer's charts provide insights into which cutting tools are best suited for specific materials. This ensures that you choose tools that can deliver optimal performance, including improved cutting speeds, longer tool life, and better surface finish.
- b. **Tool Life and Wear:** Different materials have varying degrees of hardness, abrasiveness, and thermal conductivity. Using the wrong cutting tool can result in premature wear, leading to frequent tool changes and increased production costs. Manufacturer's charts help in selecting tools with the appropriate coatings, geometries, and cutting parameters to maximize tool life.
- c. **Machining Efficiency:** Selecting the right cutting tool enhances machining efficiency by reducing cycle times, minimizing scrap, and improving overall productivity. Manufacturer's recommendations often include cutting speeds, feed rates, and depth of cut guidelines tailored to specific materials, ensuring efficient material removal while maintaining dimensional accuracy.

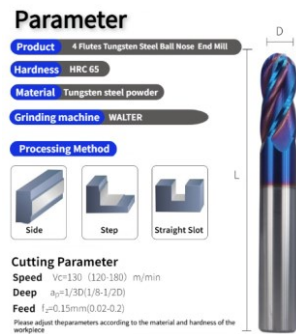


Product Information				
ITEM	R	D1	L	L1
R0.5*Φ2*50*4	R0.5	4.0mm	2.0mm	50mm
R0.75*Φ3*50*4	R0.75	4.0mm	3.0mm	50mm
R1*Φ4*50*4	R1.0	4.0mm	4.0mm	50mm
R1.25*Φ5*50*4	R1.25	4.0mm	5.0mm	50mm
R1.5*Φ6*50*4	R1.5	4.0mm	6.0mm	50mm
R2*Φ8*50*4	R2.0	4.0mm	8.0mm	50mm
R2*Φ8*75*4	R3.0	4.0mm	8.0mm	75mm
R3*Φ12*75*6	R3.0	6.0mm	12mm	75mm
R3*Φ12*50*6	R3.0	8.0mm	12mm	50mm
R4*Φ16*75*8	R4.0	6.0mm	16mm	75mm
R4*Φ16*60*8	R4.0	10mm	16mm	60mm
R5*Φ20*75*10	R5.0	10mm	20mm	75mm
R6*Φ30*75*12	R6.0	12mm	30mm	75mm
R6*Φ24*75*12	R6.0	12mm	24mm	75mm
R7*Φ28*100*14	R7.0	14mm	28mm	100mm
R8*Φ32*100*16	R8.0	16mm	32mm	100mm

- d. **Surface Finish and Dimensional Accuracy:** The choice of cutting tool significantly influences surface finish and dimensional accuracy. Using the wrong tool can result in rough surfaces, burrs, or inaccuracies in the workpiece. Manufacturer's charts help identify tools that produce superior surface finishes and precise dimensions, meeting quality standards and reducing the need for secondary operations.
- e. **Safety and Operator Comfort:** Selecting the appropriate cutting tool contributes to a safer working environment and enhances operator comfort. Properly designed tools reduce the likelihood of tool breakage, chip formation, and vibration, minimizing the risk of accidents and fatigue-related issues among machine operators. Overall, consulting manufacturer's charts ensures that cutting tools are selected based on empirical data and expertise, leading to optimized machining processes, improved product quality, and enhanced operational efficiency. It's essential to consider these recommendations alongside specific machining requirements, environmental conditions, and equipment capabilities to achieve the best results.

## 2.6 Cutting Parameters

Cutting parameters are crucial variables in machining operations that directly affect the efficiency, quality, and tool life. Here's a brief explanation of each cutting parameter:



- a. **Table feed and spindle speed:** These parameters determine the linear speed at which the workpiece moves relative to the cutting tool (table feed) and the rotational speed of the cutting tool (spindle speed). The appropriate combination of table feed and spindle speed ensures efficient material removal while maintaining surface finish and preventing tool wear.
- b. **Cutting depth:** Cutting depth refers to the thickness of the material removed by each pass of the cutting tool. It influences the forces acting on the tool, chip formation, and the amount of material removed per unit of time. Optimal cutting depth balances material removal rates with tool life and surface finish considerations.

Process Parameters and Their Levels

Process Parameters	Parameter designation	Levels		
		1	2	3
Cutting Speed(rpm)	A	2200	2400	2800
Feed rate(mm/min)	B	850	1000	1200
Depth of cut(mm)	C	0.2	0.4	0.6

- c. **Cutting speed:** Cutting speed is the linear speed of the cutting tool's outer surface relative to the workpiece material. It determines how fast the material is removed during machining and is usually measured in surface feet per minute (SFPM) or meters per minute (m/min). The appropriate cutting speed depends on the material being machined, tool material, and desired machining outcomes.
- d. **Stepover:** Stepover is a parameter used in milling operations and refers to the distance the cutting tool moves laterally between adjacent passes. It influences the surface finish, scallop height, and machining time. Optimal stepover values balance productivity with surface quality considerations.
- e. **Feed per tooth:** Feed per tooth refers to the distance the cutting tool advances along the workpiece per tooth engagement during a milling operation. It influences chip thickness, cutting forces, and surface finish. Adjusting feed per tooth helps optimize chip evacuation and prevent tool overload.

- f. **Plunge rate:** Plunge rate refers to the speed at which the cutting tool moves vertically into the workpiece during drilling or plunging operations. It affects chip formation, tool wear, and hole quality. Controlling plunge rate ensures smooth entry into the workpiece and minimizes tool deflection.

Optimizing cutting parameters requires consideration of factors such as material properties, tool characteristics, machine capabilities, and desired machining outcomes. Manufacturers often provide recommended cutting parameter ranges based on empirical data and machining experience.

## 2.7 Cutting Tool Insert

A cutting tool insert, often simply referred to as an insert, is a replaceable cutting tool element used in various machining operations. These inserts are typically made of hard materials like carbide, ceramics, cermet, or cubic boron nitride (CBN). They come in a variety of shapes, sizes, and geometries tailored to specific machining applications.

### Choose a carbide insert

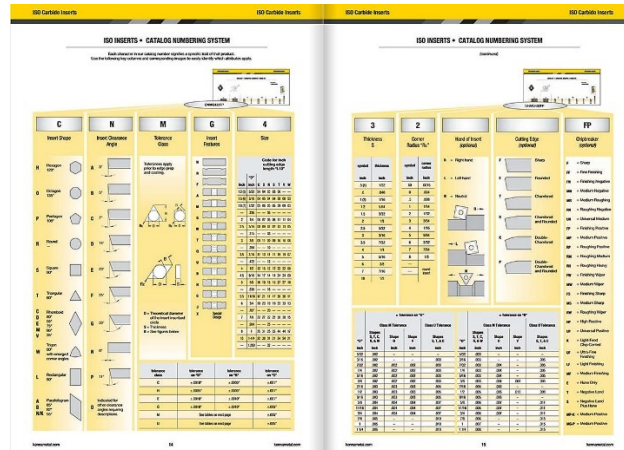
But how do you know what carbide inserts and cutter bodies to buy? And why are there so many different grades of carbide inserts out there? Good questions; we'll start with the second one first. Unlike a few decades ago, when machinists had just a few grades to choose from, there are now dozens of inserts grades, coatings, and chip-breakers available.

Many of these are tailor-made for specific materials or material groups. For instance, a shop making aerospace components can greatly increase efficiency by purchasing carbide inserts designed for tough, heat-resistant superalloys (HRSA) such as Inconel and Hastelloy. The same is true for medical shops, which tend to cut corrosion-resistant, biocompatible materials like 316 stainless steel, cobalt chrome alloy, and titanium. Automakers can dial in their processes by using inserts optimized for cast iron and low carbon steel, while oil and gas producers benefit from tooling that excels in duplex steel.

### Selecting the right carbide insert

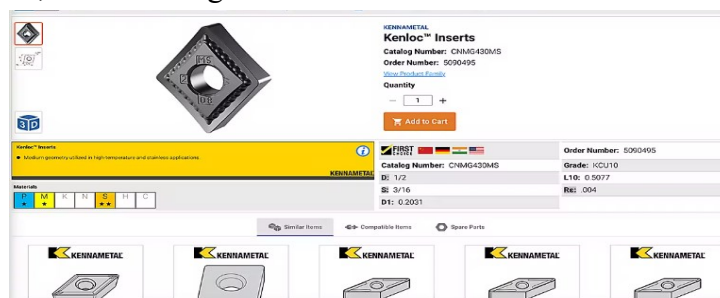
Last but not least is the whole topic of insert nomenclature. It's a deep subject, one filled with exceptions and cutting tool-specific rules. Regardless, most manufacturers follow the ANSI or ISO tool identification system (and sometimes both). We won't get into the details here except to say that it uses an alphanumeric code to specifies an insert's shape (round, square, triangular, etc.), clearance angle (neutral to positive), tolerance (some inserts are pressed to size, while others are ground), the size of the locating hole (if any) and clamping method, its size and thickness, corner radii, and various other defining features (see the chart above for an example).

Complex naming systems aside, however, choosing the right insert for your machining application isn't as difficult as it might appear. That's because cutting tool manufacturers have developed online tool advisors that walk machinists and programmers through the tool selection process.



Here are some key aspects of cutting tool inserts:

- a. **Material:** Cutting tool inserts are made from materials that exhibit high hardness, wear resistance, and toughness. Carbide inserts are the most common due to their excellent combination of these properties. Ceramics, cermets, and CBN inserts are used for specific applications requiring high-speed machining, high-temperature resistance, or machining of hard materials.



- b. **Geometry:** The geometry of a cutting tool insert includes its shape, angles, and cutting edges. Different geometries are designed for specific machining tasks such as turning, milling, drilling, threading, and grooving. Examples of insert geometries include square, triangular, round, and diamond-shaped inserts.
- c. **Insert Coatings:** Many cutting tool inserts are coated with thin layers of hard materials such as titanium nitride (TiN), titanium carbonitride (TiCN), or aluminum oxide (Al<sub>2</sub>O<sub>3</sub>) to enhance their wear resistance, reduce friction, and improve chip evacuation. Coatings can significantly extend the tool life and improve machining performance.
- d. **Mounting:** Inserts are mounted onto cutting tool holders or tool bodies using mechanical clamping mechanisms or fastening screws. The mounting method ensures secure and precise positioning of the insert within the cutting tool assembly.
- e. **Replaceability:** One of the primary advantages of cutting tool inserts is their replaceability. When the cutting edge becomes worn or damaged, the insert can be easily replaced without discarding the entire cutting tool assembly. This reduces downtime and tooling costs, making inserts a cost-effective solution for machining operations.
- f. **Application Specificity:** Cutting tool inserts are available in a wide range of grades and designs optimized for specific materials, cutting conditions, and machining

operations. Manufacturers provide detailed specifications and recommendations to help users select the most suitable inserts for their applications.

Overall, cutting tool inserts play a critical role in modern machining operations, offering versatility, precision, and cost-effectiveness.

## Self-Check Sheet 2: Select Cutting Tools

1. Write the various types of cutting tools.

**Answer:**

2. Which cutting tools are used considering in metals?

**Answer:**

3. Write the name appropriate cutting tool according to the job material.

**Answer:**

4. What are the requirements for consulting the manufacturer's chart when selecting a cutting tool?

**Answer:**

5. Explain Cutting parameters?

**Answer:**

6. Write some key aspects of cutting tool inserts.

**Answer:**

## Answer Sheet 2: Select Cutting Tools

### 1. Write the various types of cutting tools.

**Answer:**

- Face Mill
- Drill
- T-slot Cutter.
- Boring Bar
- Tapping Tool
- Insert
- Solid End Mill
- Ball Nose End Mill
- Chamfering Tools
- Fillet Tools

### 2. Which cutting tools are used considering in metals?

**Answer:**

#### Metals

- Mild steel
- Aluminum
- Alloy steel
- Bronze
- Copper
- Stainless steel
- Tool steel

#### Plastics

- General plastics
- Teflon (PTFE)
- Fiber
- Nylon

### 3. What are the requirements for consulting the manufacturer's chart when selecting a cutting tool?

**Answer:** Here's why it's important to consult manufacturer's charts when selecting cutting tools:

- a **Optimized Performance:** Manufacturer's charts provide insights into which cutting tools are best suited for specific materials. This ensures that you choose tools that can deliver optimal performance, including improved cutting speeds, longer tool life, and better surface finish.
- b **Tool Life and Wear:** Different materials have varying degrees of hardness, abrasiveness, and thermal conductivity. Using the wrong cutting tool can result in premature wear, leading to frequent tool changes and increased production costs. Manufacturer's charts help in selecting tools with the appropriate coatings, geometries, and cutting parameters to maximize tool life.
- c **Machining Efficiency:** Selecting the right cutting tool enhances machining efficiency by reducing cycle times, minimizing scrap, and improving overall productivity. Manufacturer's recommendations often include cutting speeds, feed rates, and depth of cut guidelines tailored to specific materials, ensuring efficient material removal while maintaining dimensional accuracy.
- d **Surface Finish and Dimensional Accuracy:** The choice of cutting tool significantly influences surface finish and dimensional accuracy. Using the wrong tool can result in

rough surfaces, burrs, or inaccuracies in the workpiece. Manufacturer's charts help identify tools that produce superior surface finishes and precise dimensions, meeting quality standards and reducing the need for secondary operations.

- e **Safety and Operator Comfort:** Selecting the appropriate cutting tool contributes to a safer working environment and enhances operator comfort. Properly designed tools reduce the likelihood of tool breakage, chip formation, and vibration, minimizing the risk of accidents and fatigue-related issues among machine operators.

#### 4. Explain Cutting parameters?

**Answer:** Cutting parameters are crucial variables in machining operations that directly affect the efficiency, quality, and tool life. Here's a brief explanation of each cutting parameter:

- a **Table feed and spindle speed:** These parameters determine the linear speed at which the workpiece moves relative to the cutting tool (table feed) and the rotational speed of the cutting tool (spindle speed). The appropriate combination of table feed and spindle speed ensures efficient material removal while maintaining surface finish and preventing tool wear.
- b **Cutting depth:** Cutting depth refers to the thickness of the material removed by each pass of the cutting tool. It influences the forces acting on the tool, chip formation, and the amount of material removed per unit of time. Optimal cutting depth balances material removal rates with tool life and surface finish considerations.
- c **Cutting speed:** Cutting speed is the linear speed of the cutting tool's outer surface relative to the workpiece material. It determines how fast the material is removed during machining and is usually measured in surface feet per minute (SFPM) or meters per minute (m/min). The appropriate cutting speed depends on the material being machined, tool material, and desired machining outcomes.
- d **Stepover:** Stepover is a parameter used in milling operations and refers to the distance the cutting tool moves laterally between adjacent passes. It influences the surface finish, scallop height, and machining time. Optimal stepover values balance productivity with surface quality considerations.
- e **Feed per tooth:** Feed per tooth refers to the distance the cutting tool advances along the workpiece per tooth engagement during a milling operation. It influences chip thickness, cutting forces, and surface finish. Adjusting feed per tooth helps optimize chip evacuation and prevent tool overload.
- f **Plunge rate:** Plunge rate refers to the speed at which the cutting tool moves vertically into the workpiece during drilling or plunging operations. It affects chip formation, tool wear, and hole quality. Controlling plunge rate ensures smooth entry into the workpiece and minimizes tool deflection.

#### 5. Write some key aspects of cutting tool inserts.

**Answer:** A cutting tool insert, often simply referred to as an insert, is a replaceable cutting tool element used in various machining operations. These inserts are typically made of hard

materials like carbide, ceramics, cermet, or cubic boron nitride (CBN). They come in a variety of shapes, sizes, and geometries tailored to specific machining applications.

Here are some key aspects of cutting tool inserts:

- a. **Material:** Cutting tool inserts are made from materials that exhibit high hardness, wear resistance, and toughness. Carbide inserts are the most common due to their excellent combination of these properties. Ceramics, cermet's, and CBN inserts are used for specific applications requiring high-speed machining, high-temperature resistance, or machining of hard materials.
- b. **Geometry:** The geometry of a cutting tool insert includes its shape, angles, and cutting edges. Different geometries are designed for specific machining tasks such as turning, milling, drilling, threading, and grooving. Examples of insert geometries include square, triangular, round, and diamond-shaped inserts.
- c. **Insert Coatings:** Many cutting tool inserts are coated with thin layers of hard materials such as titanium nitride (TiN), titanium carbonitride (TiCN), or aluminum oxide (Al<sub>2</sub>O<sub>3</sub>) to enhance their wear resistance, reduce friction, and improve chip evacuation. Coatings can significantly extend the tool life and improve machining performance.
- d. **Mounting:** Inserts are mounted onto cutting tool holders or tool bodies using mechanical clamping mechanisms or fastening screws. The mounting method ensures secure and precise positioning of the insert within the cutting tool assembly.
- e. **Replaceability:** One of the primary advantages of cutting tool inserts is their replaceability. When the cutting edge becomes worn or damaged, the insert can be easily replaced without discarding the entire cutting tool assembly. This reduces downtime and tooling costs, making inserts a cost-effective solution for machining operations.
- f. **Application Specificity:** Cutting tool inserts are available in a wide range of grades and designs optimized for specific materials, cutting conditions, and machining operations. Manufacturers provide detailed specifications and recommendations to help users select the most suitable inserts for their applications.

Overall, cutting tool inserts play a critical role in modern machining operations, offering versatility, precision, and cost-effectiveness.

## Task Sheet 2.1: Identify Different Types of Cutting Tools

### Working Procedure:

1. Follow OSH.
2. Read job sheet and specification sheet.
3. Collect all Tools & Equipment.
4. Wear the PPE.
5. Take Paper and Pen
6. Interpret Different CNC Milling Cutter
7. Identify the CNC Milling cutter for different Operation
8. Check the CNC Milling cutter
9. Note Cutter name
10. Clean working area.
11. Store all equipment safely.

Sl. No.	Cutter identification Number	Cutter Name
1.	1.	Face Mill Cutter
2.	2.	End Mill Cutter
3.	3.	Roughing End Mill
4.	4.	Ball mill Cutter
5.	5.	Tapper Mill Cutter
6.	6.	Tee slate Cutter
7.	7.	Carbide Tip Holder
8.	8.	Carbide Tip

## Specification Sheet 2.1 Identify Different Types of Cutting Tools

**Name of Job:** Identify different types of cutting tools

**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 different types of cutting tools 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.	Face Mill Cutter	63 x 25	No	1	
2.	End Mill Cutter	12 x 100	No	1	
3.	Roughing End Mill	12 x 100	No	1	
4.	Ball mill Cutter	12 x 100	No	1	
5.	Tapper Mill Cutter	12 x 100	No	1	
6.	Tee slate Cutter	12 x 100	No	1	
7.	Carbide Tip Holder	12 x 100	No	1	
8.	Carbide Tip	12 x 100	No	1	

## **Task Sheet 2.2: Select Cutting Tools According to Appropriate Sequence of Operation**

### **Working Procedure:**

1. Follow OSH.
2. Read job sheet and specification sheet.
3. Collect all Tools & Equipment.
4. Wear the PPE.
5. Take Paper and Pen
6. Interpret Different CNC Milling machine operation
7. Identify the CNC Milling cutter for different Operation
8. Check the CNC Milling cutter
9. Note Cutter name
10. Clean working area.
11. Store all equipment safely.

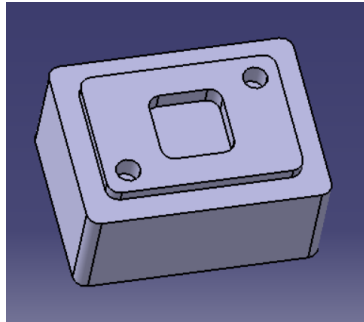
<b>Sl. No.</b>	<b>Operation Name</b>	<b>Cutter Name</b>	<b>Cutter size</b>
1.	Facing	Face Mill Cutter size	63 x 25 mm
2.	Rough profiling	Roughing End Mill	12 x 100 mm
3.	Finishing Profiling	End Mill	10 x 100 mm
4.	Square Pocketing	End Mill	10 x 100 mm
5.	Drilling	Drill	10 x 100 mm

## Specification Sheet 2.2: Select Cutting Tools According to Appropriate Sequence of Operation

**Name of Job:** Select cutting tools according to the appropriate sequence for an operation

**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 cutting tools according to the appropriate sequence for an operation

**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.	Face Mill Cutter	63 x 25	No	1	
2.	End Mill Cutter	12 x 100	No	1	
3.	Roughing End Mill	12 x 100	No	1	
4.	End Mill Cutter	10 x 100	No	1	

## Learning Outcome -3: Operate CNC Machining Center

Assessment Criteria	<ol style="list-style-type: none"> <li>1. CNC machine axes positioning is performed as per job requirement</li> <li>2. Copy, edit, load, and create files are executed</li> <li>3. Program start, program stop, and a single block, dry run, feed hold is performed</li> <li>4. Manual data input (MDI) in CNC machining center is performed</li> <li>5. Auxiliary function keys are performed</li> <li>6. Tool length compensation is performed</li> <li>7. Work offset and tool offset are performed</li> <li>8. Program is transferred from storage devices</li> <li>9. Dry and actual run is executed</li> </ol>
Conditions and Resources	<ol style="list-style-type: none"> <li>1. Workplace or Simulated Workplace</li> <li>2. CBLM</li> <li>3. Handout</li> <li>4. Laptop</li> <li>5. Multimedia Projector</li> <li>6. Paper, Pen, Pencil,</li> <li>7. Internet Facilities</li> <li>8. White Board and</li> <li>9. Audio Video Devices</li> <li>10. Necessary Tools and Equipment</li> <li>11. Necessary Materials</li> <li>12. Necessary PPE</li> </ol>
Contents	<ol style="list-style-type: none"> <li>1. CNC machine axis positioning <ul style="list-style-type: none"> <li>▪ X-axis</li> <li>▪ Y-axis</li> <li>▪ Z-axis</li> </ul> </li> <li>2. Copy, edit, load and file</li> <li>3. Program start, program stop, and a single block, dry run, feed hold</li> <li>4. Manual Data in CNC Machining Center</li> <li>5. What is the auxiliary function? <ul style="list-style-type: none"> <li>▪ F1</li> <li>▪ F2</li> <li>▪ F3</li> <li>▪ F4</li> </ul> </li> <li>6. Tool length compensation is done</li> <li>7. Work offset and tool offset <ul style="list-style-type: none"> <li>▪ What is the world</li> <li>▪ Jug handle</li> <li>▪ Jag har</li> <li>▪ Axis</li> </ul> </li> </ol>

	<ul style="list-style-type: none"> <li>▪ Edge finder</li> <li>▪ Probe tool</li> </ul> <ol style="list-style-type: none"> <li>8. Storage device <ul style="list-style-type: none"> <li>▪ Pen drive</li> <li>▪ Lan</li> <li>▪ Device manager</li> </ul> </li> <li>9. Dry and genuine run <ul style="list-style-type: none"> <li>▪ Load the program</li> <li>▪ Memory mode</li> <li>▪ Graphics mode</li> <li>▪ Start the cycle</li> </ul> </li> </ol>
Job/Task/Activity	<ol style="list-style-type: none"> <li>1. Accomplish CNC machine axis positioning according to job requirements</li> <li>2. Complete copy, edit, load and file creation</li> <li>3. Program start, program stop, and complete a single block, dry run, feed hold</li> <li>4. Complete manual data input (MDI) in the CNC machining center</li> <li>5. Complete the auxiliary function key</li> <li>6. Complete the tool length compensation</li> <li>7. Complete work offset and tool offset</li> <li>8. Transfer programs from storage devices</li> <li>9. Dry and complete the actual run</li> </ol>
Training Method	<ol style="list-style-type: none"> <li>1. Discussion</li> <li>2. Presentation</li> <li>3. Demonstration</li> <li>4. Guided Practice</li> <li>5. Individual Practice</li> <li>6. Project Work</li> <li>7. Problem Solving</li> <li>8. Brainstorming</li> </ol>
Assessment Method	<ol style="list-style-type: none"> <li>1. Written Test</li> <li>2. Demonstration</li> <li>3. Oral questioning</li> <li>4. Portfolio</li> </ol>

## Learning Experience 3: Operate CNC Machining Center

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 <b>‘Operate CNC machining center</b>
2. Read the Information sheet and complete the Self Checks & Check answer sheets on “ <b>Operate CNC machining center</b> ”	2. Read Information sheet 1: <b>Operate CNC machining center</b> Answer Self-check 1: <b>Operate CNC machining center</b> Check your answer with Answer key 1: <b>Operate CNC machining center</b>
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	3. Job/Task Sheet and Specification Sheet Job Sheet 3.1: Accomplish CNC machine axis positioning according to job requirements Specification Sheet 3.1 Accomplish CNC machine axis positioning according to job requirements  Job Sheet 3.2: Complete copy, edit, load and file creation as per specification Specification Sheet 3.2: Complete copy, edit, load and file creation as per specification  Job sheet 3.3: Perform work offset and tool offset in CNC Milling machine Specification Sheet 3.3: Perform work offset and tool offset in CNC Milling machine Job Sheet 3.4: Accomplish program start, program stop and complete a single block, dry run, feed hold as per specification Specification Sheet 3.4 Accomplish program start, program stop, and complete a single block, dry run, feed hold as per specification  Job Sheet 3.5: Perform Dry run in CNC machine as per specification Specification Sheet 3.5: Perform Dry run in CNC machine as per specification  Job Sheet 3.6: Input a part program as per specification. Specification Sheet 3.6 Input a part program as per specification.

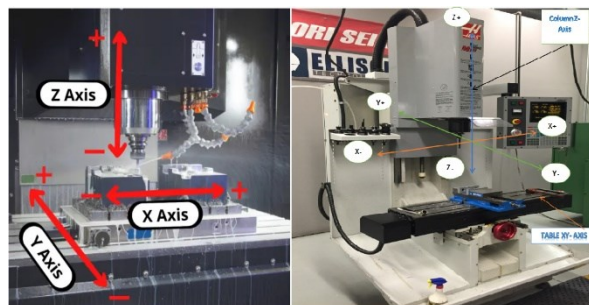
## Information Sheet 3: Operate CNC Machining Center

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

- 3.1 CNC machine axis positioning
- 3.2 Copy, edit, load and file
- 3.3 Program start, program stop, and a single block, dry run, feed hold
- 3.4 Manual Data in CNC Machining Center
- 3.5 What is the auxiliary function?
- 3.6 Tool length compensation is done
- 3.7 Work offset and tool offset
- 3.8 Storage device
- 3.9 Dry and genuine run

### 3.1. CNC Machine Axis Positioning

CNC machine axis positioning refers to the movement and control of the various axes within a computer numerical control (CNC) machine. Here's a breakdown of the primary axes commonly found in CNC machines:



#### a. X-axis

- The X-axis represents the horizontal movement of the CNC machine's cutting tool or workpiece along the length of the machine bed.
- It typically runs from left to right or right to left, depending on the machine's orientation.
- Movement along the X-axis enables machining operations to be performed across the width of the workpiece.

#### b. Y-axis

- The Y-axis represents the horizontal movement of the CNC machine's cutting tool or workpiece along the width of the machine bed.
- It typically runs from front to back or back to front, perpendicular to the X-axis.
- Movement along the Y-axis enables machining operations to be performed along the length of the workpiece.

#### c. Z-axis

- The Z-axis represents the vertical movement of the CNC machine's cutting tool or workpiece.
- It typically runs perpendicular to both the X-axis and Y-axis, moving up and down.
- Movement along the Z-axis enables machining operations to be performed at different depths within the workpiece.

These three axes working together allow for precise control of the cutting tool's position in three-dimensional space. The CNC program sends instructions to the machine controller, specifying the exact X, Y, and Z coordinates for each point along the desired toolpath. This enables the creation of complex shapes and features on the workpiece.

### **3.2. Copy, Edit, Load And File**

The process of copying, editing, loading, and filing CNC (Computer Numerical Control) programs involves several steps to prepare the machine for machining operations. Here's an overview of each step:

#### **a. Copy**

- Copying CNC programs involves duplicating the program files from one storage medium to another. This could be from a USB drive, network storage, or another location where CNC programs are stored.
- Use the appropriate file management software or file transfer methods to copy the CNC program files to the desired location accessible by the CNC machine.

#### **b. Edit**

- Editing CNC programs may be necessary to modify machining parameters, toolpaths, or other instructions based on specific job requirements or tooling changes.
- Use a CNC programming software or text editor capable of editing the CNC program files (\*.NC, \*.G-code, etc.).
- Make the necessary modifications to the program, ensuring accuracy and compliance with machining requirements.

#### **c. Load**

- Loading CNC programs involves transferring the edited program files from the storage medium to the CNC machine's control system.
- Access the machine's control interface or software platform.
- Navigate to the file management section and select the option to load or import CNC program files.
- Follow the prompts to locate and load the desired program files into the CNC machine's memory.

#### **d. File**

- Filing CNC programs refers to organizing and storing the program files in a structured manner for future use and reference.
- Create directories or folders within the CNC machine's storage system to categorize and organize program files based on criteria such as job type, part number, or machining process.

- Use descriptive filenames that clearly identify the contents and purpose of each CNC program file.
- Regularly backup CNC program files to prevent data loss and ensure availability for future use.

By following these steps, CNC operators and programmers can efficiently manage CNC program files, make necessary edits, and load programs onto CNC machines to initiate machining operations. Effective file management practices help streamline workflow, improve productivity, and maintain accurate documentation of machining processes.

### 3.3. Program Start, Program Stop, And A Single Block, Dry Run, Feed Hold

In CNC (Computer Numerical Control) machining, several commands and functions are used to control the execution of machining programs. Here's an explanation of program start, program stop, single block, dry run, and feed hold commands:

#### a. Program starts

- Program start command initiates the execution of the CNC machining program loaded into the control system.
- Once the program start command is given, the CNC machine begins to execute the instructions specified in the program, such as tool movements, spindle speeds, and feed rates.

#### b. Program stops

- Program stop command halts the execution of the CNC machining program.
- When the program stop command is activated, the CNC machine stops all tool movements, spindle rotation, and other machining operations.
- Program stop is used to pause machining operations for various reasons, such as tool changes, inspection, or adjustments.

#### c. Single block

- Single block mode allows the CNC operator to execute the machining program one block at a time.
- In single block mode, the CNC machine stops after each programmed block of instructions, allowing the operator to verify tool movements and machining operations before proceeding to the next block.
- Single block mode is useful for debugging programs, troubleshooting toolpath errors, and ensuring machining accuracy.

#### d. Dry run

- Dry run command simulates the execution of the CNC machining program without actually cutting material.
- In dry run mode, the CNC machine moves the tool along the programmed toolpath at reduced speeds, allowing the operator to verify clearance, check for collisions, and validate the machining sequence.
- Dry run is commonly used to preview machining operations, identify potential issues, and minimize the risk of tool or workpiece damage during actual production runs.

#### e. Feed hold

- Feed hold command temporarily suspends the movement of the CNC machine's axes during program execution.

- When feed hold is activated, the CNC machine stops advancing the tool along the programmed toolpath, but spindle rotation and coolant systems may remain active.
- Feed hold is used to pause machining operations quickly in response to unexpected events, such as tool breakage, workpiece interference, or safety concerns.

These commands and functions provide CNC operators with precise control over machining operations, enabling efficient program execution, error detection, and safety management throughout the manufacturing process. By utilizing these features effectively, operators can optimize machining productivity, ensure part quality, and minimize production downtime.

### 3.4. Manual Data in CNC Machining Center

In CNC (Computer Numerical Control) machining, manual data entry involves inputting specific parameters, commands, or offsets directly into the CNC machines control system. While much of CNC programming is done using CAM (Computer-Aided Manufacturing) software, manual data entry remains an essential aspect of CNC operations for various tasks. Here's how manual data entry is used in CNC machining centers:

#### a. Tool Offsets

- CNC machines require accurate tool offset data to position cutting tools correctly relative to the workpiece.
- Operators manually enter tool length offsets, tool radius offsets, and tool wear offsets into the CNC machine's control system based on measurements taken from tool presters or manual tools.

#### b. Workpiece Coordinates

- Operators may manually input workpiece coordinate offsets to align the CNC machine's reference point with specific locations on the workpiece.
- Workpiece coordinate offsets adjust the machine's positioning relative to the workpiece, ensuring precise machining at desired locations.

#### c. Program Variables

- Some CNC programs include variables that can be modified manually at the machine interface.
- Operators may adjust program variables such as feed rates, spindle speeds, tool change positions, or coolant settings based on real-time machining conditions or specific job requirements.

#### d. Incremental Positioning

- CNC machines allow operators to manually input incremental movements along individual axes.
- Operators can enter specific distances or angles to move the cutting tool or workpiece incrementally, facilitating setup, tool changes, or manual machining operations.

**e. Program Editing**

- In some cases, operators may need to manually edit CNC program code directly at the machine control panel.
- Manual program editing enables operators to make quick adjustments or corrections to program logic, toolpaths, or machining parameters without accessing external programming software.

**f. Overrides and Commands**

- CNC machines feature manual overrides and commands that allow operators to adjust machine functions in real-time.
- Operators can manually adjust feed rate overrides, spindle speed overrides, rapid traverse overrides, and other machine parameters to optimize machining performance or respond to changing conditions.

Manual data entry in CNC machining centers provides operators with flexibility, control, and adaptability in executing machining operations. By accurately inputting critical data and parameters, operators ensure efficient production, maintain machining accuracy, and respond effectively to dynamic manufacturing requirements. However, manual data entry requires careful attention to detail and proper verification to prevent errors and ensure safe and accurate machining processes.

### 3.5. Auxiliary Function

Auxiliary functions in CNC (Computer Numerical Control) machining typically refer to specific commands or operations that are not directly related to primary machining tasks but serve various supportive functions for the CNC machine's operation or control system. Here's a general overview of how these auxiliary function keys might be utilized in CNC machining:

- F1:** This function key could be programmed to perform a specific auxiliary function, such as activating coolant flow, toggling between different machining modes, or accessing a particular menu or setting in the CNC control interface.
- F2:** Similar to F1, F2 might be programmed to execute a particular auxiliary function or command, such as initiating tool changes, adjusting feed rates, or selecting different tool offsets.
- F3:** F3 could be assigned to another auxiliary function or operation, such as engaging or disengaging spindle rotation, activating tool length compensation, or pausing/resuming machining operations.
- F4:** F4 might be programmed to perform additional auxiliary functions or commands, such as toggling between different workpiece coordinate systems, selecting specific program modes, or adjusting rapid traverse rates.

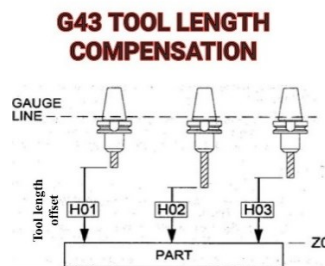
The specific functions assigned to F1, F2, F3, and F4 can vary widely depending on the CNC machine's make and model, as well as the software and control system used.

### 3.6. Tool Length Compensation

Tool length compensation (TLC) is achieved in CNC machining through a combination of **G-codes** (the main programming language for CNC machines) and **tool offset values** stored in the machine's memory. Here's a breakdown of the process:

#### a. G-Codes for Enabling/Disabling TLC

- **G43:** This G-code activates tool length compensation. It tells the machine to consider the stored tool offset value and adjust the programmed Z-axis positions accordingly.



- **G44:** This G-code is less commonly used and works opposite to G43. It instructs the machine to subtract the tool offset value from the programmed Z-axis positions.
- **G49:** This G-code deactivates tool length compensation. The machine will then use the raw programmed Z-axis positions without any adjustments.

#### b. Tool offset values

- Each tool on the CNC machine has a corresponding tool offset value stored in the machine's memory. This value represents the difference between the actual length of the tool (from the tool tip to the tool holder reference point) and a reference length programmed in the machine.

#### c. How it works

- When a tool is selected for a machining operation, the associated tool offset value is automatically loaded.
- If G43 is active, the machine adds the tool offset value to each programmed Z-axis position in the CNC program. This ensures the cutting tool tip reaches the desired depth relative to the workpiece surface, regardless of slight variations in actual tool length.

By implementing tool length compensation effectively, you can achieve precise and consistent machining results while streamlining your CNC machining workflow.

### 3.7. Work Offset and Tool Offset

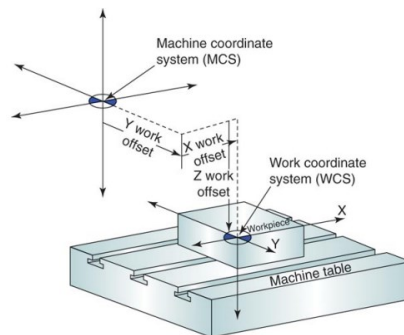
Let's break down the terms "work offset" and "tool offset," along with the other terms you've mentioned:

**a. Work offset**

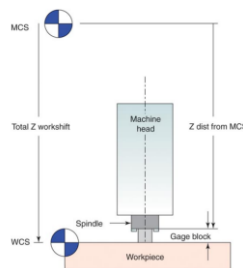
- Work offset refers to a reference point or coordinate system established on the CNC machine relative to the workpiece being machined.
- It defines the origin (zero point) from which the CNC machine calculates tool movements and machining operations.
- Work offsets allow operators to set up and machine multiple parts with different orientations or positions on the machine bed without reprogramming the entire machining program.
- Common work offset systems include G54, G55, G56, etc., each representing a unique coordinate system for positioning the workpiece.

**b. Set up work offset**

Difference between MCS and WCS on X-, Y and Z-axes

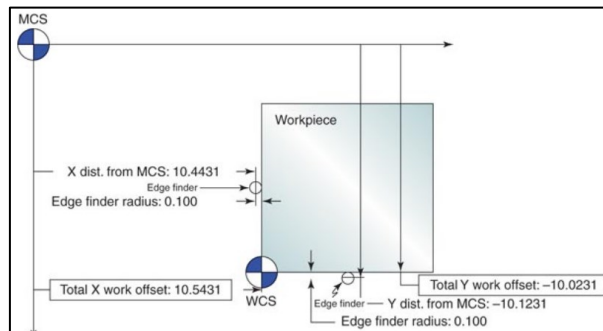
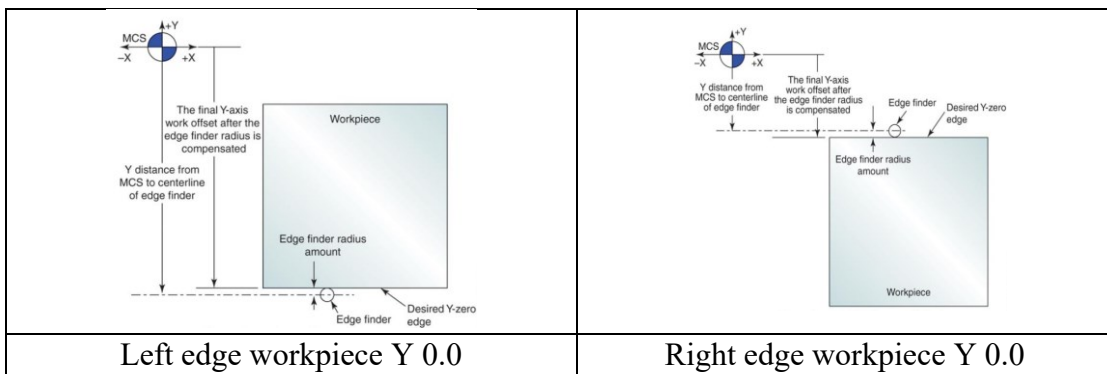
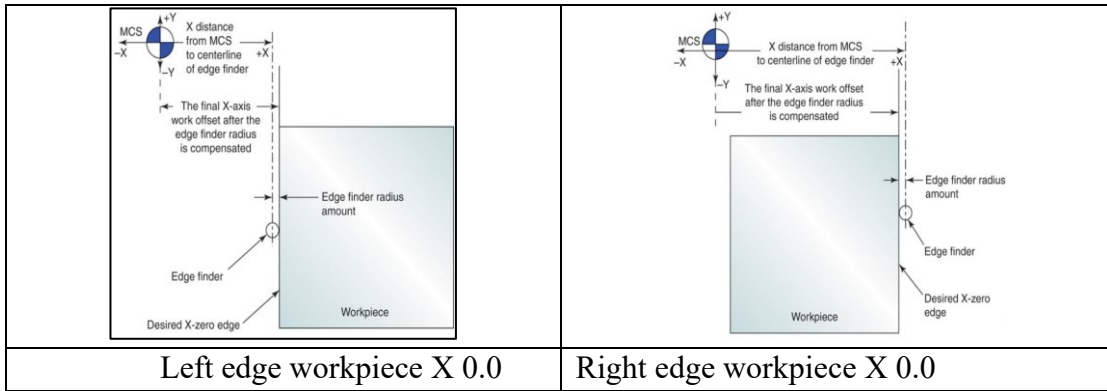


Workpiece offset, or work shift, is the distance from the MCS origin to the desired location of the WCS origin.

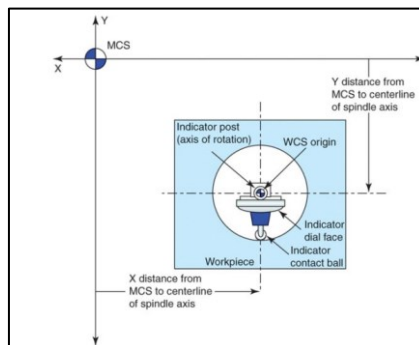


This front view shows a VMC spindle face and a gage block being used to determine the Z-axis work offset. Note the relationship of the MCS to the WCS.

<p>An edge is located with an edge finder.</p>	<p>Finding the center of a hole by sweeping with a dial indicator.</p>



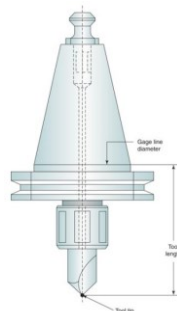
A dimensioned example of how a work offset is derived for a workpiece having the origin in the front left corner.



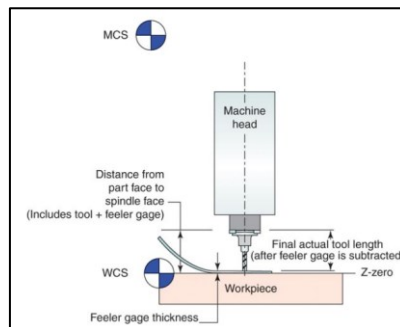
This top view shows that when a hole center is to be used as the X/Y location of the work origin, the hole center can be found by sweeping with a dial indicator. This aligns the spindle centerline with the hole centerline.

### c. Tool offset

- Tool offset is the adjustment made to compensate for variations in tool length, diameter, or geometry during machining operations.
- It ensures that the tool tip aligns precisely with the programmed toolpath coordinates.
- Tool offset values are entered into the CNC machine's control system and applied automatically during tool changes or toolpath calculations.
- Tool length and tool radius offsets are the most common types of tool offsets used in CNC machining.



Tool length is measured from the gage line on the tool holder's taper to the tool tip.

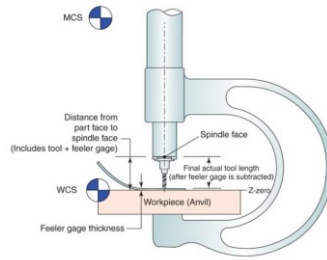


A VMC Z-axis can be used to measure tool length. The length offset is found by touching the tool tip off of the work Z-zero surface with a feeler gage and then subtracting the feeler gage thickness. The objective is to determine the dimension labeled “Final actual tool length.”

```

OFFSET / GEOMETRY                                00020 N0340
NO.      DATA      NO.      DATA
G 001    3.7894     G 009    5.2220
G 002    8.7570     G 010    3.4470
G 003    4.6930     G 011    4.1330
G 004    5.4870     G 012    4.6398
G_005    5.5100     G 013    4.8256
G 006    5.3880     G 014    4.2635
G 007    6.0820     G 015    5.7910
G 008    4.5632     G 016    6.1477
ACTUAL POSITION (RELATIVE)
X      0.0000     Y      0.0000
Z      0.0000
NO. 005 - S 0 T
16:59:12      HNDL
( WEAR )(MACRO )( MENU )( WORK )( )
  
```

A typical milling geometry offset page on the machine display screen. These numbers reflect the tool length from gage line to the tool tip in its original and unworn state.



The way a VMC Z-axis can be used to measure tool length can be compared to the way a micrometer measure



A typical milling wear offset page on the machine display screen. The number that will be entered into the wear offset will be an incremental adjustment from a baseline of zero tool length wear.

**a. Axis**

- In CNC machining, an axis refers to a direction of movement along a linear or rotational path.
- Common CNC machine axes include X, Y, Z for linear movements and A, B, C for rotational movements.
- Each axis corresponds to a specific direction of motion and is controlled by the CNC machine's control system.

**b. Edge finder**

- An edge finder is a precision tool used to locate the edges of a workpiece accurately.
- It typically consists of a cylindrical body with a spring-loaded tip that moves laterally when it contacts the workpiece edge.
- Machinists use edge finders in conjunction with CNC machines to establish work offsets and accurately position the tool relative to the workpiece edges.

If you use a milling machine to machine parts, then you're likely to be familiar with edge finders and the way they work.

Edge finders, sometimes also defined as wigglers or wobblers, are simple tools used to locate the edges of a workpiece or the center of a hole. They are also by far the most commonly used device to locate Part Zero.

They are held in a collet, end mill holder or chuck mounted in the spindle of a milling machine or drill press and come in different sizes and shapes, depending upon the inside diameter of the collet used.

## b. Wigglers

The wiggler is a very common and popular type of edge finder. It comes with a number of different points, or probes, that go in inside the body. When the spindle of the machine is set to rotate, the center finder will begin to wiggle energetically. As it starts to move closer to the edge of a part, however, the wiggling will start to subside until the tool appears almost still.

**PROS:** Not affected by collet run-out

**CONS:** Requires the machinist to understand when the two sections are aligned, which can lead to inaccuracies.



Starrett S828HZ Wiggler and  
Center Finder Complete with  
Case and 4 Attachments

## c. Mechanical edge finders

Mechanical edge finders are relatively cheap and work similarly to the wigglers. They have two sections: the body and the orbiting tip. The edge finder has three modes: orbiting, centered and offset. The edge of the part is located at the transition point between centered and offset,

**PROS:** Inexpensive and is not affected by collet run-out.

**CONS:** Requires the machinist to understand when the two sections are aligned, which can lead to inaccuracies.

OUR RECOMMENDATION



Mitutoyo 050101, Edge Finder,  
3/8" Dia. Shank, .200" Dia. Tip

## d. Electronic edge finders

Electronic edge finders are designed to light up when in contact with the edge. They pass the current from the spindle of the machine all the way through to the part and have an LED light that indicates when the finder touches the part.

**PROS:** Very easy to read.

**CONS:** Affected by the collet run out



Fowler Full Warranty 54-575-600-0  
Electronic Edge Finder with Cylindrical  
Tip, 0.200" Stylus, 1/2" Shank

#### e. 3d sensors

3D sensors, sometimes also called 3D tasters, are probably the most practical and easy to use tools to find edges, although they are not cheap. Unlike the other tools listed, they can measure in X, Y, and also Z. The tip of the sensor is connected to a dial gauge. When the dial gauge shows zero, the spindle axis is exactly on the workpiece edge



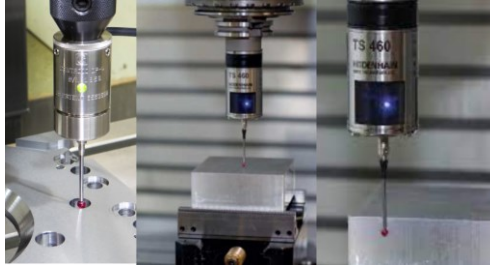
#### f. Probe tool

- A probe tool, or probing system, is a device used to measure and verify workpiece dimensions, tool lengths, and tool offsets automatically.
- It uses sensors or probes to make physical contact with the workpiece or tool and sends measurement data to the CNC machine's control system.
- Probe tools enable efficient tool setup, workpiece inspection, and in-process measurement during CNC machining operations.



#### g. Working procedure of CNC touch probe

The combination uses of a Touch Probe and Tool Setter when easily integrated into your CNC control processes and software can become instrumental in making massive improvements your resultant total customer satisfaction. Fully automated use of these tools will also make your machine shop huge financial savings with productivity, scrap and yield improvements over conventional methods.



Touch probes, or sometimes known as triggering touch probes or touch-trigger probes. They work by touch parts of stock material, component or tool to collect a number of required data points. The machine operator can use a probe like this to precisely determine the position of components or stock materials. This precision information can then be used to update machine settings, offsets and position data in your CNC control software or CAM model.

Touch probes are often used on milling machines in conjunction with an automatic tool change carousel system, but can also be used by manual exchanging a cutting tool for the probe. Touch Probes are often used in conjunction with a CNC Milling Tool Height Setter.

These terms are essential concepts in CNC machining, contributing to the precision, accuracy, and efficiency of machining processes.

### **3.8. Storage Device**

Here's an overview of each term related to storage devices:

#### **a. Pen Drive**

- A pen drive, also known as a USB flash drive, thumb drive, or memory stick, is a portable data storage device.
- It typically connects to a computer's USB (Universal Serial Bus) port and allows users to store, transfer, and access files and data.
- Pen drives are convenient for transporting files between different computers and devices due to their small size and high storage capacity.

#### **b. LAN (Local Area Network)**

- A LAN is a network that connects computers and devices within a limited geographical area, such as a home, office, or building.
- LANs enable users to share resources, such as files, printers, and internet connections, among connected devices.
- Devices on a LAN communicate with each other using Ethernet cables or wireless connections, allowing for data transfer and collaboration.

#### **c. Device Manager**

- Device Manager is a Windows utility that allows users to view and manage hardware devices installed on a computer.
- It provides information about the status, properties, and drivers of hardware devices, such as disk drives, graphics cards, network adapters, and USB controllers.

- Device Manager enables users to update device drivers, disable or enable devices, troubleshoot hardware issues, and view system resources allocated to devices.

Each of these storage-related terms plays a significant role in computing and data management, facilitating storage, connectivity, and device management in various contexts.

### 3.9. Dry And Genuine Run

It seems like you're referring to terms related to CNC machining operations. Here's an explanation of each term:

#### a. Dry Run

- A dry run in CNC machining refers to simulating the machining operation without actually cutting material.
- It allows operators to verify the toolpaths, check for potential collisions, and ensure that the machining program functions as intended.
- Dry runs are typically performed at reduced feed rates and spindle speeds to minimize the risk of damage to the machine or tooling.

#### b. Genuine Run

- "Genuine run" is not a standard term in CNC machining. It's possible that it's a typo or a term specific to certain CNC setups or processes.
- In the context of CNC machining, operators often refer to running the actual machining program on the workpiece as the "actual run" or simply executing the program.

#### c. Load the Program

- Loading the program involves transferring the CNC machining program from external storage (such as a USB drive or network) into the memory of the CNC machine's control system.
- The program contains instructions for tool movements, spindle speeds, feed rates, tool changes, and other parameters necessary for machining the workpiece.

#### d. Memory Mode

- Memory mode is a CNC machine operating mode where the machining program is stored and executed directly from the machine's internal memory.
- In memory mode, the CNC control system retrieves the program instructions and executes them sequentially to perform the machining operations.

#### e. Graphics Mode:

- Graphics mode is a feature available in some CNC control systems that allows operators to visualize the machining program using graphical representations.
- It provides a graphical interface to view toolpaths, workpiece geometry, tool positions, and other machining parameters.
- Graphics mode aids operators in understanding and analyzing the machining process, identifying potential issues, and optimizing toolpaths for efficiency and accuracy.

#### f. Start the Cycle:

- Starting the cycle initiates the execution of the machining program on the CNC machine.
- It involves activating the machine's control system to begin moving the cutting tools, adjusting spindle speeds, and performing other operations specified in the program.
- Operators monitor the machining process during the cycle to ensure that the workpiece is machined according to the desired specifications.

These terms are essential in CNC machining operations and contribute to the efficient and accurate production of machined parts.

### Self-Check Sheet 3: Operate CNC Machining Center

1. Briefly explain the axis of CNC machine to be positioned.

**Answer:**

2. Write the steps to prepare the CNC machine for machining operations process of copying, and filing programs.

**Answer:**

3. Which functions are used to control the execution of machining programs?

**Answer:**

4. Which options are used manual data in CNC machining center

**Answer:**

5. What are the auxiliary functions?

**Answer:**

6. Write the tool length compensation process.

**Answer:**

7. Shortly brief work offset and tool offset process.

**Answer:**

8. What is the storage device?

**Answer:**

9. Write the dry and genuine run in CNC program.

**Answer:**

## Answer Sheet 3: Operate CNC Machining Center

### 1. Briefly explain the axis of CNC machine is to be positioned.

**Answer:** CNC machine axis positioning refers to the movement and control of the various axes within a computer numerical control (CNC) machine. Here's a breakdown of the primary axes commonly found in CNC machines:

#### X-axis

- The X-axis represents the horizontal movement of the CNC machine's cutting tool or workpiece along the length of the machine bed.
- It typically runs from left to right or right to left, depending on the machine's orientation.
- Movement along the X-axis enables machining operations to be performed across the width of the workpiece.

#### Y-axis

- The Y-axis represents the horizontal movement of the CNC machine's cutting tool or workpiece along the width of the machine bed.
- It typically runs from front to back or back to front, perpendicular to the X-axis.
- Movement along the Y-axis enables machining operations to be performed along the length of the workpiece.

#### Z-axis

- The Z-axis represents the vertical movement of the CNC machine's cutting tool or workpiece.
- It typically runs perpendicular to both the X-axis and Y-axis, moving up and down.
- Movement along the Z-axis enables machining operations to be performed at different depths within the workpiece.

These three axes working together allow for precise control of the cutting tool's position in three-dimensional space. The CNC program sends instructions to the machine controller, specifying the exact X, Y, and Z coordinates for each point along the desired toolpath. This enables the creation of complex shapes and features on the workpiece.

### 2. Write the steps to prepare the CNC machine for machining operations process of copying, and filing programs.

**Answer:** The process of copying, editing, loading, and filing CNC (Computer Numerical Control) programs involves several steps to prepare the machine for machining operations. Here's an overview of each step:

#### Copy:

- Copying CNC programs involves duplicating the program files from one storage medium to another. This could be from a USB drive, network storage, or another location where CNC programs are stored.
- Use the appropriate file management software or file transfer methods to copy the CNC program files to the desired location accessible by the CNC machine.

#### Edit

- Editing CNC programs may be necessary to modify machining parameters, toolpaths, or other instructions based on specific job requirements or tooling changes.
- Use a CNC programming software or text editor capable of editing the CNC program files (\*.NC,\*.G-code, etc.).
- Make the necessary modifications to the program, ensuring accuracy and compliance with machining requirements.

#### **Load**

- Loading CNC programs involves transferring the edited program files from the storage medium to the CNC machine's control system.
- Access the machine's control interface or software platform.
- Navigate to the file management section and select the option to load or import CNC program files.
- Follow the prompts to locate and load the desired program files into the CNC machine's memory.

#### **File**

- Filing CNC programs refers to organizing and storing the program files in a structured manner for future use and reference.
- Create directories or folders within the CNC machine's storage system to categorize and organize program files based on criteria such as job type, part number, or machining process.
- Use descriptive filenames that clearly identify the contents and purpose of each CNC program file.
- Regularly backup CNC program files to prevent data loss and ensure availability for future use.

### **3. Which functions are used to control the execution of machining programs?**

**Answer:** In CNC (Computer Numerical Control) machining, several commands and functions are used to control the execution of machining programs. Here's an explanation of program start, program stop, single block, dry run, and feed hold commands:

#### **Program Start**

- Program start command initiates the execution of the CNC machining program loaded into the control system.
- Once the program start command is given, the CNC machine begins to execute the instructions specified in the program, such as tool movements, spindle speeds, and feed rates.

#### **Program Stop**

- Program stop command halts the execution of the CNC machining program.
- When the program stop command is activated, the CNC machine stops all tool movements, spindle rotation, and other machining operations.
- Program stop is used to pause machining operations for various reasons, such as tool changes, inspection, or adjustments.

#### 4. Which options are used manual data in CNC machining center?

**Answer:** In CNC (Computer Numerical Control) machining, manual data entry involves inputting specific parameters, commands, or offsets directly into the CNC machines control system. While much of CNC programming is done using CAM (Computer-Aided Manufacturing) software, manual data entry remains an essential aspect of CNC operations for various tasks. Here's how manual data entry is used in CNC machining centers:

##### **Tool Offsets**

- CNC machines require accurate tool offset data to position cutting tools correctly relative to the workpiece.
- Operators manually enter tool length offsets, tool radius offsets, and tool wear offsets into the CNC machine's control system based on measurements taken from tool presters or manual tools.

##### **Workpiece Coordinates**

- Operators may manually input workpiece coordinate offsets to align the CNC machine's reference point with specific locations on the workpiece.
- Workpiece coordinate offsets adjust the machine's positioning relative to the workpiece, ensuring precise machining at desired locations.

##### **Program Variables**

- Some CNC programs include variables that can be modified manually at the machine interface.
- Operators may adjust program variables such as feed rates, spindle speeds, tool change positions, or coolant settings based on real-time machining conditions or specific job requirements.

##### **Incremental Positioning**

- CNC machines allow operators to manually input incremental movements along individual axes.
- Operators can enter specific distances or angles to move the cutting tool or workpiece incrementally, facilitating setup, tool changes, or manual machining operations.

##### **Program Editing**

- In some cases, operators may need to manually edit CNC program code directly at the machine control panel.
- Manual program editing enables operators to make quick adjustments or corrections to program logic, toolpaths, or machining parameters without accessing external programming software.

##### **Overrides and Commands**

- CNC machines feature manual overrides and commands that allow operators to adjust machine functions in real-time.
- Operators can manually adjust feed rate overrides, spindle speed overrides, rapid traverse overrides, and other machine parameters to optimize machining performance or respond to changing conditions.

5. What is the auxiliary function?

Answer: Auxiliary functions in CNC (Computer Numerical Control) machining typically refer to specific commands or operations that are not directly related to primary machining tasks but serve various supportive functions for the CNC machine's operation or control system.

6. Write the tool length compensation process?

Answer: Tool length compensation (TLC) is achieved in CNC machining through a combination of **G-codes** (the main programming language for CNC machines) and **tool offset values** stored in the machine's memory. Here's a breakdown of the process:

**G-Codes for Enabling/Disabling TLC**

- **G43:** This G-code activates tool length compensation. It tells the machine to consider the stored tool offset value and adjust the programmed Z-axis positions accordingly.
- **G44:** This G-code is less commonly used and works opposite to G43. It instructs the machine to subtract the tool offset value from the programmed Z-axis positions.
- **G49:** This G-code deactivates tool length compensation. The machine will then use the raw programmed Z-axis positions without any adjustments.

7. Shortly brief work offset and tool offset process?

Answer: Let's break down the terms "work offset" and "tool offset," along with the other terms you've mentioned:

**Work Offset**

- Work offset refers to a reference point or coordinate system established on the CNC machine relative to the workpiece being machined.
- It defines the origin (zero point) from which the CNC machine calculates tool movements and machining operations.
- Work offsets allow operators to set up and machine multiple parts with different orientations or positions on the machine bed without reprogramming the entire machining program.
- Common work offset systems include G54, G55, G56, etc., each representing a unique coordinate system for positioning the workpiece.

**Tool Offset**

- Tool offset is the adjustment made to compensate for variations in tool length, diameter, or geometry during machining operations.
- It ensures that the tool tip aligns precisely with the programmed toolpath coordinates.
- Tool offset values are entered into the CNC machine's control system and applied automatically during tool changes or toolpath calculations.
- Tool length and tool radius offsets are the most common types of tool offsets used in CNC machining.

**8. What is the storage device?**

**Answer:** In Computer Numerical Control (CNC) systems, the storage device commonly used is a CNC controller or CNC control unit. This controller typically includes memory components such as Random Access Memory (RAM) and Read-Only Memory (ROM) to store program instructions, tool paths, and other necessary data for controlling the machine's movements and operations.

Additionally, CNC systems may also utilize external storage devices such as USB drives, memory cards, or networked storage solutions to transfer and store CNC programs and data. These external storage devices allow for easy backup, sharing, and management of CNC programs across different machines or systems.

**9. Write the dry and genuine run in CNC program.**

**Answer:** It seems like you're referring to terms related to CNC machining operations. Here's an explanation of each term:

**Dry Run**

- A dry run in CNC machining refers to simulating the machining operation without actually cutting material.
- It allows operators to verify the toolpaths, check for potential collisions, and ensure that the machining program functions as intended.
- Dry runs are typically performed at reduced feed rates and spindle speeds to minimize the risk of damage to the machine or tooling.

**Genuine Run**

- "Genuine run" is not a standard term in CNC machining. It's possible that it's a typo or a term specific to certain CNC setups or processes.
- In the context of CNC machining, operators often refer to running the actual machining program on the workpiece as the "actual run" or simply executing the program.

## **Job Sheet 3.1: Accomplish CNC Machine Axis Positioning According to Job Requirements**

### **Working Procedure:**

1. Wear the PPE.
2. Read job sheet and specification sheet.
3. Choose the appropriate work holding method based on the workpiece's size, shape, material, and machining operations.
4. Ensure that the workpiece is properly aligned and positioned according to the machining program's reference coordinates.
5. Power on the CNC machine and initialize the control system.
6. Load the CNC machining program into the machine's control system from external storage, such as a USB drive, network location, or internal memory.
7. Workpiece Setup according according to the machining program's reference coordinates.
8. Power on the CNC machine and initialize the control system.
9. Execute the homing sequence to establish the machine's reference position for each axis.
10. Load the CNC machining program into the machine's control system from external storage, such as a USB drive, network location, or internal memory.
11. Determine the appropriate work offset based on the job requirements and workpiece setup.
12. Install and properly secure the cutting tools in the machine's tool holders or tool changers.
13. Measure the tool lengths accurately using a tool prester or manual measurement device.
14. Use the machine's jog controls to manually move each axis to verify the workpiece's position and alignment.
15. Adjust the workpiece position and orientation as needed to align it with the machining program's reference coordinates.
16. Ensure that all safety guards, interlocks, and emergency stop buttons are functional and accessible.
17. Start the machining program to initiate the CNC machining process.
18. Monitor the machining operation closely to ensure that the tool movements, spindle speeds, feed rates, and coolant application are performed correctly.
19. After completing the machining process, inspect the machined parts for dimensional accuracy, surface finish, and adherence to quality standards.
20. Follow the manufacturer's recommended shutdown procedure to power off the CNC machine safely.
21. Clean the work area, remove any debris or chips, and perform routine maintenance tasks as necessary.

## Specification Sheet 3.1: Accomplish CNC Machine Axis Positioning According to Job Requirements

**Name of Job:** Accomplish CNC machine axis positioning according to job requirements

**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 Accomplish CNC machine axis positioning according to job requirements

**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 Tools and Equipment**

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

## **Job Sheet 3.2: Complete Copy, Edit, Load and File Creation as Per Specification**

### **Working Procedure:**

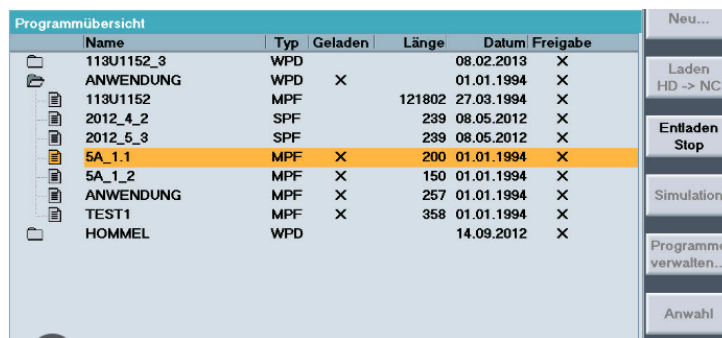
1. Follow OSH.
2. Read job sheet and specification sheet.
3. Collect all Tools & Equipment.
4. Wear the PPE.
5. Design the Part:
6. Create a 2D or 3D model of the part, specifying its dimensions, features, and tolerances.
7. Generate Toolpaths:
8. Simulate Toolpaths:
9. Post-process the CAM toolpaths to generate G-code, the machine-readable language used to control CNC machines.
10. Save G-code File:
11. Transfer the generated G-code file to a storage device such as a USB drive or network location.
12. Prepare the CNC milling machine for operation by securing the workpiece on the machine table using clamps or vises.
13. Load the saved G-code file into the CNC machine's control unit. Depending on the machine's interface, you may transfer the file via USB,
14. After complete coping, edit, load and file the clean work area.
15. Clean tools and equipment.
16. Store all equipment safely.

## Specification Sheet 3.2: Complete Copy, Edit, Load and File Creation as Per Specification

**Name of Job:** Complete copy, edit, load and file creation 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 Complete copy, edit, load and file creation as per specification

### Diagram / Design:



### Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
6	Hand Gloves	Cotton	pair	01	
7	Mask	N 95	Nos	01	
8	Safety shoe	As required	pair	01	
9	Safety goggles	Standard	Nos	01	
10	Apron	Standard	Nos	01	

### Required Tools and Equipment

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

## **Job Sheet 3.3: Perform Work Offset and Tool Offset in CNC Milling Machine**

**Job Name:** Perform work offset and tool offset in CNC Milling machine

### **Working Procedure:**

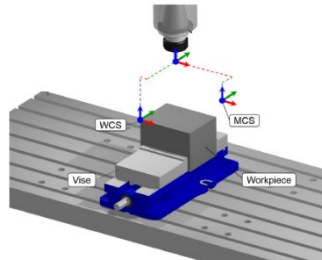
1. Follow OSH.
2. Read job sheet and specification sheet.
3. Collect all Tools & Equipment.
4. Wear the PPE.
5. Ensure that all safety precautions are observed, and the CNC machine is properly set up for machining.
6. Ensure the CNC milling machine is properly set up, powered on, and all safety precautions are in place.
7. Mount the workpiece securely on the machine table using clamps or a vice.
8. Install the appropriate edge Finder in the spindle, ensuring it's securely tightened.
9. Load the corresponding tool offset information into the CNC machine's control system if it's not already done.
10. Perform Automatic Reference Point
11. Press Jog mode
12. Perform edge Finder to touch X axis and Insert The data with G54 or G55~G59.
13. Perform edge Finder to touch y axis and Insert The data with G54 or G55~G59.
14. Perform edge Finder to touch z axis and Insert The data with G54 or G55~G59.
15. Perform Home Position.
16. Press Jog mode
17. Perform tool to touch Z axis and Insert The data with G54 or G55~G59.
18. Move the tool to a reference point on the workpiece that aligns with the desired work offset coordinate system.
19. Use the machine's control panel or software interface to zero out the appropriate axis (X, Y, Z) at the reference point.
20. Verify that the work offset has been set correctly by checking the position of the tool relative to the workpiece and the machine's coordinate system.

## Specification Sheet 3.3 Perform Work Offset and Tool Offset in CNC Milling Machine

**Job Name:** Perform work offset and tool offset in CNC Milling machine

**Conditions for the job:** You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH Perform work offset and tool offset in CNC Milling machine

**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 Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1.	Tool Holder	BT-40	Pcs	01	
2.	Edge Finde	10 mm	Pcs	01	
3.	Table Vice	12 Inch	Pcs	01	
4.	Adjustable Wrench	12 Inch	Pcs	01	
5.	Soft Hammer	2 Lb	Pcs	01	

Sl.	Name of Materials	Specification	Unit	Quantity	Remarks
1	Aluminum Bar	100 x 100 x 80	Pcs	01	

## **Job Sheet 3.4: Accomplish Program Start, Program Stop and Complete a Single Block, Dry Run, Feed Hold as Per Specification**

### **Working Procedure:**

1. Follow OSH.
2. Read job sheet and specification sheet.
3. Collect all Tools & Equipment.
4. Wear the PPE.
5. Ensure that all safety precautions are observed, and the CNC machine is properly set up for machining.
6. Initiate the program start command, typically by pressing a button or inputting a command on the control panel.
7. Selected machining program, following the programmed toolpaths and instructions.
8. Enable the dry run mode on the CNC control interface to execute the program at reduced feed rates or with the spindle turned off.
9. Perform a dry run to simulate the machining process without cutting material. Before running the machining program with the actual workpiece,
10. Monitor the machine's movements closely during the dry run to ensure that there are no collisions or other issues with the toolpaths. pause the program and execute one block (line of code) at a time for troubleshooting or verification purposes.
11. Use the single block mode feature on the CNC control interface to step through the program one block at a time.
12. Observe the machine's movements and verify the correctness of each step before proceeding to the next.
13. Pause the feed of the cutting tool while keeping the spindle running using the feed hold function.
14. Press the feed hold button on the CNC control panel to temporarily halt the
15. Stop the execution of the machining program, press the program stop button on the CNC control panel.
16. Emergency stop button in case of an emergency or if immediate cessation of machine operation is required.
17. Perform CNC machine will halt all movements and cease executing the program.
18. After complete work clean work area.
19. Clean tools and equipment.
20. Store all equipment safely.

## Specification Sheet 3.4: Accomplish Program Start, Program Stop and Complete a Single Block, Dry Run, Feed Hold as Per Specification

**Name of Job:** Perform Program start, program stop, and complete a single block, dry run, feed hold 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 Complete Perform Program start, program stop, and complete a single block, dry run, feed hold 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 Tools and Equipment

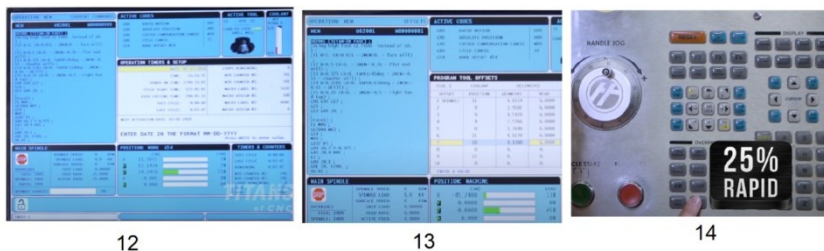
Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	CNC Milling Machine		No	01	

## Job 3.5: Perform Dry Run in CNC Machine as Per Specification

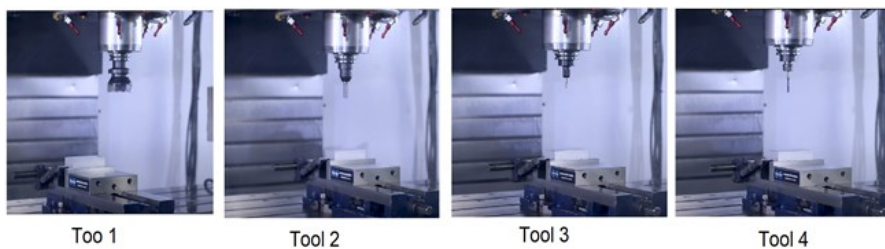
Job: Perform Dry run in CNC machine as per specification

### Procedure

1. Wear PPE.
2. Read 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. Hold or clamp job in machine Vice
9. Offset setting as required
10. Press edit mood and press program directory
11. Select program to perform dry run
12. Press offset and press work offset
13. Select G54 (001) and change Z value +6.00 Inch
14. Decrease federate 25%



15. Perform CNC Door Close
16. Press reset button
17. Press auto and cycle start (Every tool Moves 6 Inch top from Z0.0 Level)
18. Check Program running simulation and tool path movement



19. Edit Program If Required
20. After Finish dry run
21. Stop machine and Correction Z value (Before input +6Inch Value)
22. Finish the work clean all work area
23. Store tools and equipment safely.

## Specification Sheet 3.5 Perform Dry Run in CNC Machine as Per Specification

**Job Name:** Perform Dry run in CNC machine as per specification

**Conditions for the job:** You must practice safe operation procedures at all times during the perform this job. You will be required to demonstrate OSH competencies to perform dry run in CNC machine as per specification

### Diagram / Design



Definition of a rectangular.

- Tool 1: Face Mill Cutter Safe High 6 Inch
- Tool 2: End Mill Cutter Safe High 6 Inch
- Tool 3: Spot Drill
- Tool 4: Drill bit
- Roughing feedrate: 300 mm/min.
- Finishing feedrate: 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 pounds	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	Face Mill Cutter	50 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 70 X 50 mm	Pcs	08	

## Job Sheet 3.6 Input a Part Program as Per Specification.

### Working Procedure:

1. Wear PPE.
2. Read 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
9. Input Program Name and Transfer program with USB Drive 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

10. After complete Inputting program check program again.

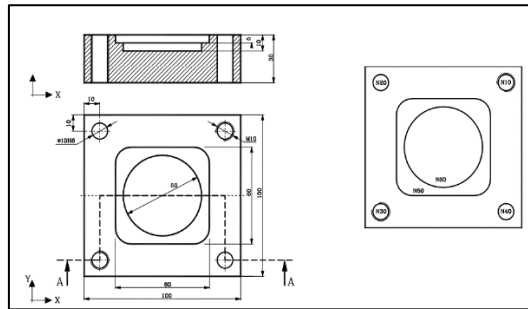
11. Shutdown machine properly.

## Specification Sheet 3.6 Input a Part Program as Per Specification.

**Job Name:** Input a part program as per specification.

**Conditions for the job:** You must practice safe operation procedures at all times during the perform this job. You will be required to demonstrate OSH competencies to Input a 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  $\text{Ø}10$  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 pounds	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 “Perform CNC Machining Center Operation”

Assessment of performance Criteria	Yes	No
Major components of CNC machining center in identified	<input type="checkbox"/>	<input type="checkbox"/>
Air pressure, coolant level and lubrication oil are checked	<input type="checkbox"/>	<input type="checkbox"/>
CNC Machining center power on & off is performed	<input type="checkbox"/>	<input type="checkbox"/>
Machine reference is performed	<input type="checkbox"/>	<input type="checkbox"/>
Override keys are adjusted as required	<input type="checkbox"/>	<input type="checkbox"/>
Hand Jog rate is performed as required	<input type="checkbox"/>	<input type="checkbox"/>
ATC function is performed	<input type="checkbox"/>	<input type="checkbox"/>
Workpiece is clamped with clamping kits	<input type="checkbox"/>	<input type="checkbox"/>
Various types of cutting tools are identified	<input type="checkbox"/>	<input type="checkbox"/>
The appropriate sequence of cutting tools for an operation is selected	<input type="checkbox"/>	<input type="checkbox"/>
Appropriate cutting tools are selected as per job materials	<input type="checkbox"/>	<input type="checkbox"/>
Appropriate cutting parameters are followed as per manufacturer chart	<input type="checkbox"/>	<input type="checkbox"/>
Different types of cutting tools inserts are selected	<input type="checkbox"/>	<input type="checkbox"/>
CNC machine axes positioning is performed as per job requirement	<input type="checkbox"/>	<input type="checkbox"/>
Copy, edit, load, and create files are executed	<input type="checkbox"/>	<input type="checkbox"/>
Program start, program stop, and a single block, dry run, feed hold is performed	<input type="checkbox"/>	<input type="checkbox"/>
Manual data input (MDI) in CNC machining center is performed	<input type="checkbox"/>	<input type="checkbox"/>
Auxiliary function keys are performed	<input type="checkbox"/>	<input type="checkbox"/>
Tool length compensation is performed	<input type="checkbox"/>	<input type="checkbox"/>
Work offset and tool offset are performed	<input type="checkbox"/>	<input type="checkbox"/>
Program is transferred from storage devices	<input type="checkbox"/>	<input type="checkbox"/>
Dry and actual run is executed	<input type="checkbox"/>	<input type="checkbox"/>

I now feel ready to undertake my formal competency assessment.

Signed:

Date:

## Development of CBLM

The Competency based Learning Material (CBLM) of ‘**Performing CNC Machining Center Operation with CAD CAM**’ (**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.

<b>SL No.</b>	<b>Name &amp; Address</b>	<b>Designation</b>	<b>Contact Number</b>
1	Aparna Barua	Writer	01817-717052
2	Uttam Kumar Das	Editor	01716-220932
3	Engr. Md. Zuvel Parves	Co-Ordinator	01737-278906
4	Engr. Md. Nazrul Islam	Reviewer	01711 273708

## Reference:

1. [https://academy.titansofcnc.com/files/Fundamentals\\_of\\_CNC\\_Machining.pdf](https://academy.titansofcnc.com/files/Fundamentals_of_CNC_Machining.pdf),
2. <https://staff.mica.edu/rmckibbin/CNC/Images/dFab%20FANUC%20Manual.pdf>,
3. <https://tech-labs.com/sites/default/files/Fanuc%20CNC%20Certified%20Education%20Training.pdf>,