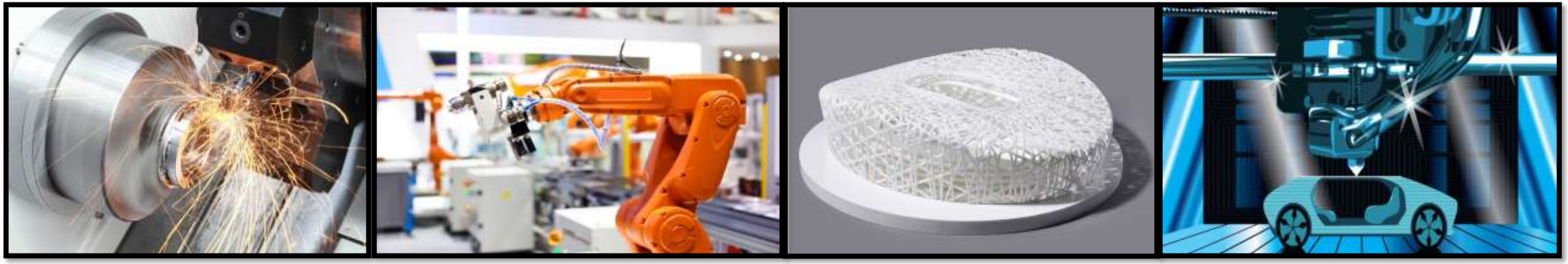
CNC Machines (ME 325)

Tishk International University – Erbil
College of Engineering
Department of Mechatronics Engineering



Academic Year 2023-2024
3rd Year (Summer School)

LECTURE 3

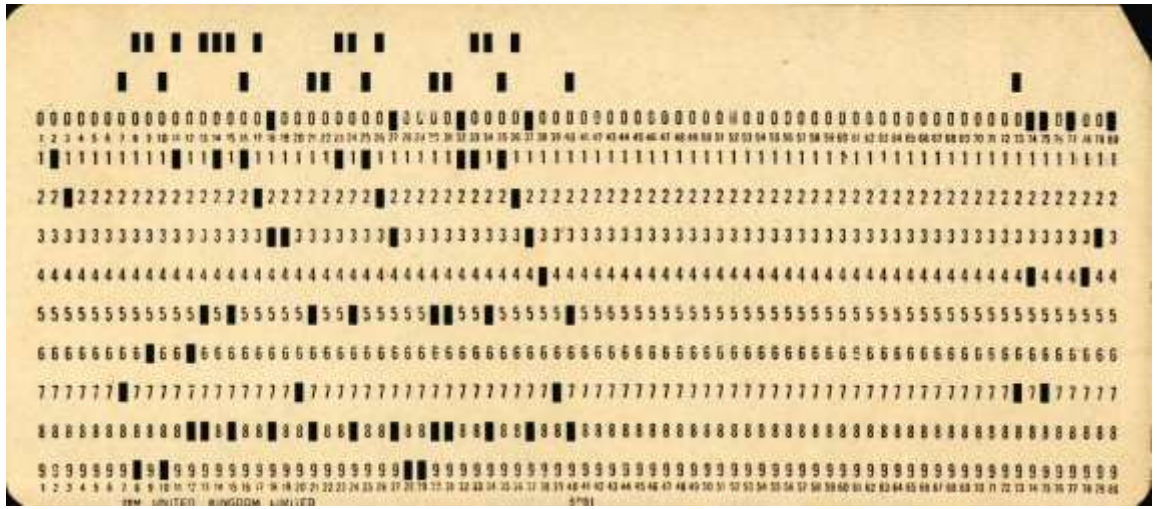
Instructor : Ahmed A. Arsalan Hadi
MSc in Mechanical & Mechatronics Engineering
Email:ahmed.ameerarsalan@tiu.edu.iq

Previous Lecture

- Important Terms in Manufacturing
- Introduction to CAD system
- CAD data models
- Data Formats
- CAD softwares
- Importance of CAD/CAM

Outline

- Numerical Control Fundamentals (NC)
- Axis of Motion
- NC vs CNC
- DNC
- Interpolations
- G code
- M code



Video

What is Numerical Control (NC)?

Form of programmable automation in which the processing equipment (e.g., machine tool) is controlled by coded instructions using numbers, letter and symbols using punched cards.

- Numbers form a set of instructions (or NC program) designed for a particular part.
- Allows new programs on same machine for different parts.
- Most important function of NC system is positioning (tool and/or work piece).

When is it appropriate to use NC?

1. Parts from similar raw material, in variety of sizes, and/or complex geometries.

2. Low-to-medium part quantity production.

3. Similar processing operations & sequences among work pieces.

4. The machine needs to be set up often for different parts.

5. Need to be sure the parts are very precise (compared to other regular machines).

Advantages of NC over conventional systems:

- Flexibility with accuracy, repeatability, high production rates, good quality.
- Reduced tooling costs.
- Easy machine adjustments.
- More operations per setup, less lead time, accommodate design change.
- Rapid programming and program recall, less paperwork.
- Faster prototype production.
- Less-skilled operator, multi-work possible.

Limitations of NC:

- Relatively high initial cost of equipment.
- Need for part programming.
- Special maintenance requirements.
- More costly breakdowns.

Advantages of CNC over conventional NC:

- Control using software (executive) rather than hard-wired
- Increased flexibility (variety of mixed operations & functions)
- Elimination of tape reader (or tape read only once per program)
- Part program storage (computer memory (multiple programs) & storage media)
- Display shows instructions being executed & other operational data
- Greater accuracy (faster control solutions)
- More adaptability (e.g., program editing (at the machine), reprogramming, tool path plotting, metric conversion, cutter dimension compensation)
- System integration capability (connect to robots & other computer- or microprocessor- based equipment, create cells)
- Machine diagnostics (gives error message or identifies problem)

Distributed (Direct) Numerical Control (DNC):

- (Direct) Central computer stores programs & directs NC operations; NC machines dependent on central computer.
- (Distributed) Central computer stores programs and transfers programs to CNC machines.
- Central computer provides management functions. Programs stored as cutter location (CL) files and post-processed for the machine assigned the job.
- Major components: central computer; bulk memory; telecommunication (EDI); CNC machine tools.

Three basic components of an NC system:

1. Input medium:

- Part program or instructions needed to drive the machine tool components
- Instructions are prepared manually or by use of computer
- Instructions include machining parameters (feed rate, cutting speed); sequence of actions (e.g., positioning & machine functions)
- Instructions are stored in the form of tape (paper, magnetic); floppy diskettes; DNC download to CNC RAM

2. Machine control unit (MCU):

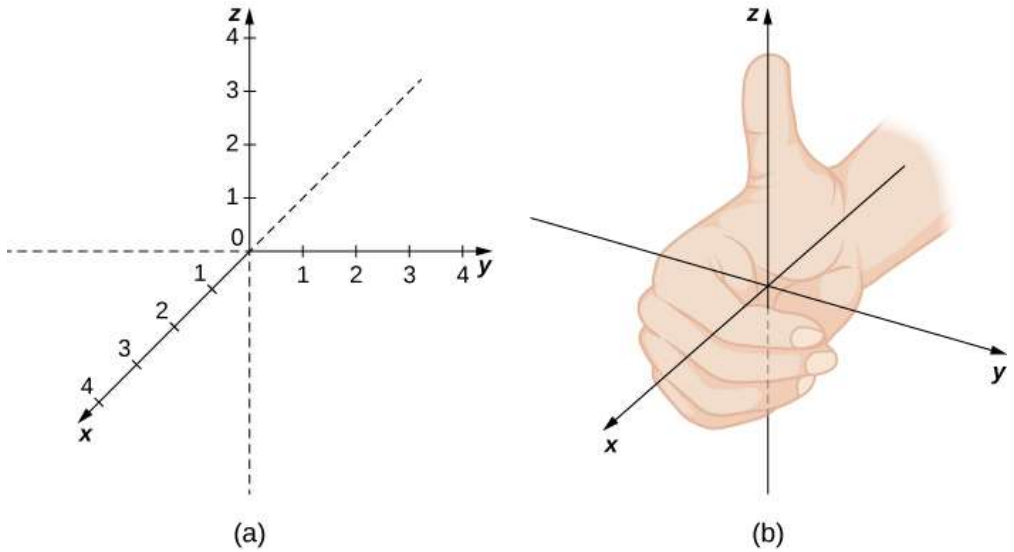
- Electronics & control hardware
- Interpret instruction set
- Execute instructions
- Monitor results & correct where appropriate

3. Machine tool:

- Mechanical structure that performs the machining, including the components that drive each axis of motion (e.g., AC or DC motor; hydraulic actuator; stepper motor — choice affects speed of response, accuracy and power capacity).

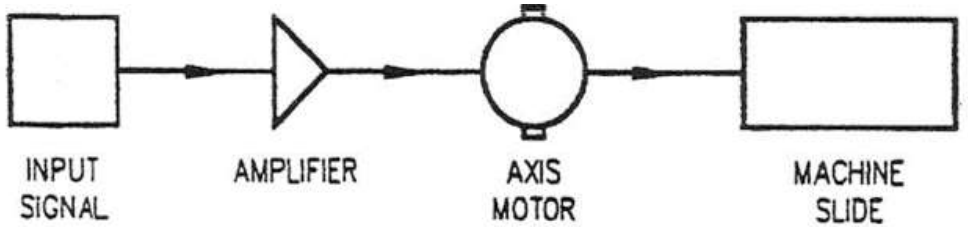
What is meant by axis of motion?

Axis of motion describes the relative motion that occurs between the cutting tool and the workpiece. Three main axes of motion for machine tools are referred to as the x, y, and z axes that form a right-hand coordinate system.

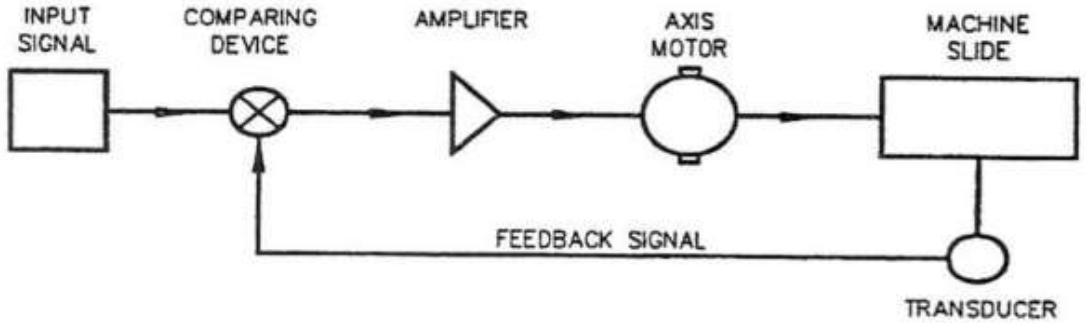


Open loop vs. closed loop control:

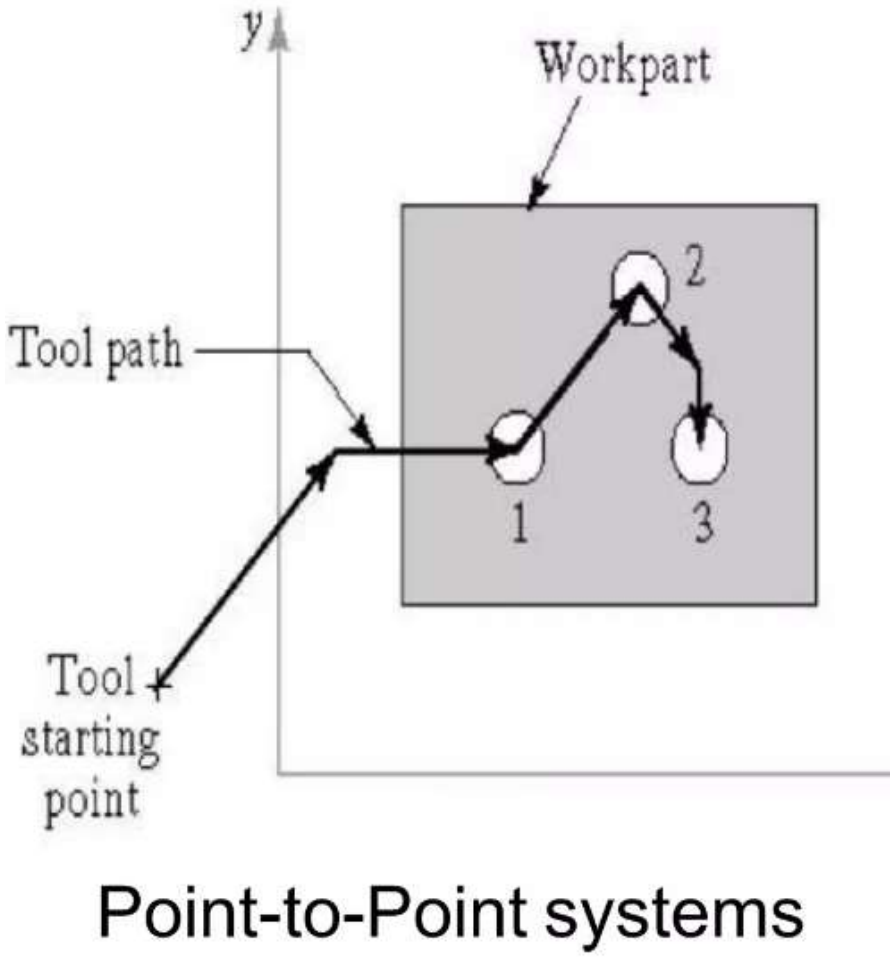
· **Open loop** — control signals are given to actuators by the MCU, but the movements and final destinations of the positioning system are not checked for accuracy.



· **Closed loop** — equipped with transducers and sensors to measure positions, compare with control signals, and correct positions as necessary.



Types of control systems:

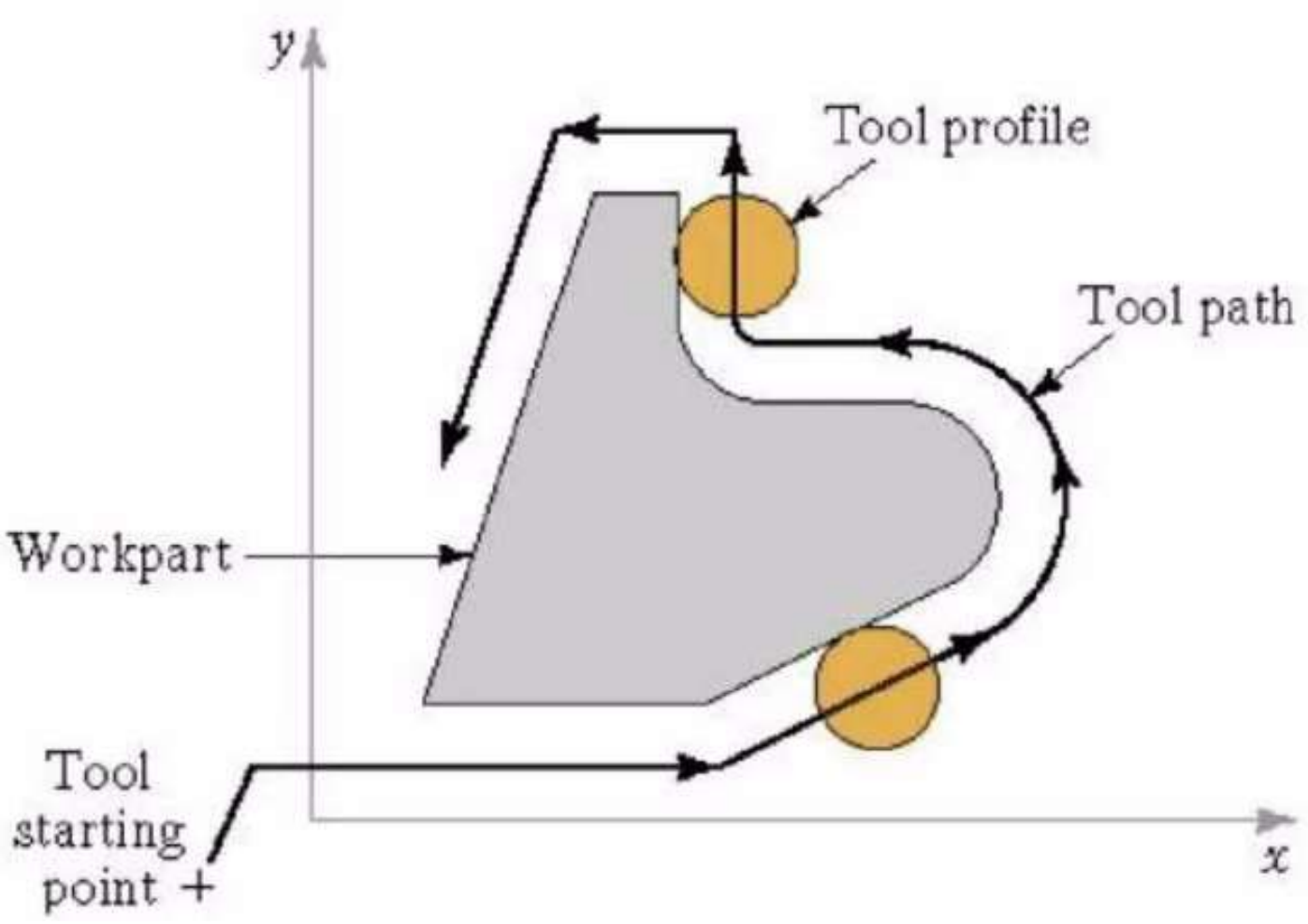


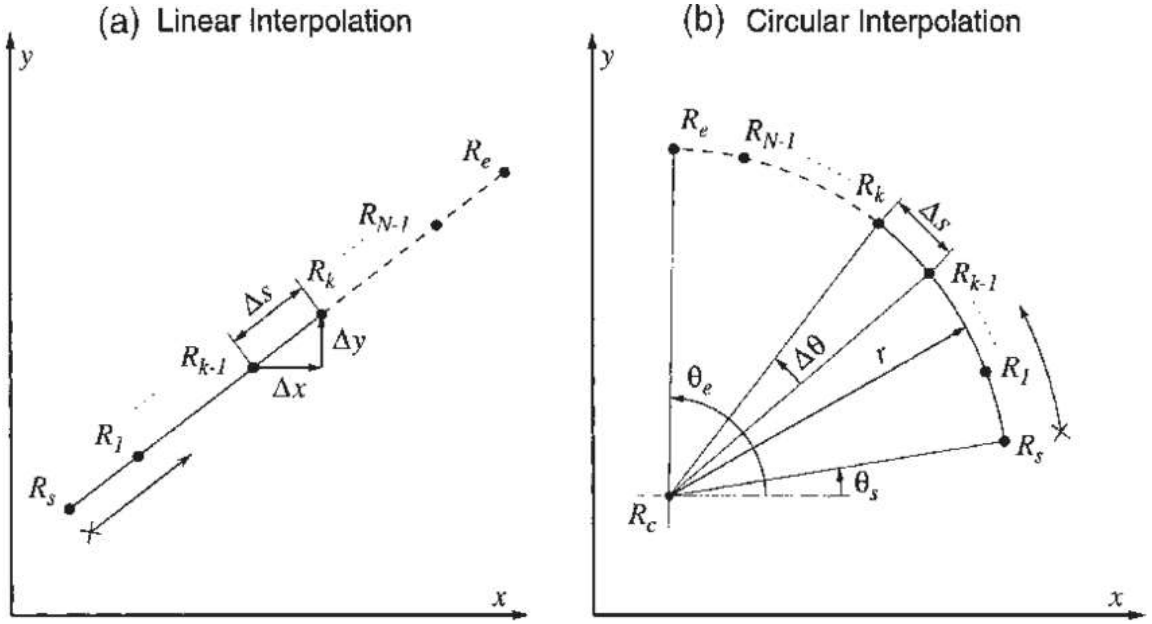
Point-to-Point systems

- **Point-to-point (positioning) system**: tool or workpiece is driven to predefined location without specifying the speed or path of motion; no tool-workpiece contact during positioning; simple, inexpensive control system; used mainly in drilling, punching, straight milling

Types of control systems:

- **Contouring (continuous path) system**: positioning (path) and speed are simultaneously controlled at specified values; contour path based on interpolation between points; complex, flexible, expensive control system; capacity for simultaneous control of more than one axis of motion; used on lathes, milling machines, grinders, welding machinery, machining centers.





Types of interpolation:

- **Linear**: tool or work piece moves in a straight line from specified start point to endpoint in two or three axes

- **Circular**: path of motion is derived from specified coordinates of end points, coordinates of arc center, radius of arc and direction of motion; tool path approximated by series of straight-line segments.

Steps in NC/CNC programming procedure:

- Interpret part drawing
- Define zero point
- Define x-, y-, z-axes
- Determine machining requirements
- Determine required operations & sequence
- Determine tooling requirements
- Determine feeds, speeds, depth(s) of cut
- Complete part program
- Complete post processor
- Store part program (e.g., punch tape, disk file)
- Verify completed program: "check it out;" is it accurate?

G-code

G-code is a common name for the programming language that is used for NC and CNC machine tools. It is defined in EIA RS-274-D. G-code is also the name of any word in a CNC program that begins with the letter G, and generally is a code telling the machine tool what type of action to perform, such as:

- rapid move
- controlled feed move in straight line or arc
- series of controlled feed moves that would result in a hole being drilled.
- change a pallet
- Set tool information such as offset.

G-code

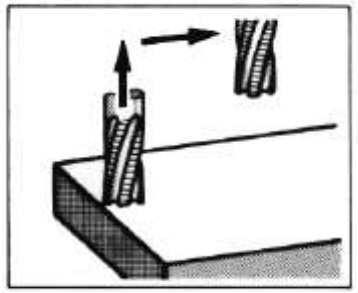
There are other codes; the type codes can be thought of like registers in a computer

- X position
- Y position
- Z position
- M code (another "action" register) F feed rate
- S spindle speed N line number R Radius
- T Tool selection
- I Arc data X axis
- J Arc data Y axis

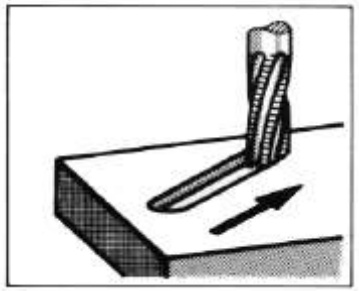
G-code files are output by CAM software such as Smartcam, Gibbscam, Featurecam, Solidworks CAM, Mastercam, etc. G-code is also output by specialized CAD systems used to design printed circuit boards. Such software must be customized for each type of machine tool that it will be used to program.

G-code

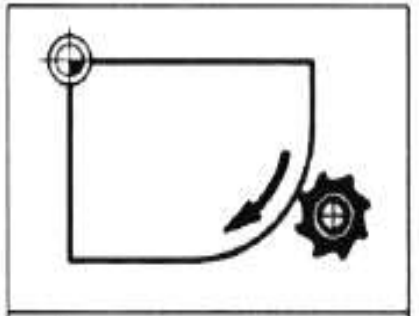
G00
RAPID TRAVERSE



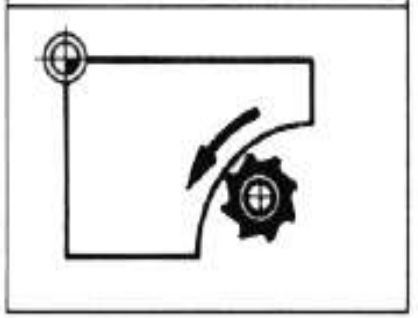
G01
LINEAR INTERPOLATION
(STRAIGHT LINE MOVEMENT)



G02
CIRCULAR INTERPOLATION
(CLOCKWISE)

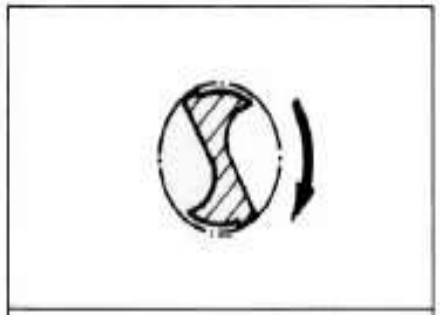


G03
CIRCULAR INTERPOLATION
(COUNTERCLOCKWISE)

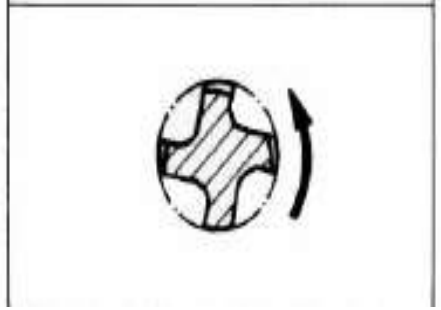


M code

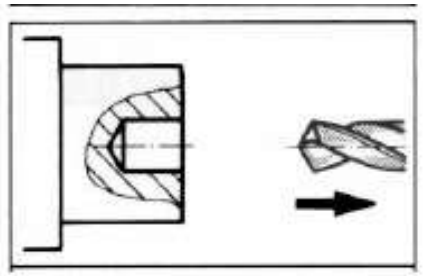
M03
DIRECTION OF ROTATION
(CLOCKWISE)



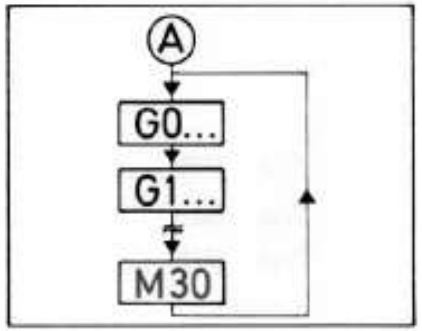
M04
DIRECTION OF ROTATION
(COUNTERCLOCKWISE)



M06
TOOL CHANGE WITH
AUTOMATIC RETRACTION



M30
END OF PROGRAM
AND
RETURN TO BEGINNING
OF PROGRAM



Next Lecture

CONVENTIONAL AND CNC MILLING OPERATIONS

Thank You