IE360: CAD/CAM

Computer Aided Design and Computer Aided Manufacturing

Lecture (10)

Manual Part Programming

Dr Ashraf Afifi

Industrial Engineering Department
King Saud University

Outline

- Basic concepts of part programming
- Syntax of a part program
- Preparatory commands (G codes)
- Miscellaneous commands (M codes)

Basic concepts of part programming:

- A part program contains **geometric information** about the part and **motion information** to move the cutting tool with respect to the workpiece.

- The first thing that a part programmer needs to define in order to describe both geometry and motion is a **coordinate system** on which point locations can be specified.

  - The NC programmer has to ensure that the orientation of the geometric coordinate system for the tool pass is identical to that of the machine tool that will read and execute the corresponding NC program. Otherwise, the machine tool using the part program will produce the wrong part.

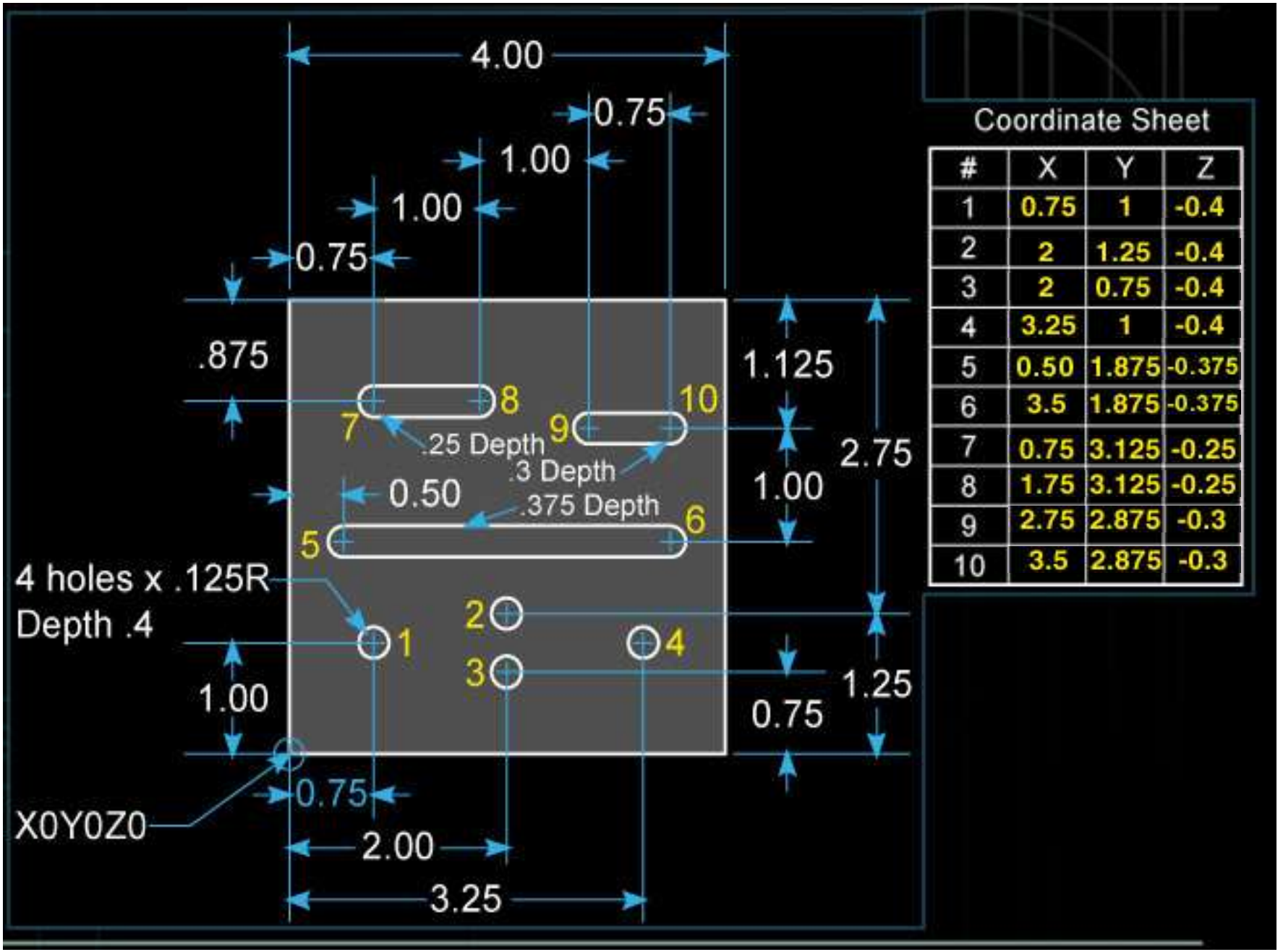
  - If the tool pass is generated from a CAD database, the coordinate system for modelling or drafting must be identical to that of the machine tool.

➢ **Program zero** allows you to specify a position from which to start or to work. Once program zero has been defined, all coordinates used in a program will be referenced from this point.

- The position selected for milling is always the lower left-hand corner and top surface of the workpiece.

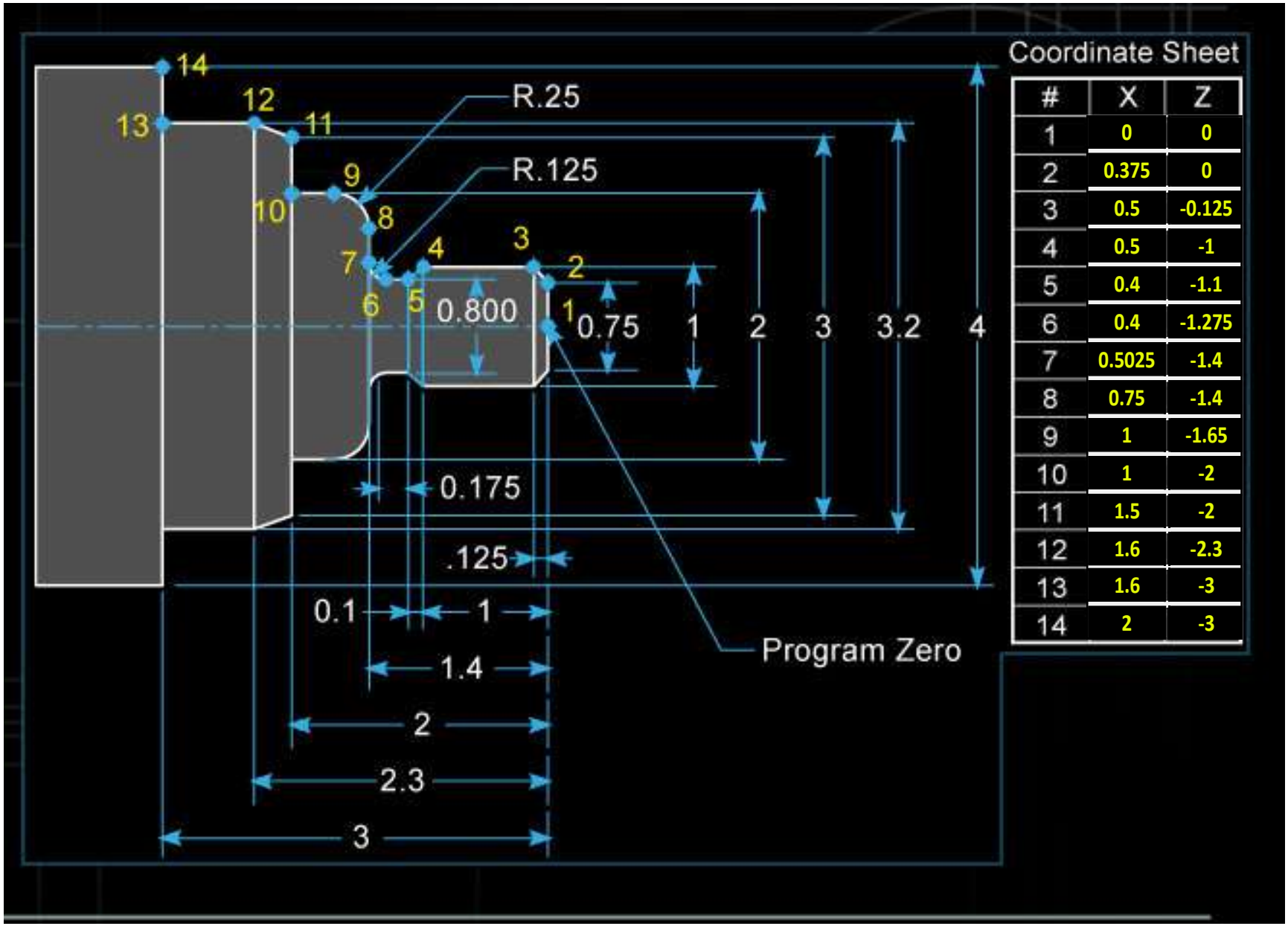
- The position used for the lathe is always the center of the part in X and the right-hand end of the finished workpiece in Z.

- The following are two examples of using the program zero to specify absolute positions in the X, Y and Z directions for milling and turning machine tools, respectively.



Coordinate Sheet

#	X	Y	Z
1	0.75	1	-0.4
2	2	1.25	-0.4
3	2	0.75	-0.4
4	3.25	1	-0.4
5	0.50	1.875	-0.375
6	3.5	1.875	-0.375
7	0.75	3.125	-0.25
8	1.75	3.125	-0.25
9	2.75	2.875	-0.3
10	3.5	2.875	-0.3



Coordinate Sheet

#	X	Z
1	0	0
2	0.375	0
3	0.5	-0.125
4	0.5	-1
5	0.4	-1.1
6	0.4	-1.275
7	0.5025	-1.4
8	0.75	-1.4
9	1	-1.65
10	1	-2
11	1.5	-2
12	1.6	-2.3
13	1.6	-3
14	2	-3

Syntax of a part program:

- A part program is a sequential list of machining instructions for the CNC machine to execute. It follows a well-defined syntax with variations due to differences between machines.

- The instructions consist of **blocks** (also called **lines**). Each block contains an individual **command** for a movement or specific action.

- Each command has an identifying **letter** (also called **word address**) followed by an associated number. The identifying letters of the commands are as follows:

  - **Sequence number (N code):** The sequence number is used to identify each block in a part program and provides a way to locate commands rapidly.

Syntax of a part program:

- **Preparatory commands (G codes):** The preparatory commands prepare the machine tool for a given operation, typically involving a cutter motion.

- **Dimension words (X, Y, and Z):** These words contain the location and axis orientation data of a cutter.

- **Feed commands (F code):** The F code is used to specify the cutter feed rates to be applied. Units are inches per minutes (ipm) by convention.

- **Speed commands (S code):** The S code is used to specify the spindle speed. Units are revolutions per minutes (rpm).

- **Tool selection command (T code):** The T code specifies which tool is to be used in a specific operation.

- **Miscellaneous commands (M codes):** M codes are used to designate a particular mode of operation, typically to switch machine functions such as coolant supply and spindle on or off.

➢ The standard sequence of words in a block is as follows:

N_, G_, X_, Y_, Z_, I_, J_, K_, F_, S_, T_, M_

where I, J, and K specify the arc center of circular tool motion, usually provided with algebraic signs.

➢ An example of a word address NC code is as follows:

N040 G00 X0.0 Y0.0 Z300.0 T01 M06

Omitted words are assumed to be zero or to be the same as the value previously defined.

- There are some restrictions to CNC blocks.
  - Each block may contain only one tool move.
  - Each block may contain any number of non-tool G-codes moves, provided they do not conflict with each other.
  - Each block may contain only one feedrate per block.
  - Each block may contain only one specified tool or spindle speed.
  - The block numbers should be sequential.
  - Both the program start flag and the program number must be independent of all other commands.
  - The data within a block should follow the standard sequence, N-block number, G-code, any coordinates, and other required functions.
  - Each block may contain only one M-code per block.

Preparatory commands (G codes):

- G-codes are preparatory functions that involve actual tool moves (for example, control of the machine). These include rapid moves, feed moves, radial feed moves, dwells, and roughing and profiling cycles.

- Most G-codes described here are **modal**, meaning that they remain active until canceled by another G-code.

- G codes can be classified into different categories as follows:

  - **Group 1: Motion commands**

    G00: Rapid positioning
    G01: Linear interpolation
    G02: Circular interpolation – clockwise (CW)
    G03: Circular interpolation – counterclockwise (CCW)

- **Group 3: Unit selection**
  - G20: Measurement in inches
  - G21: Measurement in mm

- **Group 4: Positioning system selection**
  - G90: Absolute format
  - G91: Incremental format

- **Group 5: Fixed or canned cycles** - The purpose of a fixed cycle is to execute a series of repetitive machining operations with a single block command.

  - **Turning**
    - G70: Profile finishing cycle
    - G71: Profile roughing cycle for turning and boring
    - G74: Peck drilling cycle along the *z* axis

- **Milling**

  G81: Drilling cycle

▪ **Group 6: Work coordinate setting (for milling and drilling only)**

G98: Return to the initial plane

Miscellaneous commands (M codes):

- M-codes are miscellaneous functions that include actions necessary for machining but not those that are actual tool movements. That is, they are auxiliary functions, such as spindle on and off, tool changes, coolant on and off, program stops, and similar related functions.

- M codes can be classified into different categories as follows:
  - **Group 1**
    - M30: Program end, reset to start
  - **Group 2**
    - M03: Spindle rotation normal – clockwise (CW)
    - M04: Spindle rotation reverse – counterclockwise (CCW)
    - M05: Spindle rotation stop

- **Group 3**
  M06: Tool change

- **Group 4**
  M08: Coolant on
  M09: Coolant off