446.326A CAD/CAM

NC (Numerical Control)

October 22, 2008

Prof. Sung-Hoon Ahn

School of Mechanical and Aerospace Engineering
Seoul National University

Introduction to numerical control machines

- **NC Lathe**

- **NC machining center**

Lathe (turning)

Rate of removal

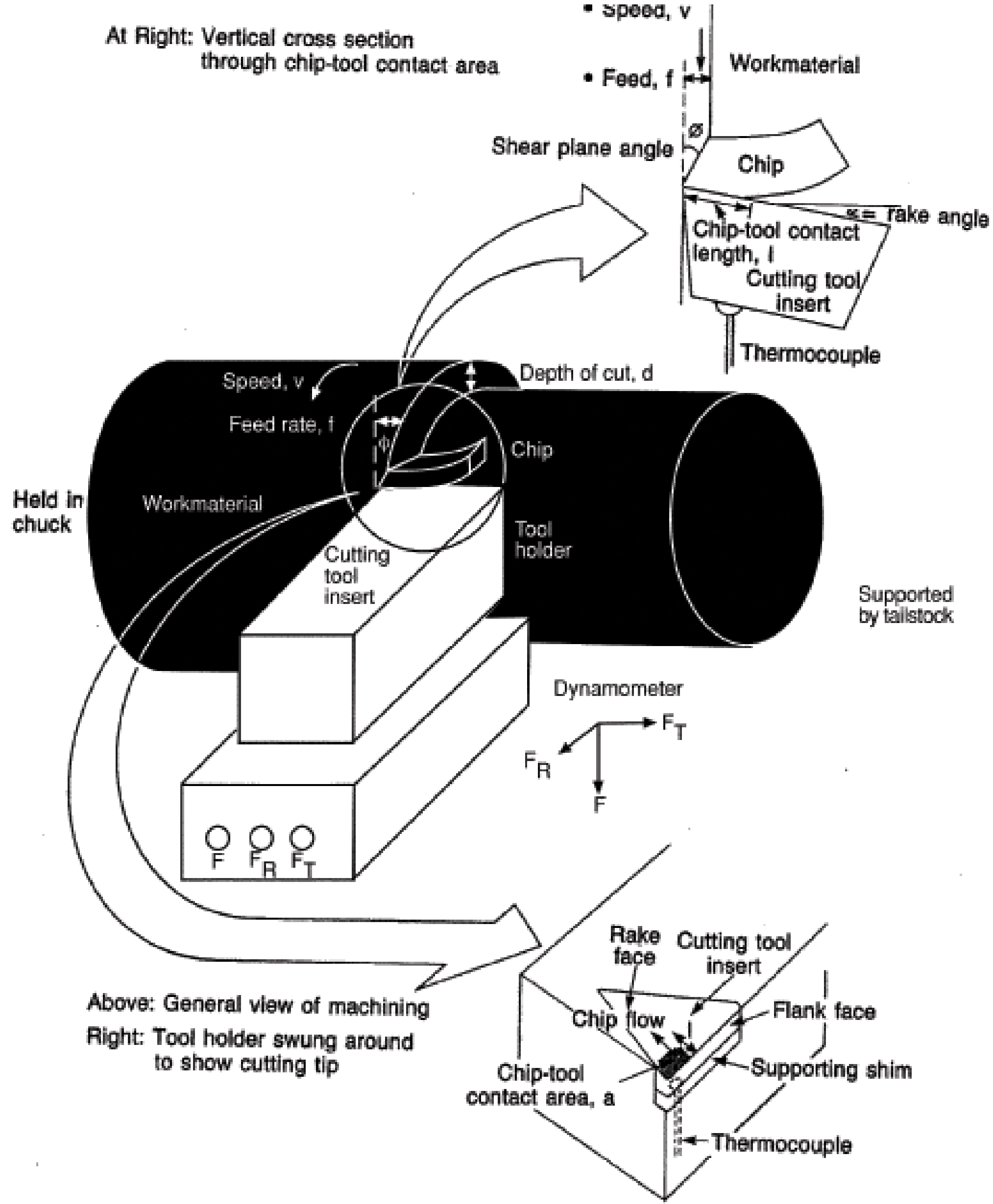
$= V f w$

Where

V: cutting speed(m/min)

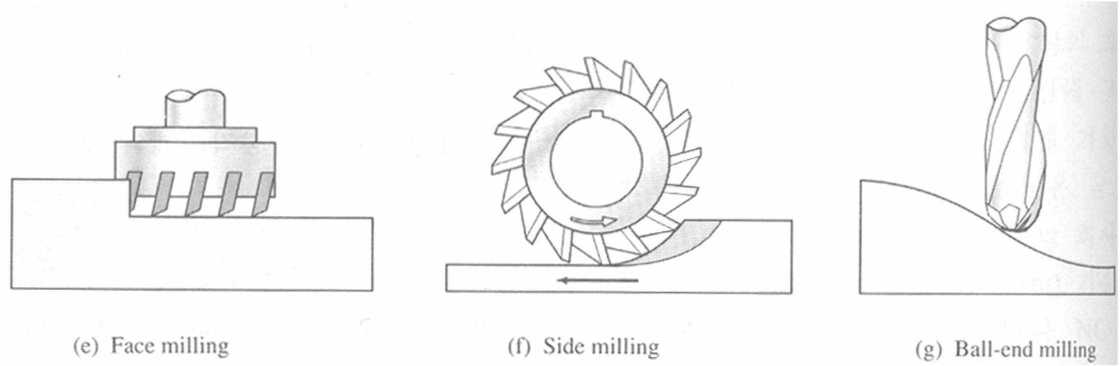
f: feed (mm/rev)

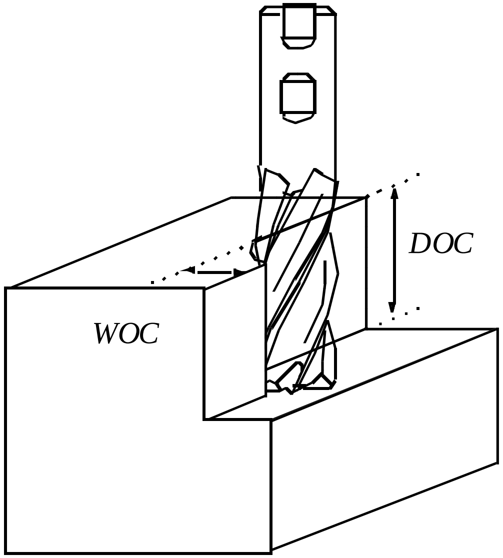
W: depth of cut (mm)

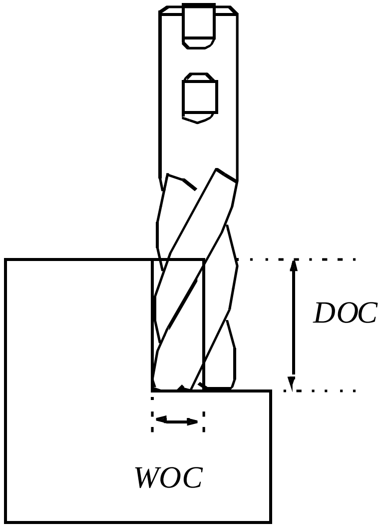


Milling

- **Face milling**
- **Side milling**
- **End milling**
  - Flat
  - Ball-nose







Depth of Cut (DOC)

Width of Cut (WOC)

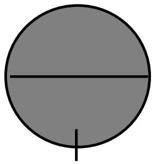
Milling

- **Cutting speed (m/min)**

  V = p D m

  Where D = Diameter of cutter (m)

  m = Revolution per minute (rpm)



Cutting Tool

- **Material Removal Rate (MRR)**

  MRR = WOC * DOC * f

  f = feed rate (mm/min) = n * m * t

- **Example**

  V = 50 m/min, t = 0.1 mm/tooth, number of tooth (n)= 2,

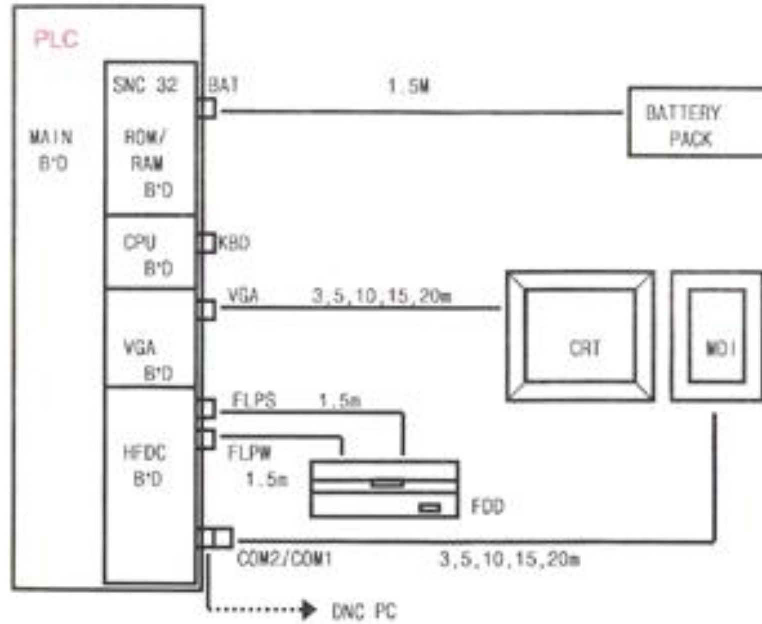
  D = 4 mm, DOC = 0.2, WOC = 3

  Cutter RPM (m) = 50000/(p x 4) = 3979

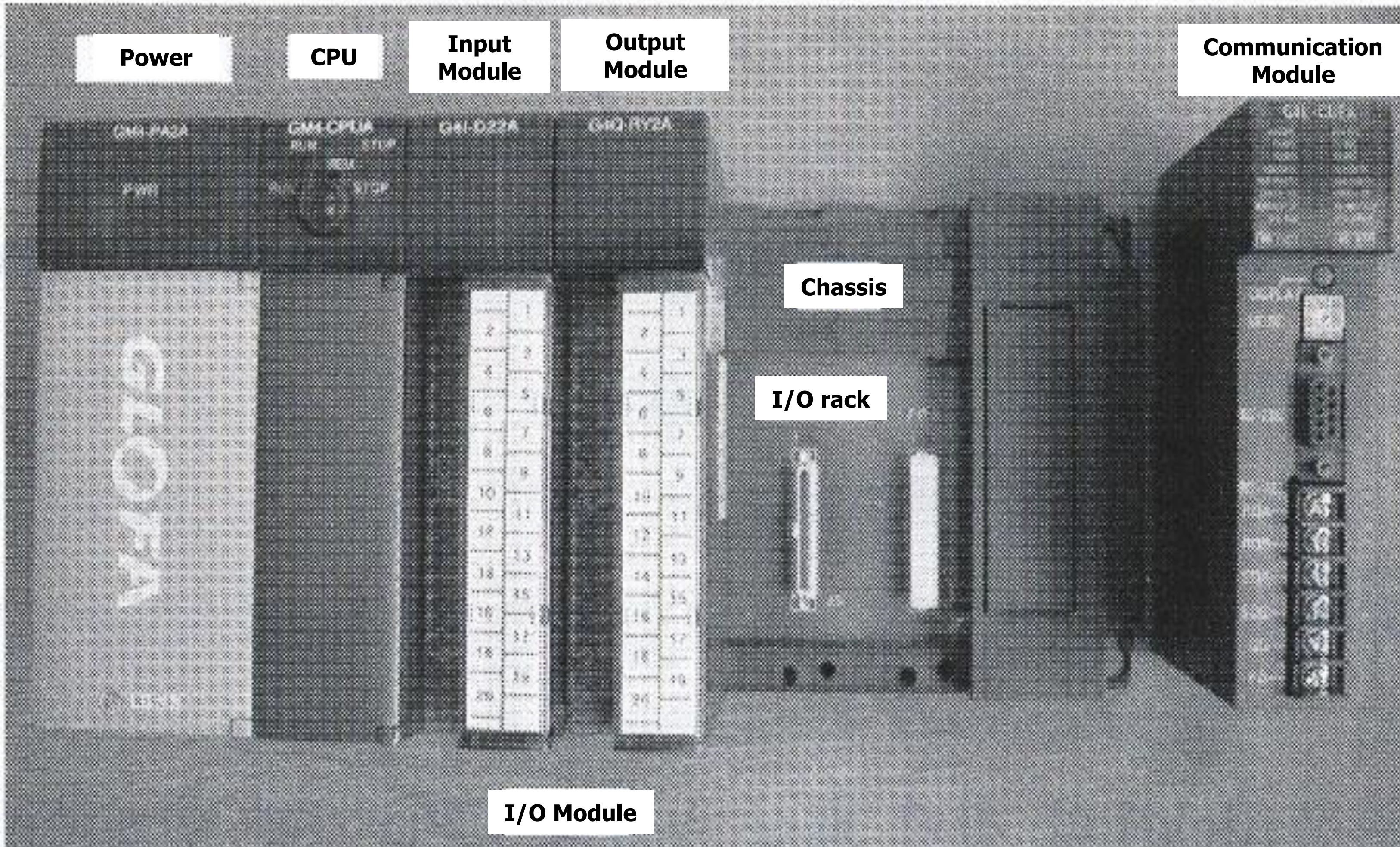
  f = 2 *3979 * 0.1 = 796 mm/min

  MRR = 3* 0.2 * 796 = 4776 mm^3/mi

CNC machining center

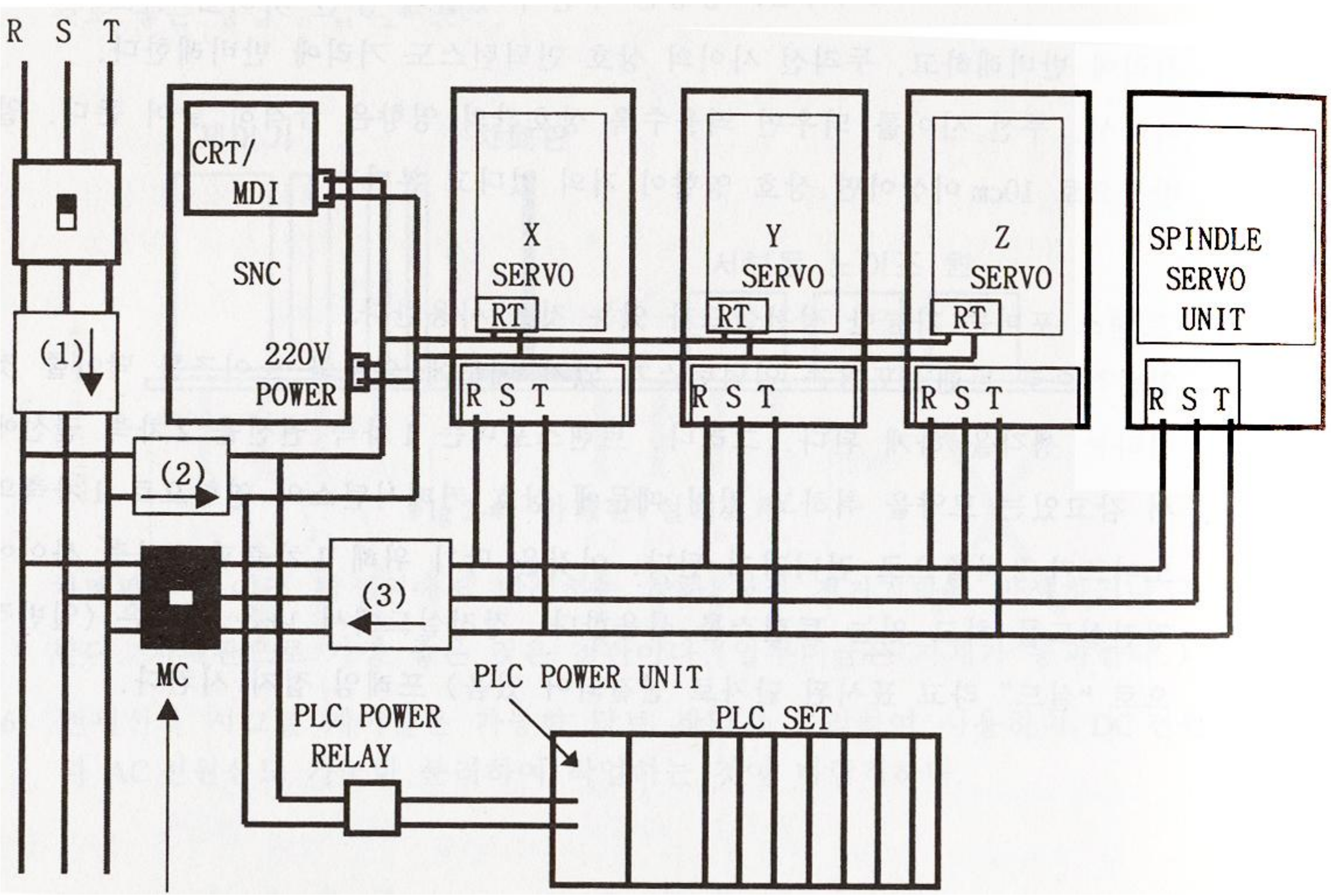


PLC (programmable logic controller)

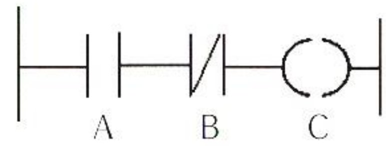
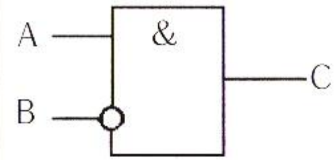
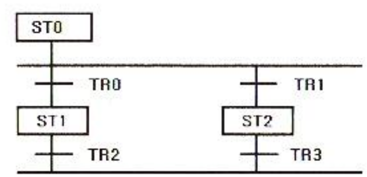


Control by PLC

- **Typical control unit of CNC machine**

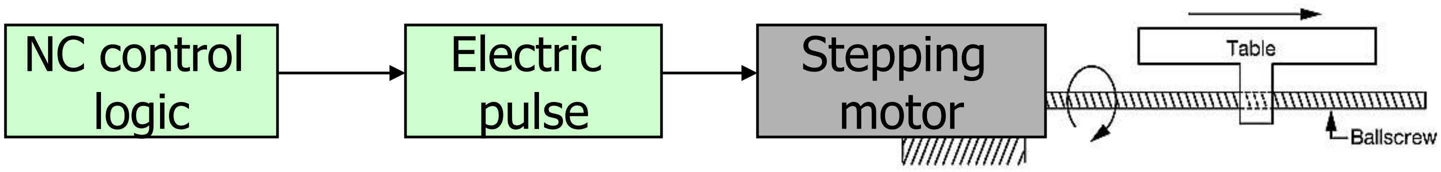


Low level programming

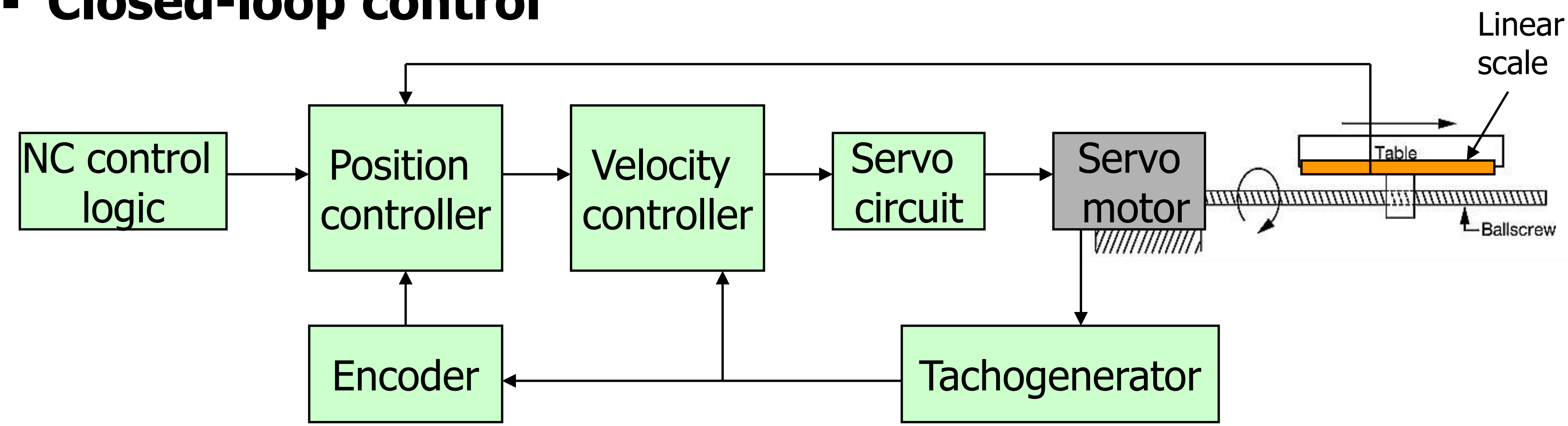
Language	Type	Example
Graphic Language	Ladder Diagram	
	Function Block Diagram	
	Sequential Function Chart	
Text Language	Instruction List	LD A ANDN B ST C
	Structured Text	c:= A AND NOT B

Control Mechanisms

- **Open-loop control**

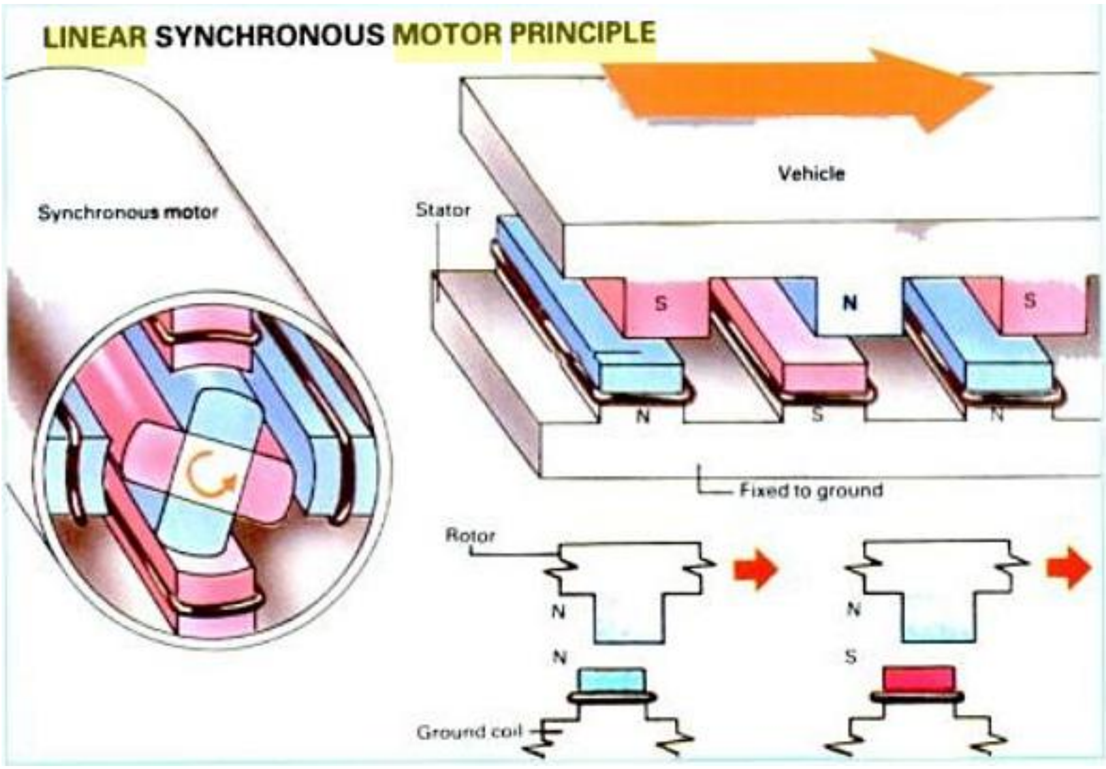


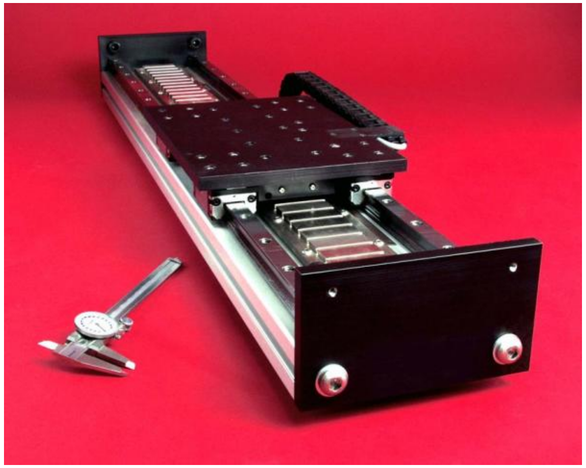
- **Closed-loop control**



Linear stage

<Rotary motor > < Linear motor >

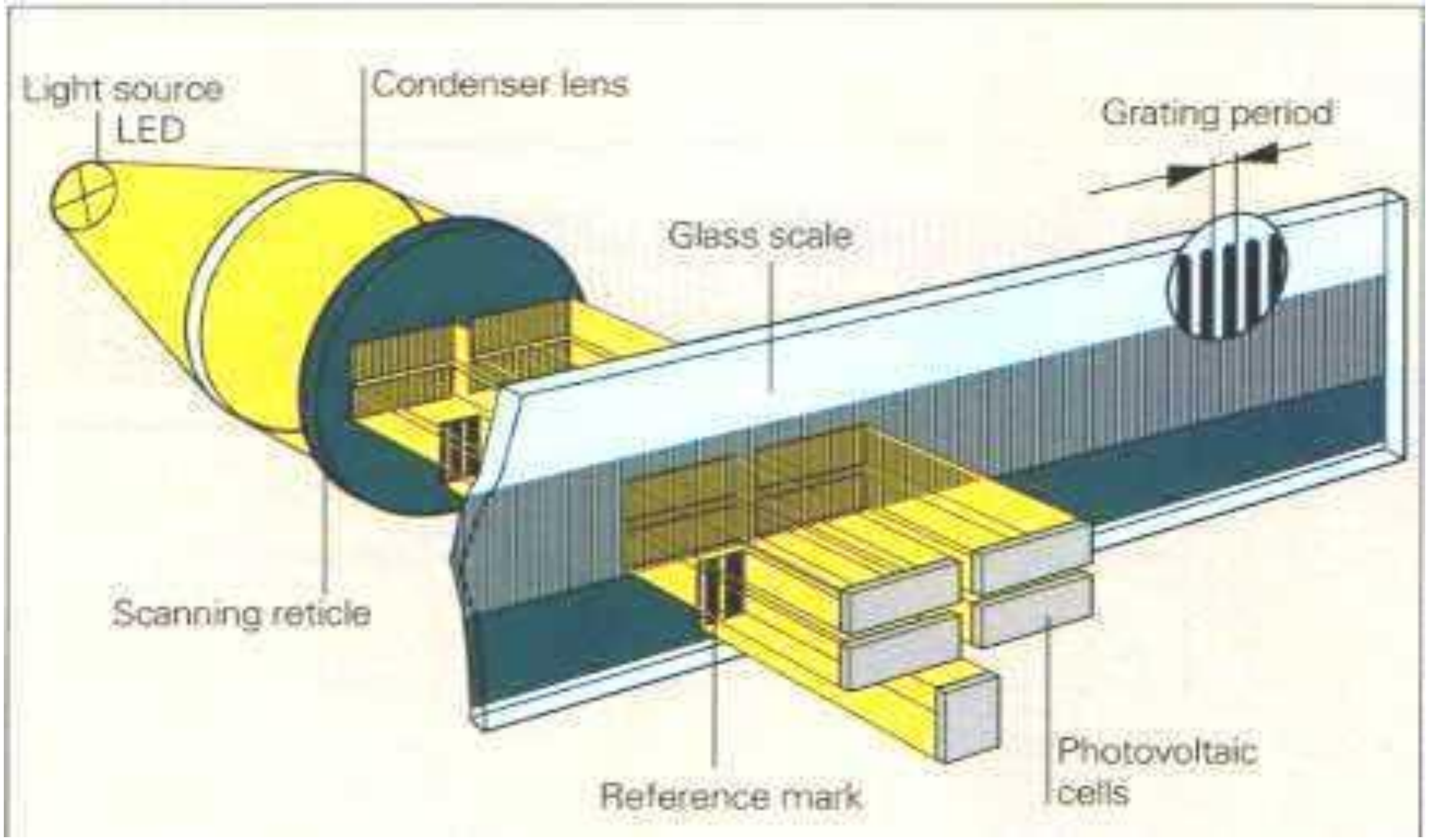




< linear stage (XY) >

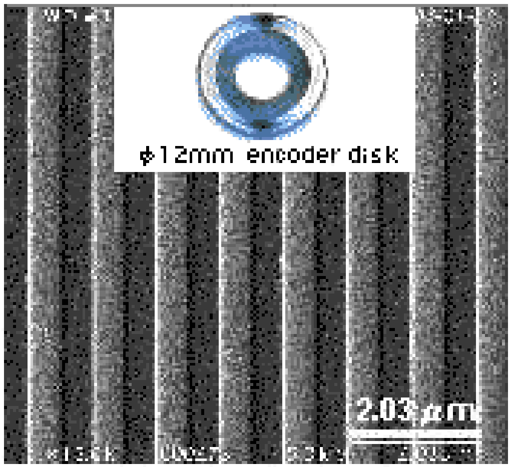
Control Mechanisms II

- **Linear scale**

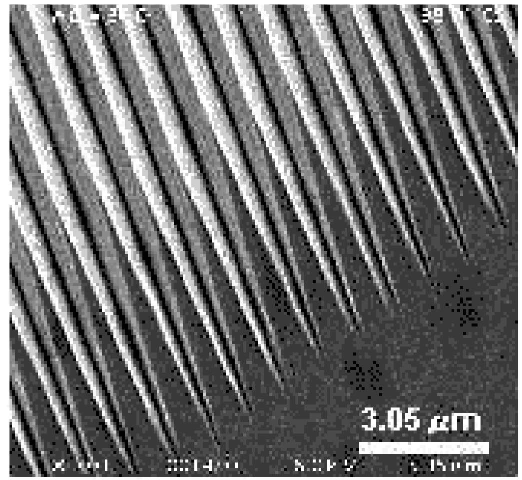


FANUC ROBOnano U*i*

Diffractive grating machined radially on the diameter 12 mm disk, Ra < 1 nanometer



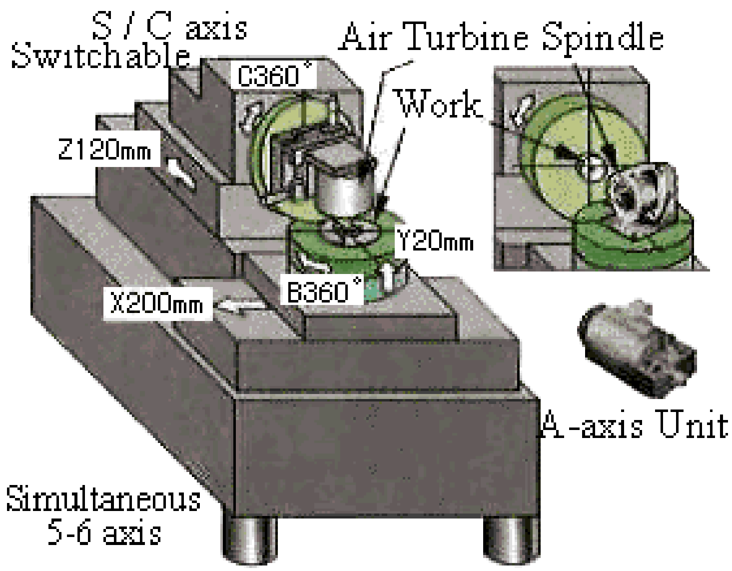
1 μm V groove grating



Edge of line “no micro bur”

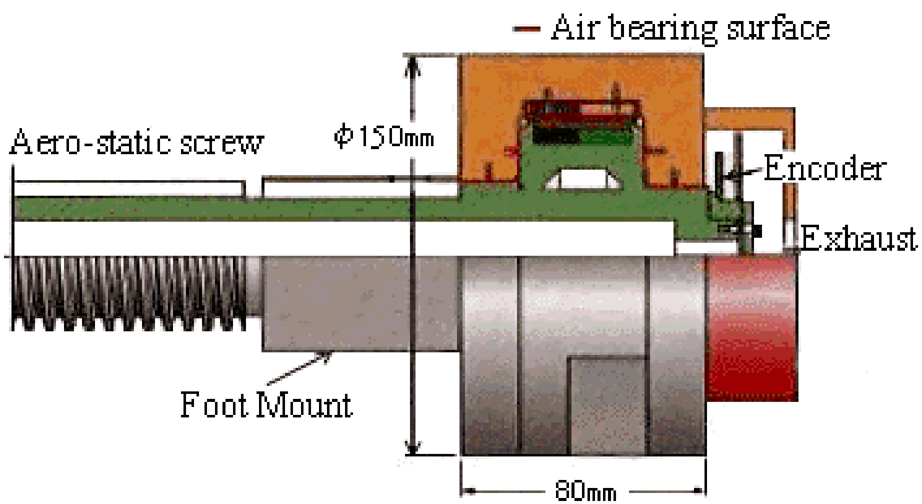
Diameter 1mm NOU mask

Cut by rotating diamond tool

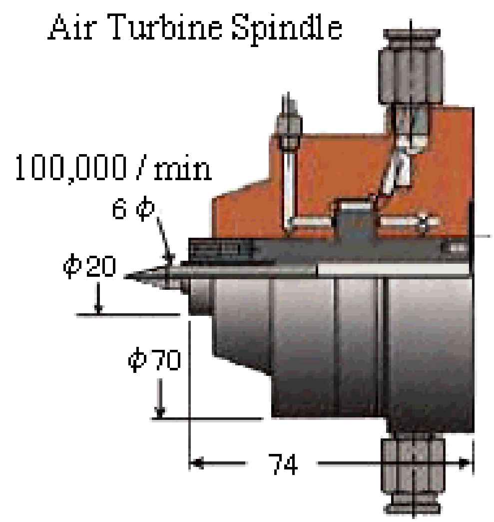


- Resolution X,Y,Z: 1nm A,B,C: 1/100,000deg.
- Building Block Structure with Super Precision Units
- Column-less 5 axes machine with turning function

Non-Friction Servo motor



Air turbine spindle



Low Amplitude Sine Tracking test
(+/-25 nm) by the high resolution pulse coder

Numerical Control

- **Use of coded numerical information in the automatic control of equipment positioning**

Part program → Control system to the machine tool

Production
Step of a part

Convert the statement into signals
that drives the machine tool

- Motion of the cutting tool
- Movement of the part being formed
- Changing cutting tools

Numerical Control (cont.)

1. **Manual part programming**

2. **Computer-assisted part programming**

3. **Part programming directly from CAD database**
   - CATIA NC
   - Pro/engineer NC
   - Commercial CAM software

Part program

- **A sequence of blocks** ← Line of words

< Example of part programming >

Words

1. **N code**
   - Line number
     ex. N001 O1234 → first line of the program and O1234 is the program number (usually the program # is located)

2. **G code**
   - Prepare the controller for a given operation
     ex. G00 X10.0 → move to positive X-direction by 10.0mm
     G00: point to point, positioning (use with combination point-to-point/contouring systems for indicating positioning operation)

3. **Dimension words (X, Y, Z, A, and B words)**
   - Location and axis orientation of a cutter A, B are for machine with more than 3 axis
     ex. Y + 500 → if the unit BLU (Basic Length Unit) is 0.001 inch,
     it means 0.5 inch moving from Y location

Words (cont.)

4. F code (feed command)

- Cutter feed rate (ipm: inch per minute)

  ex. F2.0 → move 2 inches per minute

5. S code

- Specify spindle speed

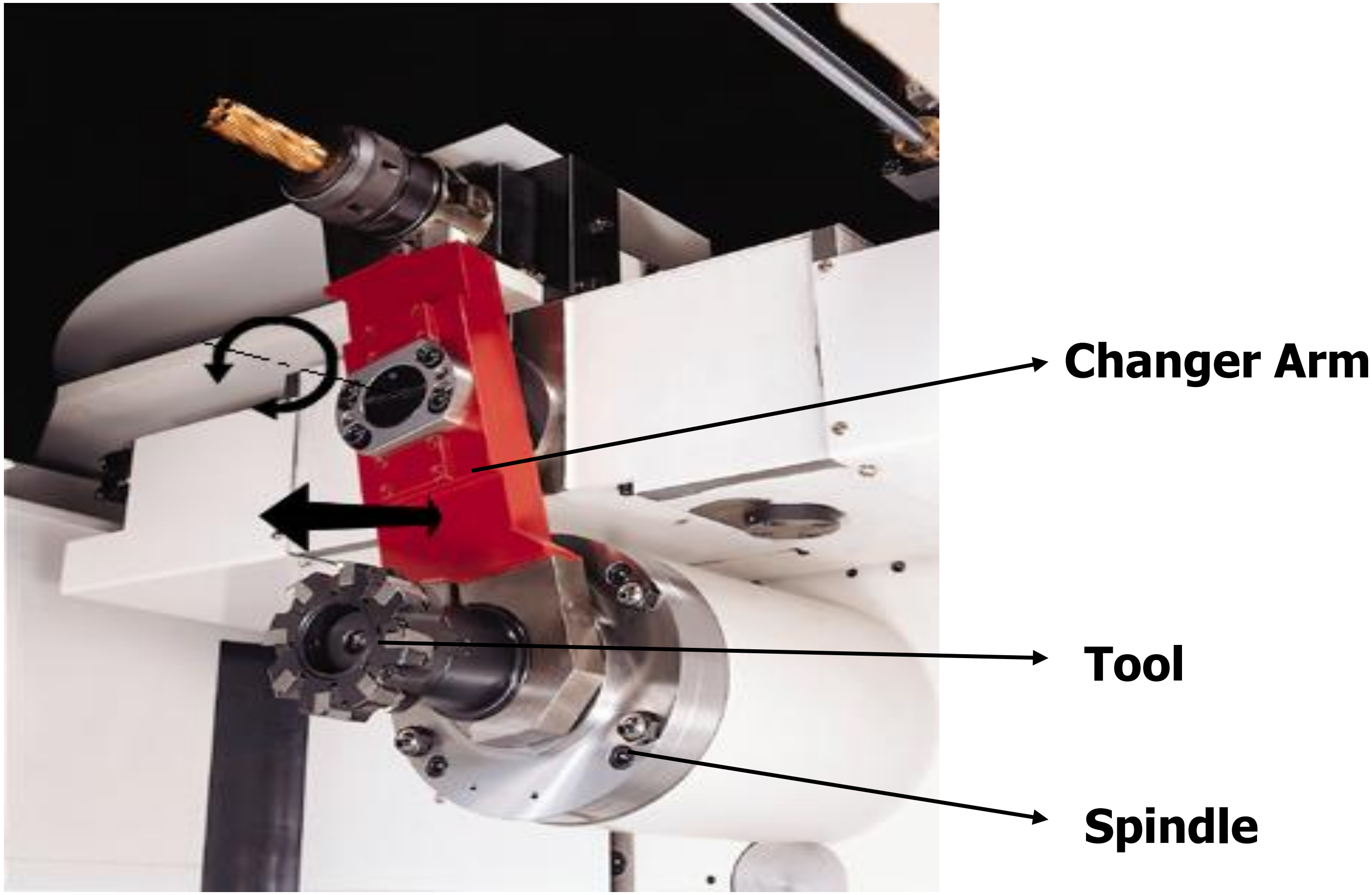
  ex. S5000 → Spindle speed is specified by 5000rpm

6. T code

- Tool selection command
- Used when the machine is equipped with a tool turret

  ex. T1 → call the tool # 1 in the tool turret

Automatic Tool Changer (ATC)



Words (cont.)

7. M code

- Miscellaneous commands
- Coolant supply, spindle on/off, etc.

  ex. M06 → tool change, executes the change of a tool (tools) manually or automatically, not to include tool selection

Words (cont.)

Code	Function	Explanation
g00	Point to point, positioning	Use with combination point-to-point/contouring systems for indicating positioning operation
g01	Linear interpolation (normal dimensions)	A mode of contouring control used for generating a slope or straight cut, where the incremental dimensions are normal, i.e., input resolution is as specified.
g02	Circular interpolation arc CW (normal dimensions)	A mode of contouring control which produces an arc of a circle by the coordinated motion of two axes. The curvature of the path (clockwise = g02, or counterclockwise = g03) is determined when viewing the plane of motion in the negative direction of the perpendicular axis. The distances to the arc center (i, j, k) are “normal dimensions”
g03	Circular interpolation arc CCW (normal dimensions)	
g04	Dwell	A programmed (or established) time delay, during which there is no machine motion. Its duration is adjusted elsewhere, usually by the f word. In this case dimension words should be set at zero

Source: Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.

Words (cont.)

Code	Function	Explanation
g05	Hold	Machine motion stopped until terminated by an operator or interlock action.
g06	Parabolic interpolation (normal dimensions)	A mode of contouring control which uses the information contained in successive blocks to produce a segment of a parabola.
g08	Acceleration	The feedrate (axes' velocity) increases smoothly (usually exponentially) to the programmed rate, which is noted later in the same block.
g09	Deceleration	The feedrate decreases (usually exponentially) to a fixed percent of the programmed feedrate in the deceleration block.
g010	Linear interpolation (long dimensions = LD)	Similar to g01, except that all dimensions are multiplied by 10. For example, a programmed dimension of 9874 will produce a travel of 98740 basic length-units. (Used only with incremental programming.)
g011	Linear interpolation (short dimensions = SD)	As g01, but dividing all dimensions by 10, e.g., 987 units for the example above.

Source: Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.

Words (cont.)

Code	Function	Explanation
g13 g14 g15 g16	Axis selection	Used to direct the control system to operate on a specific axis or axes, as in a system in which controls are not to operate simultaneously.
g17 g18 g19	XY Plane selection ZX Plane selection YZ Plane selection	Used to identify the plane for such functions as circular interpolation or cutter compensation.
g20	Circular interpolation arc CW (LD)	As g02 with long dimension distances.
g21	Circular interpolation arc CW (SD)	As g02 with short dimension distances.
g30	Circular interpolation arc CCW (LD)	As g03 with long dimension distances.
g31	Circular interpolation arc CCW (SD)	As g03 with short dimension distances.
g33	Thread cutting, constant lead	A mode selected for machines equipped for thread cutting.
g34	Thread cutting, increasing lead	As g33, but when a constantly increasing lead is required.
g35	Thread cutting, decreasing lead	As g33, but to designate a constantly decreasing lead.

Source: Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.

Words (cont.)

Code	Function	Explanation
g40	Cutter compensation - cancel	Command which will discontinue any cutter compensation.
g41	Cutter compensation - left	Displacement, normal to cutter path, when the cutter is on the left side of the work surface, looking in the direction of cutter motion.
g42	Cutter compensation - right	Compensation when cutter on right side of work surface.
g43 through g49	Cutter compensation if used; otherwise unassigned.	Compensation (g40-g49) is used to adjust for difference between actual and programmed cutter radii or diameters.
g60 through g79	Reserved for positioning only	Reserved for point-to-point systems.
g80	Fixed cycle cancel	Command which will discontinue only fixed cycle.
g81 through g89	Fixed cycle #1 through #9, respectively.	A preset series of operations which direct the machine to complete such action as drilling or boring.

Source: Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.

Words (cont.)

Code	Function	Explanation
g90	Absolute dimension programming	A control mode in which the data input is in the form of absolute dimensions. Used with combination absolute.incremental systems.
g91	Incremental dimension programming	A control mode in which the data input is in the form of incremental dimension.

Source: Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.

Words (cont.)

Code	Function	Explanation
m00	Program stop	Stops spindle, coolant, and feed after completion of the block commands. It is necessary to push a button in order to continue the program.
m01	Optional (planned) stop	Similar to m00, but is performed only when the operator has previously pushed a button, otherwise the command is ignored.
m02	End of program	Indicates completion of the workpiece. It stops spindle, coolant, and feed after completion of all instructions in the block. May include rewinding of tape.
m03	Spindle CW	Starts spindle rotation in a clockwise direction.
m04	Spindle CCW	Starts spindle rotation in a counterclockwise direction.
m05	Spindle off	Stop spindle; coolant turned off.
m06	Tool change	Executes the change of a tool (tools) manually or automatically, not to include tool selection.
m07	Coolant no. 2 on	Turns a flood coolant on.
m08	Coolant no. 1 on	Turns a mist coolant on.
m09	Coolant off	Automatically shuts the coolant off.
m10	Clamp	Automatically clamps the machine slides, workpiece, fixture, spindle, etc. (as specified by the producer).

Unassigned: m12, m17, m18, m20 to m29, m36 to m39, m46 to m99.

Source: Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.

Words (cont.)

Code	Function	Explanation
m11	Unclamp	Unclamping command.
m13 m14	Spindle CW & coolant on Spindle CCW & coolant on →	Combines spindle rotation and coolant on in the same command.
m15 m16	Motion + Motion - →	Rapid traverse or feedrate motion in either the plus or minus direction
m19	Oriented spindle stop	Causes the spindle to stop at a predetermined angular position.
m30	End of tape	Similar to m02 except that it must include rewinding of tape to the rewind-stop character, thus ready for next workpiece
m31	Interlock bypass	Temporarily circumvents normal interlock.
m32 through m35	Constant cutting speed	The control maintains a constant cutting speed by adjusting the rotation speed of the workpiece inversely proportional to the distance of the tool from the center of rotation. Normally used with turning.
m40 through m45	Gear changes if used; otherwise unassigned	

Unassigned: m12, m17, m18, m20 to m29, m36 to m39, m46 to m99.

Source: Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.

Words (cont.)

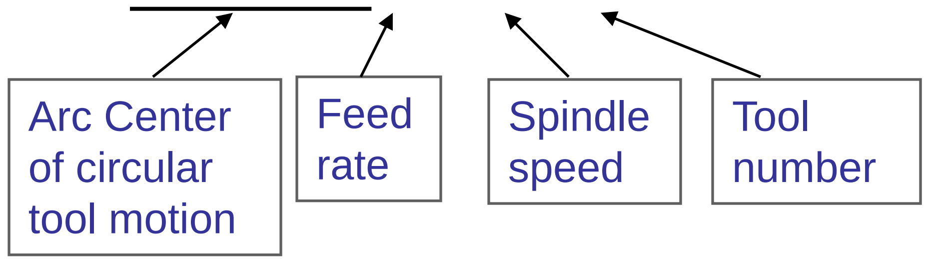
- **Fixed sequential format**
  - Each block has the same length and contains the same number of characters

- **Block address format**
  - Use change code for avoiding redundant information

- **Tab sequential format**
  - Variable length of each block
  - Insert tab key between words, EOB at the end of block
  - Omit repeated words

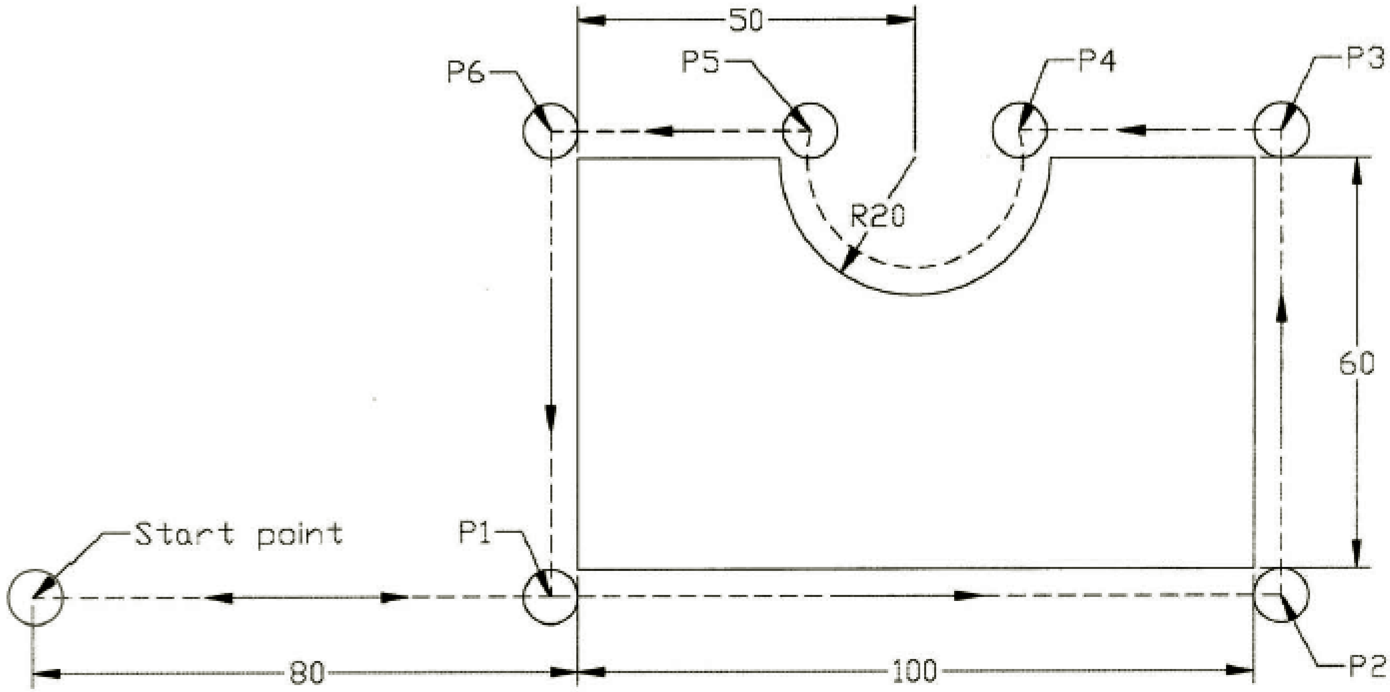
Words (cont.)

- **Word address format**
  - Used by most CNC controllers
  - N_, G_, X_, Y_, Z_, I_, J_, K_, F_, S_, T_, M_



  - N040 G00 X0 Y0 Z300 T01 M06
  - Omitted words are assumed to zero or to be the same as the value previously defined

Manual part programming example



Manual part programming example (cont).

- Dimension in mm
- Thickness of the plate 15 mm
- Bottom face z = 0
- BLU = 0.01 mm

- Constant machining feedrate of 350 mm/min is used
  Rapid traverse feedrate is 950 mm/min
- Spindle speed is 1740 rpm -> 717 magic-three code

Answer of Example

- **A cutter of 10 mm diameter is selected for this job.**
- **The cutter is initially located at the start point.**
- **We have to go through the following blocks to have the tool move along the dashed lines and arc in the direction of the arrows.**

Answer of Example (cont.)

1. Set a mode such that the coordinates are provided in the form of incremental dimension instead of absolute dimension.

   N001 G91 EOB

2. Select metric unit.

   N002 G71 EOB

3. Load the tool of diameter 10 mm above the start point by 40 mm.

   N003 G00 X0.0 Y0.0 Z040.0 T01 M06 EOB

   Note that we did not use BLU in this example

Answer of Example (cont.)

4. To move from the start point toward point P_1, two blocks given below are programmed. At the first block, the system will accelerate to the traverse feedrate of 950 mm/min. At the second block the tool approaches P_1 with the machining feedrate of 350 mm/min. At the end of these two blocks, the center of the cutter will be located at point P_1. We have to program Z dimension as well to bring the cutter down to its appropriate place.

   N004 G01 X65.0 Y0.0 Z-40.0 F950 S717 M03 EOB

   N005 G01 X10.0 F350 M08 EOB

   The command M03 starts spindle rotation while M08 starts coolant.

Answer of Example (cont.)

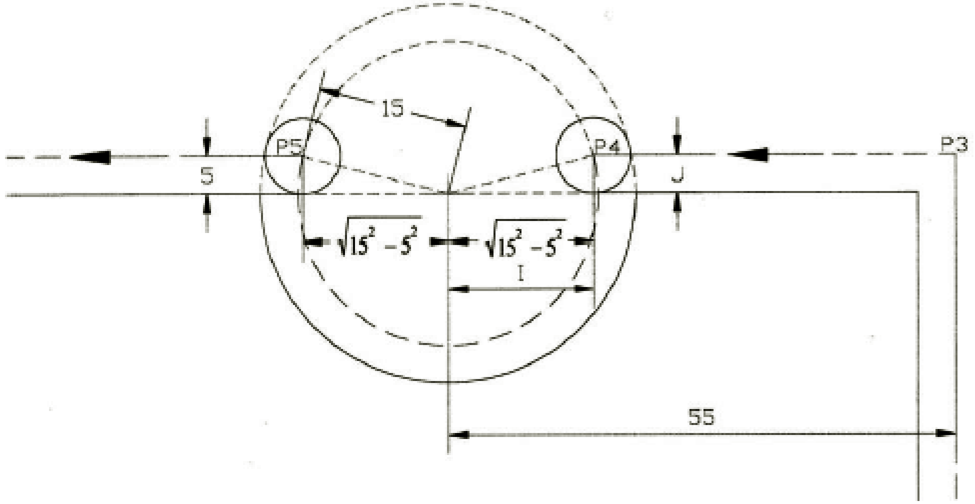
5. The following blocks will move the tool from P_1 to P_3 through P_2.

   N006 G01 X110.0 EOB

   N007 G01 Y70.0 EOB

Answer of Example (cont.)

6. The location of P_4 and P_5 are calculated using the following relations. Denoting their x and y coordinates by (X_4, Y_4) and (X_5, Y_5) respectively, the following relations are derived.



Answer of Example (cont.)

$$X_4 - X_3 = -\left(55 - \sqrt{15^2 - 5^2}\right) = -40.86$$

$$Y_4 - Y_3 = 0$$

$$X_5 - X_4 = -2\sqrt{15^2 - 5^2} = -28.28$$

$$Y_5 - Y_4 = 0$$

$$I = \sqrt{15^2 - 5^2} = 14.14$$

$$J = 5$$

Answer of Example (cont.)

Following blocks move the tool from P_3 to P_4 along a straight line, and from P_4 to P_5 along a circular arc in the clockwise direction.

N008 G01 X-40.86 EOB

N009 G02 X-28.28 Y0.0 I14.14 J5.0 EOB

In the second block, G02 activates the clockwise circular interpolation, X and Y words specify the end point of the circular arc (P_5 in this case) with respect to the starting point of the arc (P_4 in this case), and I and J specify the center of the arc with respect to the starting point.

Answer of Example (cont.)

7. $X_6 - X_5 = -\left(55 - \sqrt{15^2 - 5^2}\right) = -40.86$

$Y_6 - Y_5 = 0$

N010 G01 X-40.86 EOB

N011 G01 Y-70.0 EOB

N012 G01 X-75.0 Y0.0 Z40.0 F950 M30

M30 will turn off the spindle and coolant and rewind the tape to the beginning of the program.

Computer-assisted part programming

- **Use of high-level programming languages to define the part geometry and tool motion**
  - Define the geometry of the part
  - Instruct the cutting tool to machine along geometric elements
  - Offset is calculated automatically

Process1_Manufacturing_Program_1 - 메모장

파일(F) 편집(E) 서식(O) 보기(V) 도움말(H)

```
$$ -----------------------------------------------------------------
$$     Generated on 2005년 10월 13일 목요일 오후 6:58:41
$$     CATIA APT VERSION 1.0
$$ -----------------------------------------------------------------
$$ Manufacturing Program.1
$$  Part Operation.1

$$  Start generation of : Profile Contouring.1
LOADTL/2,1
FEDRAT/  300.0000,MMPM
SPINDL/   70.0000,RPM,CLW
GOTO  /  240.00000,  -62.50000,    0.00000
GOTO  /  240.00000,  -62.50000,  -10.00000
GOTO  /  240.00000,  -52.50000,  -10.00000
GOTO  /  239.78035,  -50.41561,  -10.00000
GOTO  /  239.15341,  -48.47321,  -10.00000
GOTO  /  237.67928,  -46.09463,  -10.00000
GOTO  /  236.23903,  -44.68498,  -10.00000
GOTO  /  234.54539,  -43.59273,  -10.00000
```

< Example of part programming >

Computer-Assisted Part Programming (cont.)

source — (geometry, cutter motions, machine instructions are coded in using a language like APT)

↓ compile — Automatically Programmed Tools

CL data — Cutter moves GOTO, GODLTA spindle, coolant, feedrate

↓ post-processor

Machine control data — (same as NC blocks in manual part programming)

APT

- **Automatically Programmed Tools**
- **Developed at M.I.T. in 1956**
- **Program statements**
  - Identification statements
    - Specify part name and specific post process
  - Geometry statements
    - Define part geometry
  - Motion statements
    - Define motions of the cutting tool with respect to the part geometry
  - Post-processor statements
    - Specify machining parameters such as feed, spindle speed
  - Auxiliary statements
    - Specify auxiliary machine-too functions

APT – Geometry Statement

$p1$ = POINT/x, y, z

$p2$ = POINT/$l1$, $l2$; intersection of two lines that are already defined

$p3$ = POINT/CENTER, $c1$; center of a circle

$p4$ = POINT/YLARGE, INTOF, $l1$, $c1$; intersection of a line and a circle, one with larger y coordinate

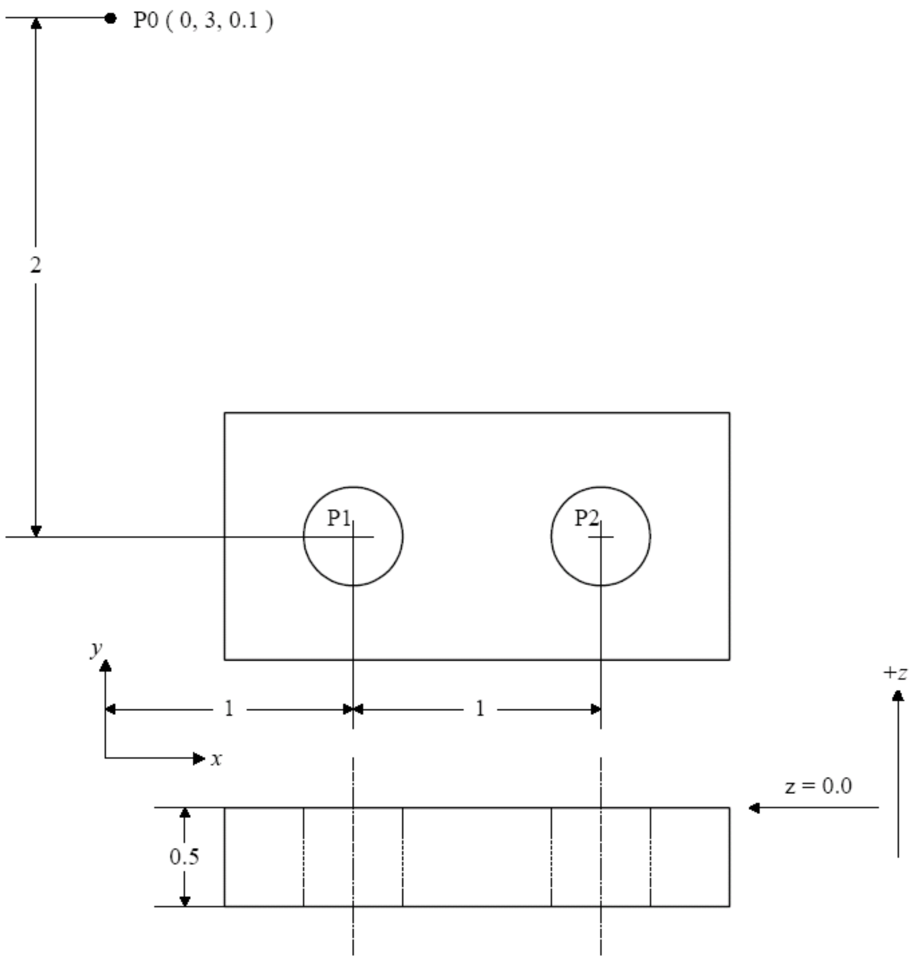
APT – Motion Statements

- **Motion statements**
  Two groups of motion statements are available
  - Point to point
  - Contouring operation

- **Point to point motion statements**
  Three motion statements exist for positing the tool at a desired point
  - FROM/point_location
  - GOTO/point_location
  - GODLTA/Δx, Δy, Δz

APT Example

- **Write an APT program**
  - To drill two holes of 0.2 in diameter on a plate
  - The home point P0 has z value of 0.1 to allow for clearance of the tool when it approaches the part.
  - The top surface of the part corresponds to z=0.
  - The center points of the holes will have the z value of 0.1.



APT – Answer of Example

p0 = POINT/0.0, 3.0, 0.1
p1 = POINT/1.0, 1.0, 0.1
p2 = POINT/2.0, 1.0, 0.1
FROM/*p0*
GOTO/*p1*
GODLTA/0, 0, -0.7
GODLTA/0, 0, 0.7
GOTO/*p2*
GODLTA/0, 0, -0.7
GODLTA/0, 0, 0.7
GOTO/*p0*

Part Programming from CAD Database

- **Use the geometry data in CAD database**
- **Defining geometry using a CAD system is easier (part with complicated curves & surfaces)**

1. **Part geometry important for machining are identified and isolated on a separate layer**
   - Additional geometry may be added to define boundaries for tool motion
   - Lathe operation -> 2D profile (2D drafting, projecting 3D geometry)
   - 2 or $2^1/_2$ axis milling, drilling -> 2.5D geometry
   - 3 or 5 axis contouring motion -> surface geometry

Part Programming from CAD Database (cont.)

2. **Define tool geometry**
   - Select from tool library
3. **Identify the desired sequence of machining operations**
   - Plan required tool path (home->home) with the proper cutting parameters
4. **X, Y, Z coordinator of the necessary points on the paths are calculated**
5. **Tool path is verified on the graphic display**
6. **CL (Cutter Location) data file is produced**
   - CL data file is post-processed to machine control data

Multi-spindle machine