MODULE - 3

COMPUTER NUMERICAL CONTROL

Introduction to CNC

CNC, short for Computer Numerical Control, refers to a manufacturing method that utilizes computerized systems to manage machinery and automated devices. This innovation has transformed several sectors, including manufacturing, engineering, and woodworking, by enhancing precision, efficiency, and the level of automation in production.

In a CNC setup, the operations and movements of machine tools are directed by a computer program. Typically composed in languages like G-code or proprietary codes from manufacturers, this program delivers comprehensive directions that detail the tool's trajectory, speed, and various parameters essential for producing the intended product or component.

Advantages of CNC Machines:

- Greater flexibility
- Enhanced productivity
- Uniform quality
- Decreased scrap rate
- Dependable operation
- Lower non-productive time
- Less manpower required
- Shorter cycle duration
- High precision
- Reduced lead time
- Just-in-time (JIT) manufacturing
- Automated material handling

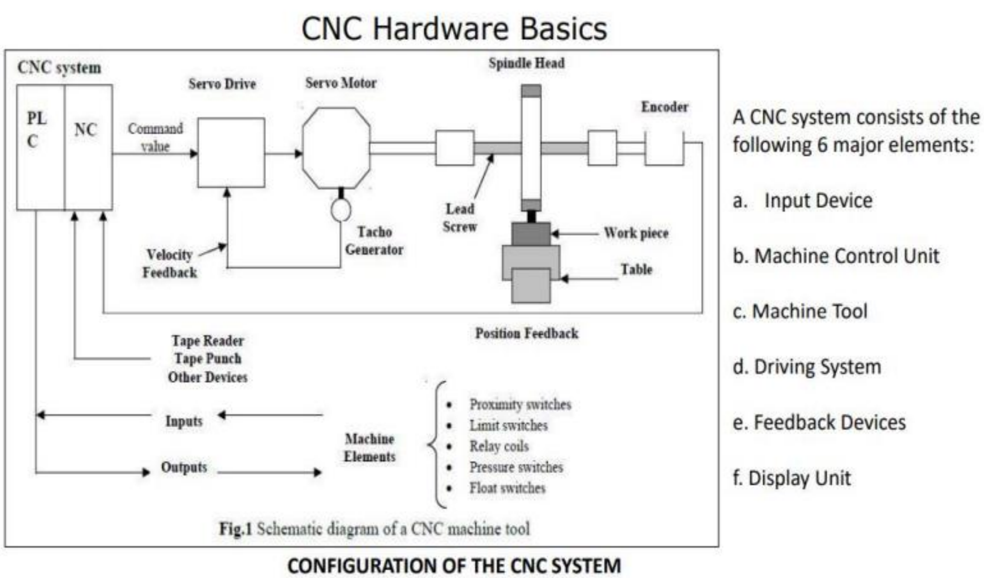
CAD Hardware

1. CPU (Central Processing Unit)
2. GPU (Graphics Processing Unit)
3. RAM (Random Access Memory)
4. Storage
5. Motherboard
6. Power Supply

This section covers the fundamental components of CNC hardware, including the configuration of CNC machine tools, actuation mechanisms, and feedback systems.

1.Structure of CNC Machine Tools

The structure serves as the mechanical foundation of a CNC machine, affecting its stability, rigidity, and accuracy.



- **Base and Frame:**
  - Acts as the machine's foundation.
  - Supports various components and stability.

- **Work Table:**
    - Holds the workpiece in position during machining.
    - Offers clamping systems to secure the workpiece effectively.
- **Spindle:**
    - Spins at high velocities while securing the cutting tool.
    - Applies forces to cut through the material.
- **Tool Changer (optional):**
    - Automatically swaps tools during machining to enhance efficiency.

2.Actuation Systems:

- Control the machine's axis movements, ensuring tools and workpieces are accurately positioned.

- **Motors:**
    - Stepper Motors: Provide precise control in incremental steps.
    - Servo Motors: Ensure high accuracy through closed-loop feedback for immediate adjustments.
- **Lead Screws and Ball Screws:**
    - Transform the rotary motion from motors into linear movement.
    - Ball screws reduce friction, resulting in smooth and accurate motion.
- **Linear Guides and Rails:**
    - Allow for smooth, accurate, and repeatable movements along the machine's axes.
- **Feedback Devices:**
    - Deliver real-time information about position, speed, and other essential parameters to maintain precision.
- **Encoders:**
    - Rotary Encoders: Measure the rotational movements of motors or spindles.

    - Linear Encoders: Track the linear movements of machine parts.
- **Limit Switches:**
    - Establish the operational boundaries of the machine.
    - Prevent excessive travel to protect both the machine and the workpiece.
- **Probes and Sensors:**
    - Identify the position of tools or workpieces.
    - Assist in zeroing and calibration tasks.
- **Auxiliary Components:**
    - Enhance the functionality and usability of the machine.
- **Coolant Systems:**
    - Provide lubrication to minimize tool wear.
    - Prevent overheating of tools and workpieces.
- **Chip Management Systems:**
    - Eliminate machining debris.
    - Ensure cleanliness and safety during operation.
- **Controllers:**
    - Decode G-code instructions and translate them into commands for motors and actuators.
- **Power Supply Units:**
    - Deliver the essential energy required for the machine's operation.

CNC TOOLING

CNC tooling encompasses the tools, accessories, and components utilized in CNC (Computer Numerical Control) machining to precisely cut, shape, and finish materials.

Automatic tool changers and workholding systems.

Automatic tool changers (ATCs) and workholding systems are crucial in modern CNC machining, boosting productivity, precision, and efficiency by automating tool changes and securely holding workpieces during operations.

Automatic Tool Changers (ATCs)

Definition: An automatic tool changer is a mechanism that automatically swaps cutting tools on a CNC machine, enabling the use of multiple tools in a single machining cycle without manual input.

Types of ATCs:

1. Carousel ATCs: Tools are organized in a circular carousel, allowing the machine to select the required tool as needed.
2. Linear ATCs: Tools are aligned linearly, and the machine moves to the specific tool position for exchange.
3. Robot Arm ATCs: A robotic arm fetches tools from storage and places them onto the spindle.

Benefits:

- Increased Efficiency: Minimizes downtime by automating tool changes.
- Versatility: Supports various machining operations without manual input.
- Improved Precision: Reduces human error in tool handling and setup.

Applications:

- CNC milling machines
- CNC lathes
- Robotic machining systems

Workholding Systems

1. Definition: Workholding systems are devices designed to firmly secure and stabilize a workpiece during machining, ensuring accuracy and consistency in operations.

Types of Workholding Systems:

2. Vises: Commonly used in milling machines to securely clamp the workpiece.
3. Chucks: Found in lathes, gripping the workpiece either externally or internally.
4. Fixtures: Custom-made devices for holding specific parts, ideal for repetitive tasks.
5. Magnetic Chucks: Use magnetic force to stabilize ferromagnetic workpieces.
6. Vacuum Fixtures: Employ suction to secure non-ferrous or complex-shaped materials..

Benefits:

- Stability:Keeps the workpiece securely positioned during machining.
- Repeatability:Ensures consistent results across parts or production cycles.
- Flexibility: Allows adaptation for various workpieces and machining tasks.

Applications:

- CNC machining centers
- Milling and turning operations
- Assembly lines

Integration of ATCs and Workholding Systems

The integration of automatic tool changers and workholding systems is essential for optimizing machining processes. Together, they enable:

- Unattended Operations: Machines can operate for longer periods without manual intervention, boosting productivity.
- Complex Machining: The ability to change tools and secure different workpieces allows for intricate machining tasks within a single setup.
- Reduced Setup Time: Fast tool changes and efficient workholding minimize the time needed for setup between jobs.

CNC programming

CNC programming entails creating instructions that control CNC machines like mills, lathes, and routers. These instructions specify the machine's movements, tool speeds, and the operations to be carried out on the workpiece.

Significance of CNC Programming

- Accuracy: CNC programming enables high degrees of precision and consistency in machining tasks.
- Productivity: Automation streamlines processes, minimizing both manufacturing time and labor expenses.
- Complexity: CNC machines can execute intricate designs and functions that would be challenging or unfeasible to accomplish manually.

CAD/CAM Software:

CAD (Computer-Aided Design) and CAM (Computer-Aided Manufacturing) software play a crucial role in CNC programming. CAD software is utilized to generate a digital representation of the component, whereas CAM software produces the tool paths and G-code commands derived from the CAD model.

Geometry Development:

The initial phase in CNC programming involves specifying the geometry of the part through the creation of a 2D or 3D model using CAD software. This model defines the part's dimensions, characteristics, and tolerances.

Toolpath Generation:

After creating the part geometry, CAM software generates the tool paths, which guide the cutting tool’s movements. It considers factors like cutting strategy, tool selection, feed rates, cutting depths, and tool changes.

Post-Processing:

Once the tool paths are generated, CAM software converts them into machine-specific G-code instructions through post-processing. This process accounts for the CNC machine's capabilities, such as axis count, spindle speed, and tool change procedures.

G-Code Programming:

G-code serves as the primary programming language for CNC machines. It consists of instructions that govern the machine's movements, initiate or halt operations, adjust speed, manage tool changes, control spindle rotation, regulate coolant, and perform various other functions.

Simulation and Verification:

Prior to running the CNC program on the machine, it's typical to utilize specialized software for simulation and verification. This process aids in spotting possible collisions, mistakes, or discrepancies in tool paths or G-code, thereby ensuring safe and precise machining.

Machine Setup:

Once the program has been verified, it is then loaded into the machine's control system. The machine is prepared with the necessary cutting tools, fixtures, and workpieces, after which the CNC program is executed to carry out the machining tasks.

<u>Glossary</u> :

Block	A single line of G Code
Canned Cycle	Complex cycle defined by a single block of code
Dwell	Program pause with a duration defined by "P" in seconds
EOB	End of block. Required at the end of every block of G Code. In Mach4 this is a carriage return
Feed rate	Velocity, set by F, at which an axis will move
Group	Collection of G codes that control the same function or mode, i.e. G90 and G91 positioning modes
Modal	Active until a code from the same group is called
Normal	A line perpendicular to a plane, pointing in the positive direction.
Origin	Point in a coordinate system where X, Y and Z are zero
RPM	Revolutions per minute
UPM	Units per minute (inches, millimeters, degrees, etc)
Word	A single word of G Code is a letter followed by a number. G01, X1.0, etc. are words

G	Preparatory function, G followed by a numerical code, specifies machining modes and functions
M	Miscellaneous function, M followed by a numerical code, defines program flow and can control auxiliary functions such as coolant. Can also perform machine specific functions and macros user or builder.
X, Y, Z, A, B, C	Movement commands followed by a numerical value, define the end point of a motion command
S	Spindle speed, followed by numerical value of desired rpm or surface speed
T	Tool call, followed by next tool number to be used
H	Tool height offset to be used, generally matches the tool number
D	Tool diameter offset to be used, generally matches the tool number
F	Followed by a numerical value to define the feed rate. The magnitude and value of which will be determined by the feed mode setting
P	Followed by a numerical value, specifies dwell time in seconds. (also used in other functions)
N	Sequence numbers. Used for program organization and go to commands

G–codes & M–codes

Mill G Codes

G00 Rapid Positioning Movement (X, Y, Z, A, B)

G01 Linear Interpolation Movement (X, Y, Z, A, B, F)

G02 Circular Interpolation Movement CW (X, Y, Z, A, I, J, K, R, F)

G03 Circular Interpolation Movement CCW (X, Y, Z, A, I, J, K, R, F)

G04 Dwell Time (P) (P=Seconds"." Milliseconds)

G09 Exact Stop, Non-Modal

G17* Selection of Circular Motion in the XY Plane (G02 or G03)

G18 Selection of Circular Motion in the ZX Plane (G02 or G03)

G19 Selection of Circular Motion in the YZ Plane (G02 or G03)

G20 Coordinate Positioning in Inches

G21 Coordinate Positioning in Metric

G28 Return to Machine Zero Via Reference Point (X, Y, Z, A, B)

G29 Navigate to Location Through

G28 Reference Point (X, Y, Z, A, B)

G40* Cancel Cutter Compensation G41/G42/G141 (X, Y)

G41 2D Cutter Compensation Left (X, Y, D)

G42 2D Cutter Compensation Right (X, Y, D)

G43 Tool Length Compensation + (H, Z)

G49* Cancel Tool Length Compensation G43/G44/G43

G52 Work Offset Positioning Coordinates

G53 Machine Positioning Coordinates, Non-Modal (X, Y, Z, A, B)

G54* Work Offset Positioning Coordinates #1 (Setting 56)

G55 Work Offset Positioning Coordinates #2

G56 Work Offset Positioning Coordinates #3

G57 Work Offset Positioning Coordinates #4

G58 Work Offset Positioning Coordinates #5

G59 Work Offset Positioning Coordinates #6

G73 High-Speed Peck Drilling Canned Cycle (X, Y, A, B, Z, I, J, K, Q, P, R, L, F)

G74 Reverse Tapping Canned Cycle (X, Y, A, B, Z, J, R, L, F)

G76 Fine Boring Canned Cycle (X, Y, A, B, Z, I, J, P, Q, R, L, F)

G77 Back Boring Canned Cycle (X, Y, A, B, Z, I, J, Q, R, L, F)

G80* Cancel Canned Cycle (Setting 56)

G81 Drilling Canned Cycle (X, Y, A, B, Z, R, L, F)

G82 Spot Drill / Counterbore Canned Cycle (X, Y, A, B, Z, P, R, L, F)

G83 Peck Drill Deep Hole Canned Cycle (X, Y, A, B, Z, I, J, K, Q, P, R, L, F)

G84 Tapping Canned Cycle (X, Y, A, B, Z, J, R, L, F)

G85 Bore In ~ Bore Out Canned Cycle (X, Y, A, B, Z, R, L, F)

G86 Bore In ~ Stop ~ Rapid Out Canned Cycle (X, Y, A, B, Z, R, L, F)

G87 Bore In ~ Manual Retract Canned Cycle (X, Y, A, B, Z, R, L, F)

G88 Bore In ~ Dwell ~ Manual Retract Canned Cycle (X, Y, A, B, Z, P, R, L, F)

G89 Bore In ~ Dwell ~ Bore Out Canned Cycle (X, Y, A, B, Z, P, R, L, F)

G90* Absolute Positioning Command

G91 Incremental Positioning Command

G92 Global Work Coordinate System

G93 Enable Inverse Time Feed Mode

G94 Disable Inverse Time Feed / Enable Feed Per Minute

G98 Return to Initial Point of Canned Cycle

G99 Return to R Plane of Canned Cycle

Mill M Codes

M00 Stop Program

M01 Optional Program Stop

M02 End of Program

M03 Turn Spindle On Clockwise (S)

M04 Turn Spindle On Counterclockwise (S)

M05 Stop SpindleM06 Tool Change (T)

M08 Activate Coolant

M09 Deactivate Coolant

M30 End Program and Reset

M31 Forward Chip Auger

M33 Stop Chip Auger

M34 Lower Coolant Spigot Position, Increment

M35 Raise Coolant Spigot Position, Decrement

M36 Pallet Part Ready

M41 Override Spindle to Low Gear

M42 Override Spindle to High Gear

M50 Execute Pallet Change

M83 Activate Auto Air Jet

M84 Deactivate Auto Air Jet

M88 Activate Coolant Through Spindle

M97 Call Local Sub-Program (P, L)

M98 Call Sub-Program (P, L)

M99 Return from Sub-Program / Routine Loop (P)

Fundamentals of part programming

Part programming in CNC involves creating a CNC program that includes instructions for machining a specific part or component. These programs, typically written in G-code, provide the necessary commands to ensure the CNC machine performs the machining operations with precision.

CANNED CYCLES

Canned cycles, also referred to as fixed or predefined cycles, are preprogrammed sequences of machining operations in CNC programming. These cycles simplify the process by automating repetitive tasks, eliminating the need to specify each movement individually. Canned cycles are especially useful for operations like drilling, tapping, boring, and other standard machining processes. Here are some of the most commonly used canned cycles in

CNC Programming Overview:

G82-Counterboring Cycle:
The G82 cycle is used when both drilling and counterboring are needed for a hole. It specifies the hole's position, the feed rate, drilling depth, counterbore size, and counterbore depth.

G83 - Peck Drilling Cycle:
The G83 cycle is designed for deep-hole drilling, allowing the drill bit to retract periodically to clear chips. It includes parameters like hole position, feed rate, drilling depth, and the peck distance (the amount the drill retracts after each cut).

G84 - Tapping Cycle:
The G84 cycle is utilized for threading operations, both internal and external. It outlines the location of the feature to be threaded, the feed rate, and details such as thread pitch, depth, and the direction of the thread (right-hand or left-hand).

G85 - Boring Cycle:
The G85 cycle is used for single-point boring, specifying the location, feed rate, boring tool size, desired bore size, and depth.

G88 - Boring Cycle with Dwell:
Similar to G85, the G88 cycle includes a dwell at the bottom of the bore to aid in chip removal and enhance surface finish.

G89 - Boring Cycle with Dwell and Feed-out:
The G89 cycle adds a controlled feed-out after reaching the bore depth, which helps remove tool marks and ensures a smoother exit from the bore.

Formatting Guidelines:

When writing G-code programs, certain rules and formatting guidelines should be followed.

The initial part of any program should include a safe startup block. This block ensures that some modes are disabled while others are set to default states.

An example of a safe startup block:

```
G00 G90 G17 G54 G40 G49 G80
```

This block sets the machine to rapid mode, uses absolute positioning in the XY plane with fixture offset 1, and cancels any active tool diameter and length offsets, as well as canned cycles.

- **G00** – Rapid mode
- **G90** – Absolute positioning mode
- **G17** – XY plane selection
- **G54** – Fixture offset 1
- **G40** – Cancel cutter compensation (tool diameter)
- **G49** – Cancel length offset
- **G80** – Cancel canned cycle

Example:
Program a rapid move to X1.0, Y3.0.

G0 G90 G54 G17 G40 G49 G80	Safe start line
T1 M6	Tool change
S2500 M3	Start spindle
G0 X1.0 Y3.0	Rapid to XY position
M30	Program end and rewind