446.326A CAD/CAM

Manual NC

October, 26, 2009

Professor Sung-Hoon Ahn
TA: Hyung-Jung Kim

School of Mechanical and Aerospace Engineering
Seoul National University

Contents

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 10.0 mm
     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 0.0254 mm, it means 12.7 mm moving from Y location

Words (cont.)

4. **F code (feed command)**
   - Cutter feed rate (mm/min, millimeters per minute)
     ex. F2.0 → move 2 mm per minute
5. **S code**
   - Specify spindle speed
     ex. S5000 → Spindle speed is specified by 5000 rpm
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

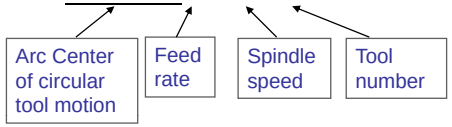
Words (cont.)

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.)

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



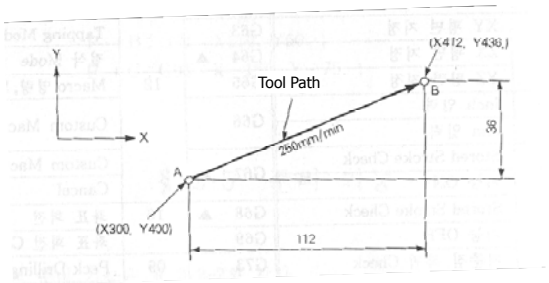
  - ex. N0010 G01 X07500 Y06250 S612
  - Omitted words are assumed to zero or to be the same as the value previously defined

```
N001 G92 G91 X0.0
Y0.0 Z20 ;
N002 G00 Z-8.0 ;
N003 G01 Z-20.0 F80 ;
N004 Y5.0 ;
N005 X10.0 ;
N006 G03 X12.5 Y12.5
R12.5 ;
N007 G02 X20 Y20
R30 ;
```

Basic preparatory commands

- **G Code**
  - G90 : Absolute dimension programming
  - G91 : Incremental dimension programming
  - G92 : Coordinate origin position setting

  - G00 : Point to point, positioning
  - G01 : Linear interpolation



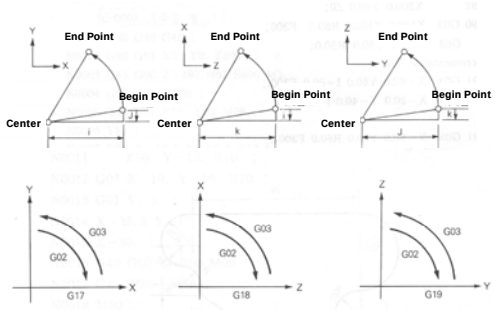
Absolute dimension
G90 G01 X412. Y436. F250;

Incremental dimension
G91 G01 X112. Y36. F250;

Basic preparatory commands (cont.)

- **G Code**
  - G17 : XY Plane selection
  - G18 : YZ Plane selection
  - G19 : ZX Plane selection

  - G02 : Circular interpolation arc CW
  - G03 : Circular interpolation arc CCW



Example

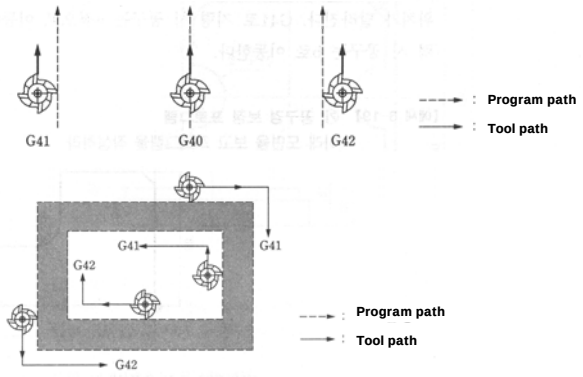
G17
G02 X-28.28 Y0.0 I14.14 J5.0

G18
G02 Y0.0 Z5.0 J5.0 K5.0

G19
G02 Z5.0 X10.0 K5.0 I5.0

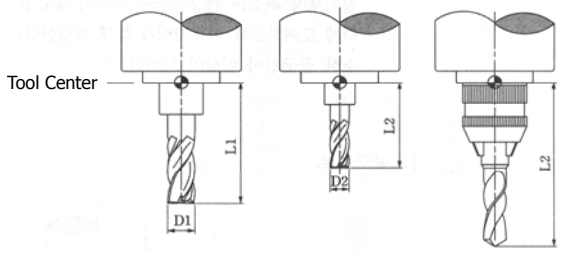
Basic preparatory commands (cont.)

- **G Code**
  - G40 : Cutter compensation for tool diameter – cancel
  - G41 : Cutter compensation for tool diameter – left
  - G42 : Cutter compensation for tool diameter – right



Basic preparatory commands (cont.)

- **G Code**
  - G43 : Cutter compensation for tool length – "+"
  - G44 : Cutter compensation for tool length – "-"
  - G49 : Cutter compensation for tool length – Cancel



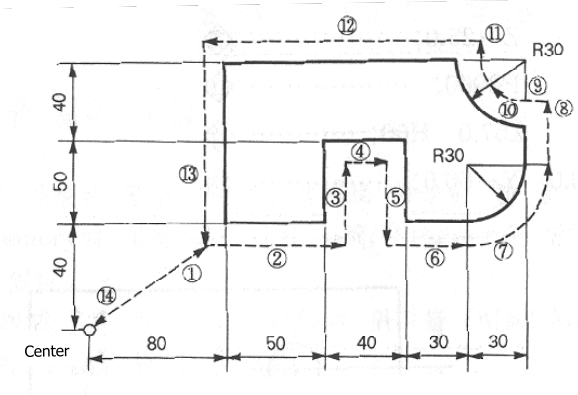
Basic miscellaneous commands

- **M Code**
  - M02 : End of program
  - M03 : Spindle CW
  - M04 : Spindle CCW
  - M05 : Spindle off
  - M30 : End of tape
    (Similar to M02, except that it is ready for the next work piece)

Manual NC Viewer

- **Gcode2000**
- **Download from the web class, and extract it on your computer**

Simple Example



Tool Diameter ϕ20

Tool position : X0.0 Y0.0 Z3.0
Feed 120
Depth : 5
① and ⑭ are not for machining

Simple Example (cont.)

- **Initialization**
  Absolute or Incremental dimension
  Set feed rate and start spindle

- **Incremental dimension programming**

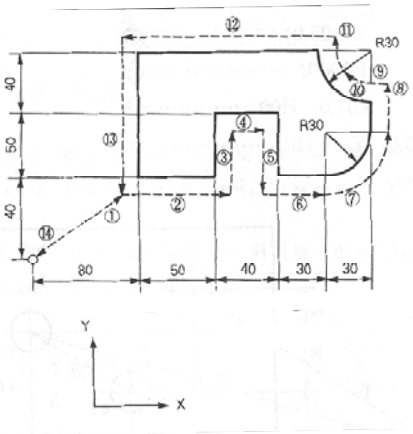
- **① Path**
  G92 G91 X0.0 Y0.0 Z3.0 ;
  G00 X70 Y30 ;
  M03;
  G01 Z-8.0 F120;

Simple Example (cont.)

- **②~ ⑥ Path**
  G01 X70.0;
  G01 Y50.0;
  G01 X20.0;
  G01 Y-50.0;
  G01 X40.0;

- **⑦ Path**
  G03 X40.0 Y40.0 I0.0 J40.0;
  (or G03 X40.0 Y40.0 R40.0;)

- **⑧~⑨ Path**
  G01 Y40.0;
  G01 X-10.0;

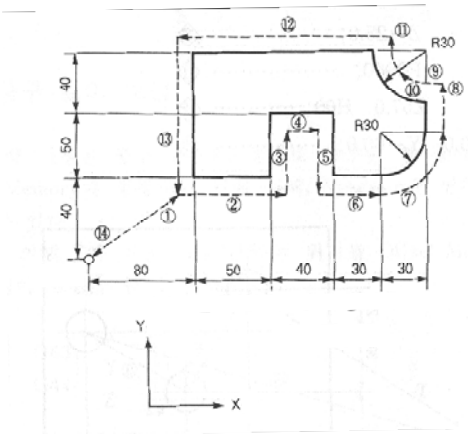


Simple Example (cont.)

- **⑩ Path**
  G02 X-20.0 Y20.0 I0.0 J20.0;
  (or G02 X-20.0 Y20.0 R20.0;)

- **⑪~⑬ Path**
  G01 Y10.0;
  G01 X-140.0;
  G01 Y-110.0;

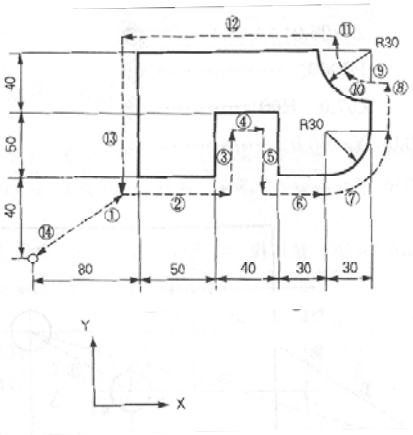
- **⑭ Path**
  G01 Z8.0;
  M05;
  G00 X-70.0 Y-30.0
  M30



Simple Example (cont.)

- **Absolute dimension programming**

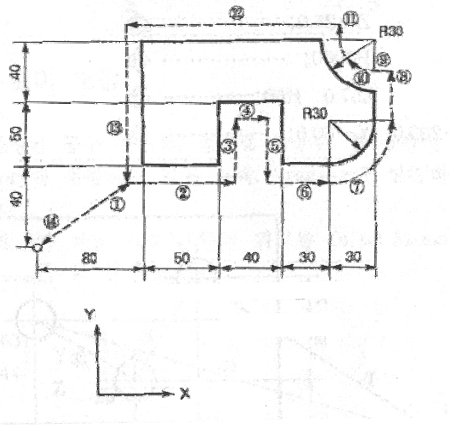
```
G92 G90 X0.0 Y0.0 Z3.0 ;
G00 X70 Y30 ;
M03;
G01 Z-5.0 F120;
G01 X140.0;
G01 Y80.0;
G01 X160.0;
G01 Y30.0;
G01 X200.0;
G03 X240.0 Y70.0 I200.0 J70.0;
G01 Y110.0;
G01 X230.0;
```



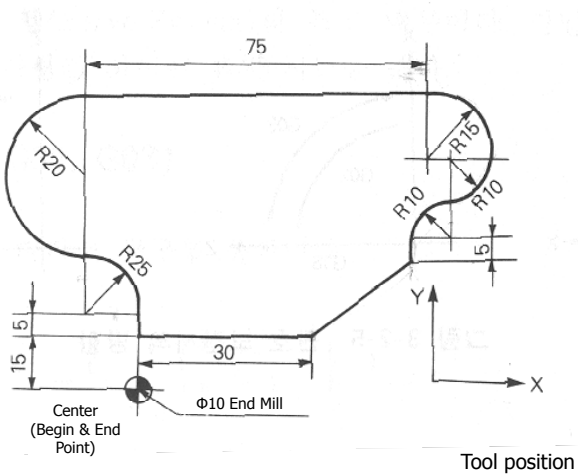
Simple Example (cont.)

- **Absolute dimension programming**

```
G02 X210.0 Y130.0 I230.0 J130.0;
G01 Y140.0;
G01 X70.0;
G01 Y30.0;
G01 Z3.0;
M05;
G00 X0.0 Y0.0
M30
```



Exercise problem 1

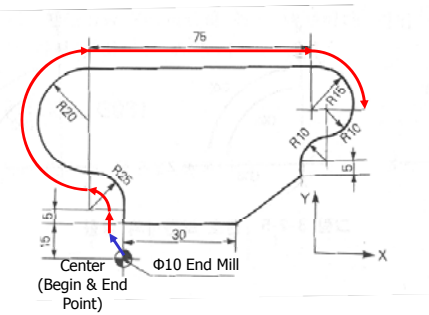


Tool position : X0.0 Y0.0 Z3.0
Feed 250
Depth : 3

Exercise 1 answer (I)

- **Incremental dimension programming**

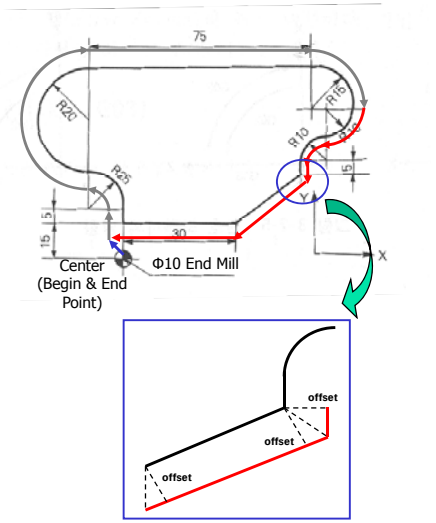
```
G92 G91 X0.0 Y0.0 Z3.0 ;
G00 X-5 Y10 ;
M03;
G01 Z-6.0 F250;
G01 Y10.0;
G03 X-20.0 Y20.0 I-20.0 J0.0;
G02 X0.0 Y50.0 I0.0 J25.0;
G01 X75.0;
G02 X20.0 Y-20.0 I0.0 J-20.0;
```



Exercise 1 answer (II)

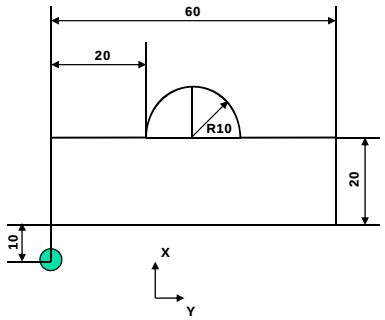
- **Incremental dimension programming**

```
G02 X-15.0 Y-15.0 I-15.0 J0.0 ;
G03 X-5.0 Y-5.0 I0.0 J-5.0 ;
G01 Y-3.0903;
G01 X-20.0  Y-31.9097
G01 X-35.0 ;
G01 Z6.0 ;
M05 ;
G00 X5 Y-10 ;
M30 ;
```



Exercise problem 2

- **Pocket machining**

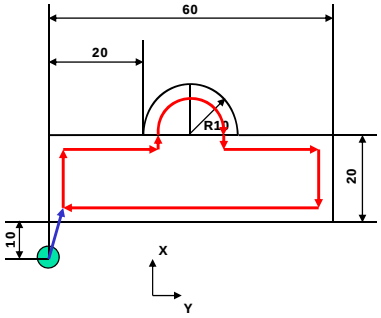


Tool position : X0.0 Y0.0 Z3.0
Feed 250
Depth : 3
Tool Diameter : 5

Exercise 2 answer (I)

- **Incremental dimension programming**

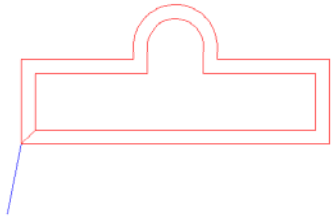
```
G92 G91 X0.0 Y0.0 Z3.0 ;
G00 X2.5 Y12.5 ;
M03;
G01 Z-6.0 F250;
G01 Y15.0;
G01 X20.0;
G01 Y2.5;
G02 X15.0 Y0.0 I7.5 J0.0;
G01 Y-2.5;
G01 X20.0;
G01 Y-15.0;
G01 X-55.0;
```



Exercise 2 answer (II)

- **Incremental dimension programming**

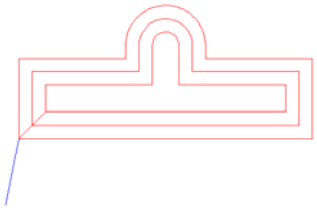
```
G01 X2.5 Y2.5;
G01 Y10.0;
G01 X20.0;
G01 Y5.0;
G02 X10.0 Y0.0 I5.0 J0.0;
G01 Y-5.0;
G01 X20.0;
G01 Y-10.0;
G01 X-50.0;
```



Exercise 2 answer (III)

- **Incremental dimension programming**

```
G01 X2.5 Y2.5;
G01 Y5.0;
G01 X20.0;
G01 Y7.5;
G02 X5.0 Y0.0 I2.5 J0.0;
G01 Y-7.5;
G01 X20.0;
G01 Y-5.0;
G01 X-45.0;
```



Exercise 2 answer (VI)

- **Incremental dimension programming**

```
G01 X2.5 Y2.5;
G01 X20.0;
G01 Y10.0;
G01 Y-10.0;
G01 X20.0;
G01 Z6.0;
M05 ;
G00 X-50.0 Y-20.0 ;
M30 ;
```