Here are the main types of CNC machines based on their operations:

- **CNC Milling Machines (Mills):** Use rotating cutting tools to remove material from a workpiece. They are highly versatile and used for drilling, cutting, and shaping, often featuring 3-axis or 5-axis, which allows for complex 3D shapes in one setup.
- **CNC Lathe Machines (Turning):** Rotate the workpiece against a single-point cutting tool. Ideal for creating cylindrical, conical, or spherical parts, such as shafts and bearings.
- **CNC Routers:** Similar to mills but generally used for larger, softer materials like wood, plastic, or aluminum. They are popular for signage, cabinetry, and furniture.
- **CNC Plasma Cutters:** Use a high-temperature, high-velocity plasma torch to cut through electrically conductive metals, providing fast, precise cuts.
- **CNC Laser Cutters:** Use a high-powered, focused laser beam to cut or engrave materials like metal, plastic, and wood with extreme precision.
- **CNC Electrical Discharge Machines (EDM):** Utilize electrical sparks to erode material from a workpiece. This is ideal for manufacturing complex cavities or working with hardened materials that conventional tools cannot cut.
- **CNC Waterjet Cutters:** Use a high-pressure stream of water mixed with abrasive particles to cut through materials, which is perfect for heat-sensitive materials like rubber, stone, or composites.
- **CNC Drilling Machines:** Specifically designed to create holes in a workpiece with high precision and speed.
- **Multi-Tasking Machines:** Combine turning, milling, and drilling in one machine to improve efficiency and reduce setup times for complex parts.

- **Open-loop control system, also known as non-feedback system, refers to a control process in which there is only forward action but no reverse connection between the control device and the controlled object.** In this type of system, the input signal is not affected by the output signal. An open loop control system operates on a set of predefined instructions or conditions without utilizing feedback from its output to adjust its actions. This means that the control action is initiated and conducted based on initial settings, without any subsequent modifications based on how the output compares to the desired outcome.

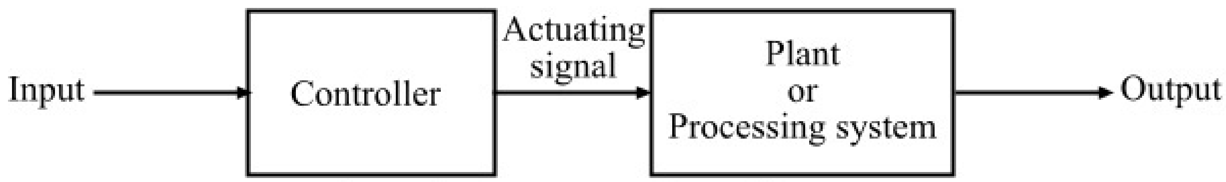


Figure 1 - Open Loop Control System

Figure 1: Open Loop Control System Block Diagram

Essentially, once an open-loop system is set in motion, it continues its operation according to the preset parameters, regardless of any discrepancies between the actual output and what was intended.

A closed-loop control system, also known as a feedback control system, dynamically adjusts its operation based on feedback from its output. This means the system continuously monitors its output, compares it to the desired outcome, and adjusts its inputs accordingly to minimize any discrepancies. This feedback loop allows the system to self-correct and adapt to changes in external conditions or to its own performance over time, enhancing precision and reliability.

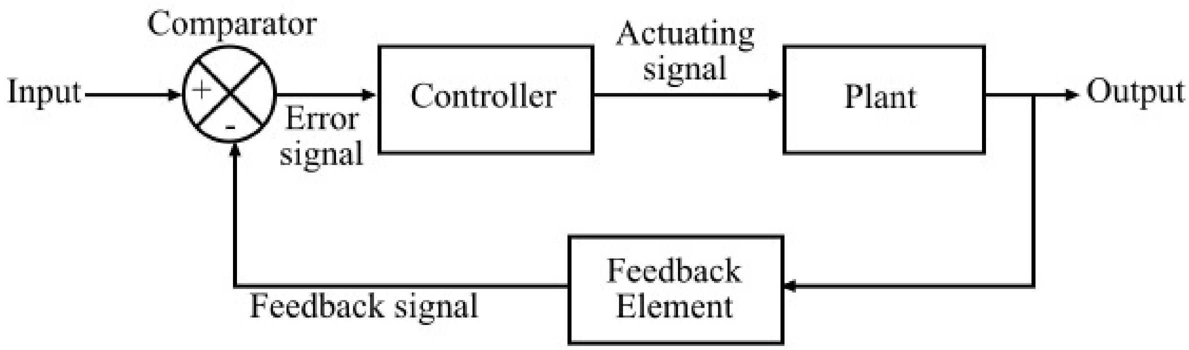


Figure 2 - Closed Loop Control System

Key components of a closed-loop control system include:

- **Sensor**: It measures the output of the system and provides the data needed for feedback.
- **Controller**: It compares the actual output with the desired output (setpoint) and decides on the action required to correct any difference.
- **Actuator**: It executes the corrective actions as determined by the controller to drive the system towards its setpoint.

As shown in Figure 2, the input signal is compared with the feedback signal from the feedback element to generate an error signal, which is processed by the controller to drive the actuator. The output of the actuator passes through the feedback element to form a feedback signal, which is returned to the comparator to complete the closed loop. The ultimate goal is to make the output signal follow the input signal.

A CNC (Computer Numerical Control) machine is a complex assembly of various components, each playing a crucial role in its functionality. Understanding the structure of a CNC machine helps in grasping how these machines achieve precision and efficiency in manufacturing processes. Below, we explore the essential parts and their functions in a typical CNC machine.

1. **Machine Control Unit (MCU):** The MCU is the brain of the CNC machine, interpreting the G-code instructions and controlling the motion of the machine. It coordinates the movements of the machine tools and ensures precise execution of machining operations.

2. **Input Device:** This component is responsible for loading the CNC programs into the machine. Input devices can include tape readers, USB interfaces, and direct computer connections.

3. **Machine Tools:** CNC machines utilize various tools, such as cutting tools, drills, and milling cutters, to perform different operations. These tools are interchangeable and can be automatically changed during the machining process.

4. **Driving System:** The driving system comprises motors, such as stepper and servo motors, which move the machine tools along the machine's axes. This system ensures accurate positioning and movement during machining.

5. **Feedback System:** Sensors within the feedback system monitor the operations and provide real-time adjustments to maintain precision. This system is crucial for achieving high accuracy in machining tasks.

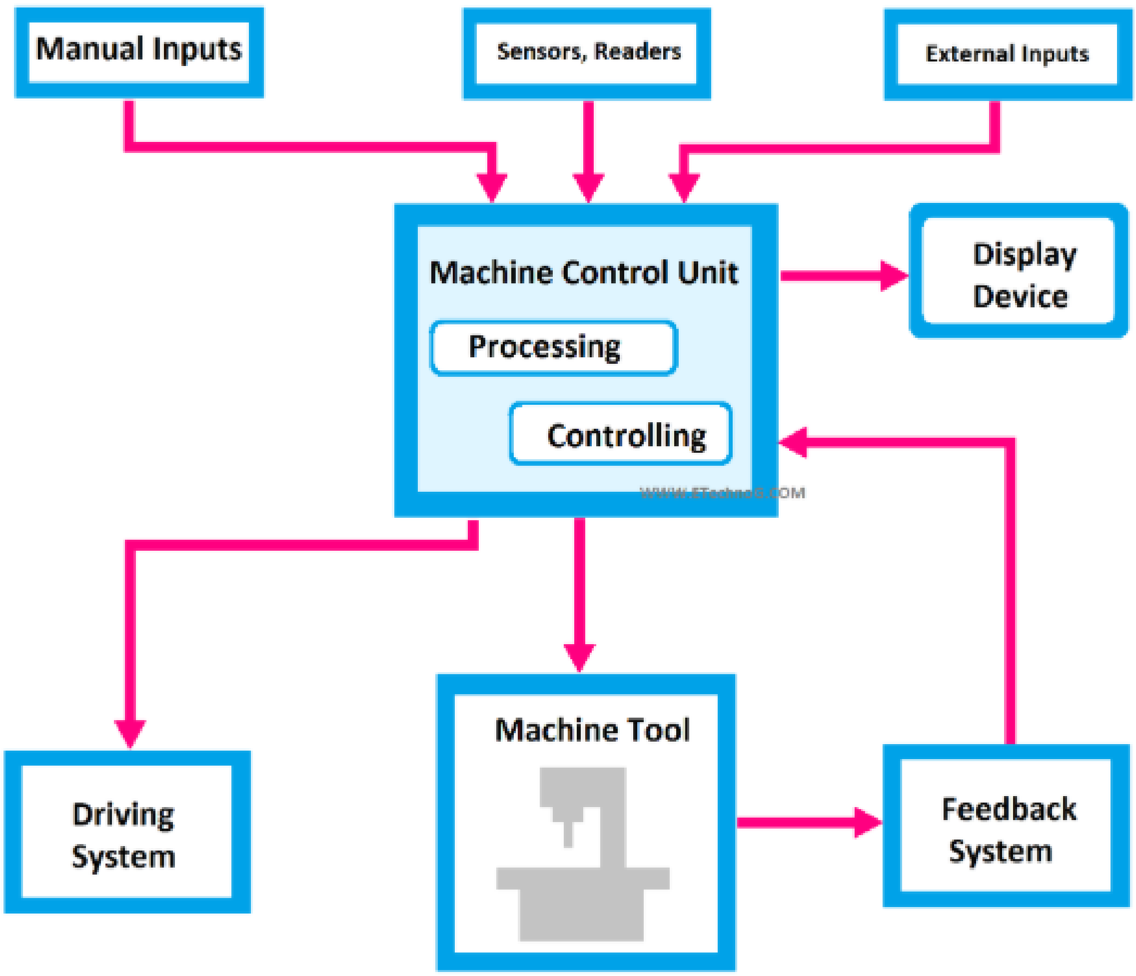
6. **Display Unit:** The display unit provides a user interface for operators to monitor and control the machining process. It displays vital information about the machine's status and operations.

7. **Bed:** The bed of the CNC machine is a sturdy base that supports all other components. It is typically made of cast iron to provide stability and reduce vibrations during machining.

8. **Headstock:** The headstock houses the spindle, which rotates the workpiece or the cutting tool, depending on the type of CNC machine. It is essential for the machine's operation and precision.

9. **Tailstock:** Used primarily in CNC turning machines, the tailstock supports long workpieces to prevent them from deflecting during machining.

10. **Control Panel:** The control panel allows operators to interact with the machine, input commands, and adjust settings as needed. It is an integral part of the CNC system, providing control over the entire machining process.



CNC Machine Block Diagram

Miscellaneous functions / M codes

M - code is Machine language code to activate miscellaneous functions.

M stands for **'Machine codes'**. This code is used for non-geometry machine functions like coolant on/off, spindle speed, tool change, pallet change etc.

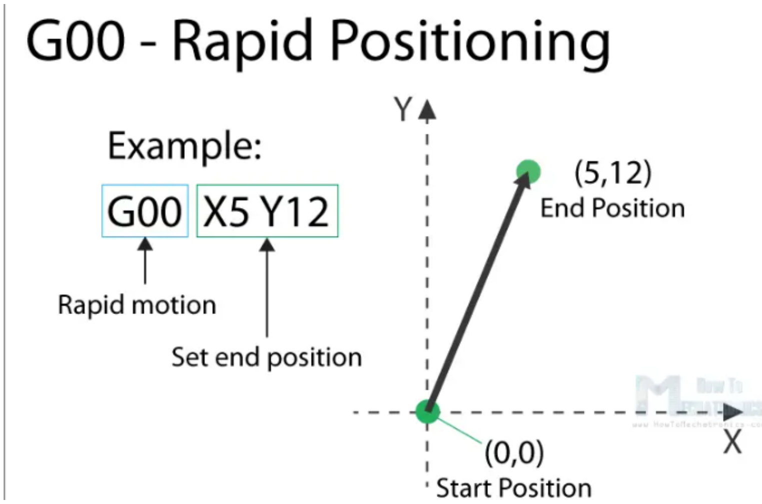
List of Miscellaneous functions (M codes)

Sl no	Functions
M00	Program stop
M01	Optional stop
M02	Program end
M03	Spindle clockwise direction
M04	Spindle counter clockwise direction
M05	Spindle stop
M06	Tool change
M17	Subprogram end
M19 OR SPOS=0	Spindle orientation
M30	Main program end

Note: Miscellaneous commands are machine manufacturer specific. Contact your machine manufacturer for more details on M functions.

G00 – Rapid Positioning

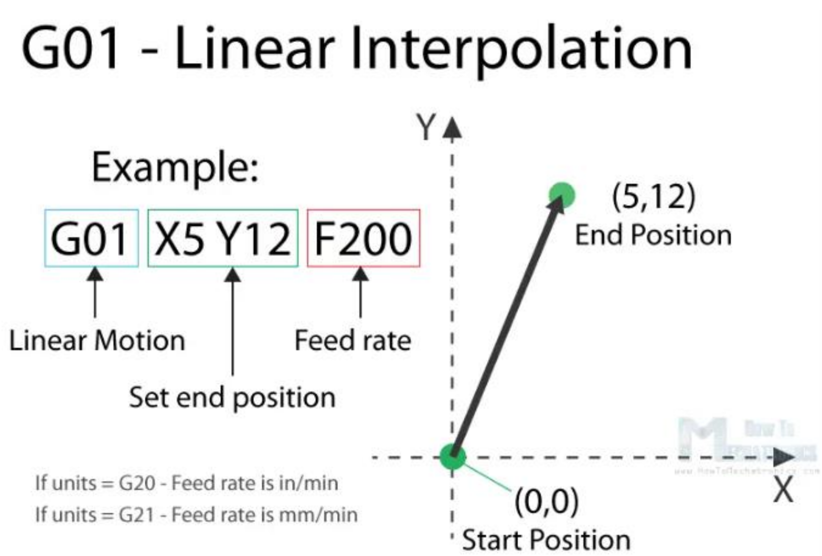
The **G00** command moves the machine at maximum travel speed from a current position to a specified point or the coordinates specified by the command. The machine will move all axis at the same time so they complete the travel simultaneously. This results in a straight line movement to the new position point.



The G00 is a non-cutting movement, and its purpose is to just quickly move the machine to the desired position to begin some kind of job, like cutting or printing.

G01 – Linear Interpolation

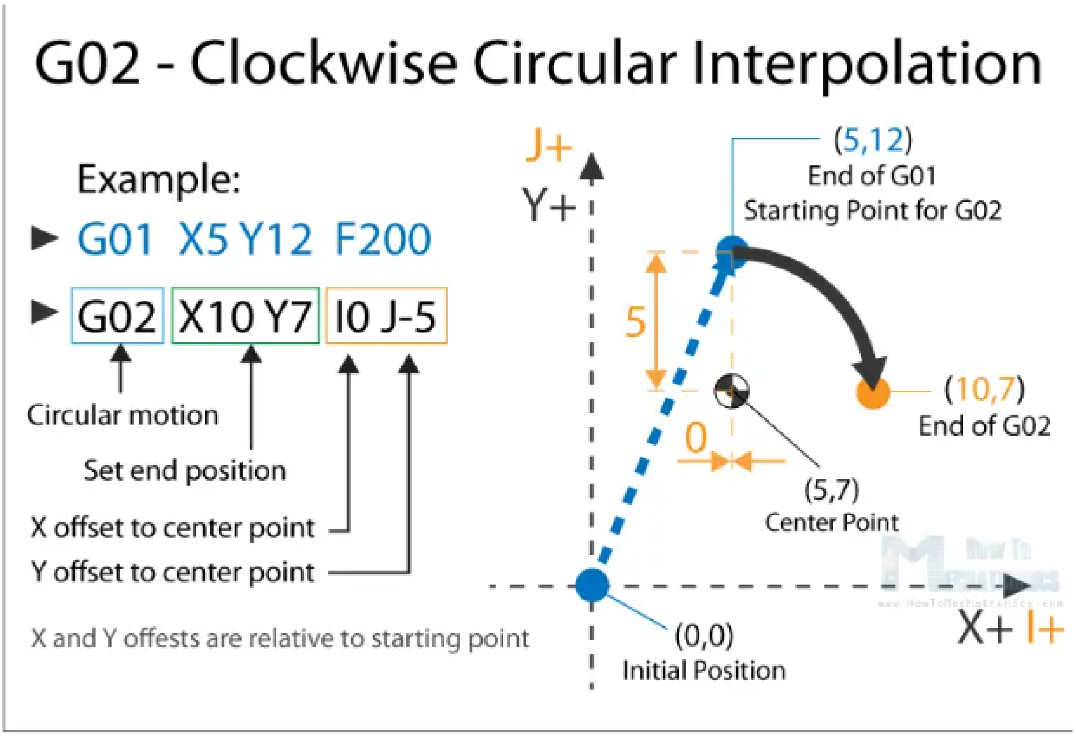
The G01 G-code command instructs the machine to move in a straight line at a set feed rate or speed. We specify the end position with the X, Y and Z values, and the speed with the F value. The machine controller calculates (interpolates) the intermediate points to pass through to get that straight line. Although these G-code commands are simple and quite intuitive to understand, behind them, the machine controller performs thousands of calculations per second in order to make these movements.

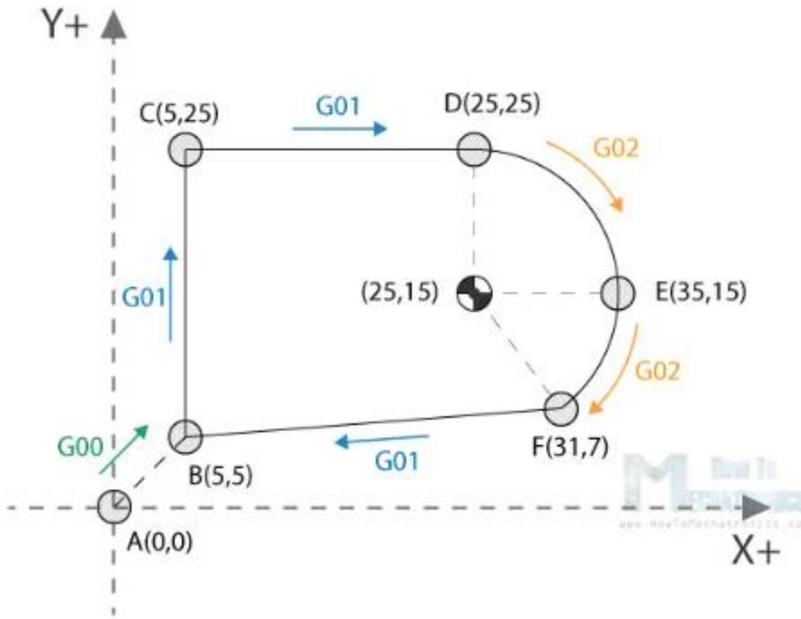


Unlike the G00 command which is used just for positioning, the G01 command is used when the machine is performing its main job. In case of lathe or mill, cutting material in straight line, and in case of a 3D printer, extruding material in straight line.

G02 – Circular Interpolation Clockwise

The G02 command tells the machine to move clockwise in a circular pattern. It's the same concept as the G01 command and it's used when performing the appropriate machining process. In addition to the end point parameters, here we also need to define the center of rotation, or the distance of the arc start point from the center point of the arc. The start point is actually the end point from the previous command or the current point



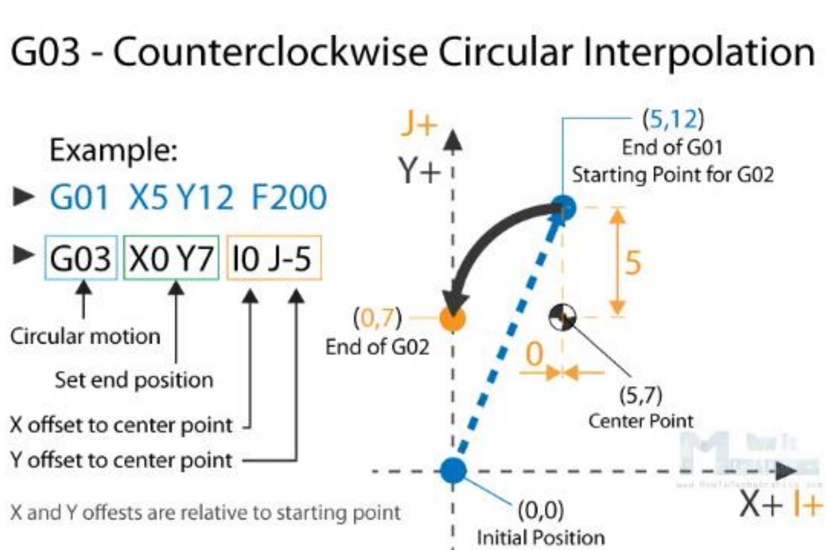


To get the toolpath for the shape shown in the image above we need to following G-code commands:

```
G00 X5 Y5            ; point B
G01 X0 Y20 F200        ; point C
G01 X20 Y0           ; point D
G02 X10 Y-10 I0 J-10     ; point E
G02 X-4 Y-8 I-10 J0      ; point F
G01 X-26 Y-2           ; point B
```

G03 – Circular Interpolation Counterclockwise

Just like the G02, the G03 G-code command defines the machine to move in circular pattern. The only difference here is that the motion is counterclockwise. All other features and rules are the same as the G02 command



G20/ G21 – Units Selection

The G20 and G21 commands define the G-code units, either inches or millimters.

G20 = inches

G21 = millimiters

We need to note that the units must be set at the beginning of the program. If we don’t specify the units the machine will consider the default set by the previous program.

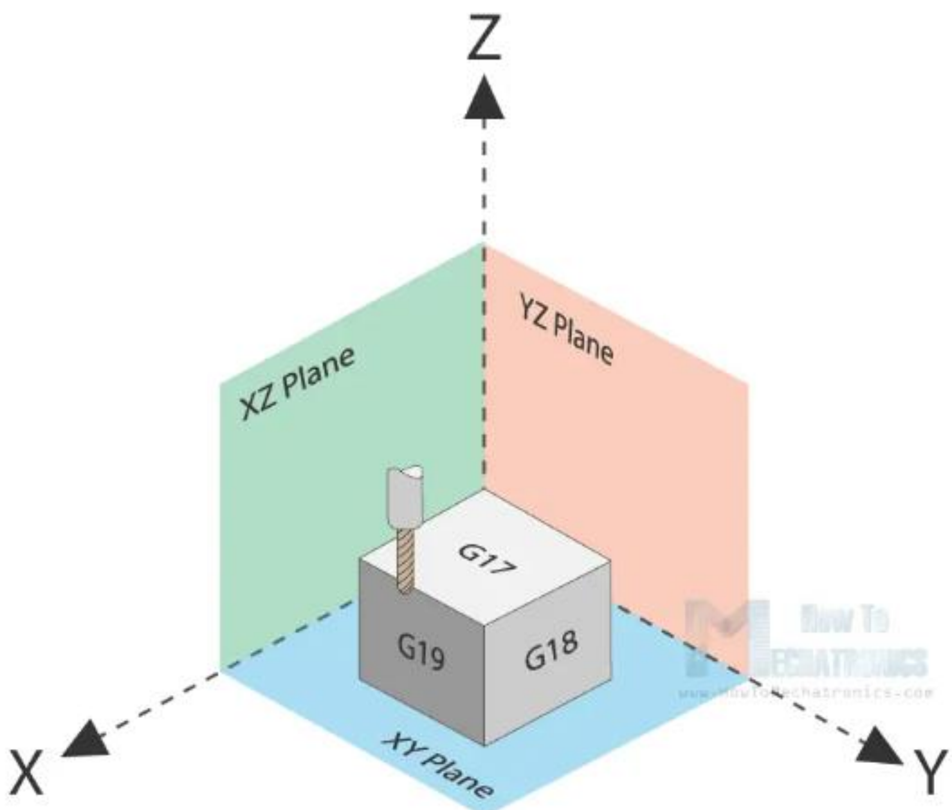
G17/ G18/ G18 – G-code Plane Selection

With these G-code commands we select the working plane of the machine.

G17 – XY plane

G18 – XZ plane

G19 – YZ plane

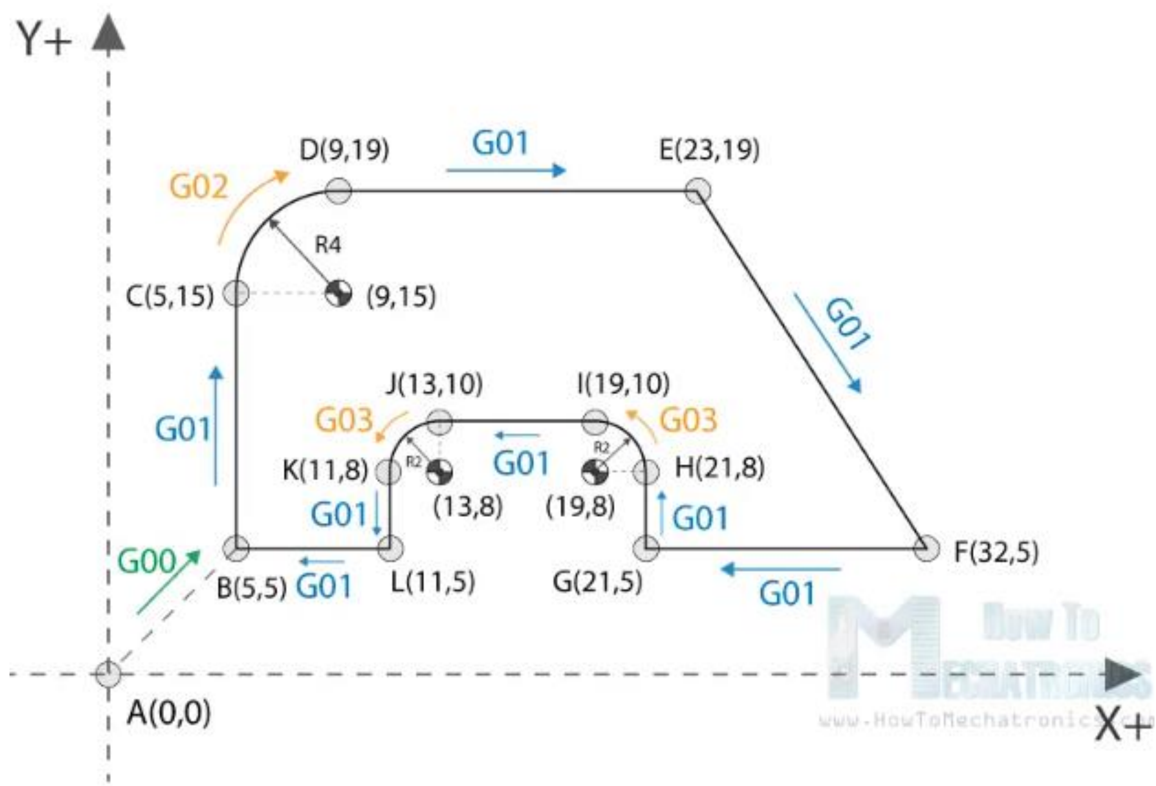


G28 – Return Home

The G28 command tells the machine to move the tool to its reference point or home position. In order to avoid collision, we can include an intermediate point with X, Y and Z parameters. The tool will pass through that point before going to the reference point.

G90/ G91 – Positioning commands

With the G90 and G91 commands we tell the machine how to interpret the coordinates. G90 is for absolute mode and G91 is for relative mode.




```
G21 G17 G90 F100
M03 S1000
G00 X5 Y5                ; point B
G01 X5 Y5 Z-1            ; point B
G01 X5 Y15 Z-1           ; point C
G02 X9 Y19 Z-1 I4 J0     ; point D
G01 X23 Y19 Z-1          ; point E
G01 X32 Y5 Z-1           ; point F
```

```
G01 X21 Y5 Z-1                ; point G
G01 X21 Y8 Z-1                ; point H
G03 X19 Y10 Z-1 I-2 J0        ; point I
G01 X13 Y10 Z-1               ; point J
G03 X11 Y8 Z-1 I0 J-2         ; point K
G01 X11 Y5 Z-1                ; point L
G01 X5 Y5 Z-1                 ; point B
G01 X5 Y5 Z0
G28  X0 Y0
M05
M30
```

Description of the G-code program:

1. Safety line: Set programming in metric system (all dimensions in mm), XY plane, absolute positioning and feed rate of 100 inches/min.
2. Spindle on clockwise at speed of 1000 RPM.
3. Rapid positioning to B(5,5).
4. Controlled motion on the same position, but lowering the tool to -1.
5. Linear cutting movement to position C(5,15).
6. Clockwise circular motion to point D(9,19), with center point at (9,15).
7. Linear cutting to point E(23,19).
8. Linear cutting to point F(32,5).
9. Same straight cutting to point G(21,5).
10. One more straight cutting to point H(21,8).
11. Counterclockwise circular interpolation to position I(19,10), with a center point at (19,8).
12. Linear cutting to point J(13,10).

13. Counterclockwise circular cutting to position K(11,8), with a center point at (13,8).
14. Linear cutting to position L(11,5).
15. Final linear cutting movement to position B(5,5).
16. Rise up the tool.
17. Go to home position.
18. Spindle off.
19. Main program end.

Fanuc G-Code List (Lathe)

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G09	Exact stop
G10	Programmable data input
G20	Input in inch
G21	Input in mm
G22	Stored stroke check function on
G23	Stored stroke check function off
G27	Reference position return check
G28	Return to reference position

G32	Thread cutting
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G70	Finish machining cycle
G71	Turning cycle
G72	Facing cycle
G73	Pattern repeating cycle
G74	Peck drilling cycle
G75	Grooving cycle
G76	Threading cycle
G92	Coordinate system setting or max. spindle speed setting
G94	Feed Per Minute
G95	Feed Per Revolution
G96	Constant surface speed control
G97	Constant surface speed control cancel

Fanuc G-Code List (Mill)

G code	**Description**
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW

G03	Circular interpolation CCW
G04	Dwell
G17	X Y plane selection
G18	Z X plane selection
G19	Y Z plane selection
G28	Return to reference position
G30	2nd, 3rd and 4th reference position return
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation + direction
G44	Tool length compensation – direction
G49	Tool length compensation cancel
G53	Machine coordinate system selection
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G68	Coordinate rotation

G69	Coordinate rotation cancel
G73	Peck drilling cycle
G74	Left-spiral cutting circle
G76	Fine boring cycle
G80	Canned cycle cancel
G81	Drilling cycle, spot boring cycle
G82	Drilling cycle or counter boring cycle
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
G90	Absolute command
G91	Increment command
G92	Setting for work coordinate system or clamp at maximum spindle speed
G98	Return to initial point in canned cycle
G99	Return to R point in canned cycle

Programming Key Letters

– O - Program number (Used for program identification)
– N - Sequence block number (Used for line identification)
– G - Preparatory function
– X - X axis designation
– Y - Y axis designation
– Z - Z axis designation
– R - Radius designation
– F – Feed rate designation
– S - Spindle speed designation
– H - Tool length offset designation
– D - Tool radius offset designation
– T - Tool Designation
– M - Miscellaneous function

Important G codes

- G00 Rapid Transverse
- G01 Linear Interpolation
- G02 Circular Interpolation, CW
- G03 Circular Interpolation, CCW
- G17 XY Plane
- G18 XZ Plane
- G19 YZ Plane
- G20/G70 Inch units
- G21/G71 Metric Units
- G40 Cutter compensation cancel
- G41 Cutter compensation left
- G42 Cutter compensation right
- G43 Tool length compensation (plus)
- G43 Tool length compensation (plus)
- G44 Tool length compensation (minus)
- G49 Tool length compensation cancel
- G80 Cancel canned cycles
- G81 Drilling cycle
- G82 Counter boring cycle
- G83 Deep hole drilling cycle
- G90 Absolute positioning
- G91 Incremental positioning

Tool Path Generation, Simulation, and Post-Processing

Tool Path Generation, Simulation, and Post-Processing are critical steps in the manufacturing process, particularly in computer-aided manufacturing (CAM) and CNC machining. These steps ensure that the manufacturing process is accurate, efficient, and results in high-quality parts.

Tool Path Generation

Definition

Tool Path Generation: The process of creating a precise path that the cutting tool follows to machine a part according to its design specifications.

Purpose

Objective: Convert the part's geometry into a sequence of movements for the cutting tool to produce the desired shape.

Impact: Ensures that the machining process accurately follows the design, optimizing material removal and tool performance.

Key Concepts

Path Strategies:

- **Contour Milling:** The tool follows the outer edges of the part.
- **Pocket Milling:** The tool removes material from within a defined area.
- **Drilling**: The tool creates holes by moving along a specified path.
- **Roughing and Finishing**: Roughing paths remove bulk material, while finishing paths refine surface details.

Tool Path Types:

- **Linear Path**: Straight-line movements of the tool.
- **Circular Path:** Tool follows circular or arc-shaped trajectories.
- **Zigzag Path:** Alternating back-and-forth movements for efficient material removal.

Factors Influencing Tool Path Generation

- Part Geometry: Complexity of the shape affects the tool path strategy.
- Tool Type: Different tools require different paths and techniques.
- Material Properties: Hardness and machinability influence the tool path design.
- Machine Capabilities: The machine's range of motion and precision impact tool path choices.

Simulation

Definition

Simulation: The process of virtually testing the tool path and machining operations to ensure that they perform as intended before actual machining.

Purpose

Objective: Identify and resolve potential issues in the tool path, such as collisions, tool interference, or incorrect machining.

Impact: Reduces the risk of errors, improves process efficiency, and prevents costly mistakes during actual machining.

Key Concepts

- Virtual Machining: Simulates the cutting process in a digital environment.
- Collision Detection: Identifies and addresses potential collisions between the tool, workpiece, and machine components.
- Material Removal Simulation: Visualizes how material is removed from the workpiece to check for accuracy and completeness.
- Tool Wear and Breakage: Simulates the effects of tool wear or breakage on the machining process.

Benefits of Simulation

- Error Detection: Find and correct issues before physical machining.
- Optimization: Fine-tune tool paths and parameters for improved performance.
- Time and Cost Savings: Reduce setup time and material wastage by preemptively solving problems

Post-Processing

Definition

Post-Processing: The final step in CAM where the generated tool path data is converted into machine-readable code, typically G-code, for execution by CNC machines.

Purpose

Objective: Translate the tool path and machining instructions into a format that the CNC machine can understand and execute.

Impact: Ensures that the machine accurately follows the tool path and performs the desired operations.

Key Concepts

- G-Code: A programming language used to control CNC machines, specifying movements, speeds, and other machining parameters.
- M-Code: Used for machine-specific functions such as tool changes and spindle control.
- Post-Processor Software: Converts tool path data from CAM software into G - code or other machine-specific formats.

Post-Processing Steps

- Code Generation: Create the machine code from the tool path data.
- Code Verification: Check the generated code for errors or inconsistencies.
- Optimization: Refine the code to improve efficiency, reduce cycle time, and enhance machine performance.

Challenges in Post-Processing

- **Machine Compatibility**: Ensuring the generated code is compatible with the specific CNC machine.
- **Error Detection:** Identifying and correcting errors in the generated code before actual machining.
- **Optimization**: Balancing between efficiency and complexity of the generated code.