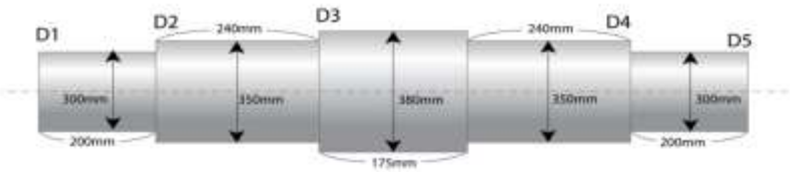
RQF LEVEL 5



MATPC501

MANUFACTURING TECHNOLOGY

Parts Production On CNC Machine

TRAINEE'S MANUAL

October 2024

PARTS PRODUCTION ON CNC MACHINE

2024

AUTHOR'S NOTE PAGE (COPYRIGHT)

Copies available from:

- *HQs: Rwanda TVET Board-RTB*
- *Web: www.rtb.gov.rw*
- **KIGALI-RWANDA**

Original published version: October 2024

ACKNOWLEDGEMENTS

The publisher would like to thank the following for their assistance in the elaboration of this training manual:

Rwanda TVET Board (RTB) extends its appreciation to all parties who contributed to the development of the trainer's and trainee's manuals for the TVET Certificate V in Manufacturing Technology, specifically for the module **"MATPC 501: Parts Production on CNC machine"**

We extend our gratitude to KOICA Rwanda for its contribution to the development of these training manuals and for its ongoing support of the TVET system in Rwanda.

We extend our gratitude to the TQUM Project for its financial and technical support in the development of these training manuals.

We would also like to acknowledge the valuable contributions of all TVET trainers and industry practitioners in the development of this training manual.

The management of Rwanda TVET Board extends its appreciation to both its staff and the staff of the TQUM Project for their efforts in coordinating these activities.

This training manual was developed:

Under Rwanda TVET Board (RTB) guiding policies and directives

Under Financial and Technical support of

COORDINATION TEAM

RWAMASIRABO Aimable

MARIA Bernadette M. Ramos

NIKUZE Bernadette

Production Team

Authoring and Review

NSANZAMAHORO Jean Jacques

NAYIGIZENTE Manasseh

Validation

HAKIZIMANA Fidele

SINGIZWA Benjamin

Conception, Adaptation and Editorial works

HATEGEKIMANA Olivier

GANZA Jean Francois Regis

HARELIMANA Wilson

NZABIRINDA Aimable

DUKUZIMANA Therese

NIYONKURU Sylvestre

KAMERE Raban

Formatting, Graphics, Illustrations, and infographics

YEONWOO Choe

SUA Lim

SAEM Lee

SOYEON Kim

WONYEONG Jeong

NDAYISABA Olivier

Financial and Technical support

KOICA through TQUM Project

TABLE OF CONTENT

ACRONYMS

2D: Two Dimension

3D : Three Dimension

3D PDF : 3D Portable Document Format

ACIS: Andy, Charles, Ian's System

CAD: Computer-Aided Design

CADD: Computer-Aided Design and Drafting

CAE: Computer-Aided Engineering

CAM: Computer-Aided Manufacturing

CBT/A: Competency-Based Training and Assessment

CD: Compact Disc

CNC: Computer Numerical Control

CPU: Computer Processing Unit

DRO: Digital ReadOut.

DVD: Digital Video Disc

G-code: Geometric Code

GUI: Graphical User Interface

KOICA: Korea International Cooperation Agency

M-code: Miscellaneous Code

RTB: Rwanda TVET Board

STEP: Standard for the Exchange of Product model datas

TQUM Project: TVET Quality Management Project

TVET: Technical and Vocational Education Training

USB: Universal Serial Bus

INTRODUCTION

This trainee's manual includes all the knowledge and skills required in Manufacturing Technology specifically for the module of **"Parts Production on CNC Machine".** Trainees enrolled in this module will engage in practical activities designed to develop and enhance their competencies. The development of this training manual followed the Competency-Based Training and Assessment (CBT/A) approach, offering ample practical opportunities that mirror real-life situations.

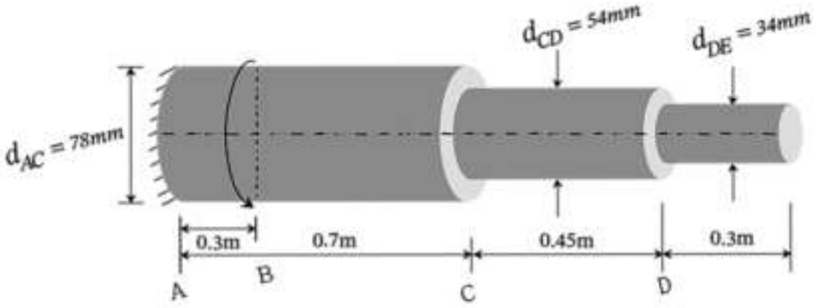
The trainee's manual is organized into Learning Outcomes, which is broken down into indicative content that includes both theoretical and practical activities. It provides detailed information on the key competencies required for each learning outcome, along with the objectives to be achieved.

As a trainee, you will start by addressing questions related to the activities, which are designed to foster critical thinking and guide you towards practical applications in the labor market. The manual also provides essential information, including learning hours, required materials, and key tasks to complete throughout the learning process.

All activities included in this training manual are designed to facilitate both individual and group work. After completing the activities, you will conduct a formative assessment, referred to as the end learning outcome assessment. Ensure that you thoroughly review the key readings and the 'Points to Remember' section.

MODULE CODE AND TITLE: MATPC501 PARTS PRODUCTION ON CNC MACHINE

Learning Outcome 1: Generate NC files
Learning Outcome 2: Set up CNC Machine
Learning Outcome 3: Execute the work


$d_{CD} = 54mm$
$d_{DE} = 34mm$
$d_{AC} = 78mm$
0.3m
0.7m
0.45m
0.3m
A
B
C
D

Indicative contents
1.1. Introduction to CNC machine **1.2. Producing NC file** **1.3. Simulating NC files**

Key Competencies for Learning Outcome 1: Generate NC file

Knowledge	Skills	Attitudes
• Explanation of key terms used in CNC machine • Description of types of CNC machine • Description of parts of CNC machine • Description of CNC programming language • Identification of methods of CNC programming • Description of CAM software • Identification of Methods of transferring data	• Applying safety precautions on CNC machine • Interpreting drawing • Interpreting CNC codes • Writing CNC program • Installing CAD and CAM software • Simulating CNC program	• Being careful while applying safety precautions on CNC machine • Being proactive to problem-solving and troubleshooting abilities during installation of CAD and CAM software • Having critical thinking while writing CNC program • Having critical thinking while interpreting drawings and CNC codes. • Having self-confidence in simulating CNC program

Duration: 30 hrs

Learning outcome 1 objectives:

By the end of the learning outcome, the trainees will be able to:

1. Explain correctly key terms used in CNC machine.
2. Apply properly safety precautions as used CNC machine operation.
3. Interpret correctly drawing used in CNC machine
4. Describe efficiently types of CNC machine used in the manufacturing industry.
5. Describe clearly CNC programming language used in machining operations
6. Identify properly methods of CNC programming used in machining operations
7. Interpret correctly CNC codes used CNC programming
8. Describe clearly CAM software used in the manufacturing industry.
9. Install correctly CAD and CAM software used in the manufacturing industry.
10. Identify properly methods of transferring data according to the product design
11. Simulate appropriately CNC program according to the product design

Resources

Equipment	**Tools**	**Materials**
● CNC lathe machine, ● CNC milling machine ● CNC router machine ● CNC drilling machine ● CNC laser cutter machine ● Computer ● Printers	● Rulers ● Squares ● Flash disk ● ARTCAM ● MARTCAM, ● MASTERCAM ● FUSION 360 ● SINUMERIK Software ● CNC Simulator Pro ● Extension cable	● CAD models of parts or components to be manufactured ● Papers ● Pens ● Rubbers ● Lubricants

Indicative content 1.1: Introduction to CNC Machine

Duration: 7 hrs

Theoretical Activity 1.1.1.: Explanation of key terms used in CNC machine

Tasks:

1: Answer the following questions:

i. What do you understand by the following terms used in CNC machine operations:

a) CNC Machine
b) Part program
c) G-code/ M-code
d) CAD/CAM
e) Axis
f) Cutting Parameters
g) Fixture
h) Rapid Traverse
i) DRO (Digital Readout)

2: Provide the answers to the asked questions by writing them on paper or flip chart,

3: Present the answers to the whole class.

4: For more clarification, read the key readings 1.1.1

Key readings 1.1.1: Explanation of key terms used in CNC machine

- **Definition of key terms used in CNC machine operations**

✓ **CNC Machine:**
A CNC (Computer Numerical Control) machine is a computer-controlled tool that automates the operation of machinery to shape materials like metal, plastic, or wood into parts or components with high precision. It uses pre-programmed software to direct the movement and functions of machine tools like lathes, mills, and routers.

✓ **Part Program:** a part program is a series of coded instructions (usually in G-code) that tells the CNC machine how to perform specific operations, such as cutting, drilling, or milling, on a workpiece. The part program defines the toolpaths, tool selection, speeds, and other parameters needed for manufacturing a part.

- ✓ **G-code / M-code**:

- **G-code**: A programming language used to control the movements and operations of a CNC machine, such as positioning, feed rate, and cutting depth. G-codes define the geometrical movements of the machine.
- **M-code**: Complementary to G-code, M-codes control non-geometrical functions like starting or stopping the spindle, coolant on/off, and tool changes.

- ✓ **CAD/CAM**:

- **CAD (Computer-Aided Design)**: Software used to design parts and products in 2D or 3D before manufacturing.
- **CAM (Computer-Aided Manufacturing)**: Software used to convert CAD designs into G-code or machine instructions that the CNC machine can execute.

- ✓ **Axis**:
  An axis refers to the directional movement of a CNC machine. CNC machines typically operate along multiple axes (such as X, Y, Z) to position tools and materials accurately in three-dimensional space. The number of axes (e.g., 3-axis, 4-axis, 5-axis) determines the complexity of movements and machining operations.
- ✓ **Cutting Parameters**:
  Cutting parameters include variables like cutting speed, feed rate, depth of cut, and spindle speed. These parameters are crucial in determining the efficiency, quality, and precision of the machining process.
- ✓ **Fixture**:
  A fixture is a device used to securely hold a workpiece in place during the CNC machining process. It ensures stability and precision during operations like cutting, drilling, or milling.
- ✓ **Rapid Traverse**:
  Rapid traverse refers to the fast movement of the CNC machine's tool or worktable between operations, without engaging in cutting. It is used to move quickly to the next position in the machining process, saving time.
- ✓ **DRO (Digital Readout)**:
  A Digital Readout is a system that displays the exact position of machine components, such as the spindle or cutting tool, in real time. It enhances precision by providing visual feedback to operators regarding the position of the axes

Practical Activity 1.1.2: Applying safety precautions on CNC machine

Task:

Go to the manufacturing workshop to implement safety precautions on the CNC machine to prevent accidents and ensure smooth operations while producing mechanical parts

1: Apply safety precautions

2: Interpret signs and symbols found on the CNC machine

3: Present your work to the trainer

4: Read the key reading 1.1.2. And ask for clarification where necessary.

5: Perform the task provided in the application of learning 1.1

Key readings 1.1.2: Applying safety precautions on CNC machine

- **Safety precautions on CNC machine**

✓ Always wear safety glasses, gloves, ear protection, and steel-toe shoes to protect against flying debris, noise, and potential injuries from machine parts.

✓ Ensure that clothing, jewelry, or long hair is secured and kept away from moving machine parts to avoid entanglement.

✓ Ensure the workpiece is securely clamped and fixtures are properly set up before starting the machine to prevent accidents from shifting parts.

✓ Always maintain a safe distance from the machine while it's operating. Avoid reaching into the machine while it’s running.

✓ Familiarize yourself with the machine's emergency stop button and always be prepared to use it in case of malfunction or emergency.

✓ Before use CNC machine, inspect the CNC machine for loose parts, damaged components, and ensure all guards are securely in place

- **Common Signs and Symbols on CNC Machines**

✓ **Emergency Stop (E-Stop**

- **Symbol:** A red button often with a white circle or hand icon
- **Meaning:** When press this button to immediately stop all machine operations in case of an emergency

✓ **Power On/Off:**

- **Symbol:** A circle (O) for "off" and a vertical line (I) for "on."

- **Meaning:** Used to turn the CNC machine on or off.
- ✓ **Safety guard warning:**
- **Symbol:** A yellow triangle with an exclamation mark or an image of a hand being cut.
- **Meaning**: Indicates the presence of moving parts or sharp tools. Always ensure safety guards are in place before operating
- ✓ **Rotating parts hazard:**
- **Symbol**: A triangle with a rotating blade or gear image.
- **Meaning**: Warns of potential dangers from moving parts that could cause injury. Avoid placing hands near these areas
- ✓ **No access or do not touch:**
- **Symbol**: A red circle with a slash over a hand or tool.
- **Meaning**: Indicates restricted areas or parts of the machine that should not be touched during operation.

Theoretical Activity 1.1.3: Description of CNC machines

Tasks:

1: Answer the following questions:

i. What should be the types of CNC machines?
ii. What do you think are the main parts of a CNC machine?

2: Provide the answers to the asked questions by writing them on paper or flip chart,

3: Present the answers to the whole class.

4: For more clarification, read the key readings 1.1.3

Key readings 1.1.3: Description of CNC machines

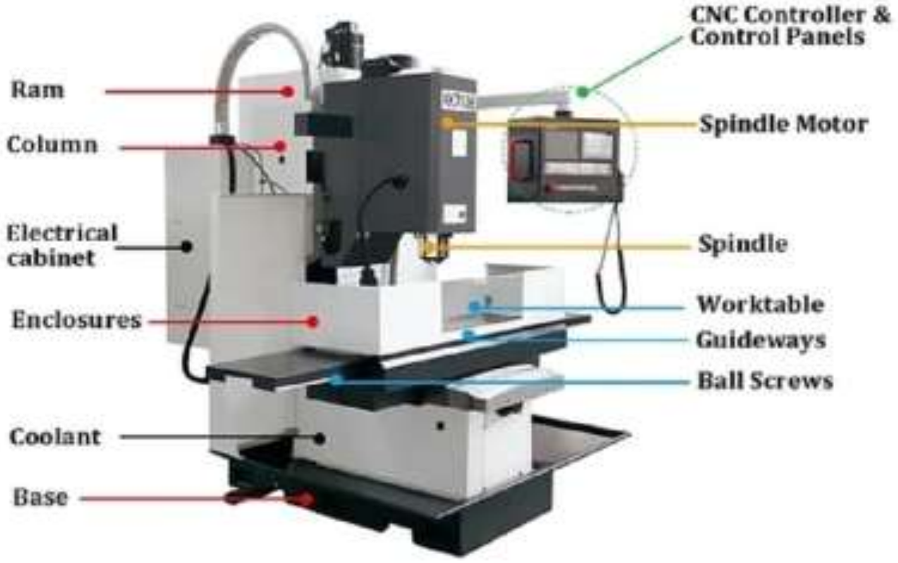
- **Types of CNC machines and their parts**
- ✓ **CNC milling machine**

CNC milling Machine is a computer-controlled machine used to precisely cut and shape materials like metal into parts with complex geometries. It operates by moving a rotating cutting tool along multiple axes (X, Y, Z), which removes material from a workpiece to create the desired shape.

Function: Performs multi-axis cutting and drilling operations on a workpiece using rotating tools.

Common Uses: Manufacturing complex 3D parts, die and mold making, and precision machining for metal parts.

Parts of CNC milling machine



✓ CNC lathe machine

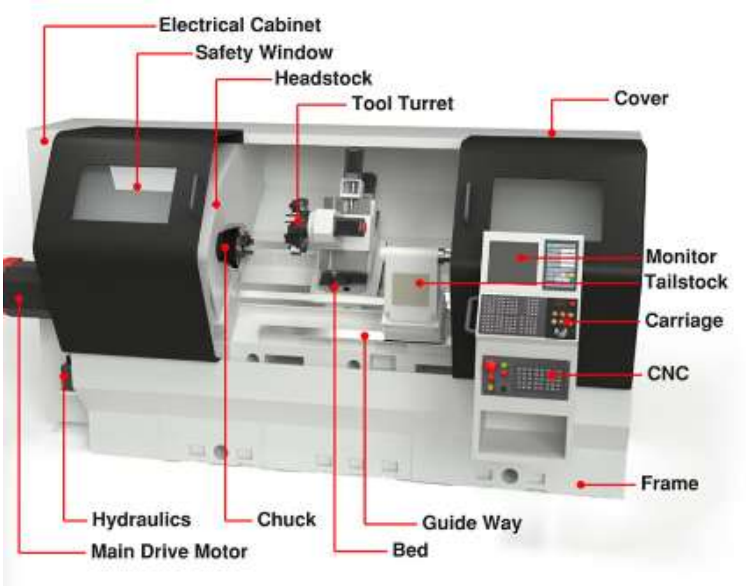
A CNC Lathe Machine used in manufacturing is a computer-controlled machine tool that primarily processes cylindrical or symmetrical parts by rotating the workpiece while cutting tools shape it.

It automates traditional lathe operations using programmed instructions (G-code or M-code) for precise, repeatable production.

Function: Rotates the workpiece while cutting tools remove material to create symmetrical objects.

Common uses: Machining cylindrical parts, such as shafts, screws, and bolts

Parts of CNC lathe machine

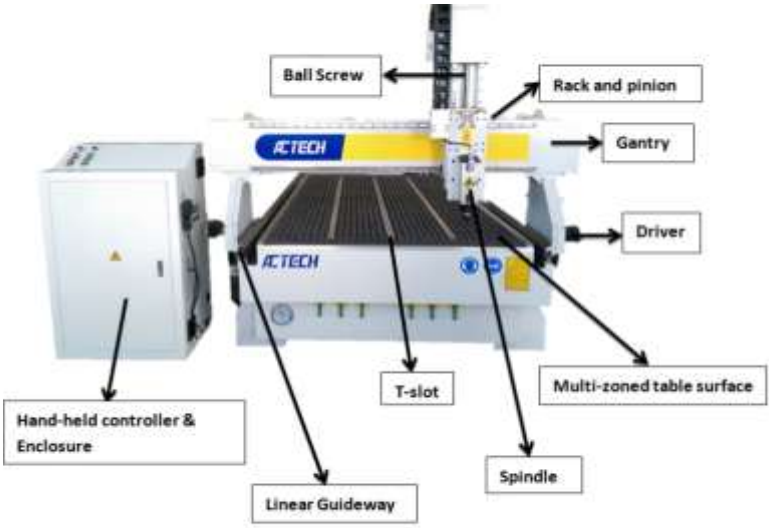


✓ **CNC router machine**

A CNC router machine is a computer-controlled cutting machine used in manufacturing to create precise and complex shapes from various materials such as aluminium, and metals. It operates by moving a rotating cutting tool (the router) along different axes (usually X, Y, and Z) to remove material from the workpiece, following a programmed path. CNC routers are highly versatile and are commonly used in industries like furniture manufacturing, cabinetry, signage, and metalworking.

Function: CNC routers are programmed using CAD/CAM software, allowing for highly accurate and repeatable cuts, essential in mass production and prototyping.

Parts of CNC router machine



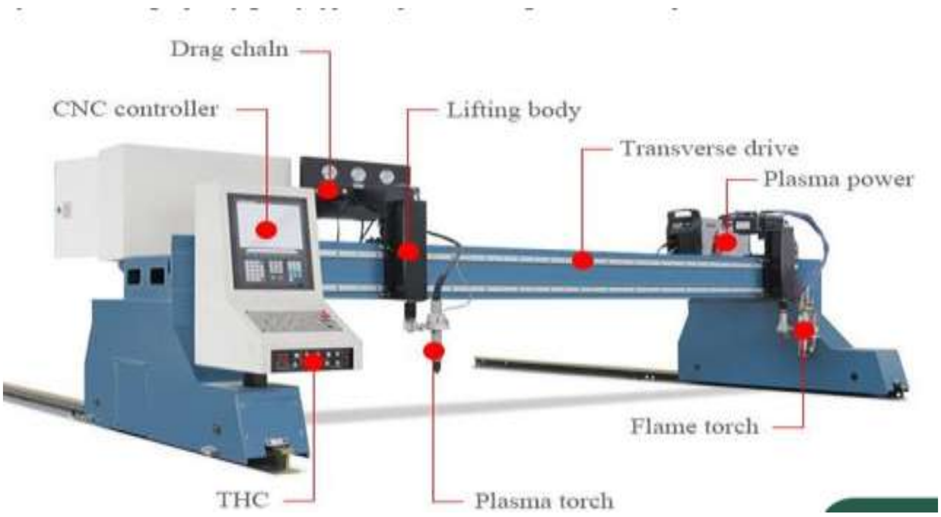
✓ **CNC plasma cutting machine**

A CNC plasma cutting machine is a type of computer-controlled machine that uses a high-velocity jet of hot plasma to cut through electrically conductive materials, such as steel, aluminium, brass, or copper. The plasma cutter generates an arc between an electrode and the workpiece, creating plasma (an ionized gas) that melts the material. The machine then blows away the molten metal with compressed gas to achieve a precise cut.

Function: Uses plasma, a high-temperature ionized gas, to cut metal

Uses: Primarily cuts materials that conduct electricity, such as metals.

Parts of CNC Plasma cutting machine



✓ **CNC electrical discharge machine (EDM)**

A **CNC electrical discharge machine (EDM)** is a type of computer-controlled machining process that uses electrical discharges (sparks) to shape or cut metal materials. It's especially effective for hard metals or materials that are difficult to machine using traditional cutting tools. EDM removes material from a workpiece by generating a series of electrical sparks between an electrode and the workpiece, which erodes the material through heat and vaporization

✓ **CNC grinding machine**

A **CNC grinding machine** is a type of computer-controlled machine tool that uses an abrasive wheel or belt to remove material from a workpiece to achieve high precision and surface finish. The grinding process involves applying friction through a rotating grinding wheel to cut, smooth, or refine the surface of the material. CNC (Computer Numerical Control) technology enables automated and highly accurate grinding operations by programming the machine's movements and tool paths.

Function: CNC grinding machines are designed for precise material removal and are capable of achieving extremely tight tolerances.

Uses: Uses a grinding wheel made of abrasive particles to cut or shape the material

Theoretical Activity 1.1.4: Identification of the aspects of CNC machines

Tasks:

1: Answer the following questions:

i. What should be the axis of CNC Machine?
ii. What are the typical specifications of CNC machines?
iii. What can be the Control features of CNC Machine?
iv. What should be the parameters of CNC machine?

2: Provide the answers to the asked questions by writing them on paper or flip chart,

3: Present the answers to the whole class.

4: For more clarification, read the key readings 1.1.4

Key readings 1.1.4: Identification the aspects of CNC machines

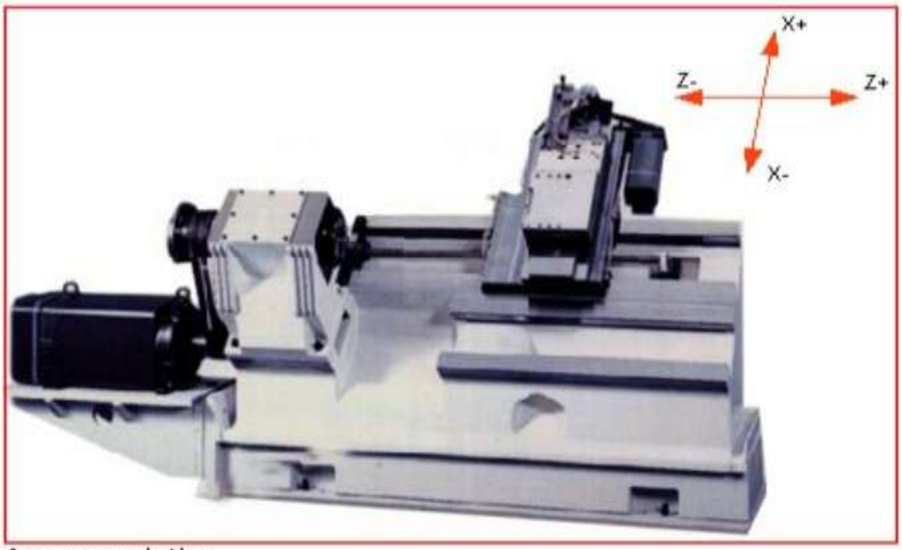
- **CNC machine axes**

CNC (Computer Numerical Control) machines operate based on the movement of axes that determine the direction and type of motion of the cutting tool or the Workpiece. Here's an overview of the common axes used in CNC machines:

- ✓ **X-Axis**: Represents the horizontal movement, typically from left to right. It controls the motion of the cutting tool or Workpiece in a linear direction along the width of the machine.
- ✓ **Y-Axis**: Represents the front-to-back movement in a horizontal plane. It controls

motion in the depth direction, perpendicular to the X-axis.

- ✓ **Z-Axis**: Represents the vertical movement, controlling the up-and-down motion of the cutting tool. This is crucial for determining the depth of the cut.
- ✓ **A-Axis**: A rotational axis around the X-axis, allowing the cutting tool or workpiece to rotate in a circular motion.
- ✓ **B-Axis**: A rotational axis around the Y-axis, which enables tilting or rotating of the workpiece or tool in specific machines like milling machines or lathes.
- ✓ **C-Axis**: A rotational axis around the Z-axis, often used in CNC turning centers (lathes) for rotating the workpiece during machining operations



Axes on a lathe

- **Typical specifications of CNC machines**

- ✓ **Axis configuration:** CNC machines typically range from 2-axis to 5-axis, determining the complexity of movements they can perform
- ✓ **Table size**: Specifies the worktable dimensions and load capacity, affecting the size and weight of parts.
- ✓ **Spindle speed**: Measured in RPM (e.g., 1,000 to 40,000 RPM), determining the cutting tool's rotation speed.
- ✓ **Spindle power**: Indicates power in kW or HP, affecting the machine's ability to cut tougher materials.
- ✓ **Travels (X, Y, Z)**: Maximum movement range along the axes, influencing part size.
- ✓ **Rapid traverse rate**: Speed at which the tool moves between cuts, enhancing efficiency.
- ✓ **Cutting feed rate**: Speed of the tool advancing into the material, typically in mm/min.
- ✓ **Tool capacity**: Number of tools the machine holds, especially with automatic tool

changers (ATC).

- ✓ **Control system**: CNC controller type (e.g., FANUC, Siemens) that manages machine operation.
- ✓ **Positioning accuracy & repeatability**: Measures precision and consistency, usually in microns.
- ✓ **Machine weight**: Affects stability and vibration damping.
- ✓ **Maximum workpiece size**: Dimensions of the largest part the machine can handle.
- ✓ **Coolant system**: For cooling during operations, preventing overheating.
- ✓ **Electrical power supply**: Voltage and power requirements, typically three-phase.

- **CNC machine control features**

- ✓ **CNC controller**: The system that interprets G-code/M-code to control machine movement and functions.
- ✓ **G-code/M-code Programming**: Controls tool movement (G-code) and auxiliary functions (M-code).
- ✓ **Human machine interface (HMI)**: A touchscreen or button interface for inputting commands and monitoring operations.
- ✓ **Manual Data Input (MDI)**: Allows manual commands for quick tasks without running a full program.
- ✓ **Feed Rate and Spindle Speed Override**: Real-time adjustments to cutting speed and spindle rotation during operations.
- ✓ **Automatic Tool Changer (ATC)**: Switches tools automatically based on program requirements.
- ✓ **Probing System**: Measures and aligns the workpiece for precision adjustments.
- ✓ **Tool Length Compensation**: Adjusts for different tool lengths to maintain consistent cutting depth.
- ✓ **Work Offset Settings**: Allows multi-part setups by setting different starting points.
- ✓ **Program Storage & Retrieval**: Stores and retrieves machining programs.
- ✓ **Interpolation**: Controls tool movement in straight lines (linear) or arcs (circular).
- ✓ **Collision Detection & Avoidance**: Prevents tool and machine part collisions.
- ✓ **Automatic Homing and Tool Zeroing**: Returns the machine to a preset position or zero point.
- ✓ **Jog Control**: Manually controls axis movement for precise positioning.
- ✓ **Canned Cycles**: Pre-programmed routines for repetitive tasks like drilling or tapping.
- ✓ **Tool Path Visualization**: Simulates the tool path for visual confirmation before machining.
- ✓ **Coolant Control**: Manages coolant systems for temperature regulation.

- ✓ **Emergency Stop (E-stop)**: Instantly stops the machine for safety.

- **CNC parameters**
- ✓ **Cutting Parameters**

  Cutting parameters for a CNC machine refer to the critical values that control the cutting process, determining how the tool interacts with the material. These parameters have a direct effect on machining performance, tool life, surface finish, and overall efficiency.
- ✓ **Positioning Parameters**

  Positioning parameters for a CNC machine refer to the settings and values that control how the machine positions its cutting tool in relation to the workpiece. These parameters are crucial for determining the accuracy and precision of the tool's movements during machining
- ✓ **Tooling Parameters**

  Tooling parameters for a CNC machine refer to the specific settings and adjustments related to the tools used in the machining process. These parameters help ensure that the CNC machine operates effectively and achieve the desired machining results.
- ✓ **Control Parameters**

  Control parameters for a CNC (Computer Numerical Control) machine are settings that govern the operation and functionality of the machine's control system. These parameters are crucial for defining how the machine executes programmed instructions and handles various aspects of machining.
- ✓ **Safety Parameters**

  Safety parameters for CNC machines are settings and precautions designed to protect operators and equipment during operation. These parameters ensure that the machine operates within safe limits and help prevent accidents, injuries, and damage.

Theoretical Activity 1.1.5: Description of major components of CNC machine

Tasks:

1: Answer the following question:

i. How can you explain the following major components of CNC machines
   a) Machine control unit
   b) Control Panel/Interface
   c) Machine tool (mechanical part)
   d) Program input device
   e) Drive system
   f) Feedback system

2: Provide the answers to the asked questions by writing them on paper or flip chart.

3: Present the answers to the whole class.

4: For more clarification, read the key readings 1.1.5

Key readings 1.1.5: Description of major components of CNC machine

- **Major components of CNC machine**

✓ **Machine control unit**

A Machine Control Unit (MCU) in a CNC machine is the central component responsible for controlling the machine's operations. It interprets the part program, which contains G-code and M-code instructions, and sends precise commands to the machine's motors and actuators to perform tasks such as cutting, drilling, or milling. The MCU manages the machine's movements along its axes (X, Y, and Z), adjusts cutting speeds, controls tool changes, and ensures accuracy and efficiency during machining. Essentially, it acts as the brain of the CNC machine, coordinating all actions based on the programmed instructions.

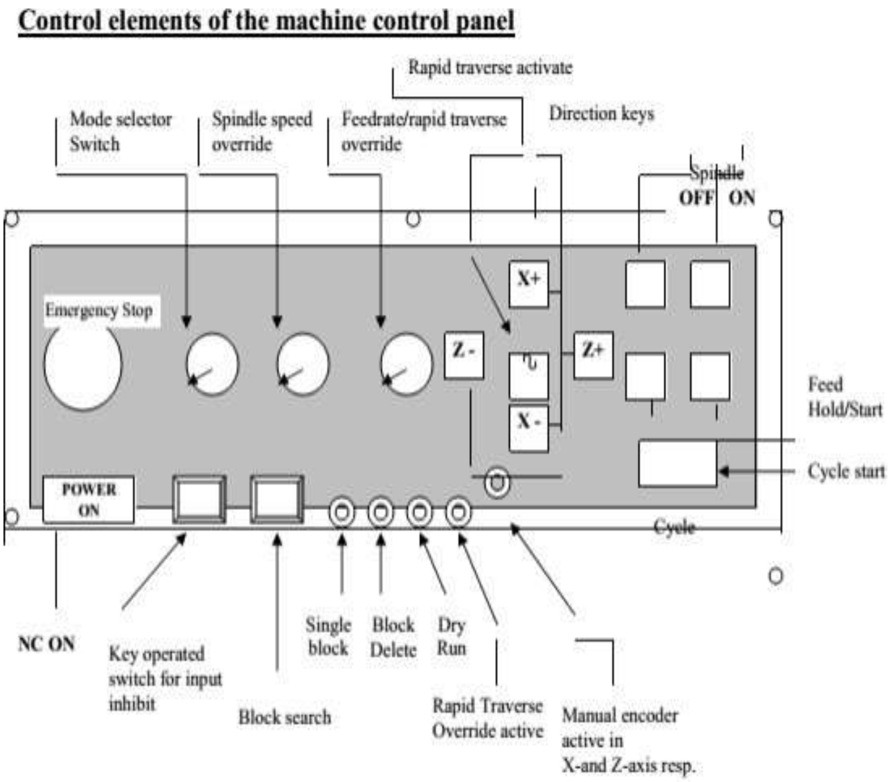


Fig.5 Machine control panel of Hinumerik 3100

✓ **Interface**

The interface for a CNC machine refers to the system that enables communication and interaction between the machine and the user, as well as between the machine and external systems. It includes both hardware and software components that allow operators to input commands, monitor operations, and control the machining process.

Key aspects of a CNC machine interface include:

- **Human-Machine Interface (HMI)**: This is the user-friendly display and control system that allows operators to input data (e.g., G-code programs), adjust settings, and monitor machine status in real-time. It can include a touchscreen or a display with a keypad or buttons.
- **Machine-Computer Interface**: Facilitates the transfer of part programs from CAD/CAM systems to the CNC machine. This is often done via USB ports, network connections, or direct serial interfaces.
- **Feedback Systems**: Sensors and encoders provide real-time data on the machine's performance, such as position, speed, and temperature, feeding this information back to the control unit and the operator for adjustments.
- **Operator Controls**: Physical buttons, dials, and joysticks on the control panel that allow manual intervention for machine setup, tool adjustments, or jogging the machine axes.

- **Software Interface**: The operating system or software environment where the part programs are loaded, simulated, and edited before execution

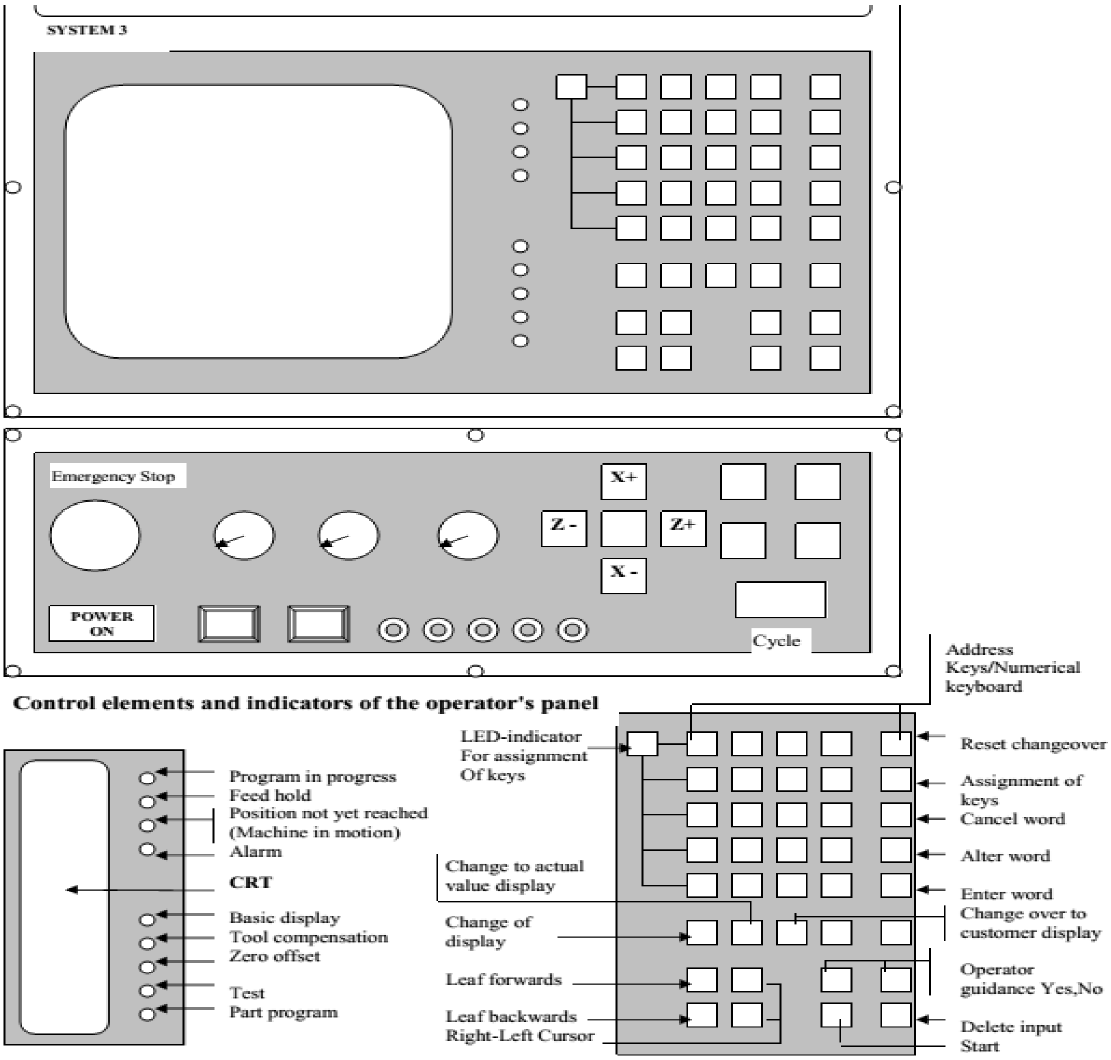


Fig.4 Operator control panel of Hinumerik 3100 system

- ✓ **Machine tool (mechanical part)**

  A Machine Tool (mechanical part) in a CNC machine refers to the specific cutting, shaping, or machining device attached to the CNC machine that performs operations on the workpiece. These tools are mechanically driven by the machine's spindle or other moving parts and are controlled by the CNC system to achieve precise cutting, drilling, milling, or other machining operations.

- ✓ **Program input device**

  A Program Input Device for a CNC machine is the hardware used to input part programs (such as G-code) into the machine's control unit. It serves as the medium through which the operator uploads or transfers the machining instructions that the CNC machine will execute to perform its operations. The program input device facilitates communication between the programming environment (often CAD/CAM software) and the CNC machine.

Common types of program input devices include:

- **Keypads and Control Panels**: Some CNC machines allow operators to manually input G-code directly through the control panel, typically using a keypad.
- **USB Ports**: External storage devices like USB drives are frequently used to transfer part programs from a computer to the CNC machine.
- **Network Connections**: CNC machines can be networked to a central computer or server, allowing part programs to be uploaded via Ethernet or wireless connections directly from CAD/CAM systems.
- **Floppy Disks (Legacy Systems)**: Older CNC machines might still use floppy disks as an input method, though this is largely outdated.
- **Direct Communication Ports**: Some machines use serial or parallel ports for direct communication with computers.

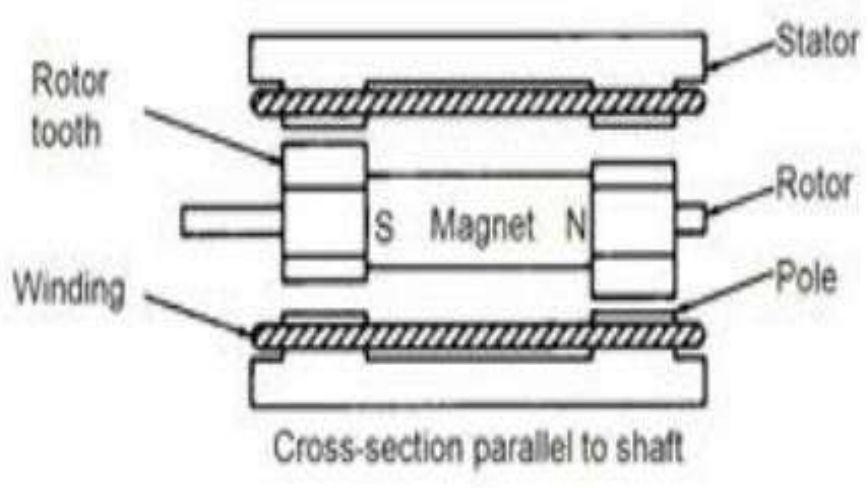
✓ **Drive system**

The Drive System in a CNC machine is a critical component that controls the movement of the machine's axes. It converts electrical signals from the Machine Control Unit (MCU) into precise mechanical movements to position the cutting tool or workpiece accurately. The drive system is responsible for moving the machine along its various axes (X, Y, Z) during the machining process.

Key components of a CNC machine's drive system include:

- **Electric Motors**:

1) **Servo Motors**: Provide high precision and control over positioning and speed. They receive feedback to adjust movement accurately.
2) **Stepper Motors**: Move in discrete steps and are typically used in applications requiring less precision than servo motors.



- **Power Supply:** Provides the electrical energy needed to operate the motors and drive the system.

- **Drive Mechanisms**:

1) **Lead screws or ball screws**: Convert the rotary motion of the motors into linear movement, ensuring smooth and precise positioning along the axes.
2) **Rack and Pinion**: A gear system used for moving larger machines, especially in X-Y motion systems.
3) **Belt Drives**: Used in some CNC machines for transferring motion, typically in light-duty applications.

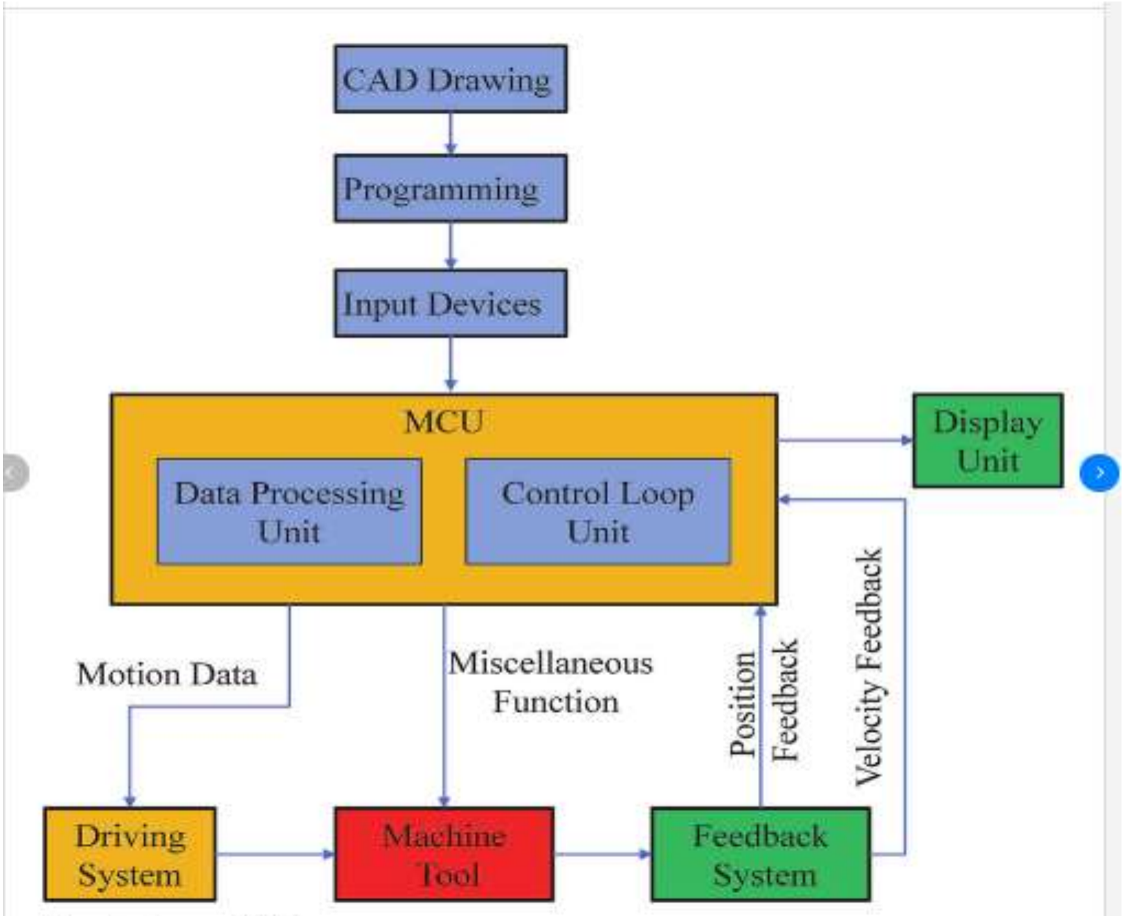
- **Feedback Devices** (in servo systems): Encoders or resolvers provide real-time feedback to the MCU about the position and velocity of the machine, ensuring precise control and movement.

✓ **Feedback system**

A Feedback System in a CNC machine is a crucial component that continuously monitors the machine's performance and provides real-time data to the Machine Control Unit (MCU). It ensures the machine operates accurately and makes adjustments as necessary. The feedback system helps maintain precision by comparing the actual position, speed, or other parameters of the machine's components with the desired values set in the program.

Key components of a feedback system:

1) **Position Feedback**: Sensors that monitor the position of the machine's tool or workpiece along its axes (X, Y, Z) to ensure that it is precisely where it should be.
2) **Velocity Feedback**: Measures the speed at which the tool or workpiece is moving to ensure it matches the desired cutting speed.
3) **Force Feedback**: In some advanced systems, sensors monitor the force applied during cutting to prevent tool breakage or ensure optimal machining conditions.

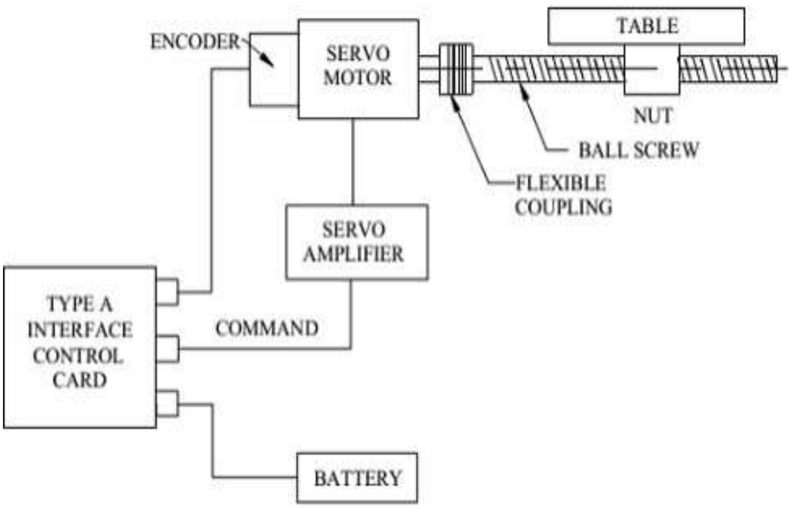


Schematic diagram of CNC.

There are two main types of feedback systems in CNC machines:

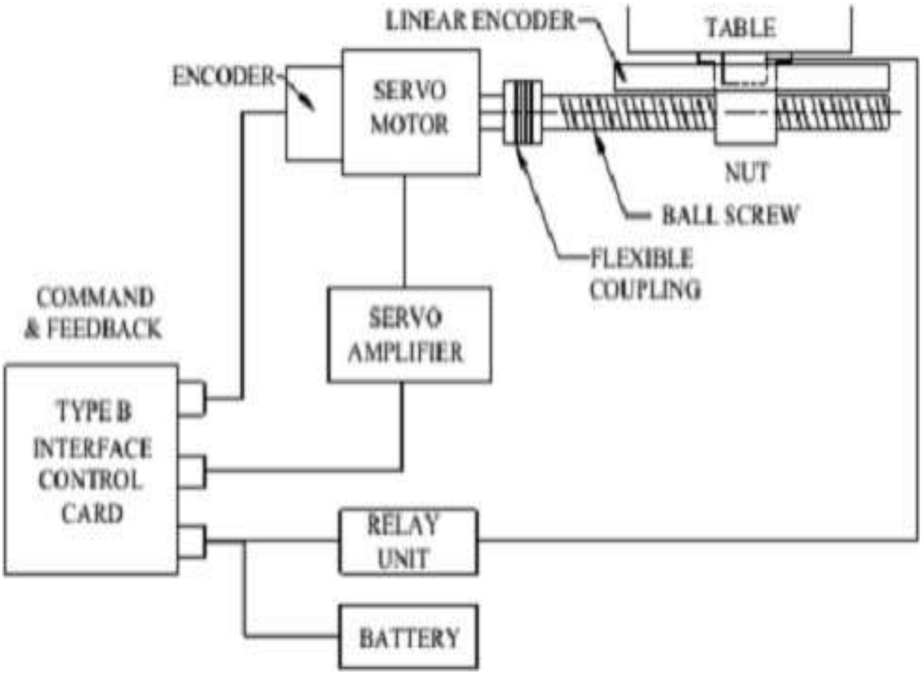
- **Closed-Loop System**

1) Detect the actual position, velocity, or movement of the machine's axes.
2) The feedback is sent to the MCU, which compares it to the programmed values. If there is a deviation, the system corrects it in real-time to ensure accuracy.



Block Diagram of an Open Loop System

- **Open-Loop System:**
  The control unit sends commands to the motors, but there is no mechanism to verify if the commands were executed correctly. These systems are generally used for simpler or lower-precision applications.



Points to Remember

- A CNC (Computer Numerical Control) machine automates machining processes through programmed commands, increasing precision and efficiency, they automate repetitive tasks, increasing production speed and reducing manual labor.
- CNC machine requires a high-performance and comes at a significant cost, which can be a barrier for some users
- CNC machines can easily switch between different tasks by changing the programmed instructions, making them versatile for various manufacturing applications
- Failure to follow safety precautions to CNC machine can lead to accidents, such as cuts, burns, or more severe injuries from moving parts or sharp tools
- Proper machine shutdown procedures lead to quickly address any malfunctions and maintain a safe working environment
- Without proper safety practices, accidents and equipment failures can disrupt workflow, leading to downtime and decreased productivity
- Remember to read the nameplate of the CNC machine before using

- The components of CNC machine work together to ensure precise and efficient machining in CNC operations
- Include information on modern motor technologies, such as direct drive systems, which can improve accuracy by eliminating mechanical backlash.
- The key components of a CNC machine include the Machine Control Unit (MCU), which acts as the machine's brain by interpreting part programs and controlling all machining operations. The interface enables communication between the machine and the user, while the drive system converts electrical signals into precise mechanical movements of the machine's axes. Finally, the feedback system monitors the machine's performance in real-time, ensuring accuracy by comparing actual conditions to programmed values and making necessary adjustments.
- The main components of a CNC machine include machine control unit, interface, machine tool, program input, drive system and a feedback system.

Application of learning 1.1.

Go to the manufacturing workshop where CNC machines are widely used and perform the following tasks:

1. Wear the required Personal Protective Equipment (PPE).
2. Interpret the signs and symbols on the CNC machines.
3. Identify the various types of CNC machines present.
4. Interpret the typical specifications of CNC machines

Indicative content 1.2: Producing NC File

Duration: 13 hrs

Practical Activity 1.2.1: Interpreting drawing

Task:

Go to your school's manufacturing workshop, where your task is to interpret the technical drawing of work piece.

1: Interpret drawing to be used in the CNC machine

2: Present your work to the trainer

3: Read the key reading 1.2.1. And ask for clarification where necessary.

4: Perform the task provided in the application of learning 1.2.1.

Key readings 1.2.1: Interpreting drawing

- **Analysis of product drawing and prototype**

✓ **2D designs**

2D designs in CNC machines refer to flat, two-dimensional representations of parts or products that guide the CNC machine in performing operations such as cutting, engraving, drilling, or milling. These designs are essential for programming the CNC machine to carry out precise and accurate machining tasks.

Analyzing 2D designs used in CNC machines involves a detailed examination to ensure the design is accurate, feasible, and optimized for CNC machining. The goal is to verify that the design can be efficiently translated into machine instructions (G-code) while achieving the desired part geometry, quality, and tolerances.

Procedure:

- **Review design details**: Verify the completeness of the drawing, including lines, symbols, notes, and annotations.
- **Standards Check**: Ensure the 2D drawing complies with relevant industry standards (e.g., ISO, ASME Y14.5).
- **Dimension Verification**: Cross-check dimensions on the 2D drawing with design specifications.
- **Scaling**: Confirm that the drawing scale is appropriate and matches the requirements.
- **Consistency Check**: Verify that all views (e.g., front, top, side) are consistent with one another.

- **Annotations**: Confirm that tolerances, material specifications, and surface finishes are correctly noted.

✓ **3D designs**

3D designs for CNC machines refer to three-dimensional models that are used to guide CNC machines in producing complex parts. These designs are typically created using Computer-Aided Design (CAD) software and include detailed information about the geometry, dimensions, and features of the part to be manufactured. Once designed, the 3D model is converted into instructions (G-code) that the CNC machine follows to produce the part from raw material.

Analysing 3D designs used in CNC machines for producing mechanical parts involves evaluating multiple aspects of the design to ensure that it can be manufactured effectively, accurately, and with the required functionality. This analysis helps identify potential issues and optimizes the design for CNC machining.

Procedure:

- **Geometry Check**: Inspect the 3D model for accuracy in shapes, curves, and alignments.
- **Dimension Comparison**: Cross-check the dimensions in the 3D model with the 2D design and design specifications.
- **CAD Standards**: Ensure the 3D design adheres to CAD modeling best practices (e.g., proper use of layers, constraints, and features).
- **Assembly Fit**: Simulate assembly with other parts to ensure proper fit, alignment, and clearance.
- **Material Analysis**: Verify that the materials specified in the 3D model are accurate and feasible.
- **Simulation**: Perform basic simulation or finite element analysis (FEA) to validate strength, thermal, or dynamic properties if applicable.

✓ **Dimension**

- Check whether all the necessary dimensions are present and clearly labeled. Missing or ambiguous dimensions can lead to machining errors
  Check the:
  i. Overall dimensions (length, width, height)
  ii. Feature dimensions (hole diameters, slots, grooves)
  iii. 3.Depths of cuts or drilled holes
- Ensure the units of measurement are consistent across the entire drawing (e.g., millimeters or inches). Inconsistent units can lead to machining inaccuracies
- Identify critical dimensions (such as fits between parts) that have tighter tolerances. Tighter tolerances require higher precision and may slow down machining or require more advanced tools
- For complex parts, consider whether 3-axis, 4-axis, or 5-axis CNC machines are

required to achieve the necessary dimensions.

Procedure:

- **Inspection Tools**: Use tools like calipers, micrometers, and coordinate measuring machines (CMM) for prototype measurements.
- **Tolerance Verification**: Check that dimensions fall within allowable tolerances as per the design.
- **Cross-Referencing**: Compare measured dimensions with the 2D drawing, 3D model, and specifications.
- **Critical Dimensions**: Focus on critical dimensions that affect functionality or assembly.
- **Documentation**: Record and report all measurements and deviations.

✓ **Cutting section**

- A cutting section is used to show internal features of a part that are not visible from the outside. It provides a view of the internal structure, dimensions, and relationships between different components
- Determine why the section view was included. It might be to clarify complex internal geometries, show assembly details, or indicate hidden features
- Check the view orientation of the section line. It will often be marked with letters or numbers that correspond to the section view on the drawing
- Assess whether the internal features can be manufactured with the available processes and tools. Consider issues like accessibility for machining or welding
- Ensure that the internal features align correctly with other parts in an assembly. Verify clearances and fits.
- Check that all features are correctly represented and that there are no discrepancies
- Ensure that all elements in the section view are clear and readable. This includes checking for legible text, clean lines, and appropriate use of sectioning symbols.
- Confirm that all necessary annotations and labels are present to explain features and dimensions

Procedure:

- **2D Section Analysis**: Examine cutting sections in the 2D design for clarity and detail (e.g., proper hatching, section line labels).
- **3D Cross-Section Analysis**: Use CAD software tools to create virtual sections for internal feature inspection.
- **Prototype Inspection**: Physically cut or use non-destructive methods like CT scans to verify internal features.
- **Dimension Check**: Measure and confirm internal dimensions with the design specifications.
- **Material Integrity**: Inspect internal surfaces for defects or inconsistencies, especially if the section exposes welded or joined areas.

Practical Activity 1.2.2: Writing CNC program

Task:

Go to your school's manufacturing workshop and write a program in CNC machine to produce the part of the circular shaft accurately and efficiently.

1: Prepare the requirement for writing CNC program to be entered into the CNC machine

2: Write CNC program on paper

2: Present your work to the trainer

3: Read the key reading 1.2.2. and ask for clarification where necessary.

4: Perform the task provided in the application of learning 1.2.2.

Key readings 1.2.2.: Writing CNC program

- **Description of CNC programming language**

  CNC (Computer Numerical Control) programming is primarily based on a language known as **G-code** (Geometric Code), which controls the movements and functions of CNC machines. This programming language communicates instructions to the machine about how to move tools, change settings, or perform various machining operations. CNC programming also uses **M-code** (Miscellaneous Code), which handles auxiliary functions.

Codes used In CNC Machine

✓ **G-code (Geometric Code)**

G-code is the primary language for CNC machines. It directs the machine on how to move, specifying toolpaths and operations like cutting, drilling, and milling.

Key Functions of G-codes:

- **Toolpath Control**: G-codes tell the CNC machine where to move along the X, Y, and Z axes, either in a straight line (linear interpolation) or along a curve (circular interpolation).
- **Speed and Feed Rates**: Define the spindle speed (RPM) and the feed rate (how fast the tool moves through the material).
- **Tool Movements**: Controls rapid or slow movements to and from the workpiece, rough cuts, and finishing passes.
  - **G00**: Rapid positioning – move the tool quickly to a specified position.
  - **G01**: Linear interpolation – move the tool in a straight line at a specified feed rate.
  - **G02**: Circular interpolation (clockwise) – cut a circular arc.

- **G03**: Circular interpolation (counterclockwise).
- **G17**: Select XY plane for 2D machining.
- **G20/G21**: Set units to inches (G20) or millimeters (G21).
- **G28**: Return to machine home position.
- **G40**: Cancel tool radius compensation

✓ **M-code (Miscellaneous Code)**

M-codes control non-motion commands, such as turning the spindle on or off, activating the coolant, or performing tool changes. They are typically used in conjunction with G-codes to manage machine operations

- **M03**: Spindle on (clockwise rotation).
- **M04**: Spindle on (counterclockwise rotation).
- **M05**: Spindle off.
- **M06**: Tool change – instructs the machine to switch to another tool.
- **M08**: Coolant on.
- **M09**: Coolant off.
- **M30**: Program end and reset – stops the program and resets the machine for the next part

✓ **Coordinate System**

CNC machines operate based on a defined coordinate system (X, Y, Z). In the program, you can use G-codes to move the tool head to specific points within this system.

- **Absolute positioning (G90)**: Coordinates are defined relative to a fixed origin.
- **Incremental positioning (G91)**: Coordinates are defined relative to the tool's current position

- **Methods of CNC programming**

✓ **Manual CNC Programming**

In manual CNC programming, a programmer writes the **G-code** and **M-code** directly, line by line, to instruct the machine on its operations. This is done without any software assistance and is typically used for simple parts or specific adjustments

Key Features:

- **Direct coding**: G-code and M-code are manually typed into the CNC controller or a programming terminal.
- **Full control**: The programmer has complete control over each machine movement and function.
- **Error-prone**: Since it is written manually, there is a higher chance of human error.
- **Used for simple geometries**: Typically used for straightforward operations such as

basic turning or milling

- ✓ **Computer-Aided Programming (CAM)**

  CAM software automates the CNC programming process by generating G-code based on a 3D CAD (Computer-Aided Design) model. The software calculates the toolpaths and outputs the necessary G-code, greatly simplifying programming for complex parts.

  Key Features:

- **Graphical interface**: CAM programs provide a user-friendly interface to design parts and generate toolpaths.
- **Automatic G-code generation**: The software automatically converts the designed part into G-code, minimizing human error.
- **Used for complex geometries**: Ideal for complex 3D shapes, contours, and multi-axis machining

- ✓ **Conversational Programming**

  Conversational programming is a type of **simplified programming** where the CNC machine's control panel offers a set of predefined functions. The operator selects machining operations (like drilling, turning, pocketing) through an easy-to-use graphical interface, and the machine automatically generates the G-code.

  Key Features:

- **Built-in functions**: Operators input parameters like dimensions, speeds, and feeds, and the machine generates the G-code.
- **No G-code knowledge required**: The operator doesn't need to know G-code as the machine generates it in the background.
- **Best for simple parts**: Conversational programming is commonly used for 2D or simple 3D parts.

- ✓ **Parametric Programming (Macro Programming)**

  Parametric programming, also known as **macro programming**, allows for more flexible and automated CNC code generation by using variables, conditional statements, and loops. It's ideal for parts that require **repetitive operations** or **complex sequences**.

  Key Features:

- **Variables**: The programmer can use variables to define dimensions or tool paths, allowing for easy adjustments and reuse of code.
- **Conditional logic**: Allows the machine to make decisions based on specific conditions, such as repeating operations based on a set number of parts.
- **Loops**: Enables repeating sequences of operations automatically, reducing programming time for batch production

✓ **Offline Programming**

Offline programming involves creating and testing CNC programs away from the machine, typically on a computer. This method allows operators to design and simulate machining processes without interrupting production. The completed program is then uploaded to the CNC machine.

Key Features:

- **Simulation**: Operators can test and verify CNC programs using simulation software to avoid errors during actual machining.
- **No downtime**: Programming is done independently from the machine, allowing it to continue running other jobs

- **Write down the CNC Program**

A CNC program consists of a series of instructions written in G-code or M-code that guides the CNC machine to perform specific tasks.

```
O1000 (Program number)
(T1 - 1/4" End Mill)
G21 (Set units to millimeters)
G17 (Select XY plane)
G90 (Absolute positioning)
G0 Z5.0 (Move the tool to a safe height above the workpiece)

(SPINDLE ON - CLOCKWISE)
M6 T1 (Select tool T1)
M3 S1200 (Start spindle at 1200 RPM)

G0 X0 Y0 (Move to the starting point)
G1 Z-5.0 F100 (Plunge into the material at 100 mm/min feed rate)
G1 X50 Y0 F200 (Cut a line to X50 at 200 mm/min feed rate)
G1 X50 Y50 (Cut a line to Y50)
G1 X0 Y50 (Cut a line to X0)
G1 X0 Y0 (Return to the starting point)

G0 Z5.0 (Retract the tool to a safe height)
M05 (Stop spindle)
M30 (End of program)
```

Explanation:

- **G21** sets the units to millimeters.
- **G17** selects the XY plane**.**
- **G90** enables absolute positioning.
- **M6 T1** selects the tool.

- **M3 S1200** starts the spindle at 1200 RPM.
- **G0** commands are used for rapid positioning.
- **G1** commands are used for linear interpolation with specified feed rates.
- **M05** stops the spindle.
- **M30** ends the program.

Example of writing CNC PROGRAM for simple mechanical part

Example of a CNC program for creating a simple mechanical part on a CNC milling machine. The part will be a rectangular bracket with two holes drilled through it. This example includes basic operations such as milling, drilling, and positioning

Part Specifications:

- **Bracket Dimensions**: 100mm x 50mm x 10mm (Length x Width x Height)
- **Hole Diameter**: 10mm
- **Hole Positions**: Centered along the width, 20mm from each end

CNC Program

```gcode
O1000 (Program for Rectangular Bracket with Holes)
G21 (Set units to millimeters)
G17 (Select XY plane for milling)
G90 (Absolute positioning)
G40 (Cancel tool radius compensation)
G49 (Cancel tool length offset)
G80 (Cancel canned cycle)

T0101 (Select Tool 1 with offset 1 - End Mill)
M06 (Tool change)
G97 S2000 M03 (Set spindle speed to 2000 RPM, Spindle ON clockwise)

G00 X0 Y0 Z5 (Rapid move to start position above the part)
G00 X0 Y0 (Rapid move to the origin of the workpiece)

M08 (Coolant ON)
```

```
M08 (Coolant ON)
G01 Z-10 F100 (Mill down to a depth of -10mm at a feed rate of 100 mm/min)
G01 X100 Y0 (Mill along X-axis to 100mm)
G01 Y50 (Mill along Y-axis to 50mm)
G01 X0 (Mill back along X-axis to 0mm)
G01 Y0 (Mill back along Y-axis to 0mm)
G00 Z5 (Rapid move up to a safe position)

(Tool Change for Drilling)
T0202 (Select Tool 2 with offset 2 - Drill Bit)
M06 (Tool change)
G97 S1500 M03 (Set spindle speed to 1500 RPM, Spindle ON clockwise)

G00 X20 Y10 (Rapid move to first hole position)
G81 Z-10 R5 (Drilling cycle to a depth of -10mm, retract to 5mm above the part)
G80 (Cancel drilling cycle)
```

```
G00 X80 Y10 (Rapid move to second hole position)
G81 Z-10 R5 (Drilling cycle to a depth of -10mm, retract to 5mm above the part)
G80 (Cancel drilling cycle)

G00 Z100 (Retract tool to safe position)

M09 (Coolant OFF)
M05 (Spindle OFF)
G28 U0 V0 (Return to home position)
M30 (End of program and reset)
```

Explanation of the CNC Program:

1. **Program Start**:
   O1000: Program number.
   G21: Set units to millimeters.
   G17: Select XY plane for milling.
   G90: Set absolute positioning mode.
   G40: Cancel tool radius compensation.
   G49: Cancel tool length offset.
   G80: Cancel any active canned cycles.
2. **Milling Operations:**
   T0101: Select Tool 1 (end mill) with offset 1.
   M06: Tool change command.
   G97 S2000 M03: Set spindle speed to 2000 RPM and turn spindle ON clockwise.
   G00 X0 Y0 Z5: Rapid move to the start position above the workpiece.
   G01 Z-10 F100: Mill down to a depth of -10mm at a feed rate of 100 mm/min.

G01 X100 Y0: Mill along the X-axis to 100mm.
G01 Y50: Mill along the Y-axis to 50mm.
G01 X0: Mill back along the X-axis to 0mm.
G01 Y0: Mill back along the Y-axis to 0mm.
G00 Z5: Rapid retract to a safe height.

3. **Drilling Operations:**
T0202: Select Tool 2 (drill bit) with offset 2.
M06: Tool change command.
G97 S1500 M03: Set spindle speed to 1500 RPM and turn spindle ON clockwise.
G00 X20 Y10: Rapid move to the position for the first hole.
G81 Z-10 R5: Drilling cycle to a depth of -10mm with retract height of 5mm.
G80: Cancel the drilling cycle.
G00 X80 Y10: Rapid move to the position for the second hole.
G81 Z-10 R5: Drilling cycle to a depth of -10mm.
G80: Cancel the drilling cycle.

4. **Program End:**
G00 Z100: Retract tool to a safe position.
M09: Coolant OFF.
M05: Spindle OFF.
G28 U0 V0: Return to home position.
M30: End program and reset.

Points to Remember

- When there is a poor analysis of product drawings and prototypes in CNC machining, it can lead to a range of issues that affect the quality, functionality, and cost of the final product
- Pay close attention to tolerances, critical dimensions, and fits.
- Ensure the dimensions are correctly interpreted for the CNC program.
- Incorrect G-code commands can result in parts being machined with incorrect dimensions, leading to unusable products.
- Inaccurate programming can cause the tool to cut too much or too little material, wasting raw materials and potentially ruining the part.
- When writing a CNC program, ensure you accurately define the sequence of operations and specify precise G-code and M-code commands to control the machine's movements and functions. Always review the program thoroughly for errors and simulate it if possible to avoid potential issues during machining

- Poor programming may require frequent machine stoppages for manual adjustments or reprogramming, which can delay production.

Application of learning 1.2.

Go to the school manufacturing workshop, where the school has received an order to produce a circular shaft with the following specifications:

- ✓ Material: Mild Steel
- ✓ Diameter: 50 mm
- ✓ Length: 150 mm
- ✓ Tolerances: ±0.05 mm for both diameter and length

Your task is to write the CNC program required to produce the shaft using a CNC lathe, by ensuring that the part meets the provided specifications.

Indicative content 1.3: Simulating NC Files

Duration: 10 hrs

Practical Activity 1.3.1: Simulating CNC program

Task:

Go to your school manufacturing workshop and Simulate CNC program
1: Apply safety precautions
2: Simulate CNC program
3: Present your work to the trainer
4: Read the key reading 1.3.1. and ask for clarification where necessary.
5: Perform the task provided in the application of learning 1.3

Key readings 1.3.1.: Simulating CNC program

- **Description of CAM software**

✓ **Definition CAM software**

CAM (Computer-Aided Manufacturing) software is a type of application used to control and automate manufacturing processes through computer-generated instructions. CAM software converts 3D models or 2D designs created in CAD (Computer-Aided Design) software into G-code, which is then used by CNC (Computer Numerical Control) machines to produce physical parts.

✓ **Commonly used CAM software and their features**

- CAM software Provides advanced toolpath strategies for milling, turning, and wire EDM
- CAM Software can used in Solid CAM
- CAM software can used in Siemens NX to Customize post-processors for a variety of CNC machines

✓ **Function of CAM software**

- CAM software translates CAD (Computer-Aided Design) models into machine-readable G-code, which controls the movements and actions of CNC machines. It automates the manufacturing of parts and components according to the design specifications.
- CAM software provides simulation of the machining process before actual production. This allows operators to check for errors, visualize the toolpaths, avoid collisions, and ensure the part will be produced correctly without damaging tools or

materials.

- CAM software helps optimize cutting strategies, reducing **machining time** and improving the lifespan of tools by calculating the most efficient cutting paths, feed rates, and speeds. This leads to reduced material waste and increased productivity
- CAM software is preferred in industries where both simple 2D and complex 5-axis operations are needed, such as in medical, aerospace, and automotive manufacturing
- CAM software enables **multi-axis machining**, which is essential for producing complex parts with intricate geometries. It supports up to 5-axis machining, allowing the CNC machine to move and cut along multiple axes simultaneously
- CAM software converts tool paths into machine-specific **G-code** or **M-code** through post-processing. This ensures compatibility with the specific CNC machine being used, allowing it to execute the machining operations according to the software-generated instructions

✓ **Converting the CAD file to CNC program**

- **Import CAD Model**

The following are the steps to import a CAD model into a CNC machine:

Step 1: Prepare the CAD Model
Step 2: Open CAM Software
Step 3: Generate Toolpaths
Step 4: Post-Process (**Generate G-Code**, **Save G-Code File**)
Step 5: Transfer to CNC Machine
Step 6: Load and Run
Step 7: Monitor and inspect

- **Generate Toolpaths**

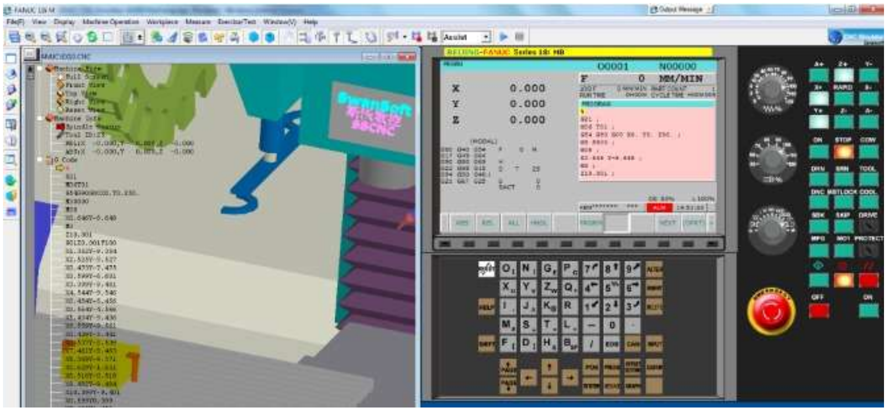
The following are the steps of Generate Toolpaths:

Step 1: Import the CAD Model
Step 2: Define Workpiece and Setup
Step 3: Select and Define Tools
Step 4: Create Toolpaths
Step 5: Simulate Toolpaths
Step 6: Post-Process Toolpaths
Step 7: Export Toolpaths
Step 8: Transfer to CNC Machine

- **Simulation Toolpaths**

Simulation and verification are crucial steps in the process of converting a CAD file into a CNC program. They ensure that the toolpaths generated for machining are accurate and will produce the desired results without errors or collisions. Here's a guide to help you through this process

1. **Run Simulation**: Use the simulation feature in your CAM software to visualize the machining process. This helps you see how the toolpaths will be executed on the model.
2. **Check for Issues**:
   - **Collision Detection**: Look for potential collisions between tools, fixtures, or the workpiece.
   - **Toolpath Accuracy**: Ensure that the toolpaths accurately follow the intended geometry and do not produce any unintended cuts.
   - **Material Removal**: Verify that the material is being removed correctly according to your design
3. **Verify Toolpaths**
   - **Review Simulation Results**: Carefully review the results of the simulation to ensure that all aspects of the machining process are correct.
   - **Adjust Toolpaths**: If any issues are found adjust the toolpaths or parameters as needed and re-run the simulation to check for improvements.



Points to Remember

- To convert a CAD file to a CNC program, the design must first be exported in a format compatible with CAM software (such as DXF or IGES), where toolpaths and machining parameters are defined. Once the toolpaths are set, the CAM software generates the G-code/M-code needed for the CNC machine to execute the design.
- When simulating NC files in a CNC program, ensure that the toolpath is accurately visualized to identify potential errors before actual machining. Verify cutting parameters and machine movements to prevent collisions and optimize efficiency during the machining process.

Application of learning 1.3.

The school plans to produce a circular shaft with dimensions of 150 mm in length and 50 mm in diameter, requiring high precision to meet product specifications. You are requested to go to the manufacturing workshop where the CNC lathe is located, and perform the following tasks:

1. Create a CAD design of the circular shaft.
2. Export the CAD file to CAM software in STEP format.
3. Set up the toolpaths for machining.
4. Generate the NC file in G-code format.
5. Simulate the NC file using the CAM software's simulation feature

Learning outcome 1 end assessment

Written assessment

1. Read carefully the statement below and answer **by TRUE** if the statement is correct or **FALSE** if the statement is wrong:
   i. A CNC machine automates the operation of machinery to shape materials like metal, plastic, or wood with high precision.
   ii. Part programs do not define toolpaths or select tools in CNC machines.
   iii. G-code controls non-geometrical functions such as turning the spindle on or off.
   iv. CAD software is used to design parts, while CAM software converts these designs into machine instructions.
   v. The number of axes in a CNC machine determines the complexity of its movements and machining operations.
   vi. Cutting parameters like spindle speed and feed rate do not affect the precision of the CNC machining process.
   vii. A fixture is used to hold a workpiece securely in place during CNC machining.

2. Match the items in **Column B,** which describe the Personal protective equipment (PPE), with their corresponding list of type of PPE in **Column C** and provide answer in **Column A**

Column A:Answer	**Column B: Description**	**Column C: Types of PPE**
1………	1.Protects the hands from cuts, burns, and chemicals	A. safety glasses
2…….	2. Protects the feet from heavy objects and sharp materials	B. Safety gloves
3……..	3.Shields eyes from debris, dust, and chemical splashes.	C. Hard Hat
4……..	4.Protects the head from falling objects and impacts.	D. Safety Shoes
5…….	5.Protect the mouth from dust	E. Earplugs
6……..	6.Reduces exposure to loud noises in the work environment	F. Emergency Stop
		G. Air Mask

3. Fill the gap of the following statement by using the correct types of CNC machine among the list provided (**CNC waterjet machine**, **CNC lathe, CNC EDM (Electrical Discharge Machine), CNC milling machine**, CNC **laser cutter**, **CNC plasma, CNC drilling)**

   a) The CNC machine used primarily for cutting and shaping materials into cylindrical shapes is called a ……………
   b) A ……………is commonly used for cutting materials with a rotating tool that moves along multiple axes.
   c) The **……………**uses electrical discharges to shape hard materials, usually for die-making.
   d) The …………uses a high-pressure stream of water mixed with abrasive particles to cut through materials.
   e) A … ………...is specialized for cutting materials in flat sheets using a laser beam.

4. Read the below statement and choose the right answer by encircling the corresponding to the correct answer:

   i. What is the purpose of sectional views in CNC drawings?

      A) To show the color of the material

      B) To provide a view of internal features

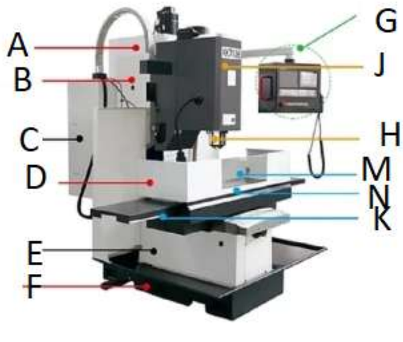
      C) To indicate tolerances

      D) To display dimensions only

   ii. What is the primary purpose of a CNC drawing?

      a) To provide aesthetic appeal to the final product.
      b) To communicate the exact dimensions and specifications for machining.
      c) To serve as a blueprint for the machinist's artistic interpretation.
      d) To document the history of the part's design and production

5. Observe the following CNC milling machine figure and name the marked parts by using the following words in bracket (**ram, column, electrical cabinet, enclosures, coolant, base, splinder motor, ball screw)**



Practical assessment

The school is facing an issue with outdated circular shafts in their milling machine. They plan to announce a tender for the production of a new circular shaft with dimensions of 150 mm in length and 50 mm in diameter, to be manufactured using a CNC machine. You are asked to:

1. Create a CAD design of the circular shaft.
2. Export the CAD file to CAM software in STEP format.
3. Set up the toolpaths for machining.
4. Generate the NC file in G-code format.
5. Simulate the NC file using the CAM software's simulation feature

References

Madison, J. (1996). CNC machining handbook: basic theory, production data, and machining

Negi, P., Ram, M., & Yadav, O. P. (2022). Basics of CNC Programming. CRC Press.

Smid, P. (2003). CNC programming handbook: a comprehensive guide to practical CNC programming.

Smid, P. (2010). CNC control setup for milling and turning: mastering CNC control systems. Industrial

Learning Outcome 2: Set up CNC Machine

Indicative contents
2.1 Applying Techniques of Setting tools in CNC machine **2.2 Transferring data to CNC machine** **2.3 Setting CNC machine parameters**

Key Competencies for Learning Outcome 2: Set up CNC Machine

Knowledge	Skills	Attitudes
• Identification of techniques of setting tools in CNC machine. • Identification of methods of transferring data to CNC machine	• Applying techniques of setting tools in CNC machines • Transferring the data to a CNC machine • Setting CNC machine parameters according to the work to be done	• Paying attention while setting tools in CNC machine. • Being careful while transferring data to CNC machine • Being accurate when setting parameters in CNC machine

Duration: 30hrs

Learning outcome 2 objectives:

By the end of the learning outcome, the trainees will be able to:

1. Identify properly techniques of setting tools in CNC machine
2. Identify properly the methods of transferring data to CNC machine as used in manufacturing industry.
3. Apply correctly techniques of setting tools in CNC machine as used in manufacturing industry.
4. Transfer properly the data to CNC machine as used in manufacturing industry.
5. Set properly CNC machine parameters according to the work to be done

Resources

Equipment	**Tools**	**Materials**
● CNC Machine ● CNC Milling ● CNC Turning ● CNC Grinding ● CNC Drilling ● CNC router ● CNC lasercutter	● Hammer ● Collets ● Wrenches ● Allen keys ● Torque wrenches ● Tool setter ● Dial Test Indicators ● Tool Balancer ● Torque Wrenches ● Cutting tool set ● SINUMERIK Software ● CNC Simulator Pro ● Autodesk fusion 360	● N/A

	- USB Drive - Memory Card - Flash disc - ARTCAM, - MARTCAM, - MASTERCAM	

Indicative content 2.1: Applying Techniques of Setting Tools in CNC Machine

Duration: 10 hrs

Practical Activity 2.1.1: Applying Techniques of Setting tools in CNC machine

Task:

Go to the manufacturing workshop and apply techniques of setting tools in CNC machine.

1: Apply safety precautions

2: Set tools in CNC machine according to the trainer demonstration

3: Present your work to the trainer

4: Read the key reading 2.1.1 and ask for clarification where necessary.

5: Perform the task provided in the application of learning 2.1

Key readings 2.1.1: Applying techniques of setting tools in CNC machine

- **Techniques of setting tools in CNC machine**

✓ **Manual tool setup**

Manual tool setup, the operator physically measures and adjusts the tool offset by hand. The tool is manually positioned, and the operator uses the CNC machine's control panel to set the tool length and diameter.

- **Procedure**

1) Position the tool close to the workpiece surface (usually in the Z-axis).
2) Slowly jog the tool down until it lightly touches the workpiece surface.
3) Use a feeler gauge or paper to determine when the tool makes contact, then set the tool offset in the machine control.

- **Application**

Manual setup is straightforward but time-consuming and prone to human error, typically used in smaller shops or for less critical jobs.

- **Accuracy**

It relies on the skill of the operator and can be less accurate compared to automated systems.

✓ **Tool Setter**

A tool setter is a device placed inside the CNC machine that automatically measures the tool length and sometimes tool diameter. It consists of a probe that the tool touches, sending data to the CNC controller to set the tool offset.

- **Procedures**

1) The tool setter is installed at a fixed position in the machine.
2) The machine's tool touches the setter, which measures the length (and diameter if applicable).
3) The CNC controller automatically records the tool offset for accurate machining.

- **Application**

  This is used in production environments where multiple tools are involved. It ensures rapid and precise tool setup, reducing human error.

- **Accuracy**

  Highly accurate and consistent, making it ideal for high-precision machining.

✓ **Probe Systems**

Probe systems are advanced measuring devices used to automatically locate the workpiece's surface, edges, or other key features. Probes can also measure the tool's length and diameter, providing input directly to the CNC controller.

- **Procedure**

1) The CNC machine moves the probe toward the workpiece or tool.
2) The probe contacts the surface or tool, and the CNC system records the position.
3) This data calculates the tool offset and/or work offsets.

- **Application**

  Probes are commonly used for both tool and workpiece setup, especially in complex parts with tight tolerances. They provide quick and accurate measurements, improving process efficiency.

- **Accuracy**

  Extremely accurate, capable of detecting even small deviations. Probes are used in high-precision manufacturing.

✓ **Edge Finders**

Edge finder is a tool used to manually locate the edge or center of a workpiece in CNC machines. It is typically a spring-loaded or mechanical device that rotates and shifts when it touches the workpiece, helping the operator find the exact position.

- **Procedures**

1) Insert the edge finder into the spindle.
2) Move the edge finder toward the workpiece while rotating.
3) When the tool touches the workpiece edge, the tip of the edge finder "jumps" or shifts. This indicates the edge has been located.
4) Record the coordinates and set the work offset accordingly.

- **Application**

  This is used in both manual and CNC machining to manually locate the edge of the workpiece for part alignment.

- **Accuracy**

  Provides moderate accuracy but is slower and more dependent on operator skill compared to automated systems like probes.

- ✓ **Optical Tool Measurement Systems**

  Optical tool measurement systems use cameras and lasers to measure the tool's length and diameter outside of the CNC machine. These systems are highly automated and provide non-contact measurement.

- **Procedures**

1) Place the tool in the optical measurement system, which uses a camera or laser to scan the tool.
2) The system captures detailed measurements of the tool's geometry, including length, diameter, and wear.
3) This data is transferred to the CNC machine to automatically set tool offsets.

- **Application**

  Typically used in high-precision machining environments for complex or delicate tools.

  It ensures highly accurate and consistent tool measurement.

- **Accuracy**

  Provides very high precision, allowing detailed measurement even for very small or complex tools.

Points to Remember

- Manual Tool Setup is straightforward but depends heavily on the operator's skill, making it prone to errors and time-consuming.
- Tool Setters automate tool length and diameter measurement, improving speed and accuracy in production environments.
- Probe Systems provide highly precise measurements for both tools and workpieces, ideal for high-precision machining.
- Edge Finders are manually operated, offering moderate accuracy for locating workpiece edges in simpler tasks.
- Optical Tool Measurement Systems use lasers and cameras to deliver extremely accurate, non-contact tool measurements, perfect for delicate or complex tools.

Application of learning 2.1

The school requires a car circular shaft with a length of 150 mm and a diameter of 50 mm. you are requested to set CNC machine by using the proper tool-setting techniques. You are requested to:

1. Position the tool
2. Install the tool setter at a fixed position
3. Locate the edge of the workpiece by using the edge finders
4. Use optical measurement system

Indicative content 2.2: Transferring Data to CNC Machine

Duration: 10 hrs

Theoretical Activity 2.2.1: Identification of methods of transferring data to CNC machine

Tasks:

1: Answer the following question:

i. What do you understand about the methods of transferring data to CNC machine?

2: Provide the answers to the asked questions by writing them on paper or flip chart

3: Present the answers to the whole class.

4: For more clarification, read the key readings 2.2.1

Key readings 2.2.1.: Identification of methods of transferring data

- **Methods of transferring data**

✓ **Manual data input (MDI):**

MDI involves entering G-codes, M-codes, and other commands manually into the CNC machine's control panel. This method is suitable for small programs or when quick changes or adjustments are needed.

Advantages of manual data input

1) Useful for quick, minor changes.
2) Time-consuming and prone to errors for longer programs.
3) Often used for simple, short tasks.

✓ **USB cable:**

Transferring CNC program files via a USB drive or directly using a USB cable from a computer to the CNC machine. USB is a common and easy method for transferring files like G-code programs to the machine.

Advantages of USB cable

1) Easy to use and widely available.
2) Portable and requires no special software.
3) Suitable for machines with USB ports.

✓ **Wireless file transfer:**

CNC machines can be equipped with Wi-Fi or Bluetooth to transfer data wirelessly from a computer or server. This is Ideal for large workshops where several machines can connect to a central system for program transfer.

Advantages of wireless file transfer

1) No physical connection needed.
2) Convenient for multiple machines.
3) Requires network security measures to prevent unauthorized access.

✓ **Network transfer:**

Transferring files via a local network (LAN) where CNC machines are connected to a central server or database. Common in large production environments where CNC machines are networked to streamline program transfer and management.

Advantages of network transfer

1) Efficient for large-scale operations.
2) Centralized storage of files, ensuring version control.
3) Requires proper network setup and maintenance.

✓ **External storage devices:**

Using devices like SD cards, external hard drives, or other portable storage devices to transfer files. This method is similar to using a USB but with **alternative storage formats**.

Advantages of external storage devices:

1) Portable and easy to use.
2) Offers large storage capacity for complex programs.
3) Useful in environments with limited network access.

- ✓ **Data transfer cables:**

  RS-232 serial cables or other data cables to transfer files from a computer directly to the CNC machine. This method is an older technique but still used in machines without USB or wireless capabilities.

Advantages of data transfer cable:

1) Reliable for older machines.
2) Limited transfer speed compared to newer methods.
3) Requires compatible software and hardware setups

Practical Activity 2.2.2: Transferring data to CNC machine

Task:

You are requested to go to a manufacturing workshop and apply methods of transferring data to CNC machine.

1: Prepare all requirement needed to be used

3: Transfer data in CNC machine according to the trainer demonstration.

4: Present your work to the trainer

5: Read the key reading 2.2.2

6: Perform the task provided in the application of learning 2.2.

Key readings 2.2.2: Transferring data to CNC machine

- **Steps of transferring data to CNC machine**

Step 1. Create or import the CNC Program

- ✓ Use CAM Software: Most CNC programs are created using Computer-Aided Manufacturing (CAM) software. This software converts CAD (Computer-Aided Design) models into G-code, which is the language understood by CNC machines.
- ✓ Import Existing G-Code: If you have a pre-existing G-code file, you can directly import it into the CNC machine's controller.

Step 2. Connect or transfer data from Computer to the CNC Machine

Step 4. Verify and Run the Program

- ✓ Simulation: Many CNC controllers allow you to simulate the program to ensure it runs as expected without damaging the workpiece.
- ✓ Dry Run: Perform a dry run with no workpiece to check for tool path errors or collisions.
- ✓ Production Run: Once satisfied with the simulation and dry run, start the production run.

- **Check program error-free**
- ✓ **Use CAM software's simulation**

Most CNC machines and CAM software have a built-in simulation feature. Run the program in simulation mode to visualize the toolpaths and verify the cutting operations without engaging the actual machine.

Look for any tool collisions, unexpected movements, or issues with the cutting sequence.

- ✓ **Dry run (air cutting)**

  Perform a dry run or air cut where the machine runs through the entire program without cutting the material. This allows you to check the machine's movements without the risk of damaging the workpiece or tools.

  Monitor the tool's path and ensure all axes move as expected.

- ✓ **Verify Toolpaths:**

  Review the toolpaths in your CAM software to ensure they match the intended design. Pay attention to entry/exit points, cutting depths, and tool changes.

- ✓ **Check for G-code errors:**

  Inspect the G-code or M-code for syntax errors or mis commands. Some CNC controllers will flag issues when loading the code, such as missing or incorrect commands.

  Look for issues like missing end-of-program codes, tool change commands, or improper use of coordinate systems.

- ✓ **Tool and workpiece offsets:**

  Double-check tool and workpiece offsets to ensure they are set correctly. Misaligned offsets can cause errors during the machining process.

- ✓ **Machine parameter validation:**

  Ensure the correct spindle speed, feed rates, and cutting parameters are used for the material and tools. Incorrect settings may cause cutting errors or tool breakage.

- ✓ **Test run with scrap material:**

  Run the program on a piece of scrap material before using the actual workpiece. This allows you to verify the program in real conditions and check for potential errors without risking valuable material.

- ✓ **Check machine alarms/warnings:**

  Pay attention to any alarms or warnings on the CNC machine during the test runs. These indicators can alert you to potential problems with the program or machine settings.

Points to Remember

- Always double-check your program for errors before running it to prevent costly mistakes.
- Check for any G-code errors, validate tool and work offsets, and confirm cutting parameters to ensure the program is error-free.
- Data transfer cables are suited for older machines but less efficient than modern alternatives.
- Manually review the G-code for errors such as incorrect coordinates, speeds, or tool selections.
- MDI is useful for quick manual input but not practical for long programs.
- Multiple methods exist to transfer data to a CNC machine, including Manual Data Input (MDI), USB drives, and wireless file transfer.
- Network transfer allows direct communication between the computer and the CNC machine for seamless data transfer.
- Perform a dry run (without cutting material) to ensure the program behaves as expected.
- USB and External Storage are simple and effective for most jobs.
- USB cables and external storage devices are widely used and convenient for data transfer.
- Use MDI for quick manual inputs but avoid for complex jobs.
- Wireless and Network Transfer are efficient for large operations with multiple CNC machines.
- Wireless and network transfer methods provide seamless and efficient file transfer in large production environments.

Application of learning 2.2

Go to the manufacturing workshop where car circular shaft having 150 mm in length and 50 mm in diameter has been designed using CAD software. You are requested to perform the following tasks:

1. Transfer the design data to the CNC lathe machine to begin the machining process.
2. Verify the program to ensure it is free of errors before starting production

Indicative content 2.3: Setting CNC Machine Parameters

Duration: 10 hrs

Practical Activity 2.3.1: Setting CNC machine parameters

Task:

Go to manufacturing workshop where CNC machine is extensively used and Set parameters of CNC machine.

1: Apply safety precaution

2: Set parameters of CNC machine according to the trainer demonstration.

3: Present your work to the trainer

4: Read the key reading 2.3.1 and ask for clarification where necessary.

5: Perform the task provided in the application of learning 2.3

Key readings 2.3.1: Setting CNC machine parameters

- **Applying the techniques of setting parameters of CNC machine**

✓ **Initialized States**

Initialized states refer to the default conditions or home positions of the CNC machine. These include the machine's axis positions (typically at zero reference points), tool offsets, and any default settings for the units (inch or metric). This ensures that the machine knows where it is positioned in space and is ready to begin operations accurately.

- Ensure that the machine is in its default "home" or initialized state before beginning any operations.
- Check that all axis positions (X, Y, Z) are zeroed, and tool offsets are set to their default values.
- Verify that the machine is using the correct unit system (inches or metric).

✓ **Canned Cycles**

Canned cycles are pre-programmed routines within CNC machines that automate repetitive tasks, such as drilling, tapping, and boring. These cycles simplify programming by allowing the operator to input just a few parameters (e.g., depth, feed rate), and the machine executes the full operation. Common canned cycles include G81 (drilling), G83 (peck drilling), and G84 (tapping).

- Use canned cycles for repetitive operations like drilling, tapping, or boring.
- Input the required parameters (e.g., depth, feed rate, retract position) into the CNC control panel.
- Canned cycles like G81 (drilling) or G84 (tapping) reduce programming effort and minimize errors.

✓ **Data entry**

Data entry refers to the process of inputting key operational values into the CNC machine, such as tool offsets, spindle speeds, feed rates, and workpiece positions. This can be done manually through the control panel or automatically through a program. Accurate data entry is essential for precise machining

- Accurately input necessary data, including tool offsets, spindle speeds, feed rates, and workpiece positions.
- Double-check manual entries to ensure precision.
- Utilize automatic data entry options when available to reduce human error.

✓ **Communications and file loading**

Communications and file loading involve transferring CNC program files (often in G-code format) from a computer or storage device to the CNC machine. This can be done via USB, network transfer, wireless systems, or older methods like RS-232 serial cables. Ensuring the file loads correctly without errors is critical for smooth operation.

- Ensure the correct G-code file is loaded onto the CNC machine before starting.
- Verify the communication method (USB, wireless, network) is functioning correctly.
- Check for any file corruption or errors after loading the file into the machine's memory.

✓ **Program protection**

Program protection is a feature that safeguards critical CNC programs from being accidentally modified, deleted, or overwritten. This can include locking programs or adding password protection to prevent unauthorized access. It ensures that only authorized personnel can make changes to essential programs.

- Implement program protection measures such as password protection or lock settings to safeguard critical CNC programs.
- Ensure only authorized personnel can modify or access sensitive CNC program files.

- Regularly backup CNC programs to avoid data loss.

✓ **User-Defined G and M Codes**

User-defined G and M codes allow operators to create custom commands that perform specific functions not covered by the machine's standard G and M codes. For example, custom codes can be programmed to start a specific probing cycle or initiate special machine operations unique to the production environment.

- Make use of custom G and M codes to simplify repetitive or specialized tasks.
- Ensure that user-defined codes are well-documented and stored in the CNC system for future use.
- Verify that any custom codes operate correctly in your specific machine environment.

✓ **Inch-Metric conversion**

Inch-metric conversion allows the CNC machine to switch between programming in inches (G20) and millimeters (G21). This is important when working with international standards or designs that use different measurement units. The machine adjusts the program and coordinate values accordingly, ensuring proper scaling and operation.

- If switching between inch and metric measurements, confirm the correct G-code is used (G20 for inch, G21 for metric).
- Check that all coordinate values are scaled properly after the unit conversion.
- Double-check the dimensions of the workpiece and tools to avoid scaling errors during machining.

- **Steps for setting CNC machine parameters**
  - **Review the drawing**: Examine the engineering drawing for material, dimensions, tolerances, and surface finish requirements.
  - **Select cutting tool:** Choose an appropriate cutting tool based on the material and machining operation.
  - **Secure workpiece**: Mount the workpiece in the CNC chuck, ensuring proper alignment and centering.
  - **Enter cutting parameters:** Set spindle speed (RPM), feed rate, and depth of cut based on material and desired outcomes.
  - **Set coolant and lubrication**: Enable the coolant system and adjust the flow rate for effective cooling during machining.
  - **Input tool offset**: Enter tool offset values into the CNC control system for accurate positioning of the cutting tool.
  - **Run simulation**: Verify toolpaths and parameters through a simulation to check for

errors or collisions.

- **Conduct test run**: Perform a test pass with minimal cutting depth to confirm settings and accuracy.
- **Monitor and adjust**: Continuously observe the machining process and adjust parameters as needed for optimal results

Points to Remember

- Always run a short, non-cutting test program to verify that conversions were applied correctly before proceeding with live machining.
- Regularly maintain the machine to keep sensors, encoders, and other critical components in working order, as this impact parameter accuracy.
- Regularly review and test custom G-codes to ensure they remain functional after machine software updates or changes.

Application of learning 2.3

Go to the manufacturing workshop where car circular shaft measuring 150 mm in length and 50 mm in diameter has been designed using CAD software and transferred in CNC lathe machine. You are requested to perform the following tasks:

1. Set up cutting parameter of CNC machine
2. Set up tool offset
3. Set up coolant and lubrication

Learning outcome 2 end assessment

Written assessment

1. Read carefully the following statement and answer by **TRUE** if the statement is **correct or FALSE** if the statement is **wrong.**
   a) Manual Tool Setup is an automated process that eliminates human error.
   b) A tool setter automatically measures tool length and sometimes diameter, sending data to the CNC controller.
   c) In manual tool setup, the operator uses feeler gauges or paper to set the tool offset.
   d) Probe systems are only used to locate workpiece surfaces and cannot measure tool length or diameter.
   e) Edge finders are used to automatically locate the edges of the workpiece without any manual input.
   f) Optical Tool Measurement Systems use cameras and lasers to measure tool geometry with high accuracy.
   g) Tool setters are highly accurate and often used in high-precision production environments.
   h) Manual Tool Setup is typically faster and more accurate than tool setters or probe systems.
   i) Edge finders rely on the operator to detect the point when the tool touches the workpiece.
2. Read and match the techniques of setting tools in CNC machine, in **column B,** with their explanation in **Column C**, and provide answer in **Column A**

Column A: Answer	Column B: Technique of Setting Tool in CNC Machine	Column C: Explanation of Techniques of Setting Tool in CNC Machine
i……	I. Manual Tool Setup	1. Automatic measurement of workpiece surfaces, edges, and tool offsets; highly accurate.
ii……….	ii. Tool Setter	2. Automates tool length and diameter measurement inside the CNC machine for quick setup.
iii……..	iii. Probe Systems	3. Non-contact, high-precision tool measurement using cameras or lasers outside the machine
iv…….	iv. Edge Finders	4. Manual tool for finding workpiece edges; cost-effective but less accurate.
v……….	v. optical Tool	5. Operator manually adjusts tool offsets; simple

	Measurement Systems	but time-consuming, prone to human error.
		6. A tool setter automatically measures tool length and sometimes diameter, sending data to the CNC controller.

3. Read the following statements and choose the correct answer by encircling the correct option:

i. Which method of transferring data to a CNC machine involves entering G-codes and M-codes manually?

a) USB Cable
b) Manual Data Input (MDI)
c) Network Transfer
d) External Storage Devices

ii. Which of the following is a widely available and portable method for transferring files to a CNC machine?

a) Wireless File Transfer
b) Data Transfer Cables
c) USB Cable
d) Network Transfer

iii. Which method of file transfer does not require a physical connection and is ideal for large workshops?

a) USB Cable
b) Wireless File Transfer
c) Manual Data Input (MDI)
d) External Storage Devices

iv. In large production environments, which method allows CNC machines to be connected to a central server for program management?

a) Network Transfer
b) External Storage Devices
c) Data Transfer Cables
d) USB Cable

v. Which method involves using devices like SD cards or external hard drives for file transfer?

a) Network Transfer
b) USB Cable
c) External Storage Devices
d) Data Transfer Cables

vi. What is an older but reliable method of file transfer used with machines that do not support USB or wireless capabilities?

a) Wireless File Transfer
b) Data Transfer Cables
c) Manual Data Input (MDI)
d) External Storage Devices

Practical assessment

The school is facing the issue of a broken lather circular shaft having the following dimensions of 150 mm in length and 50 mm in diameter. The workshop assistant decides to produce a new one on a CNC machine. You are requested to perform the following tasks:

1. Set tools in CNC machine.
2. Transfer the design data to the CNC machine to initiate the machining process.
3. Set the CNC machine parameters accordingly.
4. Verify the program to ensure it is error-free before beginning production.

References

Fanuc CNC Custom Macros, Practical Resources for Fanuc Custom Macro B Users ISBN (0-8311-).

Kumar, H., Abbas, M., Mohammad, A., & Jafri, H. Z. (2013). Optimization of cutting parameters in CNC Turning. Optimization, 3(3), 331-334.

Madison, J. (1996). CNC machining handbook: basic theory, production data, and machining procedures. Industrial Press Inc.

Smid, P. (2010). CNC control setup for milling and turning mastering CNC control systems. Industrial Press

Learning Outcome 3: Execute the Work

Indicative contents
3.1 Mounting the work piece in CNC machine **3.2 Producing pieces on CNC machine** **3.3 Performing post production activities**

Key Competencies for Learning Outcome 3: Execute the work

Knowledge	Skills	Attitudes
• Explanation of clamping methods of work-piece in CNC machine • Identification of Surface finishing techniques • Identification of cleaning methods • Identification of Protective techniques	• Mounting the workpiece in CNC machine • Producing pieces on CNC machine • Applying Surface finishing techniques for workpiece • Applying cleaning methods • Applying protective techniques on workpiece produced by CNC machine • Storing the product	• Being careful while mounting work piece in CNC machine • Having Hygiene Awareness in manufacturing workshop cleaning. • Safety Consciousness while applying protective and surface finish techniques on a produced workpiece

Duration: 30 hrs

Learning outcome 3 objectives:

By the end of the learning outcome, the trainees will be able to:

1. Explain properly the methods of clamping the work-piece used in CNC machine during workpiece production
2. Mount properly the workpiece to be produced in CNC machine
3. Produce correctly the mechanical pieces by using CNC machine.
4. Identify properly the techniques for finishing the Surface of the workpiece produced by the CNC machine.
5. Apply correctly the techniques for finishing the Surface for the workpiece produced by the CNC machine
6. Apply effectively the cleaning methods required for CNC machine work
7. Apply correctly protective techniques on the work-piece produced by CNC machine
8. Store properly the product produced on CNC machine

Resources

Equipment	Tools	Materials
• CNC Milling	• Milling vices	• Aluminium materials
• CNC Turning	• Hydraulic vices	• Titanium materials
• CNC Grinding	• Quick-change vices	• Ceramics materials
• CNC Drilling	• Clamps studs	• Coolant or Lubricant
• CNC router	• Step blocks	• Paints
• CNC laser cutter	• Masking tape	• Varnishes
• air compressor	• Wire brushes	• Bolts and nuts
• Spray guns	• Hummers	• Sandpaper
• PPEs	• Wrenches	• Abrasive wheels
	• Cutting tool set	• wax
	• Calipers	

	• Micrometres	• Epoxy • Silicone • Steels materials • Carbide

Indicative content 3.1: Mounting the Work Piece in CNC Machine

Duration: 10 hrs

Theoretical activity 3.1.1: Explanation of clamping methods of Workpiece in CNC machine

Tasks:

1: Answer the following questions:

i. What do you understand by the following:
   a) Positioning accuracy
   b) Positioning mark

ii. What methods should be used to clamp the workpiece in a CNC machine during production?

2: Provide the answers to the asked questions by writing them on paper or flip chart

3: Present the answers to the whole class.

4: For more clarification, read the key readings 3.1.1

Key readings 3.1.1.: Explanation of clamping methods of workpiece in CNC machine

- **Ways of positioning work-piece in CNC Machine**

✓ **Positioning accuracy**

Positioning accuracy refers to the ability of a CNC machine to move the tool or the workpiece to a specific location with high precision. It is a measure of how closely the actual position of the machine matches the programmed or intended position. This accuracy is crucial for ensuring that the machined part meets the required specifications, particularly for operations where tight tolerances are required. High positioning accuracy ensures that each cut, hole, or feature is made in the correct location on the workpiece, reducing errors and improving the overall quality of the part.

✓ **Positioning mark**

A positioning mark is a visual or physical reference point on a workpiece or fixture that helps align the part in the correct position during setup. These marks ensure repeatability and reduce setup time, especially when machining multiple identical parts. Positioning marks help operators align the workpiece to the machine's coordinate system quickly and accurately, ensuring that the machining operations are consistent for each workpiece

- **Methods of clamping work piece in CNC MACHINE**

✓ **Vice clamping**

Vice clamping involves holding the workpiece securely between the jaws of a vice. The vice is typically mounted on the milling or CNC machine table. It is a simple and effective method for clamping smaller or regularly shaped parts, offering strong clamping force. Vice clamping is ideal for smaller workpieces or parts that have flat surfaces and require multiple cuts or drilling operations.

✓ **Fixture Plates**

Fixture plates are flat metal plates with a grid of pre-drilled holes that allow the workpiece to be clamped using bolts, fasteners, or custom-made fixtures. They provide flexibility in positioning and are often used for securing irregularly shaped parts or multiple parts simultaneously. Fixture plates are used for complex machining tasks where precise positioning and repeatability are required, such as in batch production.

✓ **Clamping Kits**

Clamping kits are sets of T-nuts, studs, clamps, and step blocks used to securely hold a workpiece on the machine table. These kits allow for adjustable clamping forces and positioning, making them versatile for various types of workpieces and operations. Clamping kits are used in CNC milling and other machining operations that require adjustable setups for different workpieces.

- ✓ **Toe Clamps**

Toe clamps grip the workpiece from the sides, allowing the top surface to remain unobstructed for machining. They work by applying downward pressure while pulling the workpiece towards the table, ensuring secure positioning. Toe clamps are especially useful for holding irregularly shaped workpieces or when the top surface of the part needs to remain free for machining operations.

- ✓ **Vacuum Chuck Systems**

Vacuum chuck systems use suction to hold flat or thin workpieces in place by creating a vacuum between the chuck and the part. This method applies even clamping pressure without physically clamping the workpiece, preventing deformation. Vacuum chucks are ideal for holding delicate, thin, or flat materials that cannot tolerate the force of mechanical clamping methods, such as when machining large sheets or panels.

- ✓ **Magnetic Chucks**

Magnetic chucks use electromagnetic force to securely hold ferrous (magnetic) workpieces in place. This method provides quick setup, with no need for mechanical clamps around the workpiece, leaving the entire surface accessible for machining. Magnetic chucks are commonly used in milling and surface grinding operations where rapid setup and full access to the workpiece surface are required. They are particularly effective for holding steel and other magnetic materials.

Practical Activity 3.1.2: Mounting the work piece in CNC machine

Task:

Go to manufacturing workshop and mount a workpiece in CNC machine

1: Apply safety precautions

2: Mount the work piece according to the trainer's demonstration.

3: Present your work to the trainer

4: Read the key reading 3.1.2 and ask for clarification where necessary.

5: Perform the task provided in the application of learning 3.1

Key readings 3.1.2: Mounting the work piece in CNC machine

- **Steps of mounting the work piece in CNC machine**

1. **Positioning the Workpiece**

Before any machining can take place, the workpiece must be correctly positioned on the machine table or fixture. This step is crucial as it determines how accurately the machining will occur. The positioning may involve aligning the workpiece with specific reference surfaces or scribe lines.

2. **Clamping the Workpiece**

Once positioned, the workpiece needs to be securely clamped to prevent any movement during machining. This clamping is essential to counteract forces such as cutting force, gravity, and inertial forces that could cause deviation from the intended position.

3. **Selecting Appropriate Fixtures**

Depending on the type of CNC machine and workpiece geometry, appropriate fixtures should be selected. Common types include three-jaw chucks for cylindrical parts, four-jaw chucks for irregular shapes, and specialized fixtures for specific tasks.

4. **Ensuring Alignment**

For certain setups, especially when using four-jaw chucks or other adjustable fixtures, additional alignment may be necessary. This involves checking that all surfaces are aligned correctly according to specified tolerances before final clamping

5. **Tightening Clamps.**

After ensuring proper alignment, tighten the clamps or fastening mechanisms securely. Various clamping mechanisms can be used, including screw pressing plates and hydraulic clamps, depending on the requirements of the machining process.

6. **Verifying Stability**

Once clamped, it is important to verify that the workpiece is stable and secure in its position. This can involve a visual inspection and a manual check to ensure there is no movement when applying light pressure.

7. **Final Checks Before Machining**

Before starting any machining operations, conduct a final check of all settings and positions to confirm that everything is set up correctly and safely for operation

Points to Remember

- Always use the CNC machine's coordinate system and alignment tools such as dial indicators or probing systems to accurately position the workpiece. This ensures precise alignment with the programmed toolpaths, preventing dimensional errors in the finished part.

- Apply visual or physical positioning marks during setup, especially for repeat jobs. This reduces setup time and ensures repeatability, particularly when you have multiple identical parts to the machine.
- For smaller or uniformly shaped parts, use vice clamping. It's a reliable, quick, and cost-effective method for holding parts securely during machining. However, ensure that the jaws of the vice are aligned with the machine’s coordinate system.
- If you're working on irregularly shaped workpieces or need to machine multiple parts at once, use fixture plates. They provide flexibility and can secure parts with bolts, allowing for customization to different workpiece geometries.
- The best clamping method depends on the geometry of the workpiece, material type, and the specific machining operation. For high accuracy, always combine precise positioning with appropriate clamping to ensure stability and reduce setup time.
- Verify that there is adequate clearance between the cutting tool and the workpiece. This prevents any accidental contact that could cause tool breakage or damage to the workpiece.
- Confirm that the workpiece is stable and does not wobble or shift when pressure is applied. Any instability can affect machining precision and surface finish.
- Use clamping kits when flexibility and adjustable forces are required. These kits allow for easy setup changes and are ideal when you need to reposition the workpiece during machining.
- Ensure that all safety measures are in place, including proper guarding around moving parts and ensuring that operators are at a safe distance from the machine during operation.
- Always refer to any setup sheets or documentation specific to the job being performed for additional details on mounting requirements or special considerations.

Application of learning 3.1.

Go to a manufacturing workshop where a car circular shaft with dimensions of 150 mm in length and 50 mm in diameter has been designed.

Your task is to:

✓ Mount the shaft on the CNC machine for machining operations

Indicative content 3.2: Producing Pieces on CNC Machine

Duration: 12 hrs

Practical Activity 3.2.1: Producing pieces on CNC machine

Task:

Go to a manufacturing workshop and produce pieces by using CNC machine.

1: Apply safety precautions

2: Prepare a mechanical piece to be mounted

3: Produce piece using CNC machine based on the trainer's demonstration

4: Present your product to the trainer

5: Read the key reading 3.1.2 and ask for clarification where necessary.

6: Perform the task provided in the application of learning 3.2

Key readings 3.2.1: Producing pieces on CNC machine

- **Steps of Producing pieces on CNC machine**

Step 1: Start CNC machining process

✓ **Prepare the machine**

- Ensure the workpiece is mounted securely and all tools are installed correctly.
- Load the required G-code into the machine controller.
- Set the tool offset and workpiece coordinate system (WCS) to define the workpiece origin (zero point).

✓ **Start the program**

- Press the **G-code Start Button** to initiate the CNC machining process.
- Ensure that starting commands (spindle speed, coolant flow, feed rate, etc.) are correctly initialized.

✓ **Monitor initial movements:**

- Closely watch the machine's initial movements to ensure it follows the correct toolpath and no unexpected behaviors occur.

- Be ready to pause or stop the machine in case of errors.

Step 2: Inspect the Work

- ✓ **Monitoring the work:**

While the CNC machine is operating, continuously monitor the machining process to ensure it follows the programmed toolpaths and settings. This involves checking for any deviations, unusual noises, or machine errors that could affect the quality of the piece being produced. It also helps prevent tool wear or breakage that could damage the workpiece.

- ✓ **Inspect features for integrity**

Periodically pause the machine and inspect the workpiece for key features such as dimensions, surface finish, and geometry. Use measuring tools (e.g., callipers, micrometers) to verify that the part matches the design specifications. Check for any signs of defects like uneven cuts, tool marks, or misalignment that could compromise the integrity of the final piece.

- ✓ **Final quality sign-off**

Once the machining process is complete, conduct a thorough inspection of the finished part. This involves a final check of all critical dimensions, tolerances, and surface finishes. If the part meets all quality standards and design requirements, sign off on the inspection, approving the part for further use or assembly. If any defects are found, the part may require rework or rejection depending on the severity.

Step 3: Control the work done

- ✓ **Quality Control**

Quality control is the process of verifying that the produced piece meets the required standards and specifications. It involves inspecting the final product to ensure it matches the design dimensions, tolerances, and surface finishes. Any deviation from the required quality must be addressed through adjustments in the machining process.

- ✓ **Tool Wear**

Over time, the cutting tools used in CNC machines (e.g., end mills, drills) experience wear due to friction and heat during machining. Tool wear can negatively affect the accuracy and finish of the produced part. It's important to regularly inspect the tools and replace them when necessary to maintain optimal performance.

✓ **Tool Breakage**

Tool breakage occurs when the cutting tool fails during machining, which can cause damage to the workpiece and disrupt the process. Monitoring for signs of excessive wear or unusual vibrations can help prevent breakage. When a tool breaks, it must be replaced immediately, and the machine may need to be re-calibrated to ensure the new tool matches the previous setup.

✓ **Deviations from the expected dimensions**

Deviations occur when the final part does not match the design specifications, often due to issues such as incorrect tool offsets, tool wear, machine calibration errors, or material inconsistencies. Regular measurement of the part during and after machining is essential to catch and correct these deviations early.

Points to Remember

- Perform Double-check the design dimensions, tolerances, and material specifications before starting the machining process.
- Ensure the proper cutting tools are selected for the material and type of operation (e.g., drilling, milling, turning)
- Confirm the CNC machine is calibrated correctly, with accurate zero points and offsets set for the workpiece and tools.
- Apply Proper position and secure the workpiece (e.g., shaft, block) in the chuck or vice to prevent movement during machining
- Keep an eye on tool performance, cutting accuracy, and the machine's behaviour to spot any issues (e.g., tool wear, vibration)
- Regularly measure the part during machining to detect any deviations from the expected dimensions early and make corrections.
- Always wear proper personal protective equipment (PPE), such as safety glasses and gloves, and follow all safety protocols around the CNC machine.

Application of learning 3.2.

Go to a manufacturing workshop where a car circular shaft with dimensions of 150 mm in length and 50 mm in diameter has been designed. Your task is to produce the circular shaft piece by using CNC machine

Indicative content 3.3: Performing Post Production Activities

Duration: 8 hrs

Practical Activity 3.3.1: Applying surface finishing techniques for workpiece

Task:

Go to a manufacturing workshop and apply different Surface finishing techniques for the produced piece.

1: Apply safety precautions

2: Prepare finishing products/materials, tools and equipment to be used

3: Apply the finishing materials/products to the produced piece according to the trainer demonstration.

4: Present your work to the trainer

5: Read the key reading 3.3.1 and ask for clarification where necessary.

6: Perform the task provided in the application of learning 3.3

Key readings 3.3.1: Applying surface finishing techniques for workpiece

- **Techniques of applying finishing materials to produced pieces**

✓ **Grinding**

Grinding is used to remove excess material, refine dimensions, and improve surface smoothness. It is particularly useful for achieving tight tolerances and smooth finishes on CNC-machined parts. A rotating abrasive wheel is used to grind the surface of the piece, removing material in fine layers. This process is typically used for metals and hard materials. The result is a smooth, precise surface, often with tight tolerances and a fine finish.

✓ **Polishing**

Polishing enhances the surface finish of CNC-machined parts by removing fine scratches and improving smoothness, giving the part a glossy, reflective appearance. Is used to produce high-gloss finishes on CNC parts such as stainless-steel components, decorative items, or aerospace parts requiring high visual and functional quality. Polishing produces a shiny, mirror-like finish, removing minor scratches and surface imperfections.

- ✓ **Buffing**

  Buffing is a higher-level polishing technique aimed at achieving an ultra-smooth, glossy finish, often used in decorative applications. A soft cloth wheel is used with a polishing compound to buff the surface, creating a highly polished, lustrous finish. Buffing is gentler than grinding and polishing.

- ✓ **Honing**

  Honing is used to create a precise, smooth finish inside cylindrical bores or holes, improving surface texture and dimensional accuracy. Honing produces a smooth, consistent surface with high dimensional accuracy and reduced surface roughness, especially on bores and cylindrical surfaces.

- **Perform cleaning methods**

  After machining and surface finishing, it's important to clean the workpiece to remove metal chips, dust, oil, and other residues that could affect the quality of the part.

- ✓ **Cleaning with compressor:**

- Use an air compressor to blow away metal chips, dust, and fine particles from the workpiece and machine.
- Use a compressed air system to blow air over the machine's surface, including around the tool holder, workpiece area, and internal components like guide rails and spindles.
- Focus on areas where metal shavings and dust accumulate, such as the machine bed, tool changer, and chip conveyor.
- Quick and effective for cleaning tight spaces that cannot be easily accessed by hand.
- Removes debris without needing to disassemble machine parts.

- ✓ **Cleaning with cloth lugs**

  This technique is used manually wipe down machine surfaces, ensuring thorough cleaning of more accessible parts.

  Process:

- Use clean cloth lugs or rags to wipe the machine's external surfaces, worktable, and tool holders.

- Apply a mild solvent or machine-safe cleaner to remove oil, coolant residue, and stubborn dirt.
- Clean exposed mechanical parts, like tool holders or fixtures, to prevent buildup of oil or metal shaving.

- **Apply protective techniques**

✓ **Rust prevention:**

To protect metal parts from rusting due to exposure to moisture and air during and after the machining process. Rust can be highly damaging to metal components, especially those made of iron or steel. Rust prevention techniques help avoid corrosion caused by moisture and oxygen exposure.

- Apply rust inhibitors or coatings to protect the machined part, especially if it's made of ferrous materials (e.g., mild steel).
- Use rust prevention oils, anti-rust sprays, or protective waxes to coat the surface.

✓ **Waterproofing:**

Waterproofing techniques protect both CNC machines and parts from water or liquid damage, particularly in environments where coolant fluids or water-based machining processes are used

- If the workpiece will be exposed to moisture or water in its application, apply waterproofing coatings or sealants.
- Use waterproof sprays, corrosion-resistant paints, or coatings such as polyurethane.
- Helps protect the part from water damage and ensures longevity, especially in outdoor or humid environments.
- Parts can be coated with waterproof materials like silicone sprays or specialized polymers that create a water-repellent barrier.

- **Store the product**

Once the workpiece has been machined, finished, cleaned, and protected, proper storage ensures that it remains in good condition until it is needed.

✓ **Storage method**

- Store the workpiece in a dry, clean environment to prevent exposure to moisture, dust, or contaminants.
- Use protective wrapping (e.g., plastic wrap, anti-rust paper, or bags) and place the part on shelves, bins, or storage racks that are off the ground to avoid moisture.
- Store similar parts together and label them for easy identification. Ensure there is no contact with other parts that could cause scratching or deformation.

Points to Remember

- Apply surface finishing techniques such as sanding, polishing, or grinding to achieve the desired texture or appearance
- Choose abrasives and compounds that are suitable for the material being finished, as certain abrasives work better on specific metals or plastics.
- Remove machining residues such as coolant, lubricants, and metal chips
- Monitor the surface roughness and dimensional accuracy throughout the finishing process to ensure that tight tolerances are maintained.
- Different techniques will yield different levels of gloss or reflectivity; select the finishing method based on the final appearance or function of the part.
- Once the surface finishing is complete, clean the part thoroughly to remove any polishing or grinding residue.

Application of learning 3.3.

Go to a manufacturing workshop where a milling machine circular shaft with dimensions of 150 mm in length and 50 mm in diameter has been produced using CNC machine. You are asked to perform the following tasks:

1. Apply finishing techniques to the produced circular shaft
2. Store the produced parts
3. Clean the workshop by using air compressor

Learning outcome 3 end assessment

Written assessment

1. Read carefully the following statement and answer by **TRUE** if the statement **is correct or FALSE** if the statement is **wrong.**

a) Proper mounting of a work-piece is critical for achieving precision, consistency, and safety in CNC machining.

b) In the mounting process, positioning accuracy refers to how well the work-piece is aligned with the CNC machine's coordinate system.

c) A vice clamping method involves using a suction system to hold the work-piece in place.

d) Fixture plates are equipped with T-slots or mounting holes to attach various fixtures and clamping devices, providing a versatile mounting surface.

e) Magnetic chucks are suitable for holding non-ferrous work-pieces due to their magnetic force**.**

f) Tool breakage can be caused by excessive cutting forces or speeds that exceed the tool's designed capacity.

g) Dimensional deviations in a machined part are only caused by errors in the CNC program.

h) Regular monitoring and tracking of tool wear can help prevent unexpected tool failures.

i) Polishing is generally used to remove scratches and blemishes while enhancing the surface shine.

j) Honing is primarily used to improve the surface finish of external and internal cylindrical surfaces.

k) Using cloth rags is suitable for removing large particles and debris from the surface of a part.

l) Proper storage conditions include protecting the product from extreme temperatures, humidity, and corrosive elements.

2. Read and match the techniques of apply Surface finishing in **column C**, with their description in **column B**, and provide answer in **Column A**

Column A: Answer	**Column B:** Description	**Column C:** techniques of apply Surface finishing
i…….	1. Further refining and shining the surface using a soft wheel	I. Grinding
ii……..	2. Precisely removing material to improve	ii. Polishing

	surface accuracy	
iii……….	3. Using a fine abrasive to produce a shiny surface	iii. Buffing
iv……	4. Manual tool for finding workpiece edges; cost-effective but less accurate.	iv. Honing
	5. Smoothing the surface using an abrasive wheel	

1. Read the following statements and choose the letter corresponding to the right answer by encircling:

i. Which of the following is NOT a step in the preparation phase for producing a piece on a CNC machine?
   a) Installing the appropriate cutting tools
   b) Verifying the G-code program
   c) Securing the work-piece onto the machine
   d) Performing a safety check

ii. What is the purpose of setting the machine's coordinate system to zero during the setup phase?
   a) To ensure that the machine knows where to start machining relative to the work-piece
   b) To install the cutting tools correctly
   c) To check the alignment of the machine
   d) To load the G-code program into the machine's controller

iii. Which type of wear is caused by the friction between the cutting tool and the work-piece material?
   a) Adhesive Wear
   b) Chemical Wear
   c) Abrasion Wear
   d) Thermal Wear

iv. Which of the following is the primary purpose of grinding in surface finishing?
   a) To achieve a high-gloss finish on metals
   b) To remove scratches and blemishes
   c) To remove material and create a smooth surface with precise dimensions
   d) To create a reflective surface using fine abrasives

v. Which cleaning method is most appropriate for removing large particles and debris from a part's surface?
   a) Cleaning with cloth rags
   b) Cleaning with a compressor
   c) Polishing
   d) Buffing

vi. What is the primary function of rust prevention techniques?
   a) To improve the surface texture and finish
   b) To remove contaminants from the surface
   c) To prevent the formation of rust or corrosion on metal parts
   d) To enhance the appearance and remove blemishes

vii. Which method is used to make parts resistant to water damage?
   a) Passivation
   b) Using a polishing wheel
   c) Applying waterproof sealants
   d) Surface grinding

Practical assessment

Your school is facing the issue of a car circular shaft with dimensions of 150 mm in length and 50 mm in diameter. The workshop assistant has decided to design a replacement shaft. Ask trainees to:

1. Mount the work piece in the CNC machine
2. Produce circular shaft/piece
3. Perform the finishing work

References

Fanuc CNC Custom Macros, Practical Resources for Fanuc Custom Macro B Users ISBN (0-8311-)

Kumar, H., Abbas, M., Mohammad, A., & Jafri, H. Z. (2013). Optimization of cutting parameters in CNC Turning. Optimization, 3(3), 331-334.

Madison, J. (1996). CNC machining handbook: basic theory, production data, and machining

Negi, P., Ram, M., & Yadav, O. P. (2022). Basics of CNC Programming. CRC Press. procedures.

Smid, P. (2003). CNC programming handbook: a comprehensive guide to practical CNC programming.

Smid, P. (2010). CNC control setup for milling and turning mastering CNC control systems. Industrial

Smid, P. (2010). CNC control setup for milling and turning mastering CNC control systems..

Rattat, C. (2017). CNC Milling for Makers: Basics-Techniques-Applications

W. Bolton, "Programmable Logic Controllers," 2006. Published by Elsevier Newness'

.

RTB | RWANDA TVET BOARD

October 2024