CODE-534

Unit VIII: VMC G Code and M Code Programming



CLASS-1

Sparks Academy

81908 79379

Introduction to Vertical Machining Centre (VMC)

A Vertical Machining Centre (VMC) is a type of CNC (Computer Numerical Control) machine in which the spindle axis is oriented vertically. These machines are used primarily for precision machining of metal and other solid materials. VMCs are widely used in industries for milling, drilling, boring, tapping, and contouring operations.



Key Features of VMC:

1. **Vertical Spindle:**
   - The cutting tool is mounted vertically (pointing downwards).
   - Suitable for flat surface machining.
2. **CNC Control:**
   - Controlled by a computer program (G-code or M-code).
   - High precision and repeatability.

3.Automatic Tool Changer (ATC):

- Changes tools automatically during operation without manual intervention.
- Increases productivity.

4.Coolant System:

- Used to cool the cutting tool and workpiece during machining.
- Improves tool life and surface finish.

5.Enclosure:

- Fully enclosed to ensure safety and reduce noise and debris.

Basic Operations Performed in VMC:

Operation	Description
Facing	Machining flat surface perpendicular to spindle
Milling	Removing material using a rotating tool
Drilling	Creating round holes using drill bits
Tapping	Creating internal threads
Boring	Enlarging existing holes
Pocketing	Removing material from within a closed boundary
Contour milling	Following a complex path or profile

Advantages of VMC:

- High precision and accuracy
- Faster production with minimal manual intervention
- Good surface finish
- Suitable for batch production
- Integrated with CAD/CAM software

Applications of VMC:

- Automotive components manufacturing
- Aerospace parts machining
- Die and mold making
- Electronics components
- Tooling and fixture production

Concept of Axis & Coordinate System in VMC (Vertical Machining Centre)

Understanding the axis system and coordinate system is fundamental to operating a VMC effectively. CNC machines like VMC follow a Cartesian coordinate system (X, Y, Z) to define the position and movement of the tool relative to the workpiece.

1. Axis System in VMC:

Axis	Direction	Movement Description
X	Left to Right (Horizontal)	Movement of table or spindle left/right
Y	Front to Back (Horizontal)	Movement of table or spindle front/back
Z	Up and Down (Vertical)	Movement of the spindle/tool vertically

2. Additional Axes (in Advanced VMCs):

Some machines may have:

- **A-axis (Rotation around X)**
- **B-axis (Rotation around Y)**
- **C-axis (Rotation around Z)**

These are used in 4-axis or 5-axis VMCs for more complex machining.

3. Coordinate System in VMC:

VMCs use a Cartesian coordinate system to locate positions.

Machine Coordinate System (MCS):

- Set by the machine manufacturer.
- Origin (0,0,0) is fixed and usually at a corner of the machine table.
- Used internally by the CNC system.

Work Coordinate System (WCS):

- Set by the operator using a G-code command like G54 to G59.
- Origin is defined at a convenient point on the workpiece (usually top-left or center of the part).
- Allows programming relative to the part, not the machine.

1. Cutting Speed (Speed)

Definition:

Cutting speed is the speed at which the cutting tool engages the workpiece material. It is the surface speed of the rotating tool or workpiece.

Units:

- Meters per minute (m/min) – Metric
- Feet per minute (ft/min) – Imperial

Formula:

$$\text{Cutting Speed } (V) = \pi \times D \times N 1000 (\text{in m/min})$$

Where:

- V = Cutting speed
- D = Diameter of the tool/workpiece (mm)
- N = Spindle speed (RPM)

2. Spindle Speed (N)

Definition:

Spindle speed is the rotational speed of the cutting tool or workpiece, expressed in revolutions per minute (RPM).

Formula:

$$N=1000\times V/\ pi*D$$

Where:

- **V = Cutting speed (m/min)**
- **D = Diameter of tool/workpiece (mm)**
- **N = Spindle speed (RPM)**

3. Feed Rate (Feed)

Definition:

Feed is the distance that the tool advances during one revolution of the workpiece or per unit time.

Units:

- mm/rev (for turning)
- mm/min (for milling)

Formula (Milling Example):

$$\text{Feed Rate } (F) = f \times N \times z$$

Where:

- f = Feed per tooth (mm/tooth)
- N = Spindle speed (RPM)
- z = Number of teeth on the cutter
  - Increasing feed rate increases metal removal rate but may worsen surface finish and increase tool wear.

4. Depth of Cut (DoC)

Definition:

Depth of cut is the thickness of the material layer removed in one pass of the tool.

Units:

- Millimeters (mm)
- In Turning: Measured radially from the uncut surface to the cut surface.
- In Milling: Axial depth (along the Z-axis) or radial (step over in X or Y).
  - Larger depth of cut increases material removal rate but also increases cutting force and tool load.

Cutting Tools

Cutting tools are tools used to remove material from a workpiece through shear deformation. Examples include:

- Turning tools
- End mills
- Drills
- Taps
- Boring bars
- Inserts (indexable tools)

1. Identifying Cutting Tools

Each tool used in CNC or VMC operations has specific geometry, material, and usage. Identification ensures the correct tool is used for the right operation.

How Tools Are Identified

Parameter	Description
Tool Number (T-code)	Tool ID assigned in the CNC program (e.g., T01, T02, etc.)
Tool Offset Number	Links tool to its geometry and length offset (H01, H02)
Tool Type	Based on function: Drill, End Mill, Face Mill, Tap, Boring Bar, etc.
Tool Material	HSS, Carbide, Cermet, Ceramic, etc.
Tool Geometry	Tool diameter, length, flute type, insert shape
Tool Holder Type	BT, CAT, HSK, etc. – used to mount the tool in the spindle

2. Replacing Cutting Tools

When a tool wears out or breaks, it must be replaced promptly to avoid part rejection or machine damage.

Tool Replacement Process (Manual or Automatic):

Manual Replacement:

- Stop machine safely (Use M00 or Emergency Stop).
- Loosen tool holder from spindle using wrench or release button.
- Remove the old tool and insert the new tool of same type.
- Tighten the tool holder properly.
- Measure tool length offset again using tool setter or manually.
- Update tool offset data in the control panel (if needed).
- Run a trial air cut to confirm positioning.

Automatic Tool Changer (ATC):

- CNC program includes M06 tool change command (e.g., T03 M06).
- Machine automatically unloads current tool and loads new one.
- Tool magazine rotates to bring next tool into position.
- Tool offset is already stored in the control system

Best Practices in Tool Replacement:

- Use same tool geometry and material when replacing.
- Always check tool length offset after replacement.
- Replace tools before critical failure to avoid part rework.
- Label and store tools properly in tool inventory.
- Use presetting devices to save tool length data before production.

Summary:

Step	Action
Identify tool	By number, type, and condition
Inspect tool wear	Visual signs, quality checks
Replace tool	Manually or automatically (M06)
Set offset	Tool length offset (H code) using G43
Test operation	Use dry run or single block mode

1.VMC machines are mainly used for:

(a) Turning

(b) Welding

(c) Milling and Drilling

(d) Casting

2.The main advantage of a VMC over a conventional milling machine is:

(a) Lower cost

(b) Manual operation

(c) Automation and high precision

(d) Larger size

3.Which axis is typically used for vertical movement in a VMC?

(a) X-axis

(b) Y-axis

(c) Z-axis

(d) A-axis

4.Which code is generally used to call a specific tool in a VMC program?

(a) G01

(b) M03

(c) T01

(d) H01

5.The workpiece in VMC is fixed on:

(a) Spindle

(b) Tool holder

(c) Table

(d) Guide bush

6.The tool offset number is represented by which letter in CNC?

(a) T

(b) D

(c) H

(d) M

7.Tool life is reduced by:

(a) Using coolant

(b) Proper tool selection

(c) High cutting speeds

(d) Low feed rate

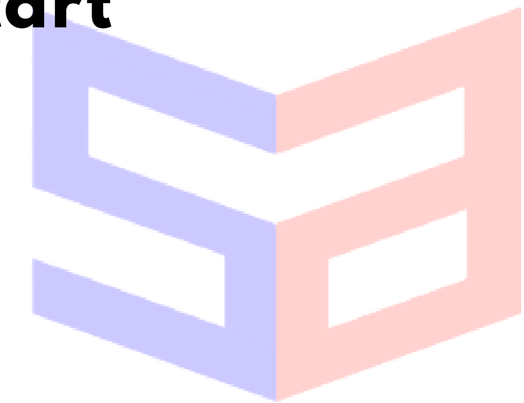
8. In an automatic tool changer (ATC), the tool is changed:

(a) By manually unscrewing

(b) By command from the CNC program

(c) After machine restart

(d) During power off



9.The most important feature to match while replacing a worn tool is:

(a) Brand name

(b) Flute color

(c) Tool geometry and size

(d) Holder shape

10.Which of the following is not a tool identification method?

(a) Tool number

(b) Tool geometry

(c) Tool color

(d) Tool holder type

11.If cutting speed is 60 m/min, and diameter is 100 mm, spindle speed (RPM) is:

(a) 120

(b) 190

(c) 600

(d) 190.99

12. Unit of depth of cut is:

(a) mm

(b) mm/min

(c) mm/rev

(d) None

13. Increasing feed rate too much causes:

(a) Better accuracy

(b) Poor surface finish and tool overload

(c) Low MRR

(d) None of these

14. For aluminum, compared to mild steel, the recommended cutting speed is:

(a) Lower

(b) Same

(c) Higher

(d) Cannot say

15.Feed in turning operation is expressed as:

(a) mm/tooth

(b) mm/min

(c) mm/rev

(d) mm^2/min

16.The Cartesian coordinate system used in VMC is based on:

(a) Polar coordinates

(b) Circular interpolation

(c) X, Y, Z linear axes

(d) A, B, C rotary axes

17.In most VMCs, the workpiece remains stationary along which axis?

(a) Z

(b) X

(c) Y

(d) None

18. Which coordinate system is affected when G54 is activated?

(a) Machine coordinate system

(b) Work coordinate system

(c) Tool coordinate system

(d) Local coordinate system

19. The coordinate system defined by the machine's software/hardware is:

(a) G92

(b) Machine coordinate system

(c) Workpiece origin system

(d) Tool length offset system

20. Which one of the following G-codes is used to cancel tool length offset?

(a) G41

(b) G49

(c) G90

(d) G80

Thank You