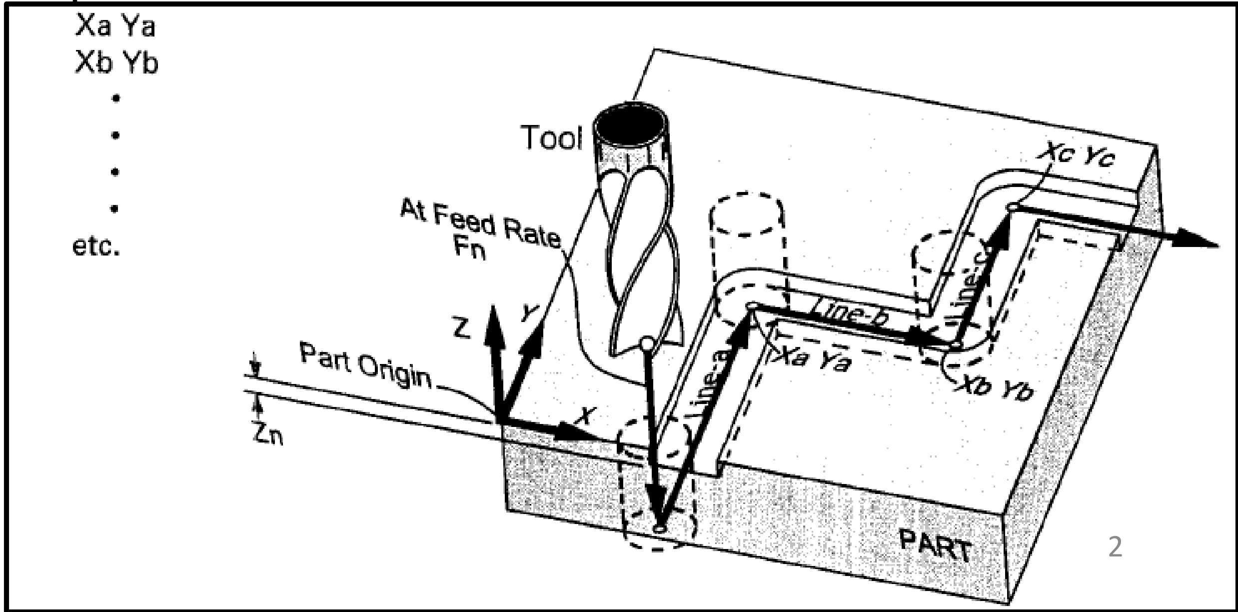
Linear Interpolation and Dwell Cycle

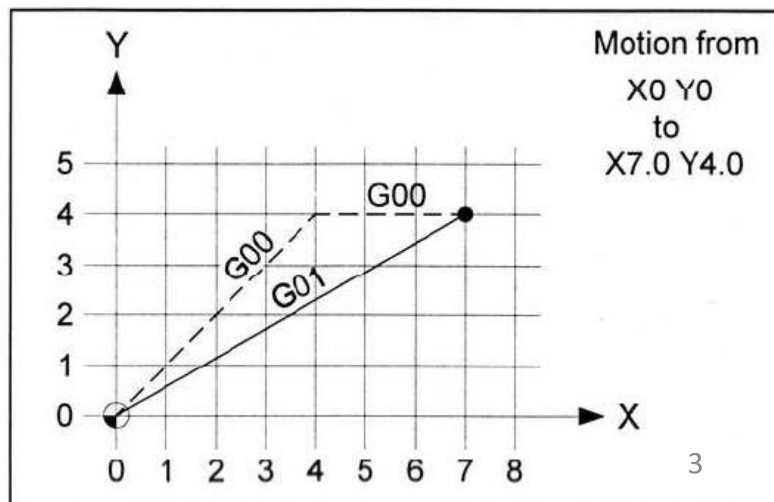
Linear Interpolation

- Linear interpolation is used in part programming to make a straight cutting motion from the start position of the cut to the its end
- Always uses shortest distance the cutting tool path can take
- Motion is always straight line connecting the contour start and end points which is important in contouring and profiling machining
- Also angle cutting is performed by linear interpolation, which requires knowing the angular path the cutter has to take
- Three types of motion can be generated in linear interpolation
  - Horizontal motion-single axis only
  - Vertical motion-single axes only
  - Angular motion- multiple axes



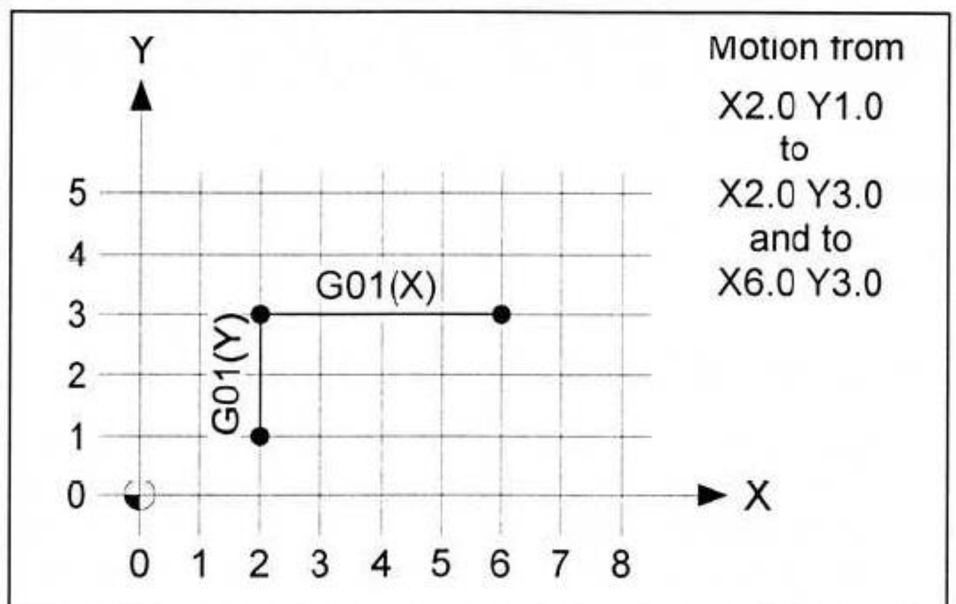
Linear Interpolation: G01

- In G01 mode, the feedrate function (F) must be in effect to do the cutting
- Programming in either G00 or G01 has the same programmed end point but the difference is the feedrate and tool path
- An alarm will occur during the first run if no feedrate function is used. The feedrate specified for the first run will be always used unless changed during the program steps
- G01 is a modal, which means they can be omitted in all subsequent linear interpolation blocks. Only change of coordinate location is required (x,y, z)
- Linear motion is motion between two points. The start point is called the ***departure***, the end point is called the ***target***
- Departure point: defined by the current tool position
- Target point: defined by the target coordinates of the current block, which might be the starting point for another point



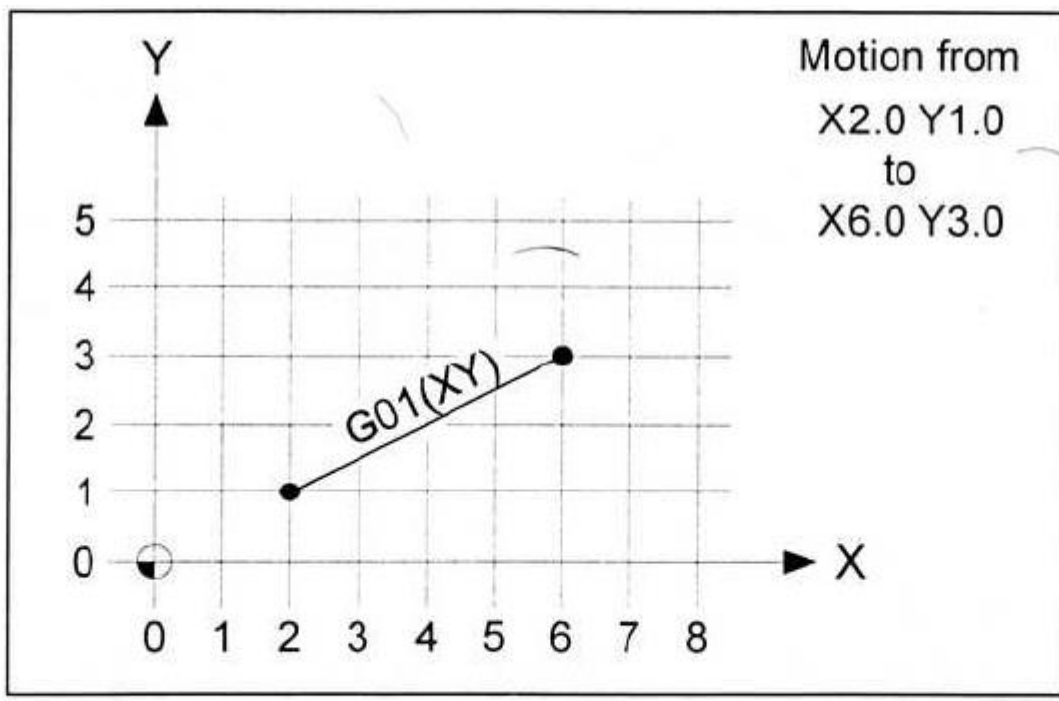
Single axis of motion

- The programmed tool motion along any single axis is always a motion parallel to that axis.
- all tool motion that are parallel to the table edge are single axis motion
- Most lathe operations (facing, shoulder turning, diameter turning) and all drilling operation are considered single axis motion operations
- A single axis motion can never me an angular motion. Workpiece is better fixed at the table axes to make most profiling and drilling operation single axes motion
- Another name is: orthogonal-horizontal or vertical only



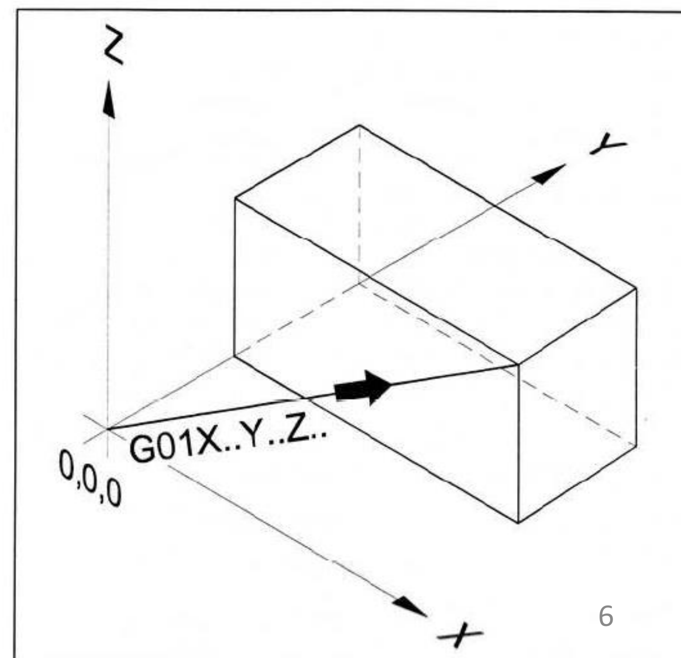
Two axes Linear Interpolation

- A linear motion can also be programmed along two axes simultaneously.
- This is the case when departure and target have at least two coordinates different form each other
- The result of this two-axis motion is a straight line (G01) at an angle calculated by the controller
- The motion is still the shortest distance between the points



Three-axes of motion

- A simultaneous linear motion along three axes is possible on all CNC machines
- However, this kind of programming is not always easy specially with complex parts which requires extensive calculation for the tool path
- 3-axes are NOT done manually in most cases. A CAD/CAM software is commonly used (Pro/E)



Feedrate for linear operations

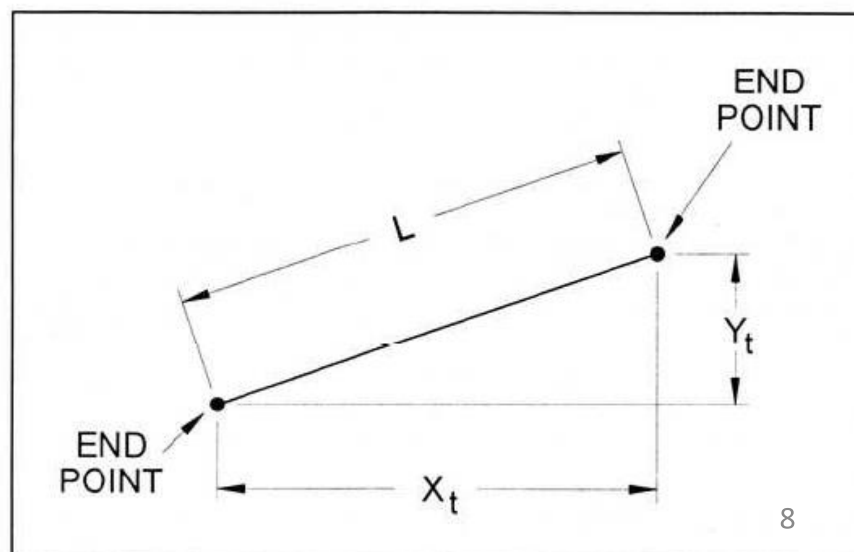
- The actual cutting feedrate for tool motion can be programmed in two modes
  - Per time (mm/min or inch/min)
  - Per spindle revolution (mm/rev or inch/rev)
- The selection depends on the machine type and dimensional units used. For milling and profiling operations we use feedrate per time., while for turning operations we use spindle revolution
- There is a range for feedrate for every CNC machine
- The feedrate values in table below represents the controller ranges depending on the technology in making the machine . It is unlikely to have such large feedrate close the maximum limits of the CNC machines

Minimum motion increment	MILLING
0.001 mm	0.0001 - 240000.00 mm/min
0.001 degree	0.0001 - 240000.00 deg/min
.0001 inch	.0001 - 240000.00 in/min

Feedrate Ranges

Individual axes feedrate

- The machine controller translates the 2 or 3 axes feedrate into individual axes feedrate. This is not important during programming but it is good to explain how the controller behaves
- In order to keep the linear motion as the shortest motion between two points, the CNC unit must always calculate for each axis individually
- The computer will speed up on one axis and hold back the other axis , at the same time, depending on the angle of motion
- A straight line is actually a jagged line of small increments of x and y values that are very small to see



Example: Feedrate calculation

- Example:
  - G 70
  - G00 X 10.0 Y 6.0
  - G01 X 14.5 Y 7.25 F 12
- Find actual travel:
- Xt = 14.5-10.0 = 4.5 inch
- Yt= 7.25-6.0 = 1.25
- Zt= 0
- L is the compound motion calculated by Pythagorean theorem

$$L = \sqrt{X_t^2 + Y_t^2 + Z_t^2}$$

$$L = \sqrt{4.5^2 + 1.25^2 + 0^2} = 4.6703854$$

Feedrate calculation, continued

- Feed rates for x and y

$$F_x = \frac{X_t}{L \times F}$$

$F_x = 4.5 / 4.6703854 \times 12 = 11.562215$

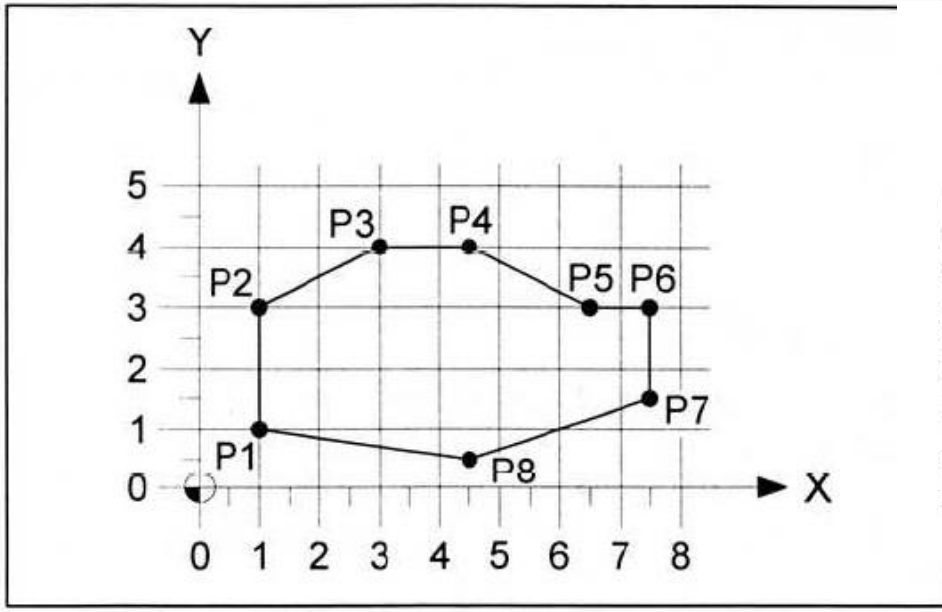
$$F_y = \frac{Y_t}{L \times F}$$

$F_y = 1.25 / 4.6703854 \times 12 = 3.2117263$

$$F_z = \frac{Z_t}{L \times F}$$

$F_x = 0 / 4.6703854 \times 12 = 0.0$

Example: linear interpolation



Example 1 :

```
(CLOCKWISE DIRECTION FROM P1)

G90 ...                          (ABSOLUTE MODE)
G01 X1.0 Y3.0 F...                   (P1 TO P2)
X3.0 Y4.0                            (P2 TO P3)
X4.5                                 (P3 TO P4)
X6.5 Y3.0                            (P4 TO P5)
X7.5                                 (P5 TO P6)
Y1.5                                 (P6 TO P7)
X4.5 Y0.5                            (P7 TO P8)
X1.0 Y1.0                            (P8 TO P1)
...
```

Example 2 :

```
(COUNTERCLOCKWISE DIRECTION FROM P1)

G90 ...                          (ABSOLUTE MODE)
G01 X4.5 Y0.5 F...                   (P1 TO P8)
X7.5 Y1.5                            (P8 TO P7)
Y3.0                                 (P7 TO P6)
X6.5                                 (P6 TO P5)
X4.5 Y4.0                            (P5 TO P4)
X3.0                                 (P4 TO P3)
X1.0 Y3.0                            (P3 TO P2)
Y1.0                                 (P2 TO P1)
...
```

Dwell time programming: G04-code

- Dwell is the *pause* in the program
- It is an intentional time delay applied during program processing
- In this time any motion is stopped while all other program commands and functions remain unaffected
- When the designated time expires, the control system resumes processing the program with the block immediately following the block command that contains the dwell
- Programming a dwell is very easy and useful in two main applications:
  - During actual cutting when the tool is in contact with the material
  - For operation of machine accessories, when no cutting takes place
- Each application is equally important to programmers but there NEVER used simultaneously

Application for cutting

- When cutting is removing material, it is in contact with the machined mart.
- A dwell can be applied during machining for a number of reasons
  - Breaking chips while drilling, counter boring, grooving or parting off
  - Eliminate physical marks left on the part by end thrust of the cutting tool in case of turning, thrust is attributed to the tool pressure during cutting
  - Control deceleration of the cutting feed on a corner during fast feedrates.
- The Dwell command forces the machining operation to be fully completed in one block
- The programmer still has to supply the exact period of time required for the pause and must be sufficient, not too short and not too long

Applications for accessories

- Usually after certain miscellaneous functions (M-function)
- The dwell will allow completion of an accessory task during the dwell time, tailstock calibration, part catcher installation . Cleaning purposes are NOT allowed as well as any other manual operations using dwell time
- The machine spindle may either be stationary or rotating but it is NOT important if spindle is rotating or not because the tool is not in contact with the workpiece
- Dwell is also used when changing spindle speed usually after gear range change

Tailstock

Part catchers

The G04 command

- Similar to G01, it is has to be associated with a certain word address format, in this case the amount of time to dwell
- The correct addresses for dwell are X, P, or U (address U is only for lathe machines).
- The time selected is either in milliseconds or seconds
- Dwell command does not carry over to the following block unlike G01 or other command. It will perform the dwell ONLY for the same block (line of commands)
- Dwell is used in fixed cycles (drilling operations) without the G04 command. This is a special case for fixed cycle operations which will be covered in future lectures
- The structure for dwell function is:
  - X5.3 (for all machines except fixed cycles)
  - U5.3 (for lathe machines only)
  - P53 (all machines including fixed cycles)
- Remember 1 second = 1000 ms

Examples of dwell format

- G04 X 2.0   (preferred for long dwells)
- G04 P2000 (preferred for short or medium dwell, this address does NOT accept decimal points)
- G04 U 2.0 (lathe only-in seconds)
- in this example, the dwell is 2 seconds or 2000 milliseconds.
- G04 X0.5
- G04 P500
- G04 U0.5
- In this example, the dwell is 500 milliseconds
- In a CNC program, the dwell may appear in the following way, note how dwell is separate block and does not affect the following block:
- N21 G01 Z-1.5 F12.0
- N22 G04 X0.3
- N23 Z-2.7 F8.0
- Do NOT confuse the X address with the X-axis, there will be NO motion if the dwell command is used, CNC machine use the (T) letter for tool change only

Examples o f dwell command

- The addresses X and U can also be programmed in milliseconds without decimal points
- G04 X2.0 is equal to G04 X2000
- Leading zero suppression is assumed in the formal without decimal point (trailing zeros are required):
- X1 = X 0001 (1 millisecond)
- P10 = P0010 (10 milliseconds)
- P100 = P0100 (100 millisecond)
- If decimal point format is used, five digits before the decimal and three digits after decimal is the maxim mum allowed digits:
- Range of : 0.001 – 99999.999 seconds

Dwell time selection and minimum dwell time

- Dwell time is commonly in the order of few seconds (less than 10). And most often much less than a one second.
- Time delay for accessory operation is recommended by machine manufacturers to prevent machine damage
- Time delay during cutting is the machinist responsibility and requires attention NOT to over program dwell duration
- Make sure to calculate the minimum dwell that can do the job for higher productivity on the machine
- The minimum dwell is the time (in seconds) required to complete one revolution of the spindle:

$$\text{min. dwell} = \frac{60}{\text{rev/min}}$$

- Example: calculate the minimum dwell for spindle rotating of 420 rev/min:
- Divide the rev/min into 60 (60 seconds in one minute)
- 60/420 = 0.143 seconds dwell
- G04 X0.143
- G04 P143
- G04 U 0.143
- It is a common practice to round up the minimum dwell time to avoid machine limits and ease of operations. For the preceding example, a typical roundup is:
- G04 X0.2

Time equivalent

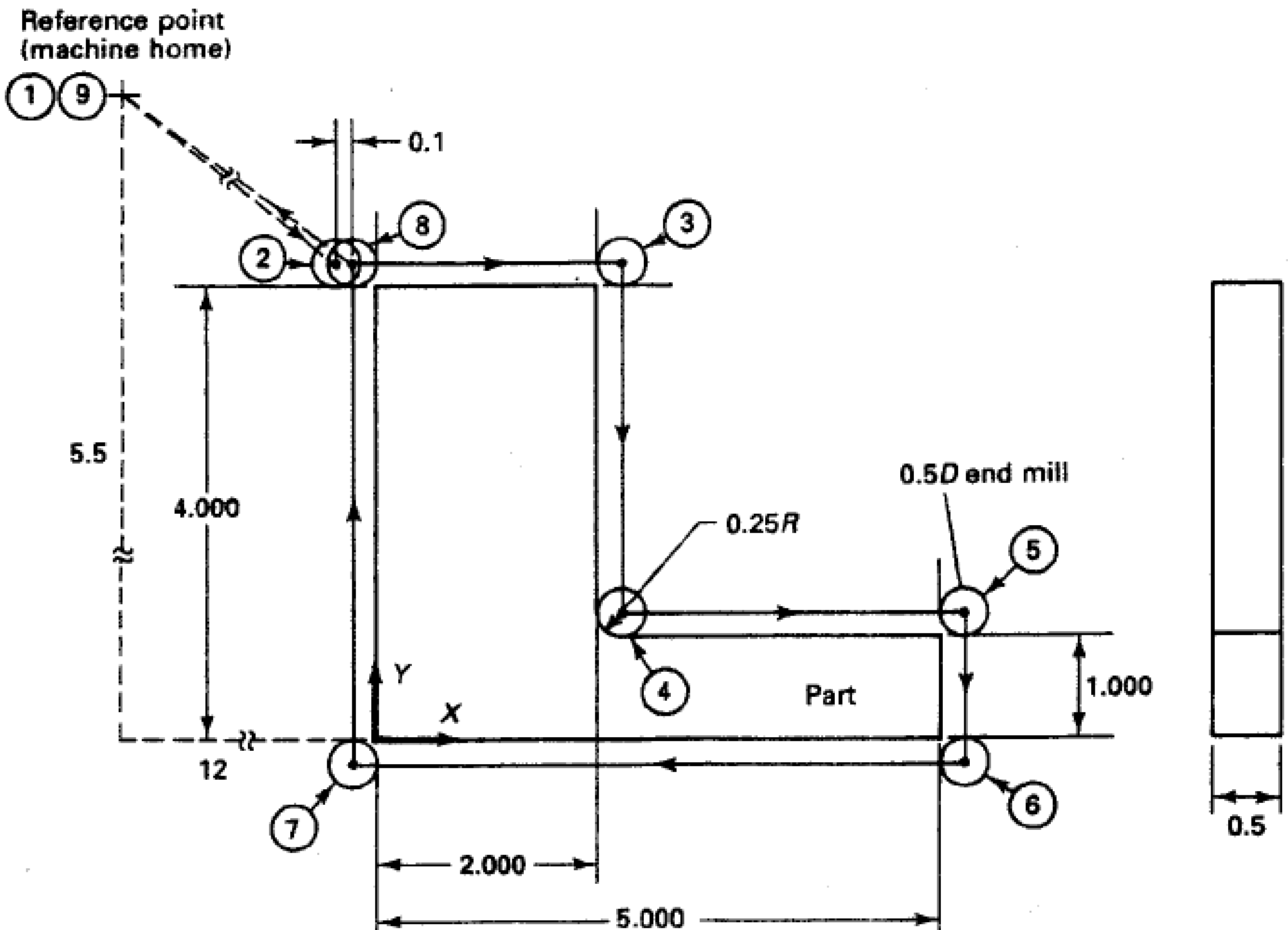
- The dwell time for a required number of spindle revolutions is the time equivalent

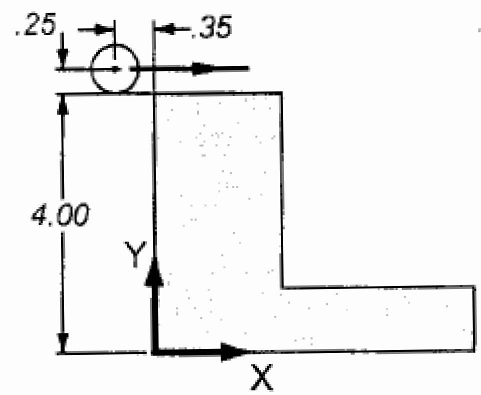
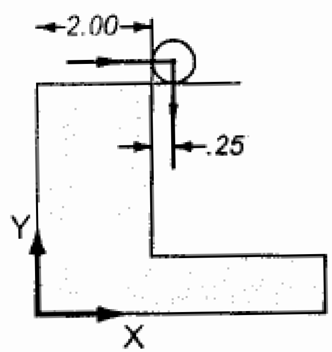
$$Dwell_{\sec} = \frac{60 \times n}{rev/\min}$$

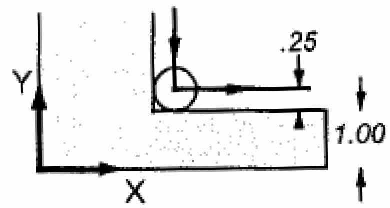
- n= required number of spindle revolution
- example: calculate the dwell time for full three spindle revolutions at spindle speed of 420 rev/min.
- Dwell = 60 x 3/ 420 = 0.249
- The program block representing the three spindle revolutions in terms of dwell time will be:
- G04 X0.429
- G04 P429
- G04 U0.429

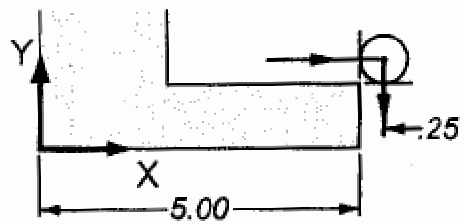
Example

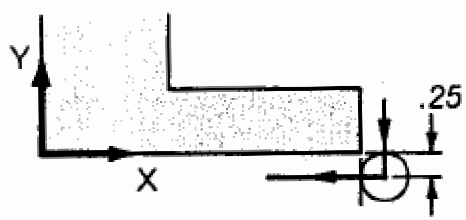
Tool	Operation	Tooling	Speed (rpm)	Feed (ipm)
1	Profile mill contour × 0.52 deep	0.5 *D* end mill	1200	8

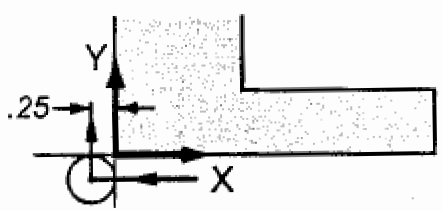


		Absolute coordinates	
Position		**X**	**Y**
②	 X = −.25 − .1 = −.35 Y = 4.00 + .25 = 4.25	−.35	4.25
③	 X = 2.00 + .25 = 2.25 Y = 4.25	2.25	4.25

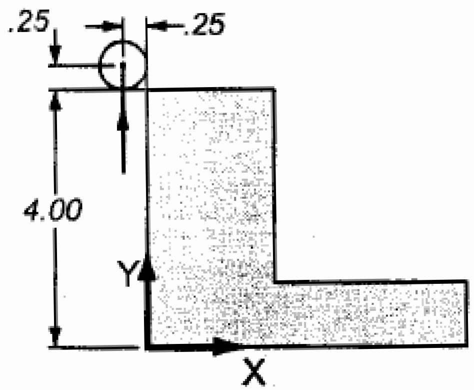
<table>
<tr><th></th><th></th><th></th><th></th></tr>
<tr><td>④</td><td>

X = 2.25
Y = 100 + .25 = 1.25</td><td>2.25</td><td>1.25</td></tr>
<tr><td>⑤</td><td>

X = 500 + .25 = 5.25
Y = 1.25</td><td>5.25</td><td>1.25</td></tr>
<tr><td>⑥</td><td>

X = 5.25
Y = −.25</td><td>5.25</td><td>−.25</td></tr>
<tr><td>⑦</td><td>

X = −.25
Y = −.25</td><td>−.25</td><td>−.25</td></tr>
</table>

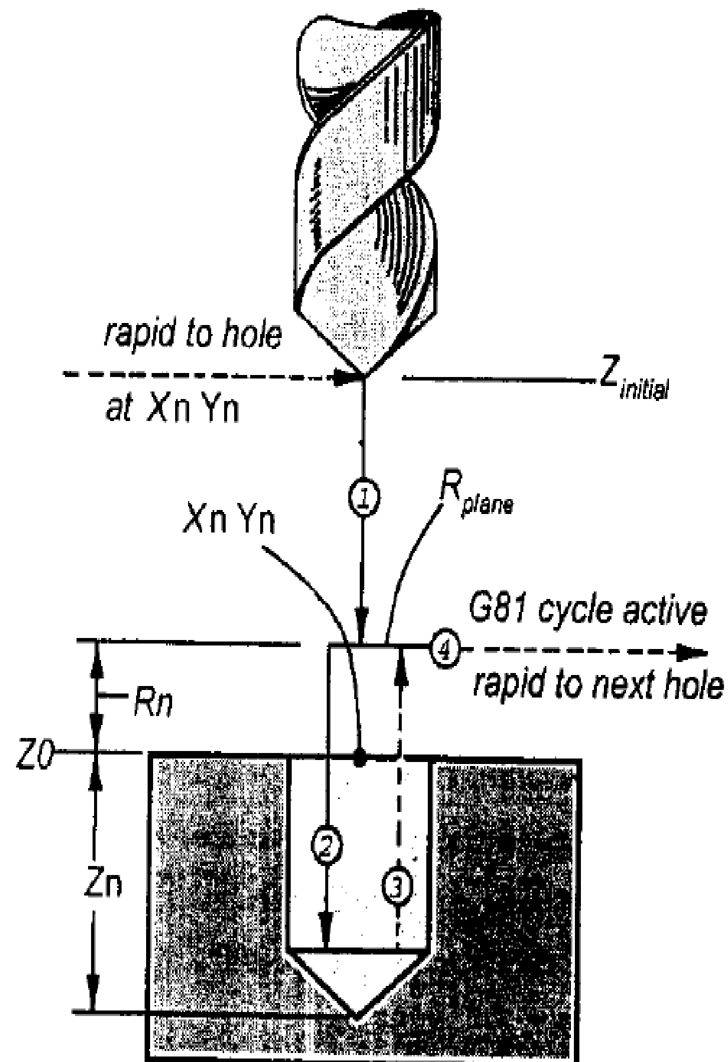
⑧	 X =.25 Y = 4.00 + .25 = 4.25	−.25	4.25

Introduction to Hole operations

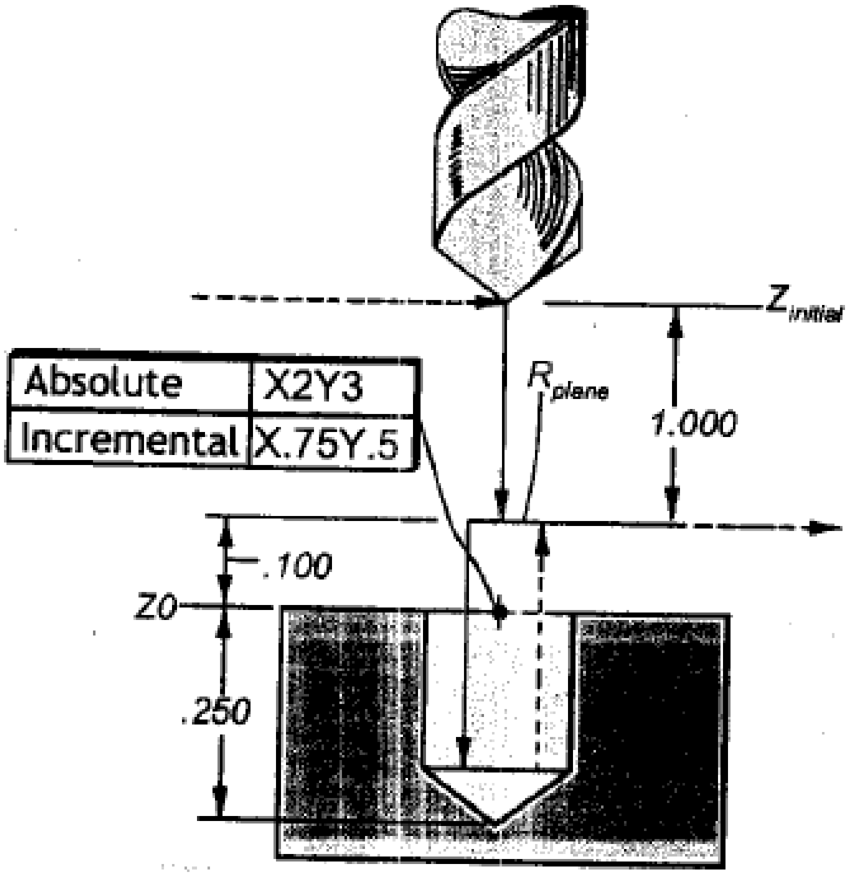
CAD/CAM

Drill center, or ream cycle

- A G81 cause the machine to rapid the tool from an initial position to a specified “plane” before drilling
- Then drill to a defined depth at a given feedrate
- Then move out of the drilled hole to the original starting “plane”
- Then rapid motion to the center of the new next hole defined by coordinates (X,Y)
- Any drilling operation can be either in absolute or incremental dimensions specified before drilling
- Command :
- G81 X_n Y_n Z_n R_n F_n
- X_n Y_n : nemeric vlaue (n) specify the location of hole drilling
- Z_n : numeric value (n) specify the depth of the drill
- Rn :
  - Absolute: specifies the distance to R-plane from Z0
  - Incremental: specifies the distance below the Z-initial to Rn



Drill center or ream example



Absolute Coordinates

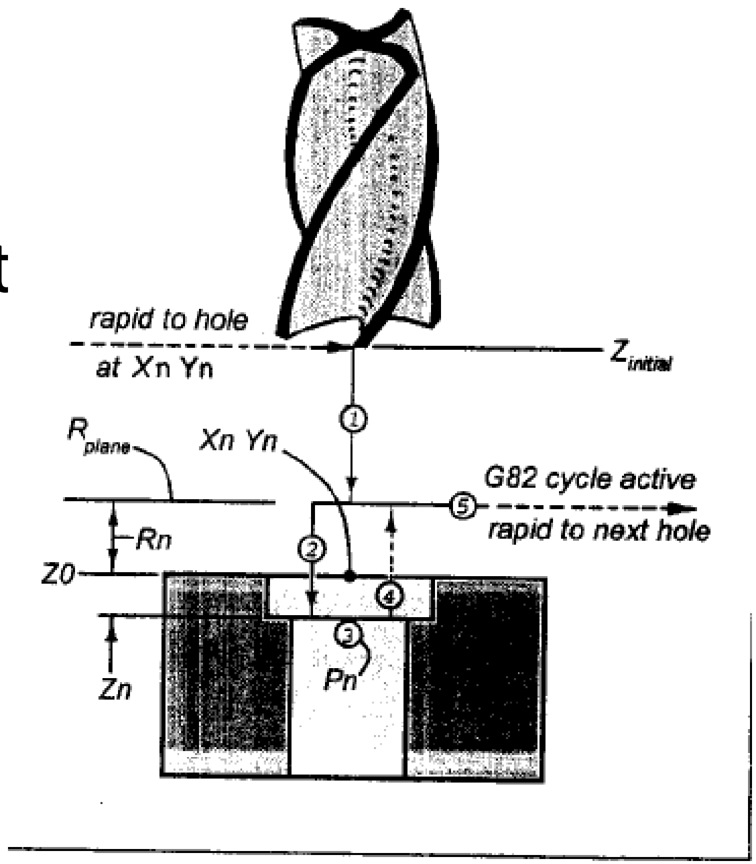
N0060 G90
N0070 G81 X2. Y3. Z-.25 R.1 F7

Incremental Coordinates

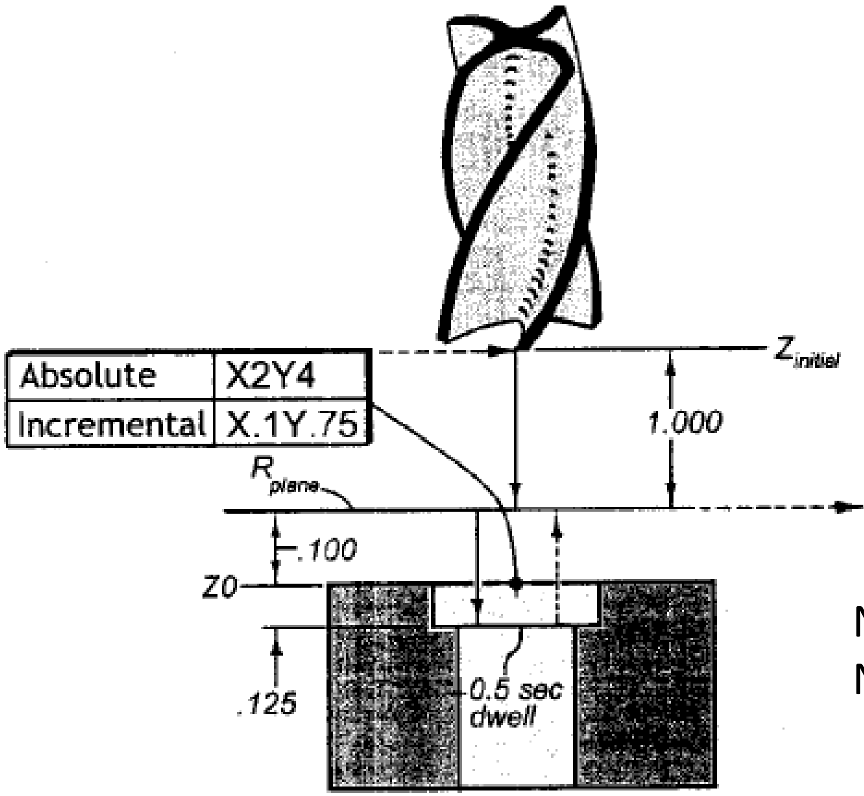
N0060 G91
N0070 G81 X.75 Y.5 Z-.35 R-1. F

Counter bore or spot face cycle

- Word address command: G82
- G82 cycle causes the machine to rapid the tool from Z-initial to R-plane, then bore the hole to a depth of Zn at feed rate of Fn, then dwell for Pn seconds at depth Zn, then rapid back to R-plane and finally rapid movement to the next hole if a new X, Y are given in the next block
- Complete command block:
  – G82 Xn Yn Zn Rn Pn Fn



Counter bore or spot face example



Absolute Coordinates

N004 G90
N005 G82 X2.0 Y4.0 Z-0.125 R0.1 P500 F6

Incremental Coordinates

N004 G91
N005 G82 X0.1 Y0.75 Z-0.225 R-1.0 P500 F6