SYNTEC

Instruction Guide of Lathe Programming

By：SYNTEC
Data：2006/11/10
Ver：7.7

The record of version update

item	The content	Date	Author	The latest version
01	the first craft	2005/10/01	Jerry	V7.1
02	1.add G68,G69 2.Modify G10	2006/01/25	Jerry	V7.2
03	1. modify the specification of G01,G04,C 2. modify the feedrate F in the examples and define its unit mm/rev	2006/06/06	Jerry	V7.3
04	1.add the specification of G65 G66 G67	2006/07/18	Jerry	V7.4
05	1. add the specification of G12.1 G13.1	2006/07/20	Jerry	V7.5
06	1. add the specification of G07.1	2006/10/05	Jerry	V7.6
07	1. mdify M99 descriptions	2006/11/10	Jerry	V7.7

Menu

Lather Program Instruction Description

A、 G Code Instruction Description

1.1、G Code List

Function Name	G code			Index
	Type A	Type B	Type C	
Positioning(Rapid traverse)	G00	G00	G00	5
Linear interpolation(cutting feed)	G01	G01	G01	7
Circular interpolation(CW)	G02	G02	G02	11
Circular interpolation(CCW)	G03	G03	G03	11
Dwell	G04	G04	G04	16
Cylinder interpolation	G07.1	G07.1	G07.1	17
Exact stop	G09	G09	G09	19
Programmable data input	G10	G10	G10	20
Start polar coordinates interpolation	G12.1	G12.1	G12.1	22
Cancel polar coordinates interpolation	G13.1	G13.1	G13.1	22
XpYp plane selection	G17	G17	G17	25
ZpXp plane selection	G18	G18	G18	25
YpZp plane selection	G19	G19	G19	25
Outer/internal diameter drilling cycle	G90	G77	G20	26
Threading cycle	G92	G78	G21	31
Endface turning cycle	G94	G79	G24	36
Return to reference position	G28	G28	G28	41
Return from any reference position	G30	G30	G30	42
Skip function	G31	G31	G31	43
Thread cutting	G32	G33	G33	31
Cancel tool nose radius compensation	G40	G40	G40	58
Tool nose radius compensation left	G41	G41	G41	58
Tool nose radius compensation right	G42	G42	G42	58
Local coordinate system setting	G52	G52	G52	68
Machine coordinate system setting	G53	G53	G53	69

Function Name	G code			Index
	Type A	Type B	Type C	
Workpiece coordinate system selection	G54 ~G59.9	G54 ~G59.9	G54 ~G59.9	71
Single Marco calling	G65	G65	G65	73
Marco modal calling	G66	G66	G66	73
Marco modal call cancel	G67	G67	G67	73
Input in imperial system	G20	G20	G70	74
Input in metric system	G21	G21	G71	74
Fine cutting cycle	G70	G70	G72	74
Stock removal in turning	G71	G71	G73	74
Stock removal in facing	G72	G72	G74	75
Pattern repeating	G73	G73	G75	80
End face peck drilling	G74	G74	G76	87
Outer diameter/internal diameter drilling	G75	G75	G77	91
Multiple threading cycle	G76	G76	G78	95
Canned cycle for drilling cancel	G80	G80	G80	106
Cycle for face drilling	G83	G83	G83	108
Cycle for face tapping	G84	G84	G84	111
Cycle for face boring	G85	G85	G85	113
Cycle for side drilling	G87	G87	G87	108
Cycle for side tapping	G88	G88	G88	111
Cycle for side boring	G89	G89	G89	113
Coordinate system setting/max. spindle speed setting	G50	G92	G92	115
Feedrate per minute(mm/min.)	G98	G94	G94	120
Feedrate per revolution(mm/rev.)	G99	G95	G95	135
Constant surface speed control	-	G96	G96	135
Constant surface speed control cancel	-	G97	G97	135
Return to initial point	-	G98	G98	-
Return to R point	-	G99	G99	-

1.2、Command Description

1.2.1 G00：positioning

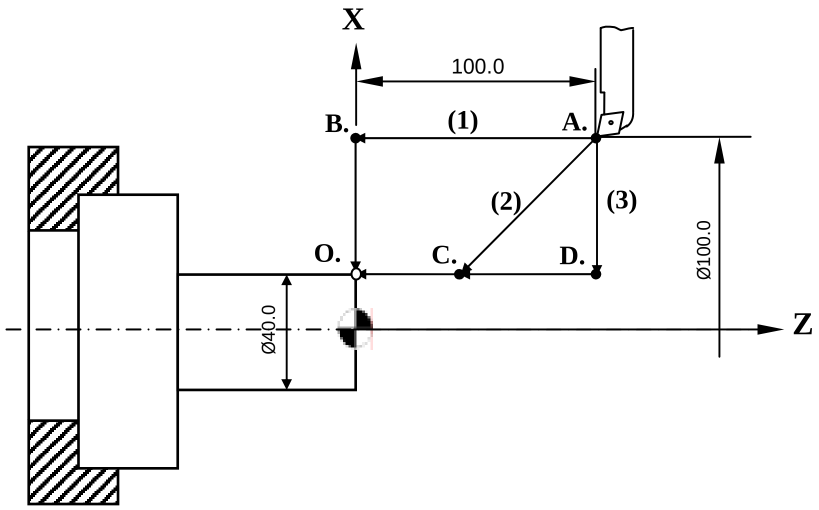
Format：

G00 X(U)___ Z(W)___；

X、Z：specified position(absolute mode)
U、W：specified position(increment mode)

Description：The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate。There is no any cutting action in this command。The main aim is saved the movement time in no cutting status；in the lathe program, it is usually used in the tool from machine zero point to start cutting point, or from end point to machine zero point。In absolute mode(G90), tool moves to specified position in coordinate system ；in increment mode(G91), tool moves to specified position by specified distance。

Example：



Program description：There are several ways to make tool move from point A to point O。There are three kinds as below：

1. Absolute mode：

(1). G00 Z0.0；// A.→B.
X40.0；// B.→O.

(2). G00 X40.0 Z0.0；//A.→C.→O.

(3). G00 X40.0；//A.→D.
Z0.0；//D.→C.→O.

2. Incremant mode：

(1). G00 W-100.0；// A.→B.
U-60.0；// B.→O.

(2). G00 U-60.0 W-100.0；//A.→C.→O.

(3). G00 U-60.0；//A.→D.
W-100.0；// D.→C.→O.

3. Combination of absolute mode and increment mode：

(1). G00 Z0.0； or G00 W-100.0；
U-60.0； X40.0；

(2). G00 X40.0； or G00 U-60.0；
W-100.0； Z0.0；

(3). G00 X40.0 W-100.0； or G00 U-60.0 Z0.0；

1.2.2 G01：Linear Interpolation

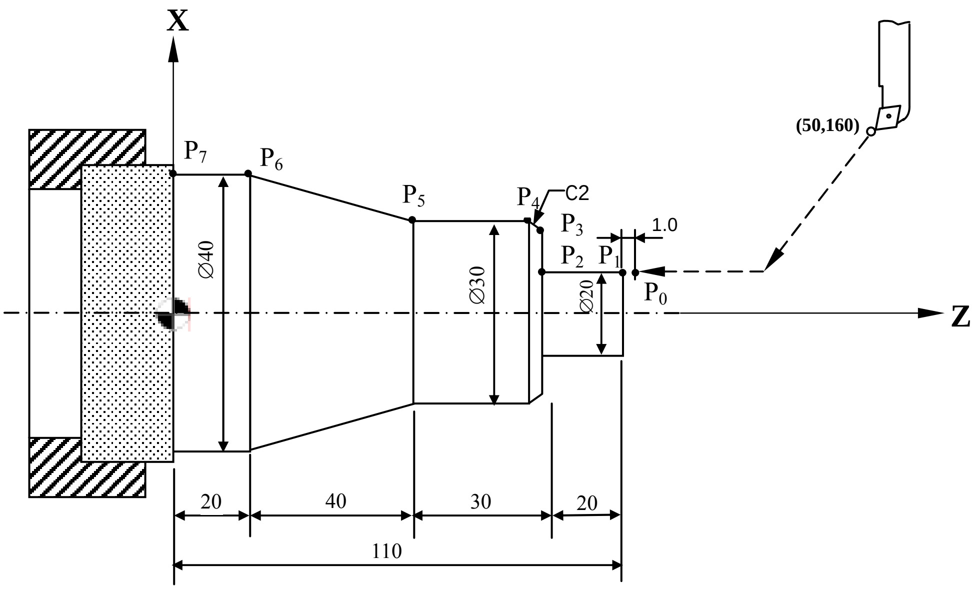
Format：

G01 X(U)___ Z(W)___ F___；

X、Z：specified position(absolute mode)
U、W：specified position(increment mode)
F：Feedrate ※ G94：mm/min . in/min ← **default**
G95：mm/rev . in/rev

Description：Tools do linear interpolation and move to specified position at the feedrate specified in F function。It can process：outer(internal)diameter、endface、outer(internal)tank..ect。

Example：



Program description：

G92 X50.0 Z160.0 S10000；
//set the program zero point, max. speed 10000 rpm
T01；//use tool NO. 1
G96 S130 M03；
//constant surface speed, surface speed 130m/min, spindle rotate CW
M08；//cutting liquid ON
G00 X20.0 Z111.0；//positioning to specified point P_0
G01 Z90.0 F600；//linear interpolation $P_0 \rightarrow P_2$

```
      X26.0；//P2→P3
      X30.0 Z88.0；//P3→P4
      Z60.0；//P4→P5
      X40.0 Z20.0；//P5→P6
      Z0.0；//P6→P7
G00 X50.0；//return the tool
      Z160.0；//return to zero point
M05 M09；//spindle stops, setting liquid OFF
M30；//program end
```

1.2.3　G02、G03：Circular Interpolation

Format：

$$\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} \ X(U)\_\_ \ \ Z(W)\_\_ \ \begin{Bmatrix} R\_\_ \\ I\_\_ \ \ K\_\_ \end{Bmatrix} \ F\_\_ \ ;$$

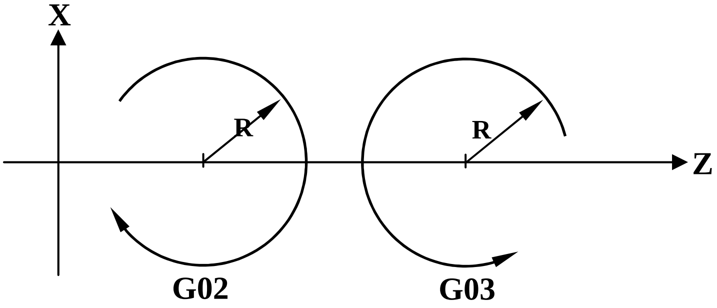
G02：Circular Interpolation(CW)
G03：Circular Interpolation(CCW)
X(U)、Z(W)：end point of the arc
R：radius of arc(under 180°)
I、K：X(Z) axis distance from starting point of arc to centered。There are differences in positive and negative，It depends on the direction
F：Feedrate of cutting

Description：The G02、G03 command will move a specified tool along a circular arc on XpZp plane，the parameter setting as below：

Data setting			Command	Definition
1	Tool direction		G02	CW
			G03	CCW
2	End position	G90	X、Z	The end position of specified arc
		G91	U、W	Vector value from starting point to end point
3	Distance from starting point to centered		Two axes among I、J、K axis	Vector value from arc starting point to centered
	Radius of arc		R	Radius of arc
4	Feedrate		F	Feedrate along the arc

PIC：

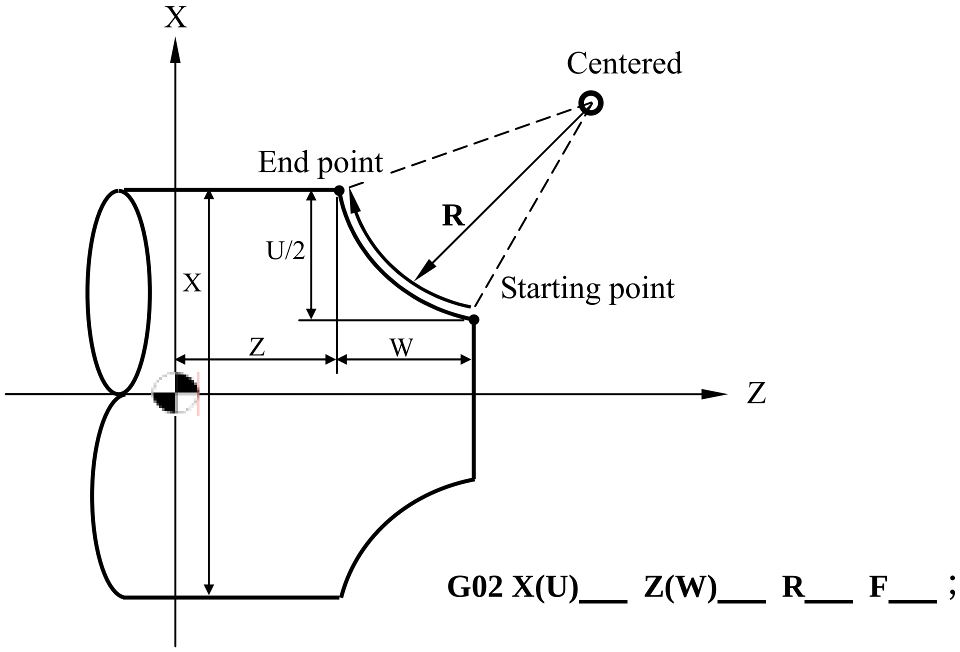
1. G02/G03 Direction Decision



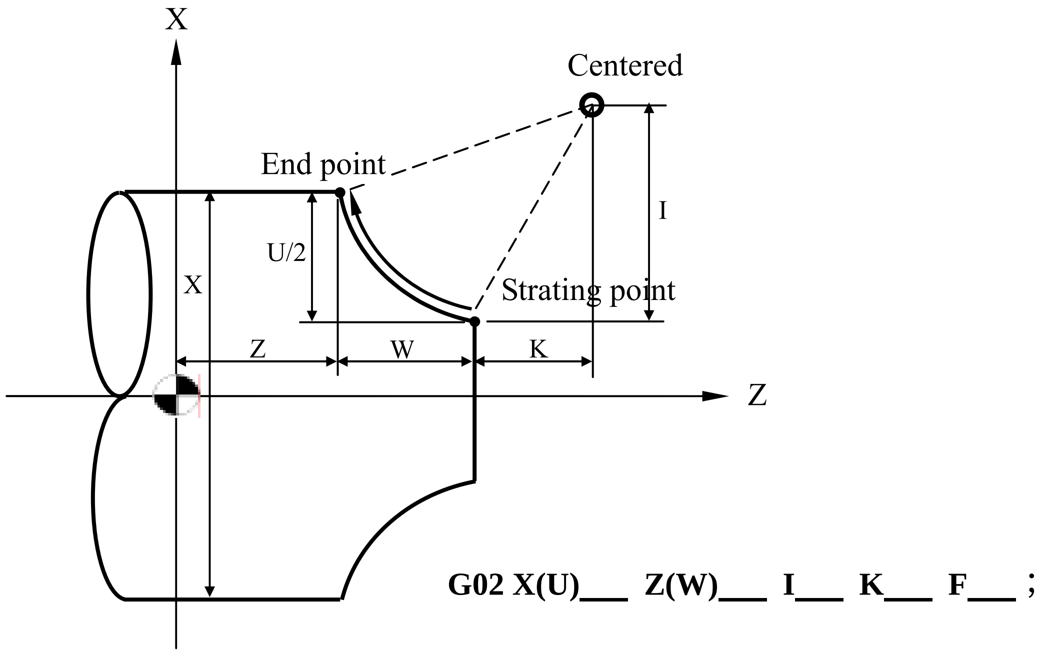
2. Parameter setting in process

(1). G02 circular interpolation

a. Use R value

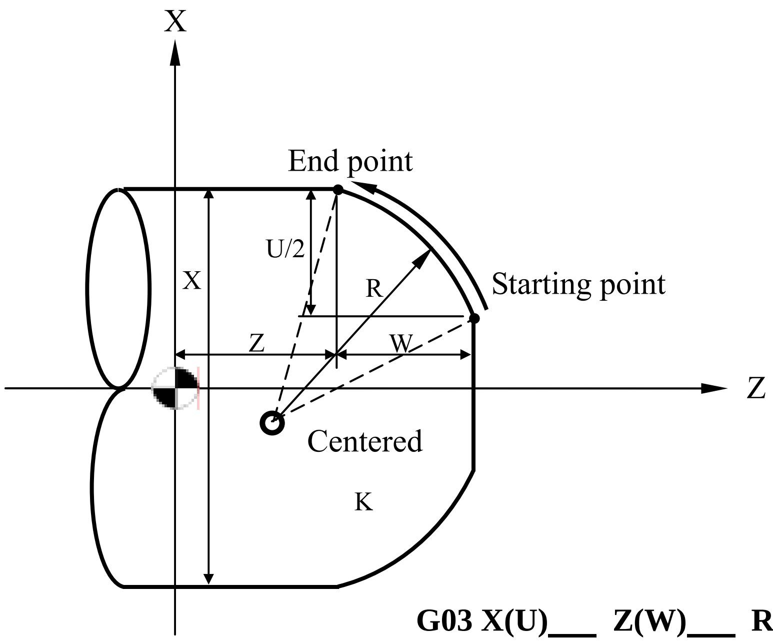


b. Use I、K



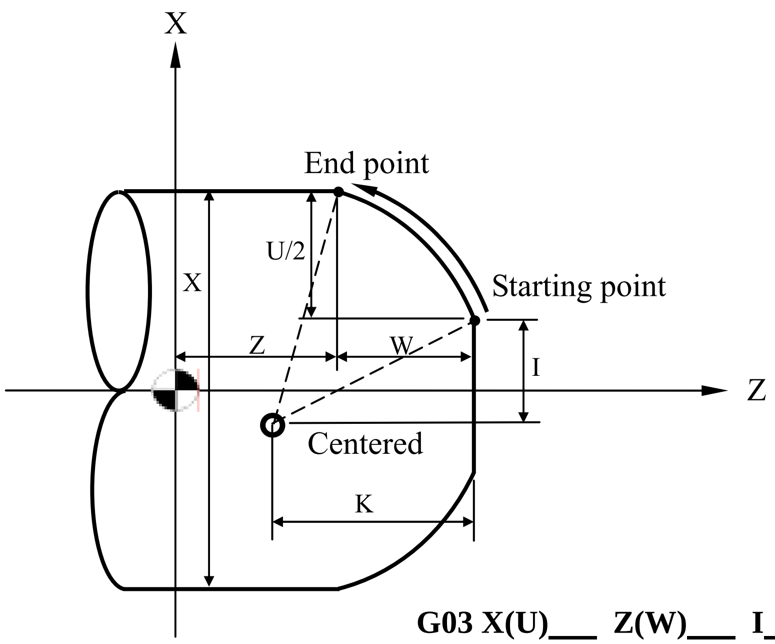
(2). G03 circular interpolation

a. Use R value



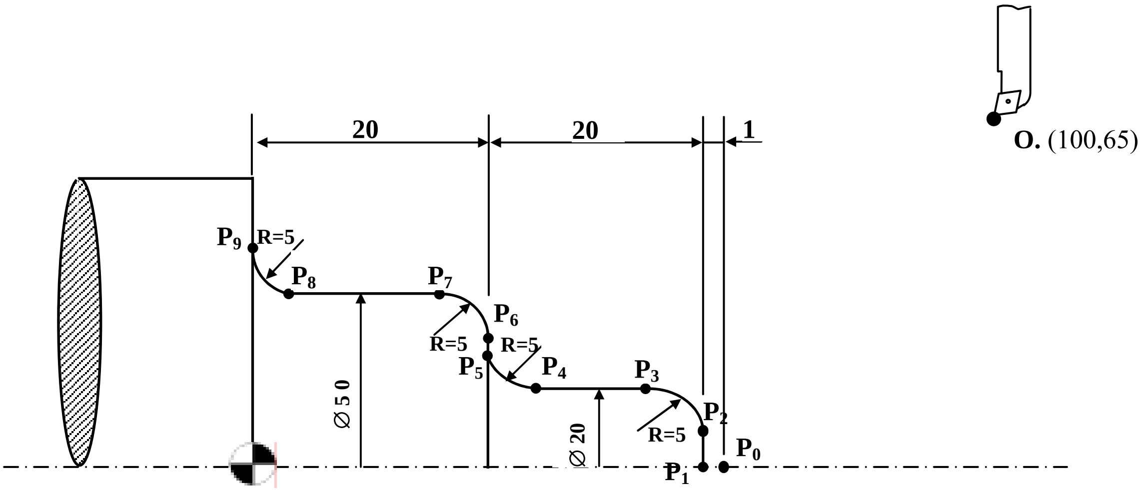
G03 X(U)___ Z(W)___ R___ F___ ;

b. Use I、K



G03 X(U)___ Z(W)___ I___ K___ F___ ;

Example one：



Program description：

N001 T01；//use tool NO.1

N002 G92 S10000；//spindle max. speed 10000 rpm

N003 G96 S130 M03；

//constant surface speed, surface speed 130 mm/min, spindle rotate CW

N004 M08；//cutting liquid ON

N005 G00 X0.0 Z41.0；//positioning **O.**→P_0

N006 G01 Z40.0 F600；//linear interpolation, feedrate 600 mm/rev, P_0→P_1

X10.0；//P_1→P_2

N007 G03 X20. Z35.0 R5.0；//circular interpolation CCW P_2→P_3, radius 5mm

N008 G01 Z25.0；//P_3→P_4

N009 G02 X30.0 Z20. R5.0；//circular interpolation CW P_4→P_5, radius 5mm

N0010 G01 X40.0；//P_5→P_6

N0011 G03 X50.0 Z15.0 R5.0；//circular interpolation CCW P_6→P_7, radius 5mm

N0012 G01 Z5.0；//P_7→P_8

N0013 G02 X60.0 Z0.0 R5.0；//circular interpolation CW P_8→P_9, radius 5mm

N0014 G00 X100.0；//tool escape, escape from workpiece

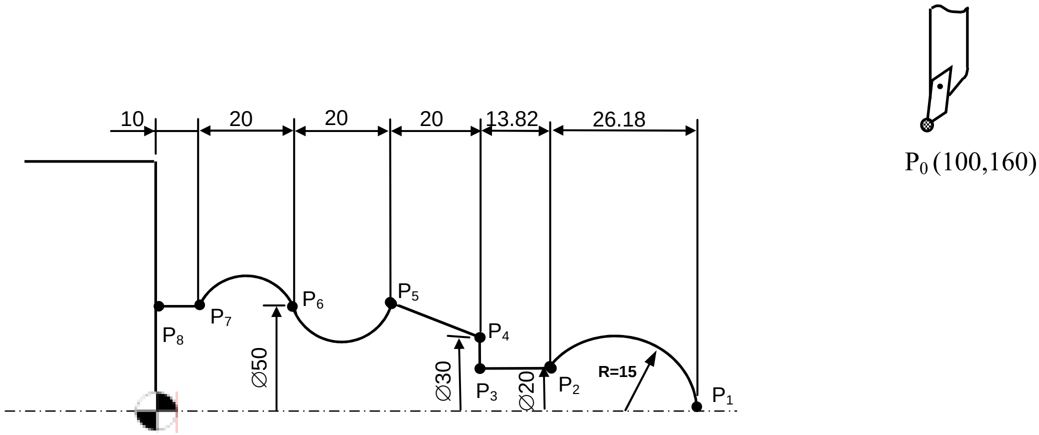
N0015 G00 Z65.0；//return to initial point

N0016 M09；//cutting liquid OFF

N0017 M05；//spindle stops

N0018 M30；//program end

Example two：



Program description：

N001 T01；//use tool NO.1

N002 G92 S10000；//spindle max. speed 10000 rpm

N003 G96 S130 M03；

//constant surface speed, surface speed 130 mm/min, spindle rotate CW

N004 M08；//cutting liquid NO

N005 G00 X0.0 Z110.5；//positioning, close to the starting point

N006 G01 Z110.0 F500；//linear interpolation, feedrate 500mm/min

N007 G03 X20.0 Z83.82 K-15.0；

//circular interpolation CCW, $P_1 \rightarrow P_2$, radius 15 mm

N008 G01 Z70.0；//linear interpolation, $P_2 \rightarrow P_3$

X30.0；//$P_3 \rightarrow P_4$

X50.0 Z50.0；//$P_4 \rightarrow P_5$

N009 G02 X50.0 Z30.0 R10.0；

//circular interpolation CW, $P_5 \rightarrow P_6$, radius 10 mm

N0010 G03 X50.0 Z10.0 R10.0 ；

//circular interpolation CCW, $P_6 \rightarrow P_7$, radius 10 mm

N0011 G01 Z0.0；//linear interpolation, $P_7 \rightarrow P_8$

N0012 M09；//cutting liquid OFF

N0013 G00 X100.0；//tool escape, escape from workpiece

Z160.0；//return to initial point

N0014 M05；//spindle stops

N0015 M30；//program end

1.2.4 G04：Dwell

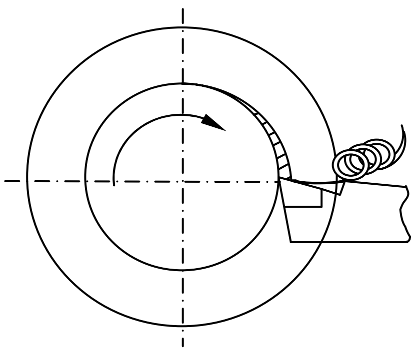
Format：

$$G04 \left\{ \begin{array}{l} X(U)\_\_\_ \\ P\_\_\_ \end{array} \right\} ;$$

X(U)、P：dwell time

Description：We can use G04 command to let the tool dwell a specified time when we process to the specified position。It can cut the offscourings of iron, improve the precision, let the surface more luminosity(as below)。When G04 command match with G94 or G95 in usage, the unit of time is second。※G04 command is only effective in single block。

PIC：



Example：

G04 X0.5；//dwell 0.5s

G04 U0.5；//dwell 0.5s

G04 P500；//dwell 0.5s, ※Notice：P__ is not allowed to be decimal point

＊Referenced formula：

$$T = \frac{Z \times 60}{N}$$

T：dwell time (s)

Z：dwell coils

N：rev/min

(Notice：Syntec controller didn't offer the Command to wait for Coils Number finish)

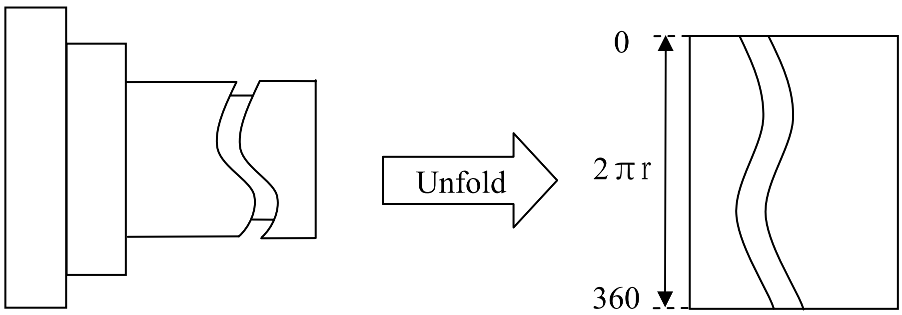
1.2.5 G07.1：Cylinder Interpolation

Format :

G19 Z0 C0; //select the working platform
G07.1 C__; //start the cylinder difference, C__the cylinder radius
、
、 //the description of the route
、
G07.1C0; //end the cylinder difference

Description:G07.1 start the cylinder difference, G02/G03-> circular interpolation function, G40/G41/G42-> tool nose radius compensation function. Because of the difficulty of the calculation of the vector in the center of a cirle,we use the way of R_radius address. Feedrate F_ is linear velocity in the surface of the cylinder. About the way of feed we must swith it into G94 in the lathe system in the first,for the C-axis is the main shaft probably.

PIC：



EX:

Program
G28 U0 W0;
T0202;
G97 S1000; // set up the rotational speed of the main shaft
G00 X50.0 Z0.;
G94 G01 X40.0 F100.;
G19 C0 Z0; // choose CZ the working platform

```
G07.1 C20.0;                  // start G07.1， the radius is 20.0
G41;                          // start process
G01 Z-10.0 C80.0 F150.0;
G01 Z-25.0 C90.0;
G01 Z-80.0 C225.0;
G03 Z-75.0 C270.0 R55.0;
G01 Z-25.0;
G02 Z-20.0 C280.0 R80.0;
G01 C360.0;
G40;                          // end process
G07.1 C0;                     // cancel G07.1
G01 X50.0;
G00 X100.0 Z100.0;
M30;
```

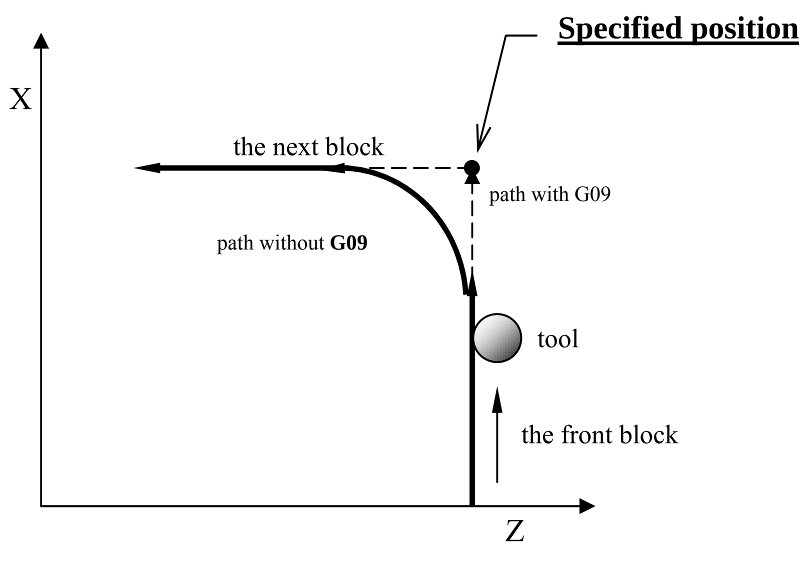
1.2.6 G09：Exact Stop

Format ： G09 X__ Z__ ；

X、Z：specified corner position

Description：When we process the corner，because the tool moves too fast or servo system delays。We can not cut the exactly corner and the error occurs。But in the situation that we need a right-angled，we can use G09 to make it。It can let the tool deceleration when tool approaches to the corner。When the tool reach to specified position，then the next block will be executed。

PIC：



1.2.7 G10：Programmable Data Input

Format：

G10 P___ X___ Z___ R___ Q___；

or

G10 P___ U___ W___ C___ Q___；

P：offset number

Tool wear offset value：P =number of tool wear offset

Tool geometry offset value：P=10000 + number of tool geometry offset value

X：offset value on X axis(absolute)

Y：offset value on Y axis(absolute)

Z：offset value on Z axis(absolute)

U：offset value on X axis(incremental)

V：offset value on Y axis (incremental)

W：offset value on Z axis (incremental)

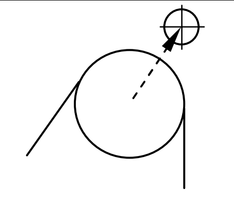
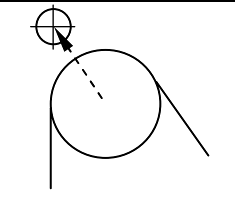
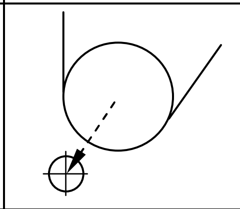
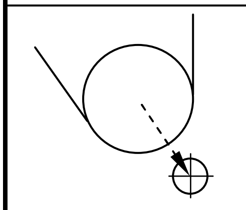
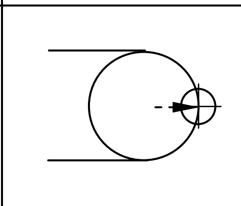
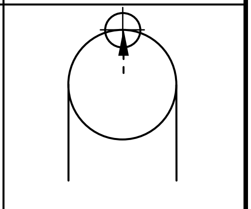
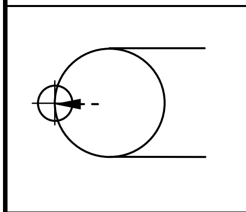
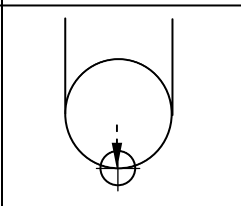
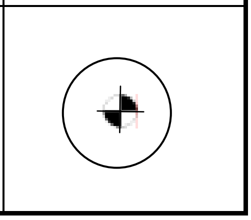
R：tool nose radius offset value(absolute)

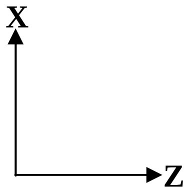
C：tool nose radius offset value (incremental)

Q：imaginary tool nose number(setting method is in next page)

Description：G10 command is programmable data input command。When we write the program， we can use this command to change the tool offset value。

＊Imaginary tool nose setting：

Imaginary tool nose NO.1	Imaginary tool nose NO.2	Imaginary tool nose NO.3
		
Imaginary tool nose NO.4	Imaginary tool nose NO.5	Imaginary tool nose NO.6
		
Imaginary tool nose NO.7	Imaginary tool nose NO.8	Imaginary tool nose NO.9
		



1.2.8　G12.1、G13.1：Start/Cancel polar coordinates interpolation

Format:

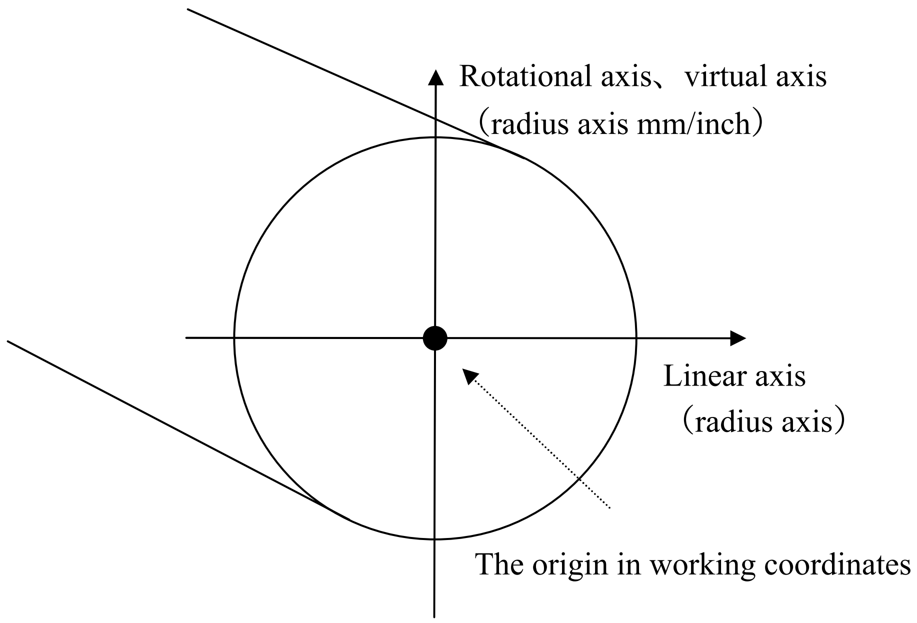
G12.1: Start polar coordinates interpolation

(Start linear circular or circular interplolation in rectangular coordinate and rectangular coordinate is composed of linear and rotational shaft)

G13.1: Cancel polar coordinates interpolation

Description:

1. The function of the polar coordinates interpolation transfers the program instructions of the patterns in the rectangular coordinate to linear motion (knife motion) and rotational motion (workpiece motion). This way is usually used in cutting end face and milling cam shaft in lathe.
2. the plane of the polar coordinates interpolation:
   G12.1 starts polar coordinates interpolation and selects the plane of the polar coordinates interpolation (below). The polar coordinates interpolation is completed in the plane.



When start power or reset the system, cancel polar coordinates interpolation(G13.1).

With G12.1,the planes (chose by G17、G18 or G19)used before are canceled but with G13.1 they are retored. If we reset the system polar coordinates interpolation is canceled and use G17、G18 or G19 to assign the plane.

3. We can use G code with polar coordinates interpolation

G01	linear interpolation
G02，G03	circular interpolation
G04	pause
G40，G41，G42	tool nose radius compensation
G65，G66，G67	program calling

4. Circular interpolation in polar coordinate plane:
   In the polar coordinate plane the arguments of the arc's radius with Circular interpolation(G02 or G03) are I and J.

5. The motion along the axis of the plane of the un-polar coordinates interpolation in polar coordinates interpolation:
   The knives move along these axes and have no relationship with polar coordinates interpolation.

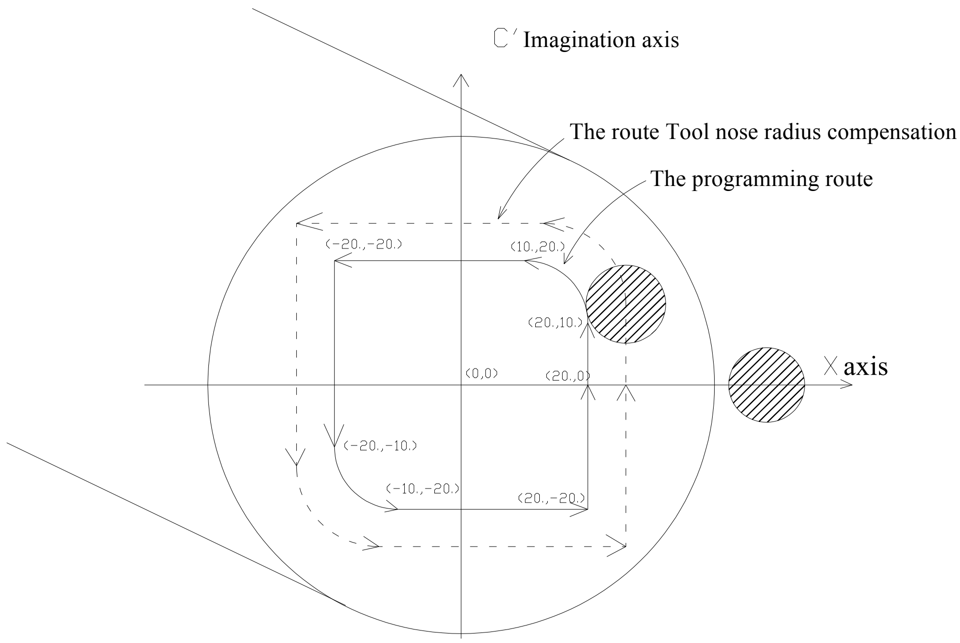
6. The display of the coordinates in polar coordinates interpolation:
   Linear axes(X) and rotational axes(C) show their real location by radius axes and others show theirs by parameters.

Restriction :

1. the coordinates in polar coordinates interpolation:
   We should set new working coordinates before G12.1 and the center of the rotational aeis is the origin in the coordinate . With G12.1 we cannot change coordinates absolutely(G92、G52、G53、G54~G59 and so on.).

2. The instructions of the tool nose radius compensation
   With G41 or G42 we cannot start or cancel G12.1 or G13.1, only can do that with G40.

3. Restart the programs:
   We cannot excute "restart" in the programs with G12.1.
4. The compliation of diameter and radius:
   We compile linear axes(X) and rotational(C) all with the compliation of radius.
   Ex:



程式說明：

```
N001 T0101
 ⋮
N009 G00 X110. C0 Z_;          //positon
N010 G40 G94;
N011 G12.1;                    //start  polar coordinates interpolation
N012 G42 G01 X20. F_;
N013 C10.;
N014 G03 X10. C20. R10.;
N015 G01 X-20.;                //use Cartesian coordinate
N016 C-10.;                    //X－C'plane
N017 G03 X-10. C-20. R10.;     //edit program
N018 G01 X20.;
N019 C0
N020 G40 X110.;
N021 G13.1;                    //cancel polar coordinates interpolation
 ⋮
N100 M30
```

1.2.9 G17、G18、G19：Plane Selection

Format：

G17；XpYp plane selection

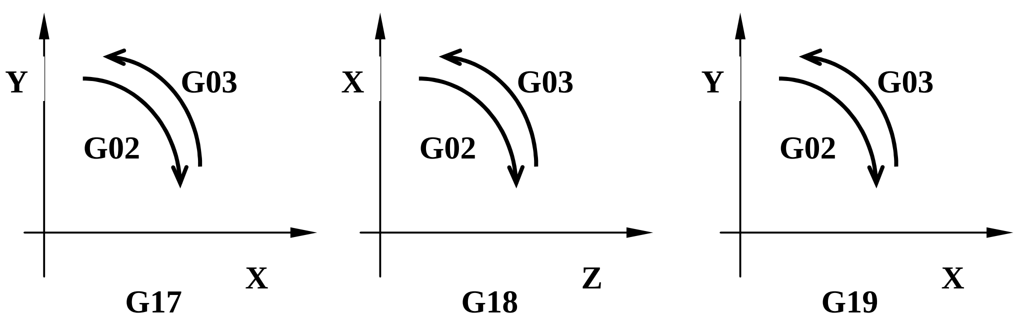
G18；ZpXp plane selection ←controller defaut

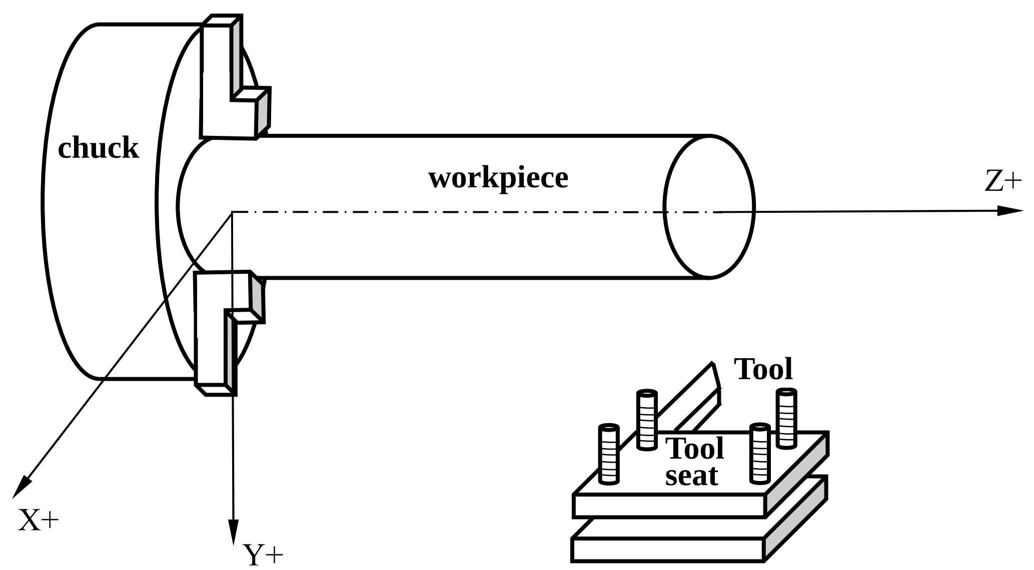
G19；YpZp plane selection

Description：

When use circular interpolation command 、 tool radius compensation command, we must use G17、G18、G19 to select the cutting plane。

PIC：





1.2.10 G20：Outer(Internal) Diameter Cutting Cycle

Format：

1. Linear cutting cycle： **G20 X(U)___ Z(W)___ F___**；

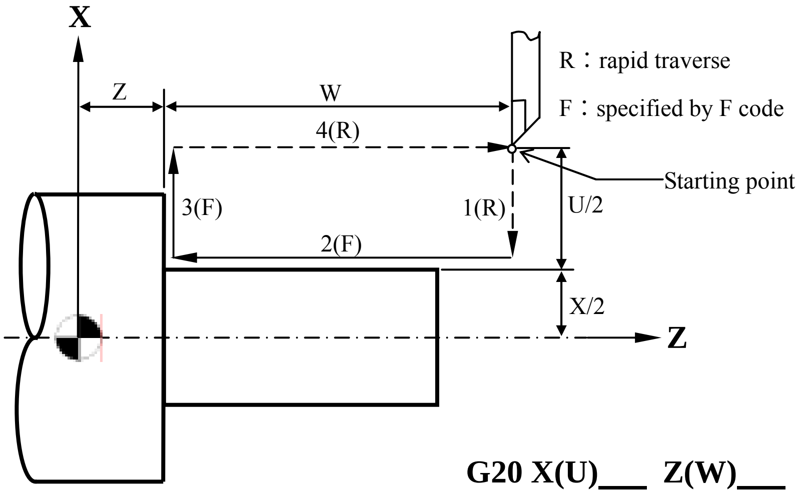
2. taper cutting cycle： **G20 X(U)___ Z(W)___ R___ F___**；

X、Z：end position of cutting(absolute)
U、W：end position of cutting(incremental)
R：difference radius value between starting point and end point
F：feedrate

Description：G20 command use in outer(internal)diameter cutting and taper cutting cycle。We can use only one block to let the program repeat many times by cycle function, and make the program more simple。

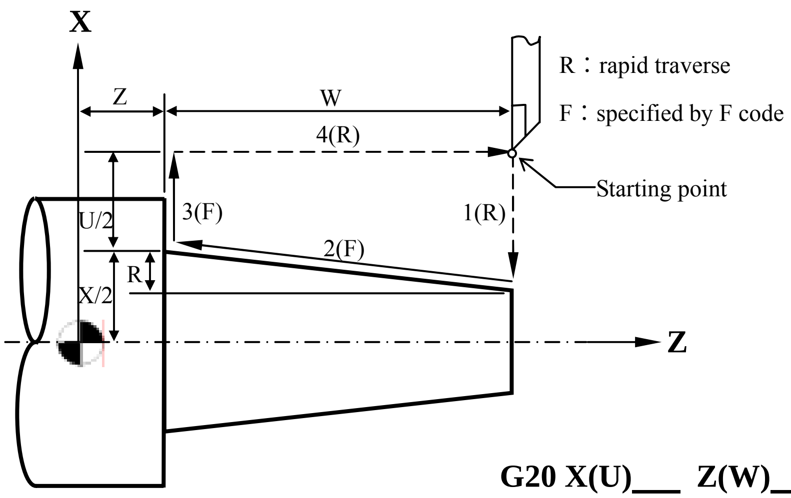
PIC：

1. Linear cutting cycle



G20 X(U)___ Z(W)___ F___；

2. Taper cutting cycle



G20 X(U)___ Z(W)___ R___ F___；

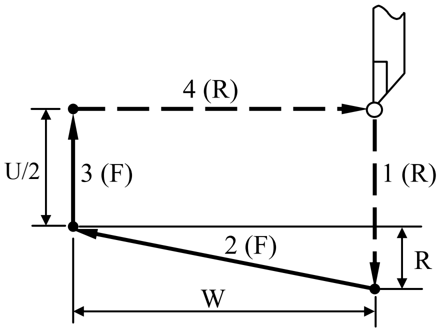
＊action description：

0. positioning the tool to starting point before cycle start；
1. after executing G20 command, tool move to specified X(U) position in X direction；
2. then the tool starts cutting to the specified X(U)、Z(W) position in specified feedrate；

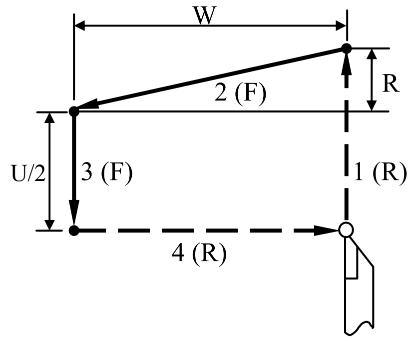
3. after cutting, tool return to starting point；
4. after reaching the starting point, tool will repeat cutting in the path by changed X(U) value；
5. when cut to specified size, the tool will stop at starting point, and the tool will wait the next cycle。

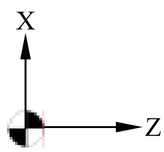
※ when we use increment mode, the relationship of U、W、R(plus or minus) and the tool path as below：

(a). U＜0, W＜0, R＜0

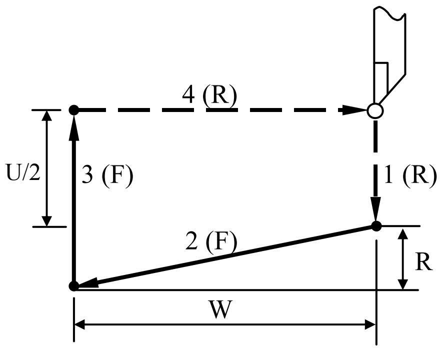


(b). U＞0, W＜0, R＞0

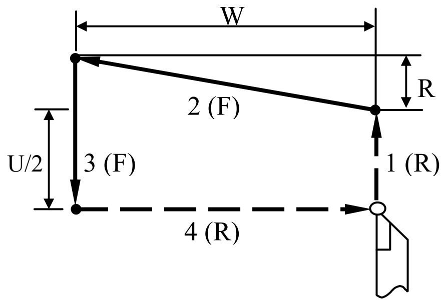




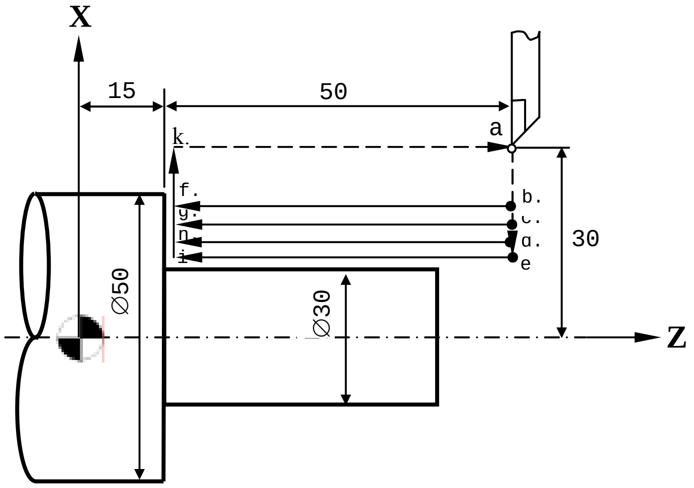
(c). U＜0, W＜0, R＞0, at ∣R∣≦∣U/2∣



(d). U＞0, W＜0, R＞0, at ∣R∣≦∣U/2∣



Example one： Straight cutting cycle



Program description：

N001 G92 S5000；//max. speed 5000 rpm

N002 T01；//use tool NO. 1

N003 G96 S130 M03；

//constant surface speed, surface speed 130 m/min, spindle rotate CW

N004 M08；//cutting liquid ON

N005 G00 X60.0 Z65.0；//positioning to **a.**(starting point)

N006 G20 X45.0 Z15.0 F600；

//execute Straight cutting cycle, feedrate 600 μm/rev, a.→b.→f.→k.→a.

X40.0；//a.→c.→g.→k.→a.

X35.0；//a.→d.→h.→k.→a.

X30.0；//a.→e.→i.→k.→a.

N007 G28 X60.0 Z70.0；

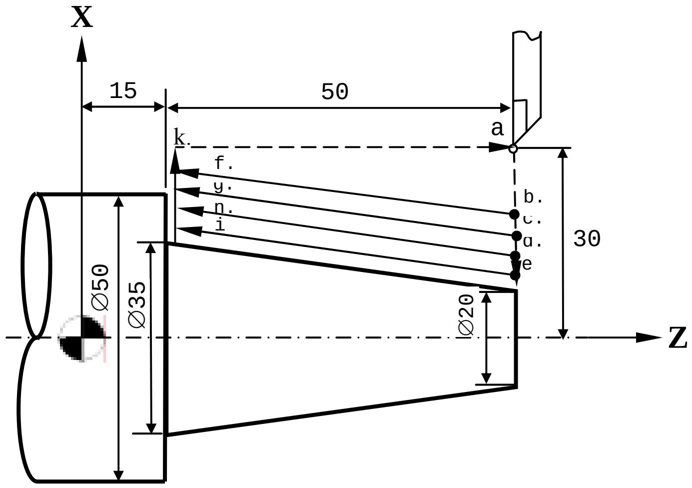
//positioning to specified mid-point then return to machine zero point

M09；//cutting liquid OFF

N007 M05；//spindle stops

N008 M30；//program ends

Example two： Taper cutting cycle



Program description：

N001 G92 S5000；//max. speed 5000 rpm

N002 T01；//use tool NO.1

N003 G96 S130 M03；

//constant surface speed, surface speed 130 m/min, spindle rotate CW

N004 M08；//cutting liquid ON

N005 G00 X60.0 Z65.0；//positioning to **a.**(starting point)

N006 G20 X53.0 Z15.0 R-7.5 F600；

//Taper cutting cycle, feedrate 600 μm/rev, a.→b.→f.→k.→a.

X48.0；//a.→c.→g.→k.→a.

X42.0；//a.→d.→h.→k.→a.

X35.0；//a.→e.→i.→k.→a.

N007 G28 X60.0 Z70.0；

// positioning to specified mid-point then return to machine zero point

N008 M09；//cutting liquid OFF

N009 M05；//spindle stops

N0010 M30；//program ends

1.2.11　G21：Thread Cutting Cycle

Format：

1.straight thread cutting cycle：**G21 X(U)___ Z(W)___ H___ {F___ / E___}** ；

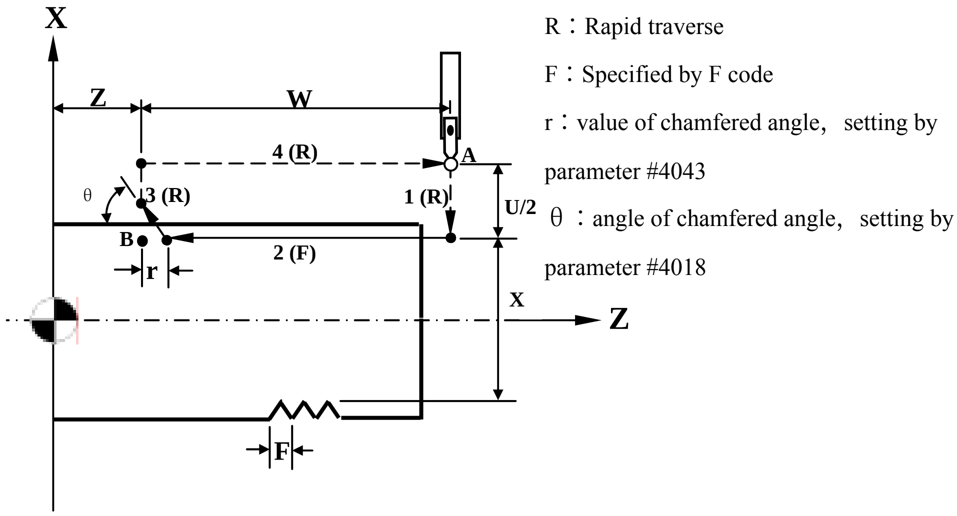
2.taper thread cutting cycle：**G21 X(U)___ Z(W)___ R___ H___ {F___ / E___}** ；

X、Z：end point of cutting(absolute)
U、W：end point of cutting(incremental)
R：difference radius value between starting point and end point
F：screw lead of Metric system(unit：mm/tooth)
E：screw lead of English system(unit：tooth/mm)
H：number of teeth
(ex：H3 for cutting the screw of 3 teeth type。In case of H command, F：pitch of teeth)

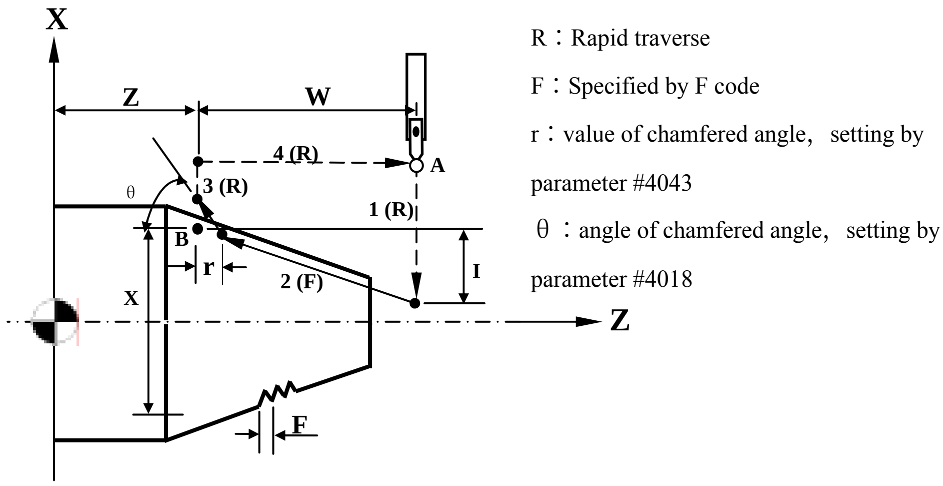
Description：G21 command is thread cutting cycle。It simplifies many repeating thread cutting blocks into one single block。

PIC：

1. Straight thread cutting cycle：**G21 X(U)___ Z(W)___ F___；**



2. Taper thread cutting cycle：**G21 X(U)___ Z(W)___ R___ F___；**

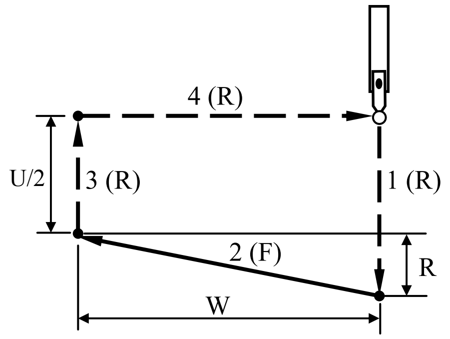


＊ Action description：

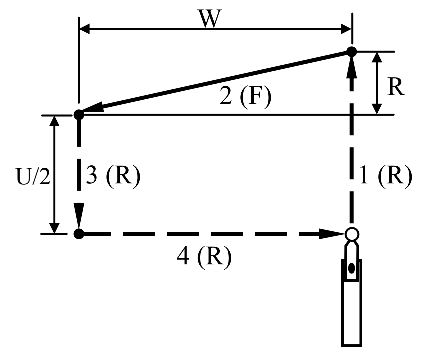
6. We should positioning the tool to starting point before cycle starts；
7. After executing G24 command, tool moves at the X axis direction and reaches to the specified X(U) position；
8. Then tool start cutting to the specified X(U)、Z(W) by specified F code；
9. After cutting, the tool returns to starting point；
10. After reaching to the starting point, tool will repeat cutting in the path by changed X(U) value(the changed value is the value that we cutting each time, it can reference tool feed value table in G33)；
11. When tool cut to specified size, the tool will stop at starting point, and wait to the next cycle。

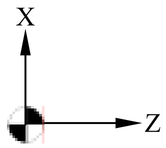
※ When we use increment mode, the relationship of U、W、R(plus or minus) and the tool path as below：

(a). U<0, W<0, R<0

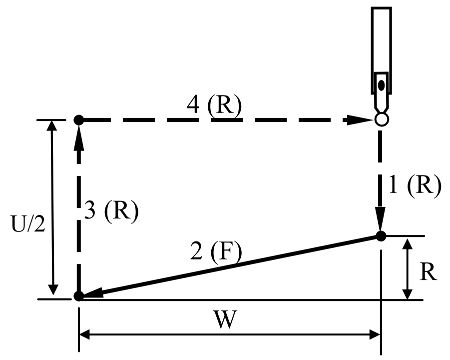


(b). U>0, W<0, R>0

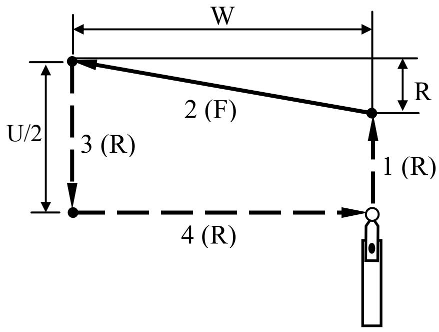




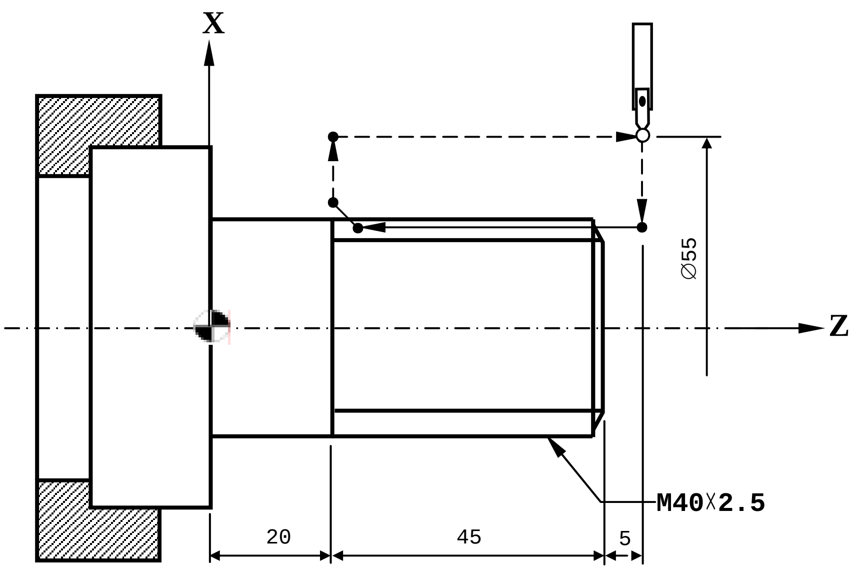
(c). U<0, W<0, R>0, at ∣R∣≦∣U/2∣



(d). U>0, W<0, R>0, at ∣R∣≦∣U/2∣



Example one： straight thread cutting cycle, 3 teeth type



Program description：

N001 T03；//use tool NO.3

N002 G97 S600 M03；//constant rotate speed, 600 rpm CW

N003 G00 X50.0 Z70.0；//positioning to the starting point of cycle

N004 M08；//cutting liquid ON

N005 G21 X39.0 Z20.0 H3 F2.5；//execute thread cutting, 3 teeth type, first cycle

X38.3；//second cycle

X37.7；//third cycle

X37.3；//fourth cycle

X36.9；//fifth cycle

X36.75；//sixth cycle

N006 G28 X60.0 Z75.0；

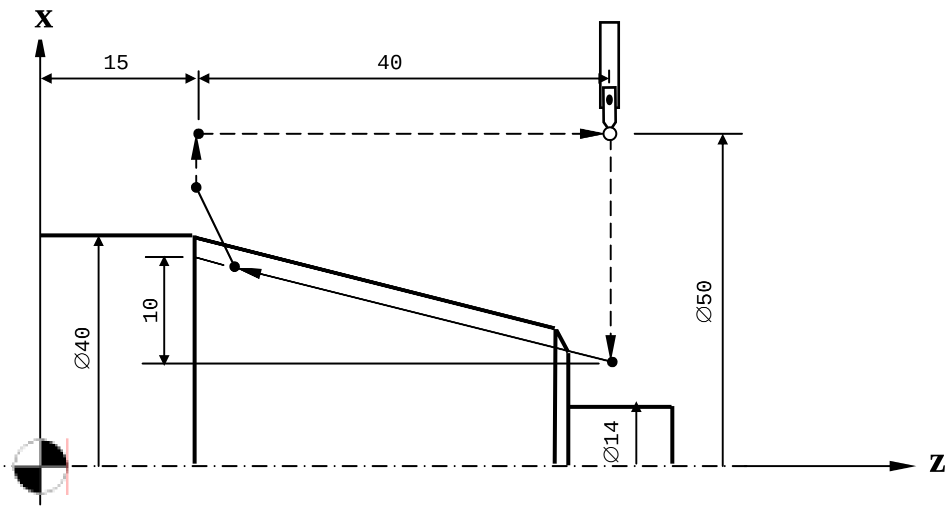
//positioning to specified mid-point and return to machine zero point

N007 M09；//cutting liquid OFF

N008 M05；//spindle stops

N009 M30；//program ends

Example two：Taper thread cutting cycle, single tooth type



Program description：

N001 T03；//use tool NO.3

N002 G97 S600 M03；//constant rotate speed, 600 rpm CW

N003 G00 X50.0 Z55.0；//positioning to the starting point of cycle

N004 M08；//cutting liquid ON

N005 G21 X39.0 Z15.0 R-10.0 F2.5；//execute taper thread cutting cycle, first cycle

X38.3；//second cycle

X37.7；//third cycle

X37.3；//fourth cycle

X36.9；//fifth cycle

X36.75；//sixth cycle

N006 G28 X60.0 Z70.0；

// positioning to specified mid-point and return to machine zero point

N007 M09；//cutting liquid OFF

N008 M05；//spindle stops

N009 M30；//program ends

1.2.12 G24：End Face Turning Cycle

Format：

1. straight end face cutting cycle：**G24 X(U)___ Z(W)___ F___**；

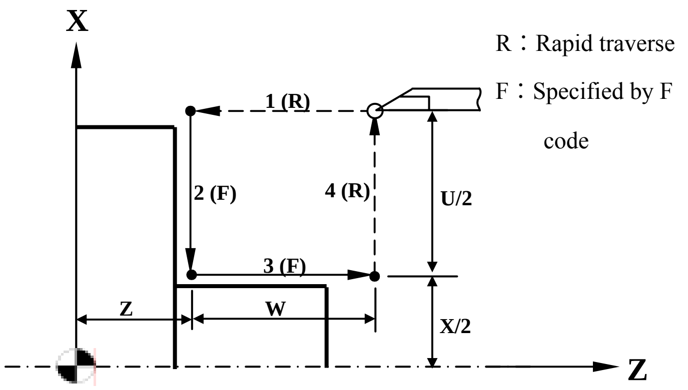
2. taper end face cutting cycle：**G24 X(U)___ Z(W)___ R___ F___**；

X、Z：end position of cutting(absolute)
U、W：end position of cutting(incremental)
R：difference length from starting point to end point
F：feedrate

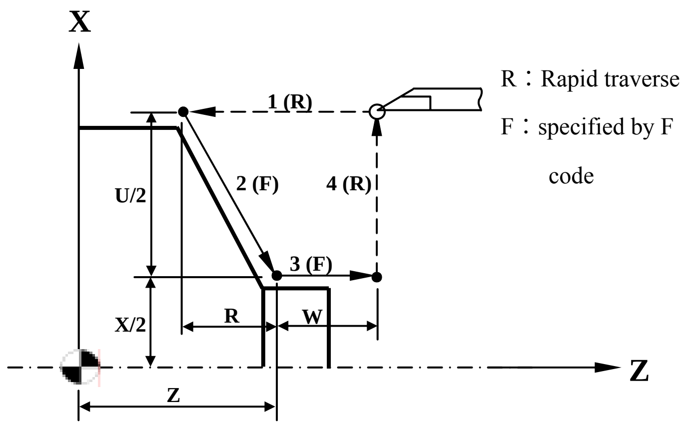
Description：G24 command is end face cutting cycle。It simplifies many repeating end face cutting blocks into one single block。

PIC：

1. Straight end face cutting cycle：**G24 X(U)___ Z(W)___ F___；**



2. Taper end face cutting cycle：**G24 X(U)___ Z(W)___ R___ F___；**



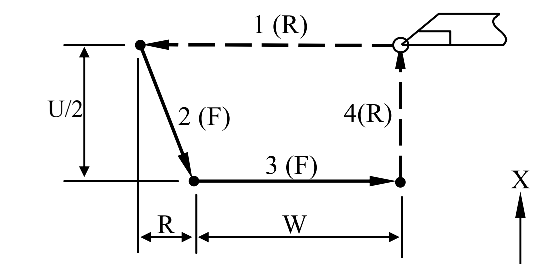
＊ Action description：

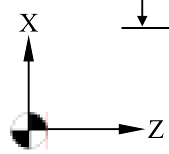
12. We should positioning the tool to starting point before cycle starts；
13. After executing G24 command, the tool will move at Z direction and reach the specified Z(W) position；
14. Then the tool will cut to specified X(U)、Z(W) by specified feedrate；
15. After finishing cutting, the tool returns to starting point；
16. After reaching to the starting point, tool will repeat cutting in the path by changed Z(W) value；
17. When reach to the specified size, the tool will stop at starting point, and wait the

next cycle。

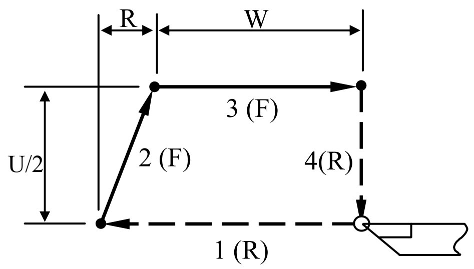
※ when we use increment mode, the relationship of U、W、R(plus or minus) and the tool path as below：

(a). U<0, W<0, R<0

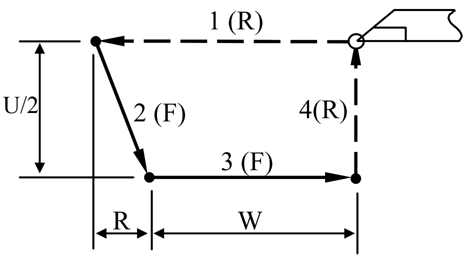




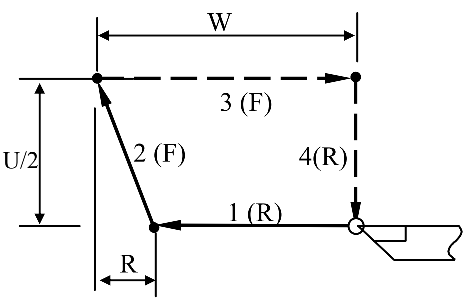
(b). U>0, W<0, R<0



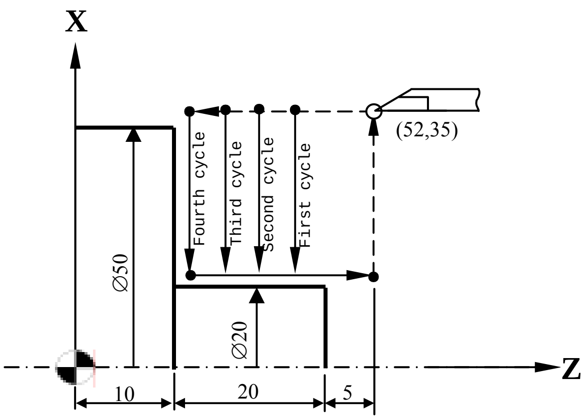
(c). U<0, W<0, R>0, at ︱R︱≦︱w︱



(d). U>0, W<0, R>0, at ︱R︱≦︱w︱



Example one：Straight end face cutting cycle



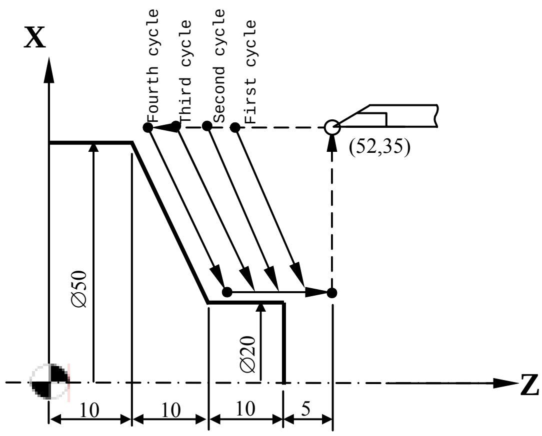
Program description：

N001 G92 S3000；//max. rotate speed 3000 rpm

N002 T01；//use tool NO. 1

N003 G96 S130 M03；//constant surface speed, surface speed 130 m/min
N004 M08；//cutting liquid ON
N005 G00 X52.0 Z35.0；//positioning to starting point of cycle
N006 G24 X20.0 Z25.0 F600；
//execute straight end face cutting, feedrate 600 μm/rev, first cycle
Z20.0；//second cycle
Z15.0；//third cycle
Z10.0；//fourth cycle
N007 G28 X70.0 Z40.0；
//positioning to specified mid-point, then return to the machine zero point
N008 M09；//cutting liquid OFF
N009 M05；//spindle stops
N0010 M30；//program ends

Example two：Taper end face cutting cycle



Program description：
N001 G92 S3000；//max. rotate speed 3000 rpm
N002 T01；//use tool NO.1
N003 G96 S130 M03；//constant surface speed, surface speed 130 m/min
N004 M08；//cutting liquid ON
N005 G00 X52.0 Z35.0；//positioning to starting point of cycle
N006 G24 X20.0 Z32.0 R-10.0 F600；
//execute taper end face cutting cycle, feedrate 600 μm/rev, first cycle
Z28.0；//second cycle
Z24.0；//third cycle
Z20.0；//fourth cycle

N007 G28 X70.0 Z35.0；

//positioning to specified mid-point, then return to machine zero point

N008 M09；//cutting liquid OFF

N009 M05；//spindle stops

N0010 M30；//program ends

1.2.13 G28：Reference point return

Format：

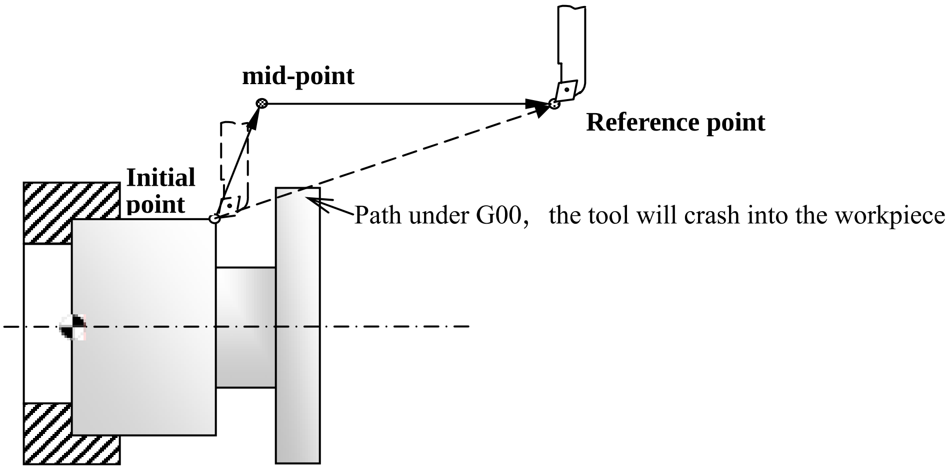
G28 X(U)___ Z(W)___；

X、Z：specified mid-point(absolute)
U、W：specified mid-point(incremental)

Description：when G28 command is executed, tool will move to specified mid-point and then return to reference point(machine zero point) by the speed of G00；the main aim is warded the workpiece in order not to crash。
In absolute mode, it is the absolute value to the mid-point；in increment mode, it is the increased value from starting point to mid-point。

※ Notice：Before G28 command is executed, we must cancel tool compensation function in order to sure the returned action correctly。

PIC：



1.2.14 G29：Return from reference point

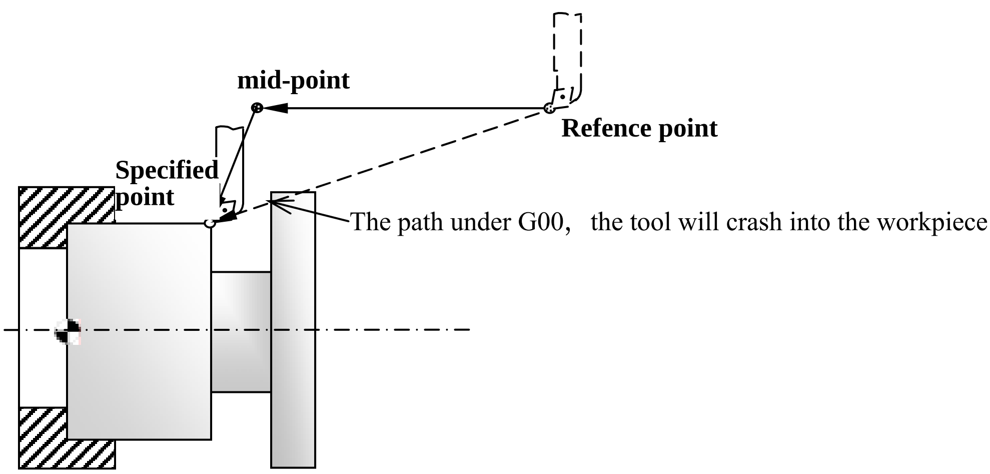
Format：

G29 X(U)___ Z(W)___ ；

X、Z：specified point(absolute)
U、W：specified point(incremental)

Description：G29 command only be executed after you execute G28 command，It can move to specified position through mid-point from reference point。G29 command can not use alone，because G29 does not specify own mid-point，it use the mid-point that G28 specify，so we must execute G28 before executing G29。
In absolute mode，it is the absolute value to the mid-point；in increment mode，it is the increased value from starting point to mid-point。

PIC：



1.2.15 G30：Any reference point return

Format：

G30 Pn X(U)__ Z(W)__ ；

X、Y、Z：coordinate value of mid-point；

P**n**：specify the reference point(setting parameter #2801 ~ #2860)

P1：machine zero point；

P2：second reference point；

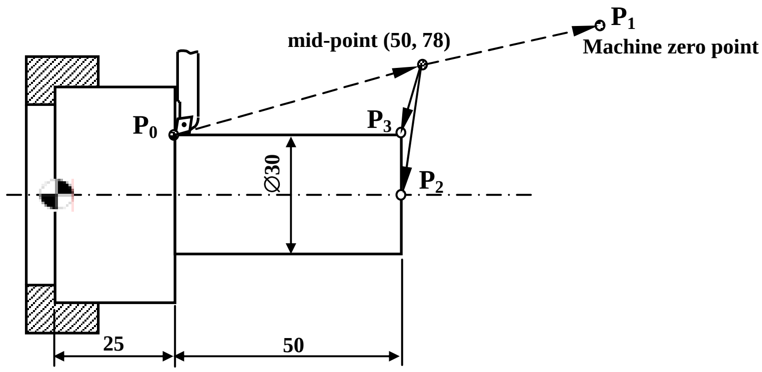
P_default is P2；

Description：

In order to be convenient in tooling exchange and inspection。We specify another reference point from the machine zero point by parameter。The machine need not to return to machine zero point when tool exchange, and increase the efficiency。Usage of this command is the same as G28 command expect the point that tool return is different。G30 command usually use in tool exchange position differ from origin point。Movement mode is G00 mode。

<Notice>This command usually use in auto tool exchange。We should cancel tool compensation function before executing G30 in safety。

Example：



Program description：

Path one ………G30 P01 X50.0 Z78.0；// P_0→mid-point→P_1

Path two ………G30 P02 X50.0 Z78.0；// P_0→mid-point→P_2

or G30 X50.0 Z78.0；//default P_2

Path three ………G30 P03 X50.0 Z78.0；// P_0→mid-point→P_3

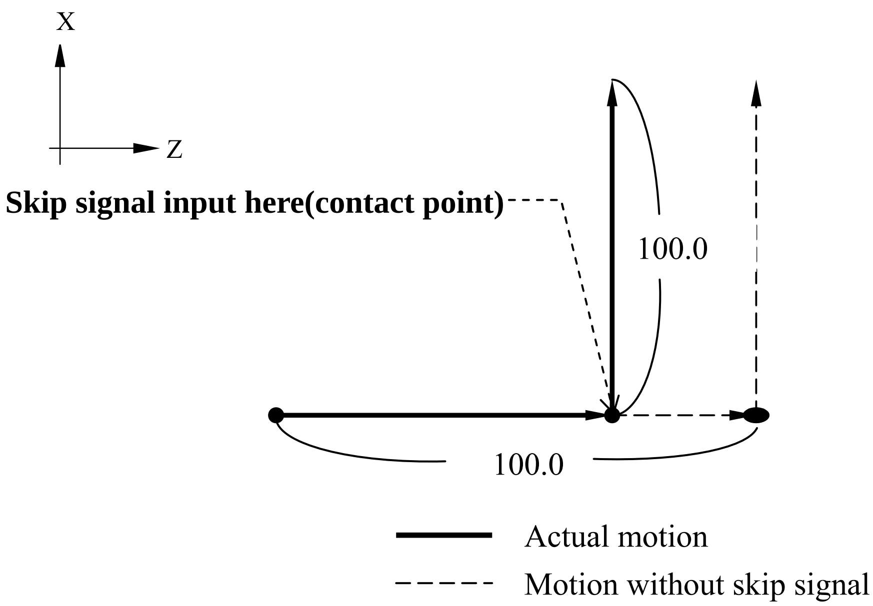
1.2.16 G31：Skip Function

Format：G31 X(U)__ Z(W)__ F__；

X、Z：specified position(absolute)
U、W：specified position(incremental)
F：feedrate

Description：Skip function is used in unknown end of program。It specifies the end point, when tool reaches the contact point, machine will receive the signal then LADDER C BIT ON。G31 command will record the present position of machine and interrupt action of G31, and to execute the next block。

Example one：Increment mode

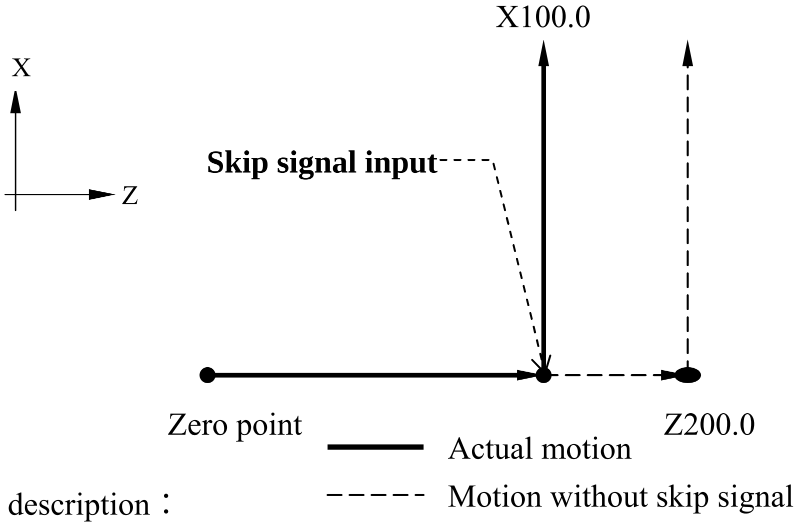


Program description：

N001 G31 W100.0 F100；//origin path until run into contact point

N002 U100.0；//use contact point to be the relative coordinate and change the path to specified position

Example two：Absolute mode for one axis

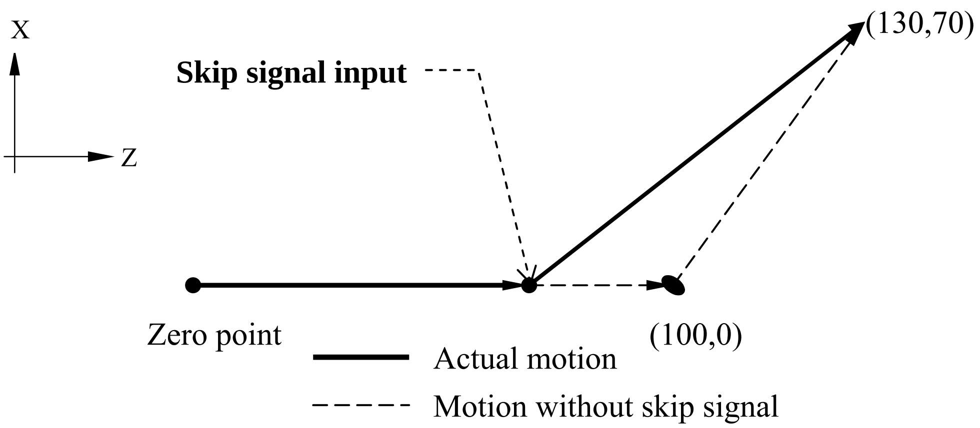


Program description：

N001 G31 Z200.0 F100；// origin path until run into contact point

N002 X100.0；// use zero point to be the relative coordinate and change the path to specified position

Example three：Absolute mode for two axes



Program description：

N001 G31 Z100.0 F1000；// origin path until run into contact point

N002 Z130.0 X70.0；// use zero point to be the relative coordinate and change the path to specified position

1.2.17 G33：Thread cutting

Format：

(1)continuous thread cutting： **G33 Z(W)___ Q___ $\begin{Bmatrix} F\_\_\_ \\ E\_\_\_ \end{Bmatrix}$;**

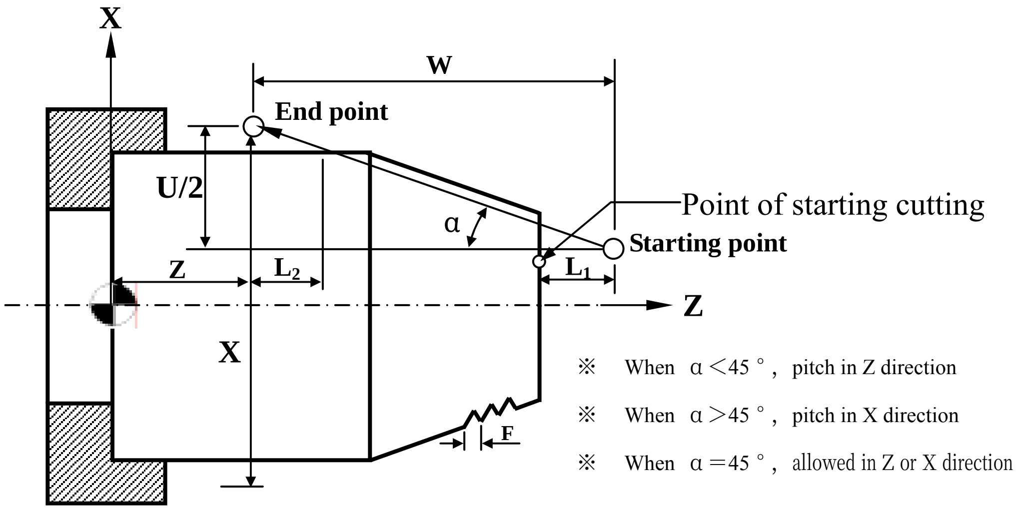
(2)circular threading： **G33 X(U)___ Z(W)___ Q___ $\begin{Bmatrix} F\_\_\_ \\ E\_\_\_ \end{Bmatrix}$;**

(3)multiple-thread cutting： **G33 X(U)___ Q___ $\begin{Bmatrix} F\_\_\_ \\ E\_\_\_ \end{Bmatrix}$;**

X、Z：specified position(absolute)
U、W：specified position(incremental)
F：lead in longitudinal direction←common thread、Metric system
E：lead in longitudinal direction←pricise thread、English system
Q：the shift of the threading start angle, this function can let the tool starting cutting point be the same when cutting rotating workpiece。Use in multiple-thread cutting, we can use **default value Q= 0**° when single-thread cutting, (range：0.001~360.000°)

Description：G33 command executes multiple-thread cutting、circular threading、continuous thread cutting, and it based on spindle rotates and tool feed executing synchronously。

PIC：



Notice：

※ **Input unit and modal of E、F value as below table：table 1. Metric system、table 2. English system**

Input unit	A(0.01mm)			B(0.001mm)			C(0.0001mm)		
Command position	F(mm/rev)	E(mm/rev)	E(pc/inch)	F(mm/rev)	E(mm/rev)	E(pc/inch)	F(mm/rev)	E(mm/rev)	E(pc/inch)
Min. command unit	1(-0.001) (1,-1.000)	1(-0.0001) (1,-1.0000)	1(-1) (1.-1.0)	1(-0.0001) (1.-1.0000)	1(-0.00001) (1.-1.00000)	1(-1) (1.-1.00)	1(-0.00001) (1.-1.00000)	1(-0.000001) (1.-1.000000)	1(-1) (1.-1.000)
Command range	0.001 9999.999	0.0001 9999.9999	0.1 9999999.9	0.001 999.9999	0.00001 999.99999	0.01 999999.9	0.00001 99.99999	0.000001 99.999999	0.001 99999.999

Table 1. input by Metric system

Input unit	A(0.00inch)			B(0.0001inch)			C(0.00001inch)		
Command position	F(inch/rev)	E(inch/rev)	E(pc/inch)	F(inch/rev)	E(inch/rev)	E(pc/inch)	F(inch/rev)	E(inch/rev)	E(pc/inch)
Min. command unit	1(-0.00001) (1,-1.00000)	1(-0.000001) (1,-1.000000)	1(-1) (1.-1.000)	1(-0.000001) (1.-1.000000)	1(-0.000000 1) (1.-1.000000 0)	1(-1) (1.-1.0000)	1(-0.0000001) (1.-1.000000 0)	1(-0.00000001) (1.-1.00000000)	1(-1) (1.-1.00000)
Command range	0.00001 999.99999	0.000001 99.999999	0.001 99999.999	0.000001 99.999999	0.0000001 9.9999999	0.0001 9999.9999	0.0000001 9.9999999	0.00000001 0.99999999	0.00001 999.99999

Table 2. input by English system

【Note 1】If the conversion feedrate is over than Max. cutting feedrate, the pitch will vary, the pitch is not the specified one。

(1). Slant thread cutting command and spiral thread cutting command can not use in constant surface speed mode。

(2). The spindle speed should be fixed from coarse cutting to fine cutting。

(3). If we use dwell in thread cutting, the thread will be damaged。So we can not use dwell when thread cutting。If we push down the dwell button, the thread cutting will be ended(not in G33 mode)。And it will stop in the next block。

(4). In the beginning of thread cutting, the varying cutting feed rate will be compared with the limitation of cutting speed。The alarm of error operation will be occurred if it excess the speed limitation【Note 1】。

(5). In the thread cutting, it is possible that the varying cutting speed excess the limitation cutting speed for keeping the constant pitch。

(6). The limitation of spindle speed is as below：

$$1 \leqq \text{Revolution}(\mathbf{R}) \leqq \frac{\text{Max. feedrate}}{\text{Lead of thread}}$$

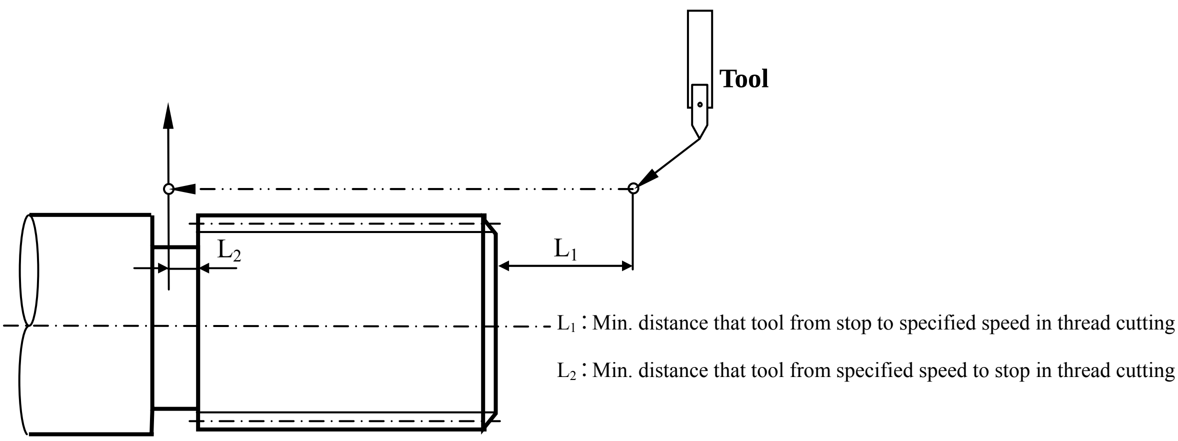
R：spindle rotate speed(rpm)
Lead of thread(F)：mm or inch
Feedrate：mm/min or inch/min

(7). At the near of start and end thread cutting point, the incorrect pitch length will occur due to the delayed of servo system。Therefore the thread length we want should be the specified thread length(L_1、L_2) plus the thread length。

※L_1、L_2 computer formula as below：

$$L_1 \quad \frac{S \times P}{400}$$

$$L_2 \quad \frac{S \times P}{1800}$$



(8). The external speed control is effective during the thread cutting。But the feed rate of external speed control and revolving of spindle could not be allowed executed in synchronous。

(9). In non-synchronous feed(G94) command, the thread cutting command will become synchronous feed type。

(10). During the thread cutting, manual adjustment of speed is effective, too。 If you manually adjust the speed during thread cutting, it will produce an incorrect thread cutting due to delay of servo system。

(11). It will execute thread cutting and temporarily cancel tool nose radius compensation when there is any thread cutting command executed during tool nose radius compensation。

(12). During the G33 command executing, if you change to other automatic modes it will not execute thread cutting。 And terminate automatic spinning after single block executing。

(13). During the G33 command executing, if you change to manual mode it will not execute thread cutting。 And terminate automatic spinning after single block executing。 During the spinning of single block, it will not execute thread cutting。 And terminate automatic spinning after single block executing。

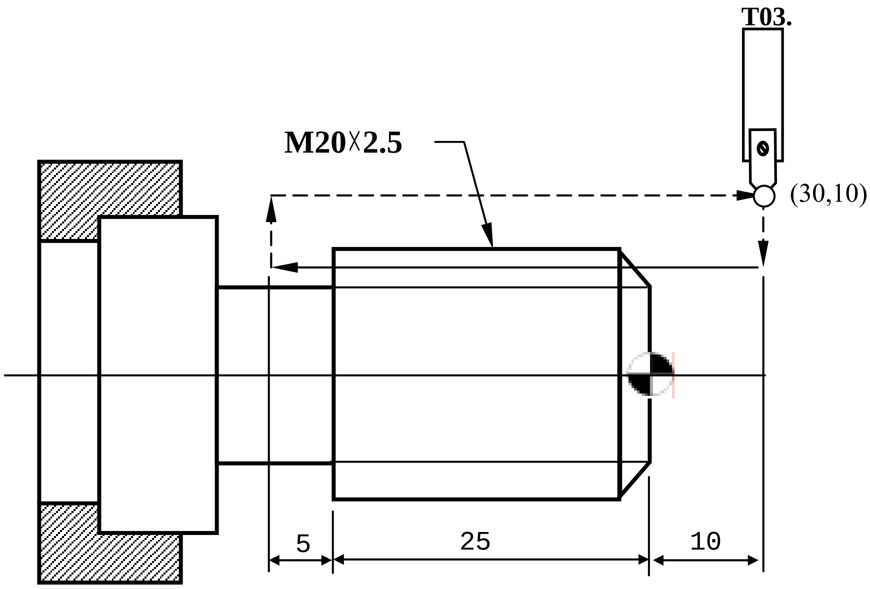
(14). During the thread cutting, it begins to move till the appearance of synchronous signal per one revolution from spinning encoder。 But in case of a thread cutting on a system which there is another thread cutting command, it will start to move and not wait the appearance of synchronous signal per one revolution from backward encoder。 Therefore, please do not execute duplicated system of thread cutting command。

(15). Tool feed value of thread cutting reference table：

English system		depth of tooth h = 0.6403P				P = Pitch		
Thread number per inch		8	10	12	14	16	18	24
Pitch of thread(in)		0.1250	0.1000	0.0833	0.0714	0.0625	0.0556	0.0417
Height of thread 0.6403P(in)		0.0800	0.0640	0.0533	0.0457	0.0400	0.0356	0.0267
Numbers of cutting and the value of cutting(diameter)	1	0.0472	0.0394	0.0354	0.0315	0.0315	0.0315	0.0315
	2	0.0276	0.0276	0.0236	0.0236	0.0236	0.0236	0.0157
	3	0.0236	0.0236	0.0236	0.0197	0.0197	0.0118	0.0062
	4	0.0200	0.0157	0.0157	0.0118	0.0052	0.0043	
	5	0.0200	0.0157	0.0083	0.0048			
	6	0.0158	0.0060					
	7	0.0058						

Metric system		depth of tooth = 0.06495P				P = Pitch		
Pitch of thread(mm)		4.0	3.5	3.0	2.5	2.0	1.5	1.0
Height of thread 0.6495P(mm)		2.598	2.273	1.949	1.624	1.299	0.974	0.650
Numbers of cutting and the value of cutting(diameter)	1	1.5	1.5	1.2	1.0	0.9	0.8	0.7
	2	0.8	0.7	0.7	0.7	0.6	0.6	0.4
	3	0.6	0.6	0.6	0.6	0.6	0.4	0.2
	4	0.6	0.6	0.4	0.4	0.4	0.16	
	5	0.4	0.4	0.4	0.4	0.1		
	6	0.4	0.4	0.4	0.15			
	7	0.4	0.2	0.2				
	8	0.3	0.15					
	9	0.2						

Example one：



Program description：

T03；//use tool NO.3

G97 S1000 M03；//spindle rotate CW 1000 rpm,　constant rotate speed

M08；//cutting liquid ON

G00 X30.0 Z10.0；//positioning to starting point of cutting

X19.0；//

G33 Z-30.0 F2.5；//

G00 X30.0；//

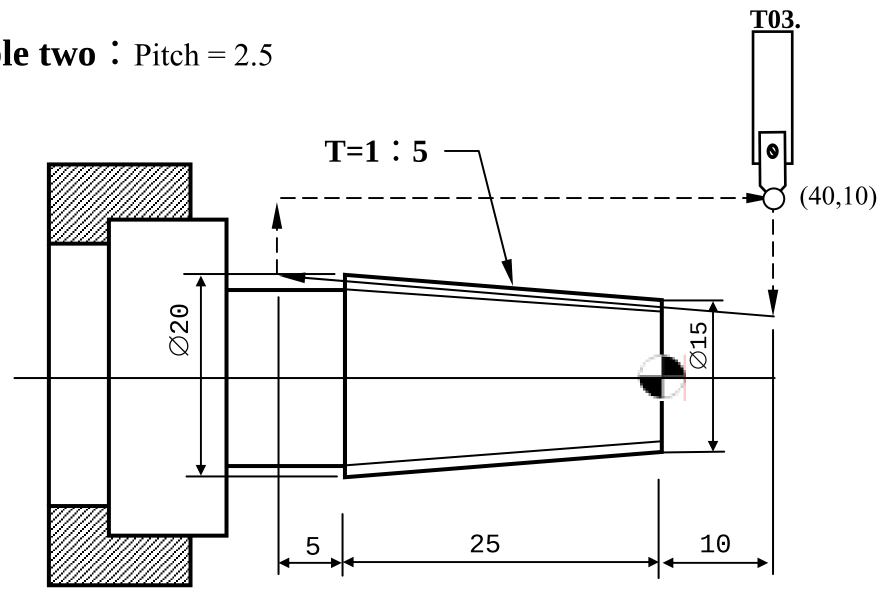
Z10.0；//

X18.3；//

First cutting 1.0 mm

G33 Z-30.0 F2.5；//
G00 X30.0；//
Z10.0；// **Second cutting 0.7 mm**
X17.7；//
G33 Z-30.0 F2.5；//
G00 X30.0；// **Third cutting 0.6 mm**
Z10.0；//
X17.3；//
G33 Z-30.0；//
G00 X30.0；// **Fourth cutting 0.4 mm**
Z10.0；//
X16.9；//
G33 Z-30.0 F2.5；//
G00 X30.0；// **Fifth cutting 0.4 mm**
Z10.0；//
X16.75；//
G33 Z-30.0 F2.5；//
G00 X30.0；// **Sixth cutting 0.15 mm**
Z10.0；//
G28 X50.0 Z30.0；
//positioning to specified mid-point, then return to machine zero point
M09；//cutting liquid OFF
M05；//spindle stops
M30；//program ends

Example two：Pitch = 2.5



Program description：

T03；//use tool NO.3
G97 S1000 M03；//spindle rotate CW 1000 rpm, constant rotate speed
M08；//cutting liquid ON
G00 X40.0 Z10.0；//positioning to starting point of cutting

X12.0；//

G33 X20.0 Z-30.0 F2.5；//
G00 X40.0；//
Z10.0；//
X11.3；//

First cutting 1.0 mm

G33 X19.3 Z-30.0 F2.5；//
G00 X40.0；//
Z10.0；//
X10.7；//

Second cutting 0.7 mm

G33 X18.7 Z-30.0 F2.5；//
G00 X40.0；//
Z10.0；//
X10.3；//

Third cutting 0.6 mm

G33 X18.3 Z-30.0 F2.5；//
G00 X40.0；//
Z10.0；//

Fourth cutting 0.4 mm

X9.9；//

G33 X17.9 Z-30.0 F2.5；//
G00 X40.0；//
Z10.0；//
X9.75；//

Fifth cutting 0.4 mm

G33 X17.75 Z-30.0 F2.5；//
G00 X40.0；//
Z10.0；//

Sixth cutting 0.15 mm

G28 X50.0 Z30.0；

//positioning to specified mid-point,　and return to machine zero point

M09；//cutting liquid OFF

M05；//spindle stops

M30；//program ends

Tool Compensation Function(T Function)

Format：

T ＊＊；(two code form)
T ＊＊ ＊＊；(four code form)

Description：

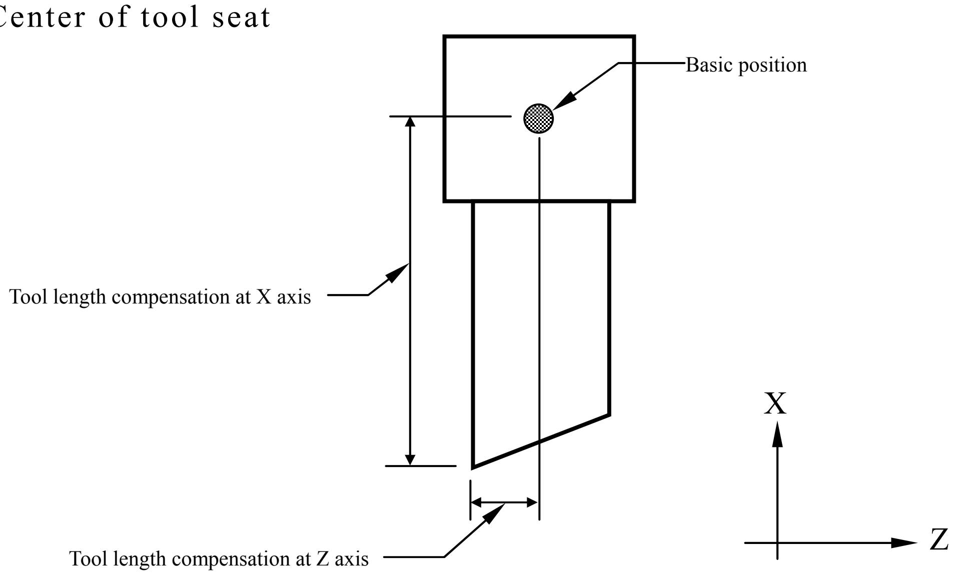
Two code form：for tool number、tool length compensation and wear compensation selection。
Four code form：the front two code for tool number, the back two code for tool length and wear compensation。

When we select compensation value and not to execute compensation action in executing T function command, then tool compensation action will be executed。When there is movement action in a block compensation action will be executed。

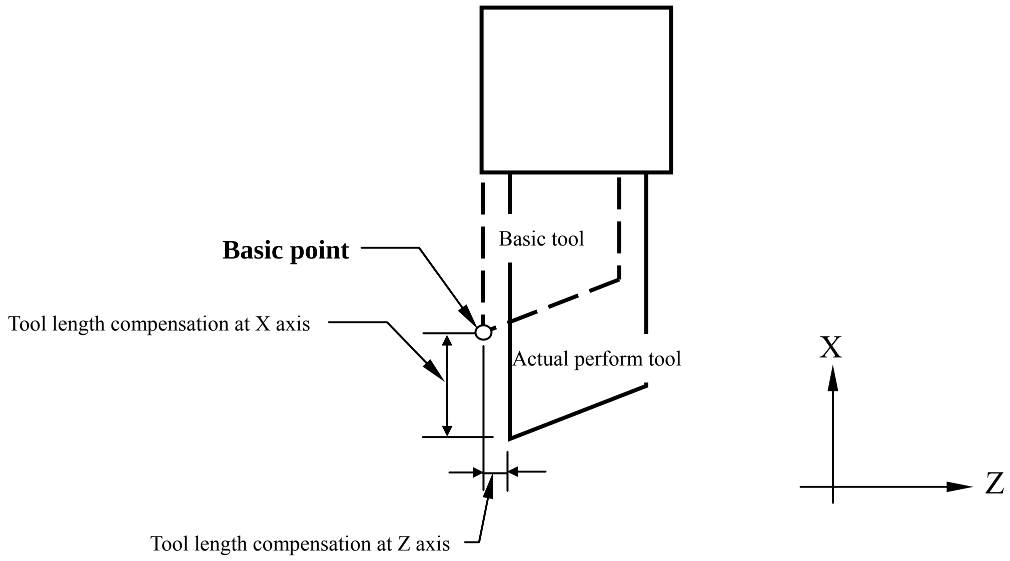
a. Modal Of Tool Length Compensation：

a-1 Tool length compensation

Do tool length compensation at the basic position of program。
The basic position of program：center of tool seat or tool nose of basic tool：
(1) Center of tool seat



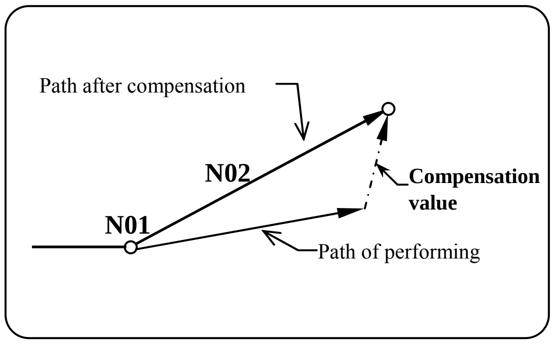
(2) Tool nose of basic tool



b. Principle Of Tool Length Compensation：

b-1. Tool compensation starts

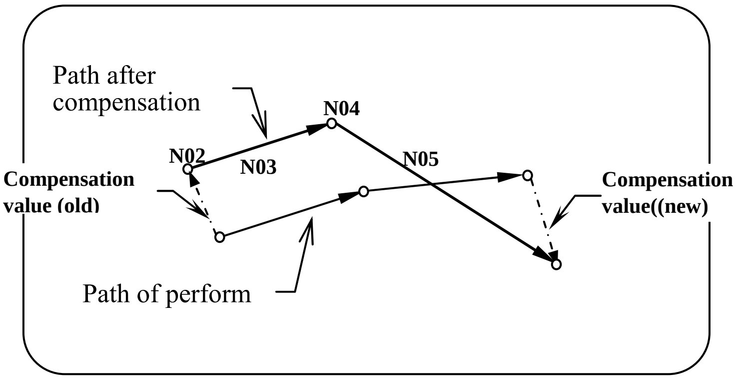
Tool compensation action is start after executing T command and executing movement command



N01 T0101；
N02 X10.0 Z10.0；

b-2. Number change of tool length compensation

When number of tool changes, offset of performance is the tool compensation value plus the number of new tool

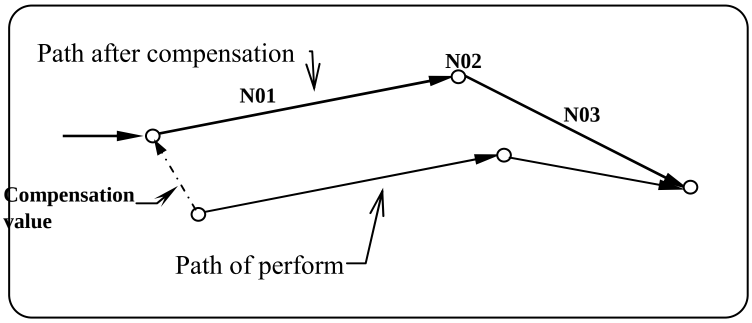


N01 T0100；
N02 G01 X10.0 Z10.0 F200；
N03 G01 X13.0 Z15.0 F300；
N04 T0200；
N05 G01 X13.0 Z20.0 F205；

b-3. Tool length compensation cancel

(1) Number of compensation is 0

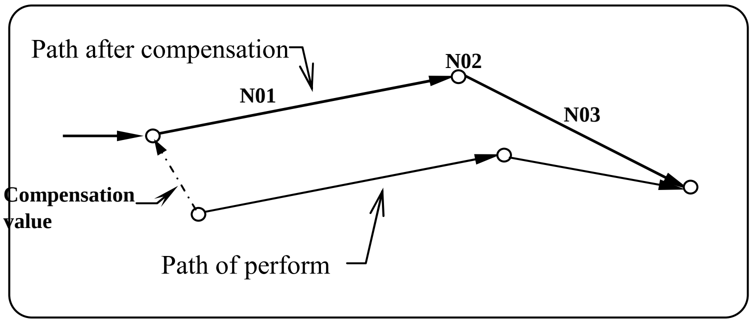
When number of compensation is “0” in T command , compensation cancels。



N01 X10.0 Z10.0 F100；
N02 T0000；
N03 G01 X10.0 Z20.0；

(2) Compensation value of command is “0”

When compensation value of tool length compensation number is “0” in T function, the compensation cancel。

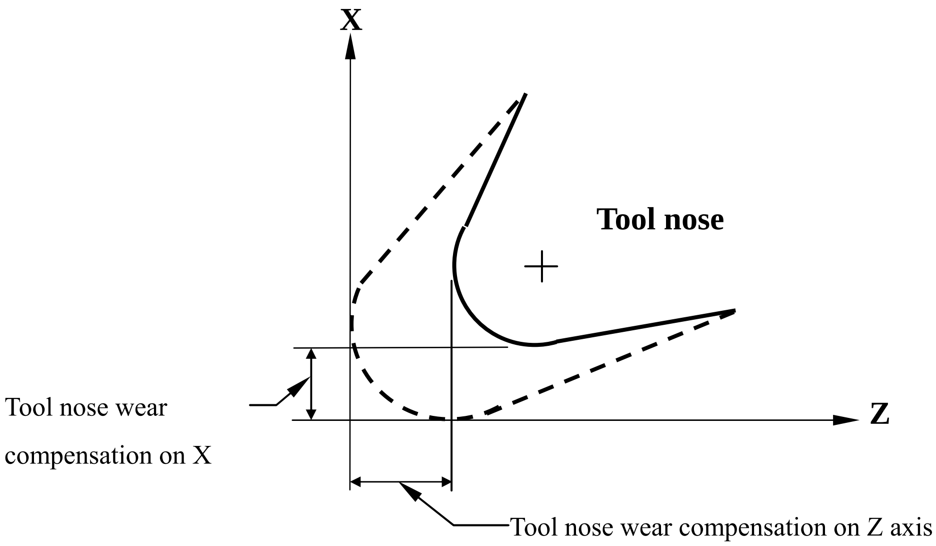


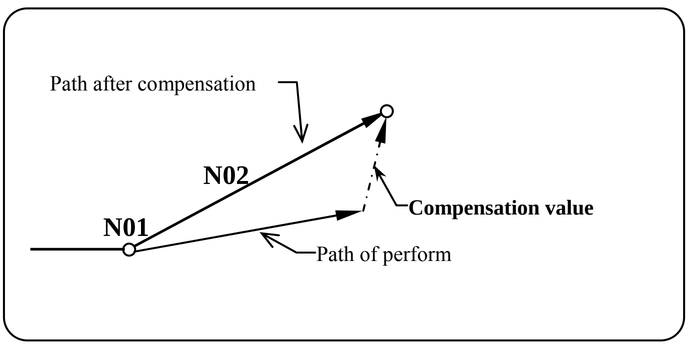
N01 G01 X10.0 Z10.0 F100；
N02 T0100；
N03 G01 X10.0 Z20.0；

c. Tool Nose Wear Compensation：

Tool nose wear compensation value setting

It can compensate when tool nose wears, this compensation value will plus geometric compensation。**Geometric compensation = tool length compensation + wear compensation**, When we specify the number of compensation, then geometric will be executed。





N01 T0102；
//start tool NO.1 compensation, the number of compensation is 2
N02 X10.0 Z10.0；

1.2.18 G41、G42、G40：Tool Nose Radius Compensation

Format：

$$\left\{ \begin{matrix} \textbf{G41} \\ \textbf{G42} \end{matrix} \right\} \ \textbf{X(U)\_\_\_ \ Z(W)\_\_\_} \ ;$$

G40；compensation cancel

X、Z：specified position(absolute)

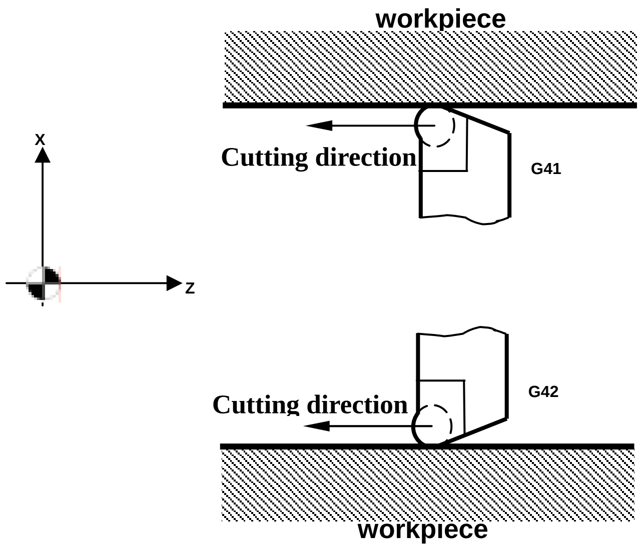
U、W：specified position(incremental)

Description：We grind a small and round nose on the tip of tool, it can increase intension of tool tip、increase the life of tool、decrease the stress、help to release the hot and improve the smooth of surface。It is called tool nose, and its radius is called tool nose radius。But when we use tool nose to cut corner、slant line or an arc, errors will occur because of the arc of tool tip, we can not perform the exactly shape of workpiece, we can use G41、G42 to adjust the error of tool nose, it can compute the error of tool nose radius exactly and compensate it。

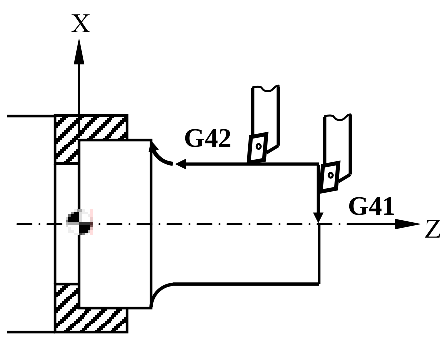
G code	Function	Position of tool
G40	Tool nose compensation cancel	Tool moves along the path of program
G41	Tool nose compensation left	Tool offsets right a specified value to the path of program
G42	Tool nose compensation right	Tool offsets left a specified value to the path of program

PIC：

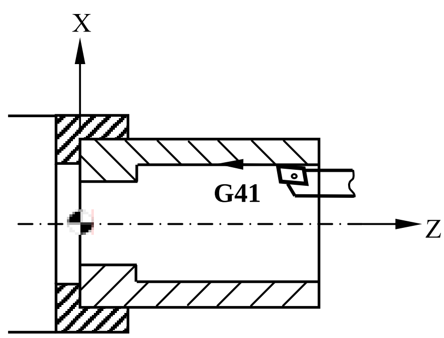
1. **Relationship between tool feed direction and workpiece, setting of compensation：**



2. Compensation setting of actually perform

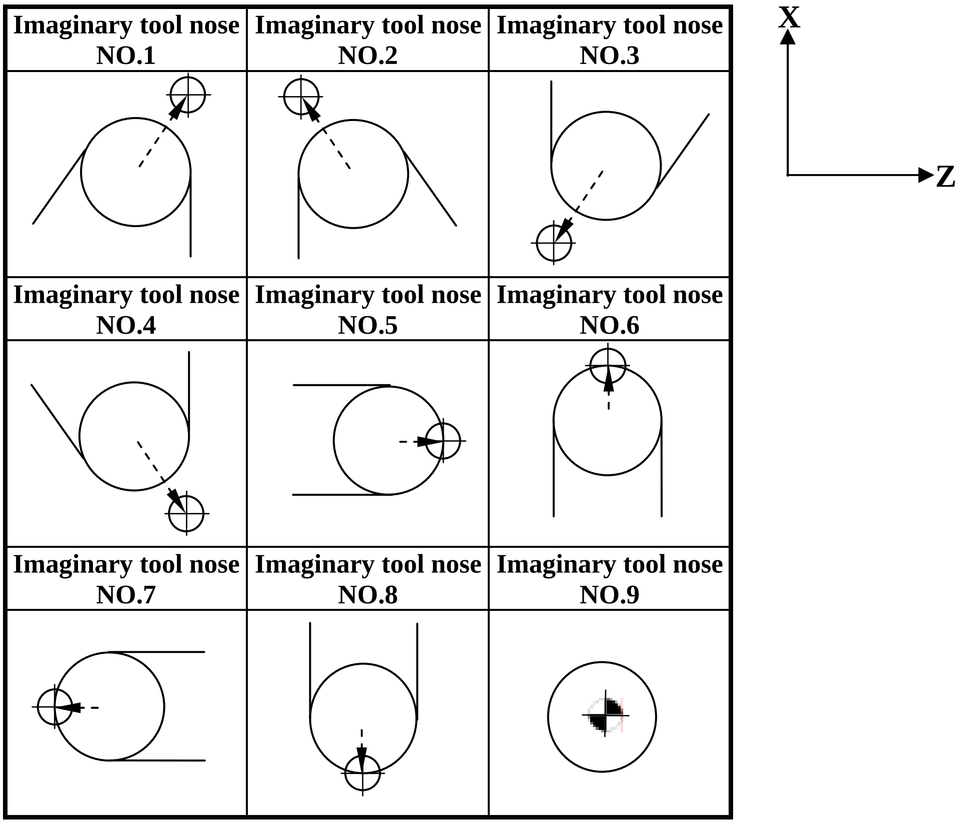


a. cutting outer radius and end surface



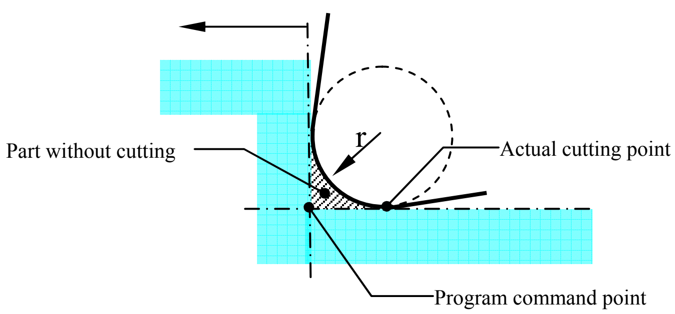
b. cutting internal radius

3. Imaginary tool nose number setting：

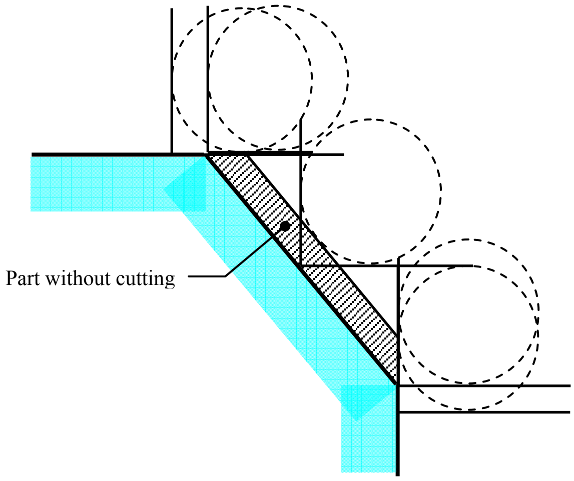


4. Compensation without tool nose：

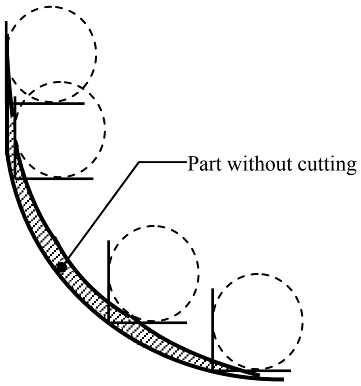
(1). Surface cutting：



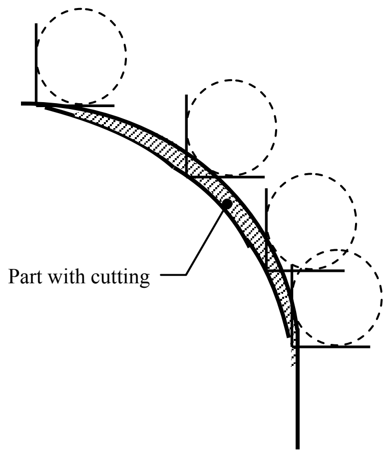
(2). Corner or slant surface：



(3). Cutting arc：



a.cutting internal of a cycle



b.cutting outer of a cycle

※ Tool Radius (R) compensation：

1. Compensation Starts：

When a block which satisfies all the following conditions is executed in start mode, the system enters the offset mode. Control during this operation is called start-up

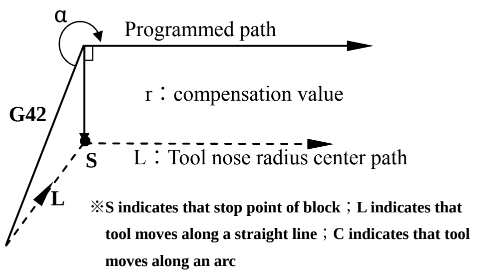
(a). G41 or G42 is contained in the block, or has been specified to set the system enters the offset mode；

(b). The offset number of tool nose compensation is not “ 00 ”；

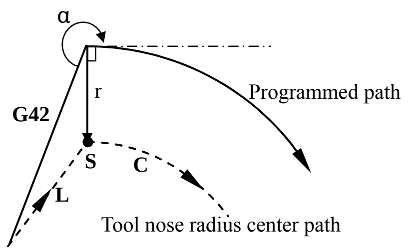
(c). X or Z moves are specified in the block and the move distance is not zero

(a). Inner Side (180°≧ α)

(i). Linear→Linear

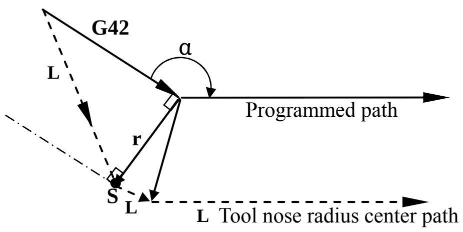


(ii). Linear→Circular

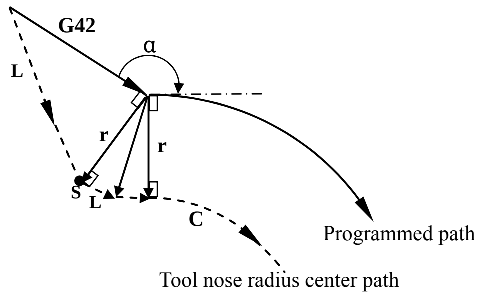


(b). Outer Side (90°≧ α <180°)

(i). Linear→Linear

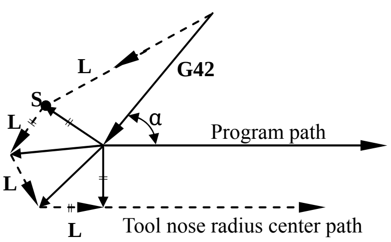


(ii). Linear→Circular

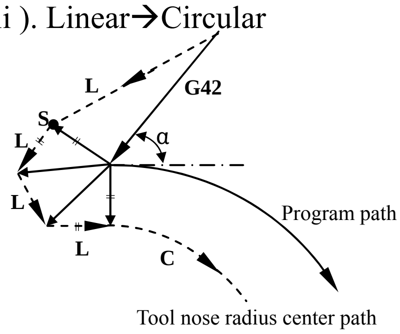


(c). Outer Side (α <90°)

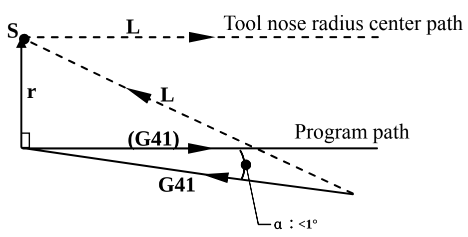
(i). Linear→Linear



(ii). Linear→Circular



(d). At corner(<1°) outer Linear→Linear Perform(α <1°)

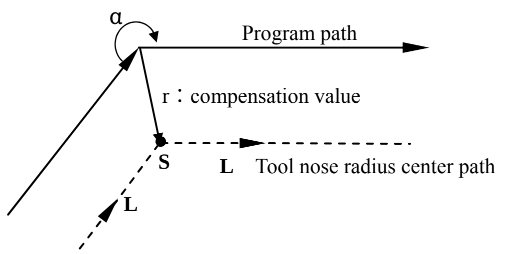


2. Compensation mode：

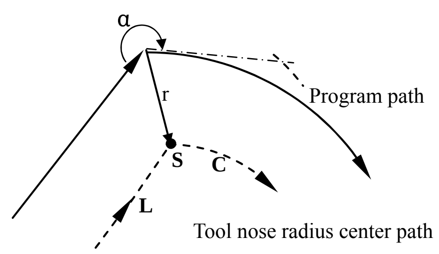
In compensation mode, it uses compensation even during positioning；In compensation mode, it does not specify movement block(M Function or dwell .etc.) it can not be specified continuity；If it is specified continuity, over cutting or not enough cutting will occur。

(a). Inner Side (180°≦α)

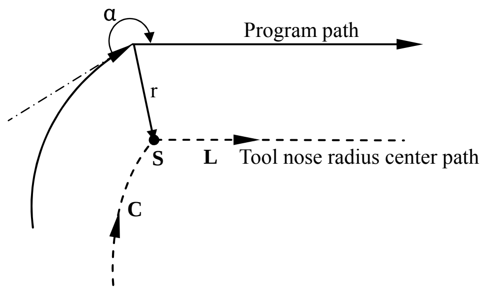
(i). Linear→Linear



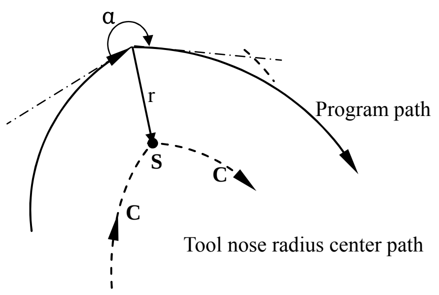
(ii). Linear→Circular



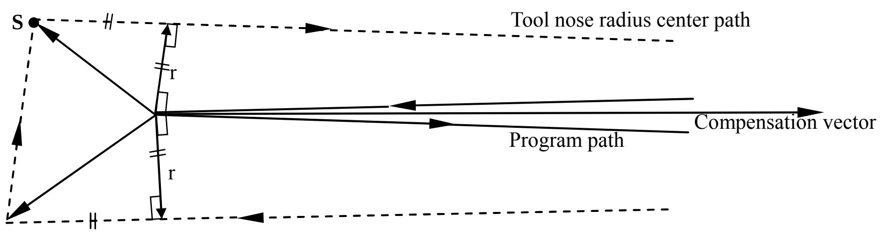
(iii). Circular→Linear



(iv). Circular→Circular

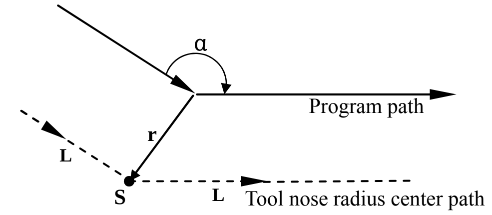


(v).Perform inner side at little angle(<1°) and enlarge compensation vector

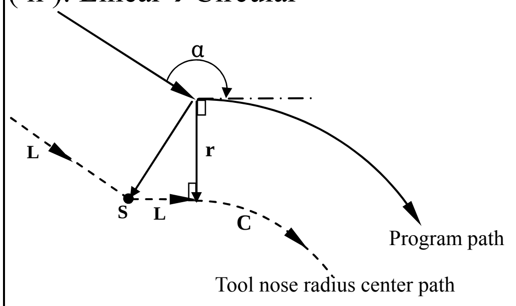


(b). Outer Side(90°≦ α ＜180°)

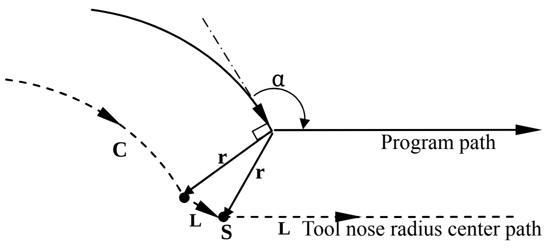
(i). Linear→Linear



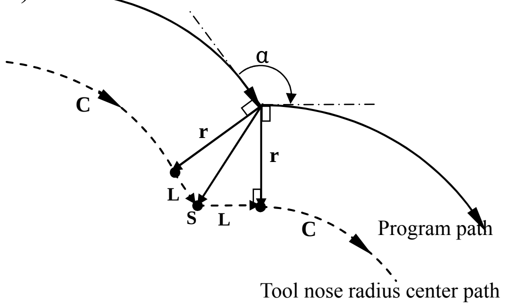
(ii). Linear→Circular



(iii). Circular→Linear

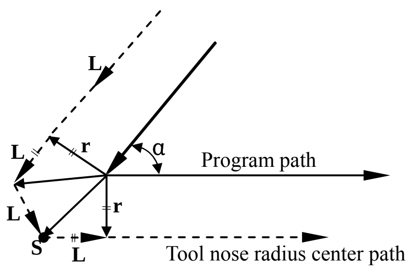


(iv). Circular→Circular

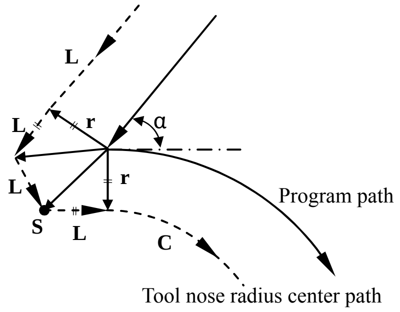


(c). Outer Side (α ＜90°)

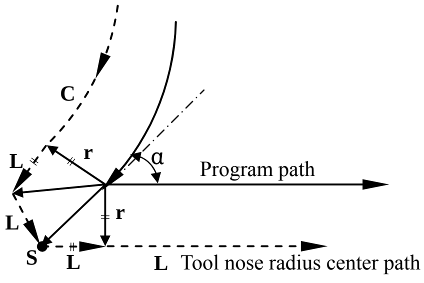
(i). Linear→Linear



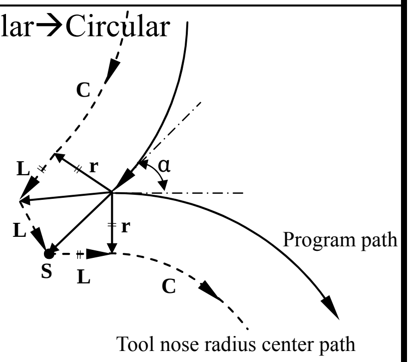
(ii). Linear→Circular



(iii). Circular→Linear



(iv). Circular→Circular



3. Compensation Cancel

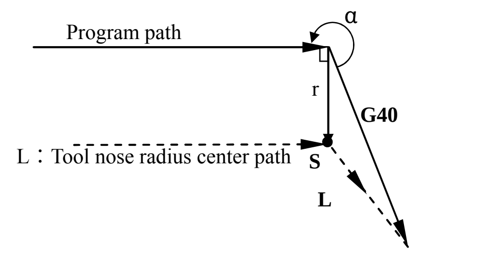
In compensation mode, when block satisfied below following conditions, system will enter cancel mode：

a. Specify G40

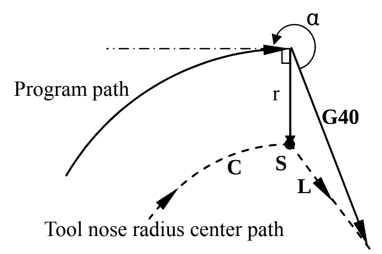
b. The number of tool nose radius compensation is specified to “0”

(a). Inner Side (180°≦ α)

(i). Linear→Linear

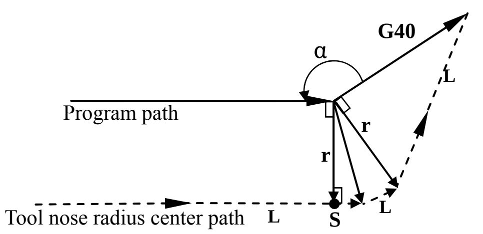


(ii). Linear→Circular

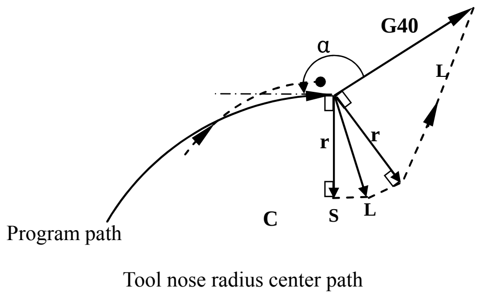


(b). Outer Side (90°≦ α <180°)

(i). Linear→Linear

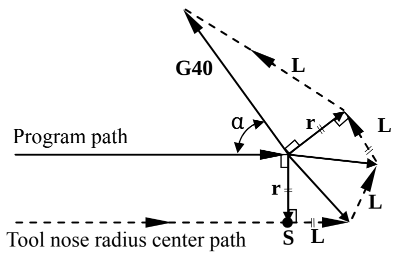


(ii). Linear→Circular

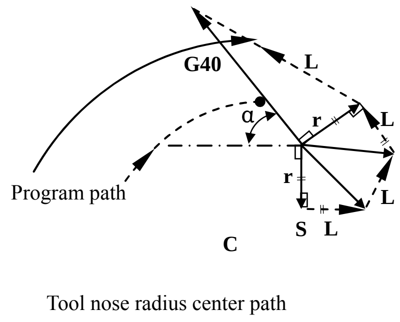


(c). Outer Side (α <90°)

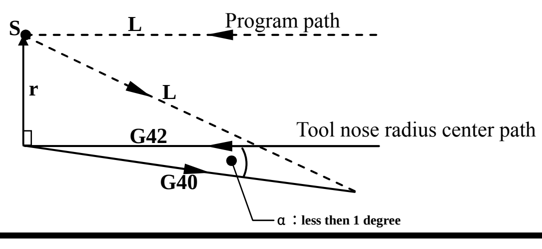
(i). Linear→Linear



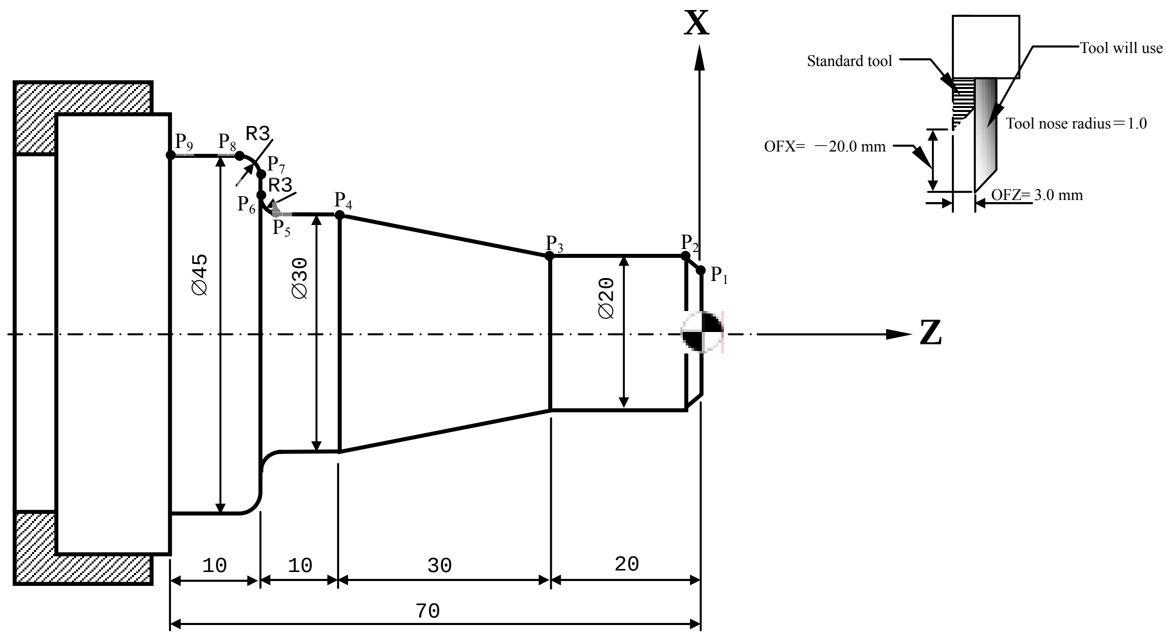
(ii). Linear→Circular



(d). Tool movement around the outside linear→linear at an acute angle less than 1 degree(α <1°)



Example one：

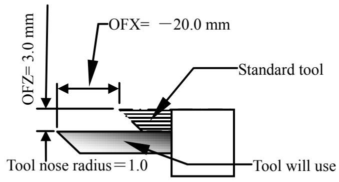


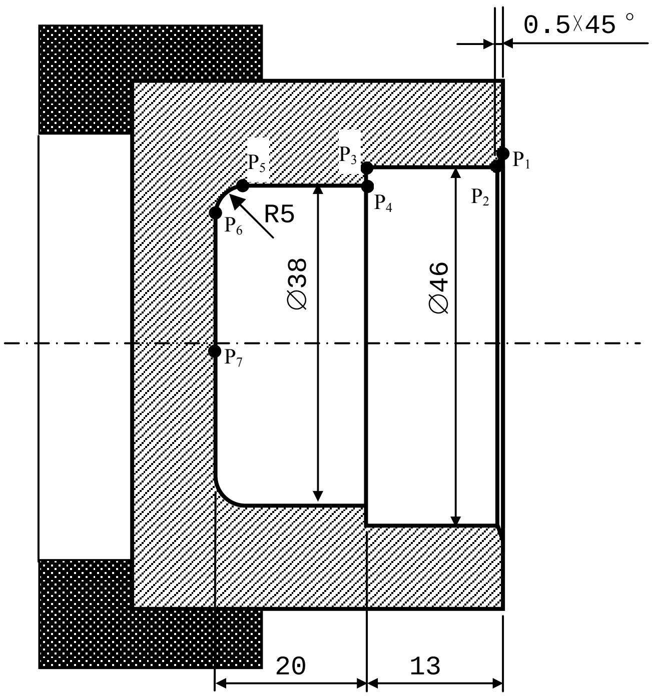
Program description：

```
    T02；//use tool NO.2
    G92 S10000；//max. rotate speed, 10000rpm
    G96 S130 M03；//constant surface speed, spindle rotate 130 m/min CW
    M08；//cutting liquid ON
    G42 X21.0 Z0.0；//tool compensation start-up, move to P1
    G01 X25.0 Z-2.0 F600；//linear interpolation, feedrate 600 μm/rev,
P1→P2
        Z-20.0；// P2→P3
        X30.0 Z-50.0；// P3→P4
        Z-57.0；P4→P5
    G02 X36.0 Z-60.0 R3.0；// P5→P6
    G01 X39.0；// P6→P7
    G03 X45.0 Z-63.0 R3.0；// P7→P8
    G01 Z-70.0；// P8→P9
        X60.0；//return the tool
    G28 X70.0 Z-60.0；
          //positioning to specified mid-point, then return to machine
 zero point
    M09；//cutting liquid OFF
```

```
M05；//spindle stops
M30；//program ends
```

Example two：





Program description：

T02；//use tool NO.2

G92 S1000；//max. rotate speed, 10000rpm

G96 S130 M03；//constant surface speed, spindle rotate 130 m/min CW

M08；//cutting liquid ON

G41 X47.0 Z0.0；//start tool compensation, move to P_1

G01 X46.0 Z-0.5 F600；// linear interpolation, feedrate 600 μ m/rev, $P_1 \rightarrow P_2$

Z-13.0；//$P_2 \rightarrow P_3$

X38.0；//$P_3 \rightarrow P_4$

Z-28.0；//$P_4 \rightarrow P_5$

G03 X28.0 Z-33.0 R5.0；//circular interpolation CCW, radius 5 mm, $P_5 \rightarrow P_6$

G01 X-1.0；//linear interpolation

M09；//cutting liquid OFF

G28 Z20.0；//positioning to specified mid-point, then return to machine zero point

M05；//spindle stops

M30；//program ends

1.2.19 G52：Local Coordinate System Setting

Format：G52 X__ Y__ Z__ ；

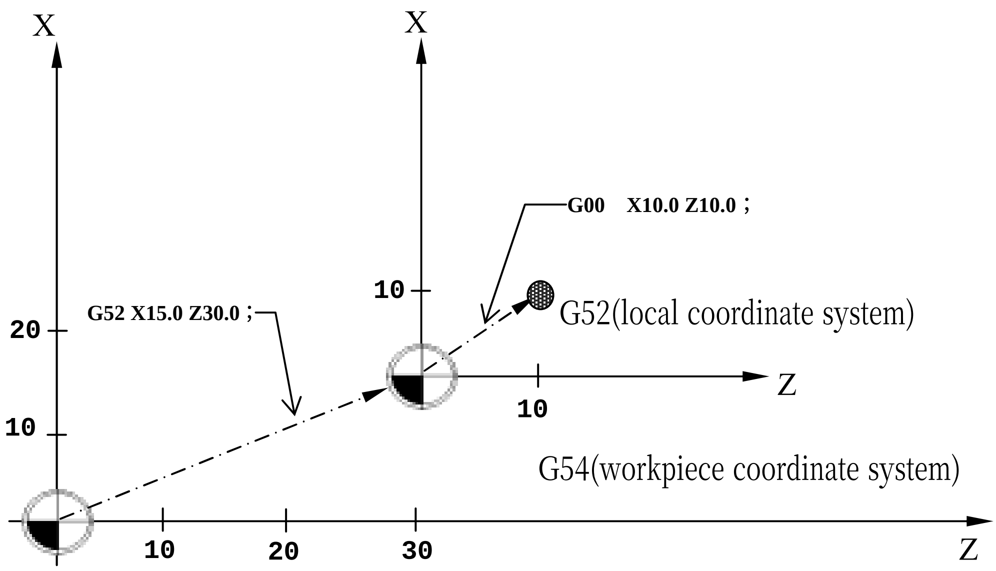
X、Y、Z：setting the local coordinate system

Description：

When you specify a work coordinate system(G54~G59.9)。When perform the workpiece, it need to establish another sub-coordinate, this sub-coordinate is called **local coordinate system**。

G52 X0.0 Z0.0：**cancel** local coordinate

Coordinate System：



Program description：

N001 G54；//specify workpiece coordinate system G54
N002 G52 X15.0 Z30.0；//specify the zero point of local coordinate system to X15.0 Z30.0 of workpiece coordinate system
N003 G00 X10.0 Z10.0；//positioning to X10.0 Z10.0 of local coordinate
N004 G52 X0.0 Z0.0；//local coordinate system cancel

1.2.20 G53：Machine Coordinate System

Format：

G53 X___ Y___ Z___ ；

X：move to specified X in machine coordinate。

Y：move to specified Y in machine coordinate。

Z：move to specified Z in machine coordinate。

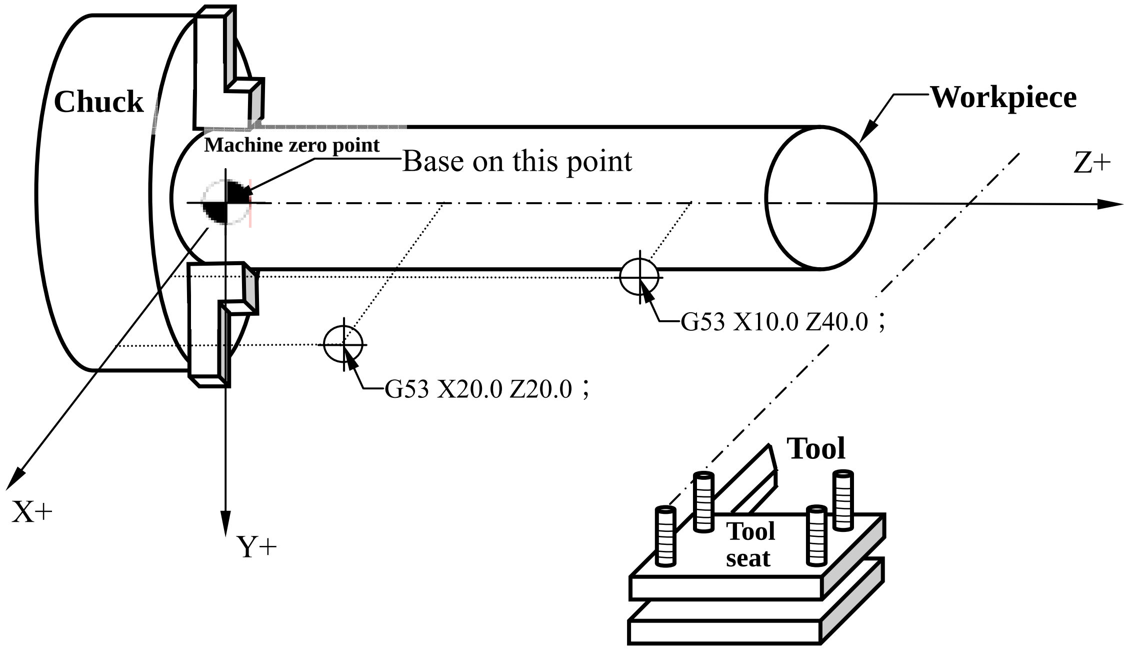
Description：

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point。 A machine tool builder sets a machine zero point for each machine。When a position has been specified as a set of machine coordinates, the tool moves to that position by means of rapid traverse。

<Notice>：

1. G53 command is only effective in specified block；
2. G53 is only effective in absolute mode, not effective in increment mode；
3. Before specify G53, we should cancel relative tool radius、length or position compensation；
4. Before G53 command is specified, manual reference position return must be performed。

Example：



Program description：

1. G53 X20.0 Z20.0；//move to specified position in machine coordinate
2. G53 X10.0 Z40.0；//move to specified position in machine coordinate

1.2.21 G54...G59.9：Workpiece Coordinate System

Format：

G54
G55
G56
G57
G58
G59 X__ Y__ Z__ ；
G59.1
G59.2
：
：
G59.9

G54： First workpiece coordinate system
: :
: :
G59： Sixth workpiece coordinate system
G59.1：Seventh workpiece coordinate system
:
:
G59.9：15th workpiece coordinate system

X、Y、Z：move to specified position in specified workpiece coordinate system；

Description：

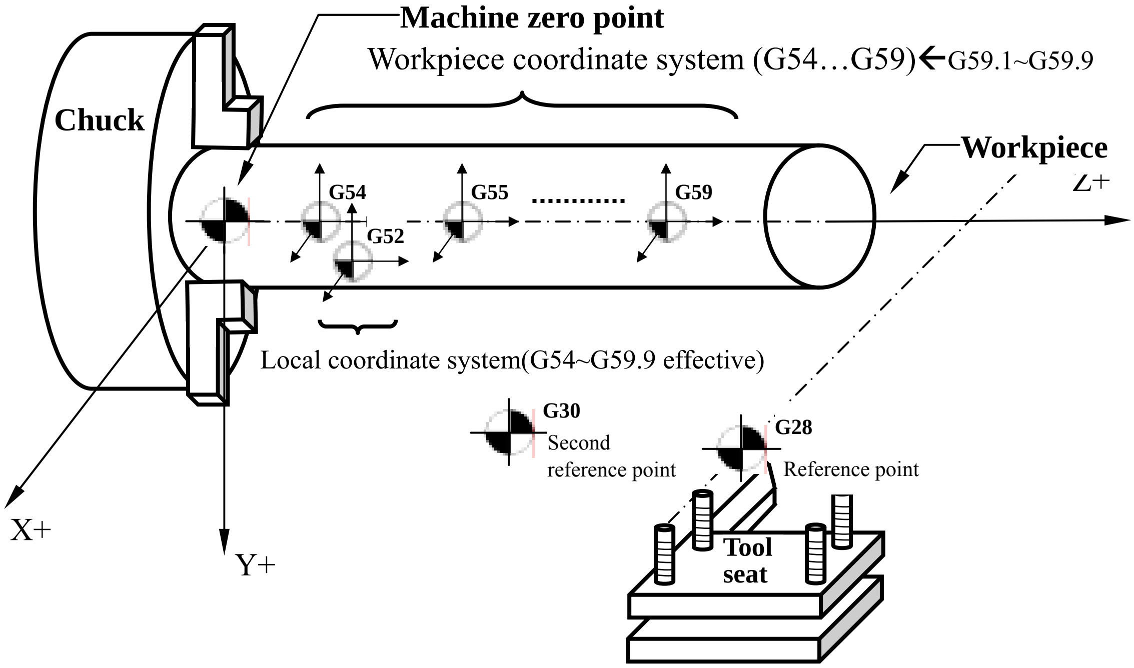
When we operate the lathe, we may repeat perform different position in same workpiece。By specifying G code from G54 to G59 and G59.1 to G59.9, one of the workpiece coordinate system 1 to 15 can be selected。It can be set by parameter #3229「disable workpiece coordinate system」(0：enable ； 1：disable)。

※How to set G54……G59.9：

"Selecting workpiece coordinate system" in controller operation interface, set

54 …G59.9 one by one。

Example：



1.2.22 G65：Simple Marco Call

Format：

G65 P__ L__ ；

P：number of the program to call；

L：times of repeating；

Description：

After G65, specify at address P the program number of the custom Marco to call, but it is effective in one block with G65；please refer SYNTEC『OPEN CNC Macro Develop Tool Guide』。

Ex:

```
G65 P10 L20 X10.0 Y10.0 //call the second program repeatedly and execute 20
                        //times,and take the values of X10.0 Y10.0 into the
                        //second program to calculate them
```

1.2.23 G66、G67：Modal Marco Mode

Format：

G66 P__ L__ ；Modal Marco **call**

G67 ；Modal Marco **cancel**

P：number of the program to call；

L：time of repeating；

Description：

After G66, P_ call th subprogram of the number to execute and L_specify G65 to repeat it several times. After finishing the moving single section, it will execute the contents of the G66 again until exexuting G67 (if there executes variable calculation in the called subprogram you must notice there are the pre-calculated problems about variables).

Ex:

```
G91
G66 P10 L2 X10.0 Y10.0 //call O0010 two times and put the value of X10.0
                       //Y10.0 into the program to calculate them
X20.0  //move X axis to 20.0, after finish and executeG66 P10 L2 X10.0 Y10.0
Y20.0  //move Y axis to 20.0, after finish and executeG66 P10 L2 X10.0 Y10.0
G67    //cancel the model marco mode
```

1.2.24 G70/G71：English/Metric Unit Setting

Format：

G70；
G71；

Description：

G70：English unit system
G71：Metric unit system
After changing English/Metric, workpiece coordinate offset、tool data、system parameter、and reference position are still correct。System will transfer the unit automatically。After transferring, unit will change too：

Coordinate display、unit of speed
Incremental JOG unit
MPG JOG unit

1.2.25 Decimal Point Input

When parameter input by decimal point, it will be specified to common unit, mm、inch、sec …etc.。If it inputs by whole number, then the unit is specified by system default Min. unit, such as μm、ms…etc.。

Example：

Decimal point： ○○.○○
Whole number： ○○○○

1.2.26 Multiple Repetitive Cycle

This option canned cycles to make CNC programming easy。For instance, the data of the finish work shape describes the tool path for rough machining。And also, a canned cycles for the thread cutting is available。

1.2.27 G72：Finishing Cycle

Format：

G72 P(ns) Q(nf) ；

ns：Sequence number of the first block for the program of finishing cycle
nf：Sequence number of the last block for the program of finishing cycle

Description：

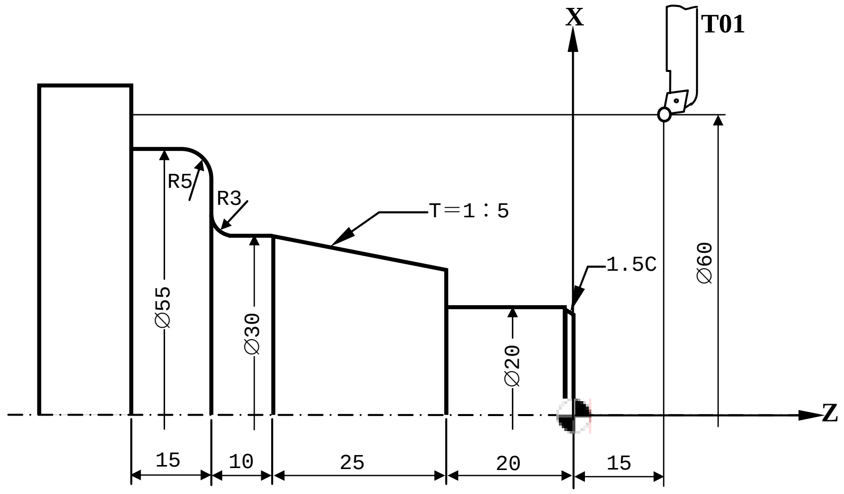
G72 command is finishing cycle, this command must use together with stock removal cycle in next block。In general, finishing cycle is written behind stock removal cycle in the program, the range it executes only includes "**P(ns)**" to "**Q(nf)**"。

After G73 / G74 / G75 cutting cycle, we must match G72 command to reach to specified size。

※ Notice：

1. F、S and T functions specified in the block G73、G74 and G75 are not effective but those specified between sequence number "**ns**"→"**nf**" are effective in G72。
2. When the cycle machining by G72 is terminated, the tool is returned to the start point and the next block is read。
3. In blocks between "ns" and "nf" referred in G72 through G75, the subprogram can not be called。

Example One：



Program description：

T01；//use tool NO. 1

G92 S5000；//Max. rotate speed 5000 rpm

G96 S130 M03；

//constant surface speed, surface speed 130 m/min, spindle rotate CW

G00 X60.0 Z15.0；//positioning to start point

M08；//cutting liquid ON

G73 U2.0 R1.0；//cut 3.0 mm in X axis direction, tool returned value 1.0 mm

G73 P01 Q02 U0.8 W0.1 F300；

//execute stock removal in turning, sequence number N01→N02, left 0.8mm for finishing allowance in X axis direction, left 0.1mm for finishing allowance in Z axis direction, feedrate 300 μm/rev

```
N01 G00 X17.0；
    G01 Z0.0；
        X20.0 Z-1.5；
        Z-20.0；
        X25.0；
        X30.0 Z-45.0；          shape of cutting
        Z-52.0；
    G02 X36.0 Z-55.0 R3.0；
    G01 X45.0；
    G03 X55.0 Z-60.0 R5.0；
N02 G01 Z-70.0；
```

G72 P01 Q02；//execute fine cutting cycle, sequence number N01→N02

M09；//cutting liquid OFF

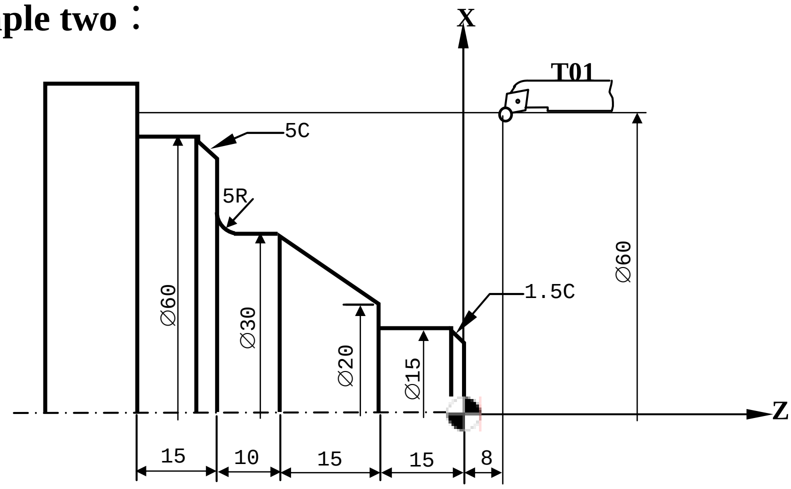
M28 X60.0 Z20.0；

//tool positioning to specified mid-point, then return to machine zero point

M05；//spindle stops

M30；//program ends

Example two：



Program description：

T01；//use tool NO. 1

G92 S5000；//Max. rotate speed 5000 rpm

G96 S130 M03；

//constant surface speed, surface speed 130 m/min, spindle rotate CW

G00 X60.0 Z8.0；//positioning to start point

M08；//cutting liquid ON

G74 W3.0 R1.0；//cut 3.0mm in Z axis direction, tool returned value 1.0 mm

G74 P01 Q02 U0.8 W0.2 F600；

//execute stock removal in facing, the sequence number N01→N02, left 0.8mm for finishing allowance in X axis direction, left 0.2mm for finishing allowance in Z axis direction, feedrate 600 μm/rev

```
N01  G00 Z-55.0；
     G01 X60.0；
         Z-45.0；
         X50.0 Z-40.0；
         X40.0；
     G03 X30.0 Z-35.0 R5.0；          shape of cutting
     G01 Z-30.0；
         X20.0 Z-15.0；
         X15.0；
         Z-1.5；
N02      X12.0 Z0.0；
```

G72 P01 Q02；//execute fine cutting cycle, the sequence number N01→N02

M09；//cutting liquid OFF

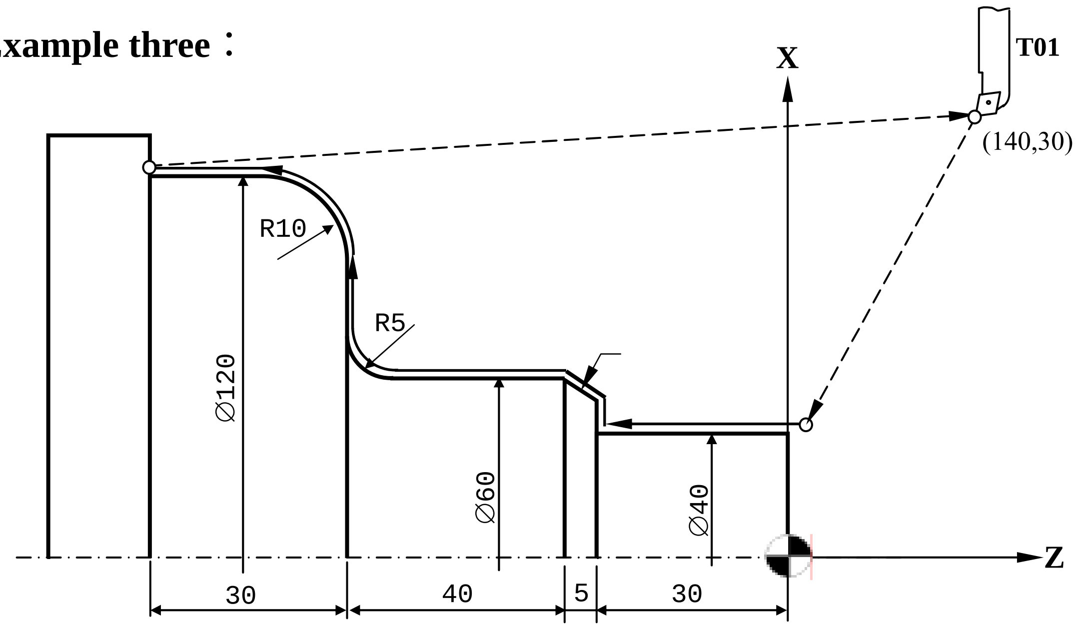
G28 X60.0 Z10.0；

//positioning to specified mid-point, then return to machine zero point

M05；//spindle stops

M32；//program ends

Example three：



Program description：

T01；//use tool NO.1

G92 S5000；//max. rotate speed 5000 rpm

G96 S130 M03；

//constant surface speed, surface speed 130 m/min, spindle rotate CW

G00 X140.0 Z30.0；//positioning to start point

```
    M08；//cutting liquid ON
    G75 U15.0 W15.0 R3.0；
    //cut 15.0mm in X axis direction, cut 3.0mm in Z axis direction, repeat 3 times
    G75 P01 Q02 U0.8 W0.2 F300；
    //execute pattern repeating cutting, the sequence number N01→N02, left 0.8mm
      for finishing allowance in X axis direction, left 0.2mm for finishing
      allowance in Z axis direction, feedrate 300 μm/rev
N01 G00 X40.0 Z5.0；
    G01 Z-30.0；
        X50.0；
        X60.0 Z-35.0；
        Z-70.0；                        shape of cutting
    G02 X70.0 Z-75.0 R5.0；
    G01 X100.0 ；
    G03 X120.0 Z-85.0 R10.0；
N02 G01 Z-105.0；
```

G72 P01 Q02；//execute fine cutting cycle, the sequence number N01→N02

```
    M09；//cutting liquid OFF
    G28 X140.0 Z30.0；
          //positioning to specified mid-point, then return to machine zero point
    M05；//spindle stops
    M30；//program ends
```

1.2.28 G73：Stock Removal in Turning

Format：

G73 U<u>Δd</u> R<u>e</u> ；

G73 P<u>(ns)</u> Q<u>(nf)</u> U<u>Δu</u> W<u>Δw</u> F___ S___ T___ ；

Δd：depth of cut in X axis direction, it can be specified by the parameter#4013 and the parameter is changed by the program command

e：escaping amount, it can be specified by the parameter#4012 and the parameter is changed by the program command

ns：sequence number of the first block for the program of finishing shape

nf：sequence number of the last block for the program of finishing shape

Δu：distance and direction of finishing allowance in X direction

Δw：distance and direction of finishing allowance in Z direction

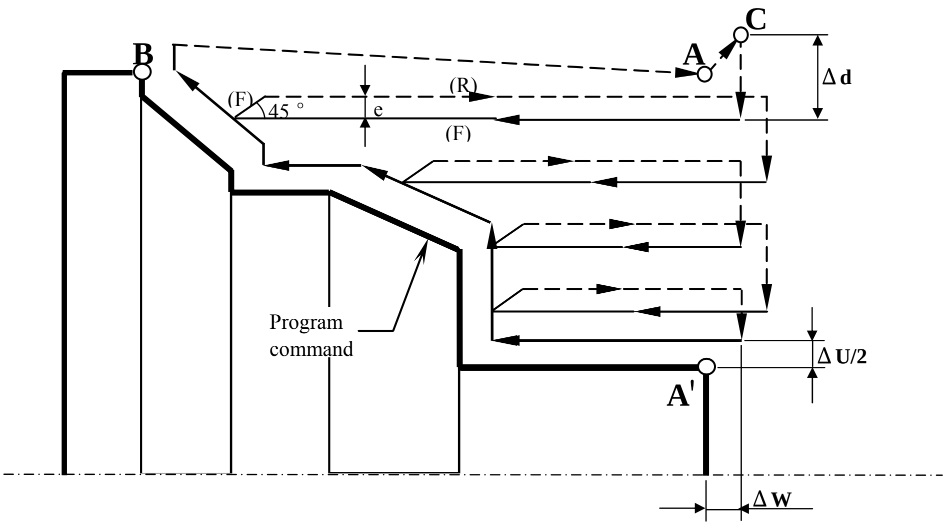
F：feedrate **T**：number of the tools

S：spindle rotate speed

Description：G73 command is stock removal in turning, it can perform workpiece to specified shape, and left a specified value for finishing allowance。This cutting cycle need to define block range of workpiece path、depth of cutting each time and distance and direction of finishing allowance。

PIC：

1. **TYPE I：**there is only X axis motion command in first block “ns”, it usually use in end face performing。 Each block must satisfy that cut value must be decrease or increase next block to last block in X axis and Z axis。



Description：

(1). Tool should be positioning to **point A(start point)** before cycle starts；

(2). After executing G73, tool offsets to point C by specified finishing allowance **(ΔU/2 for X axis, ΔW for Z axis)**；

(3). Tool move to X axis in Δd distance, and then feed the outline face；

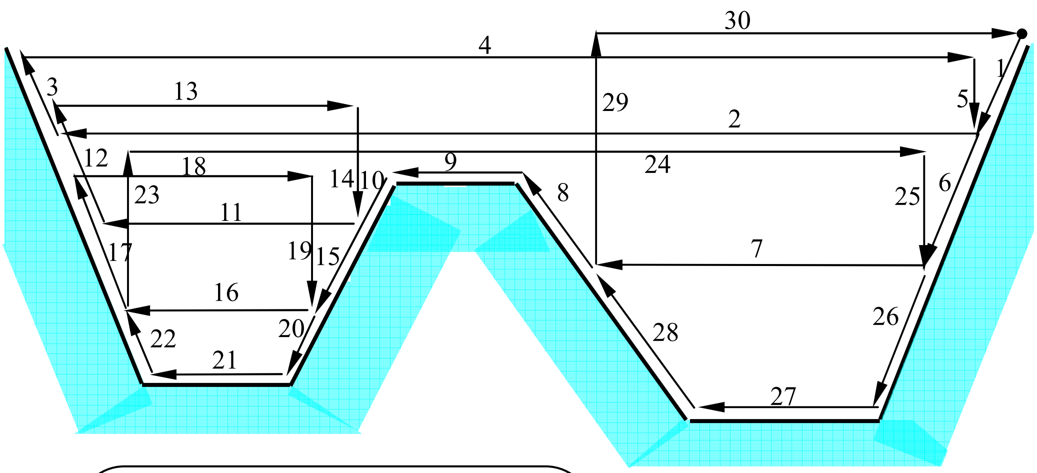
(4). Then escape e distance in X axis direction by 45°, Z axis feed in reverse direction and return to the start point that parallel to X axis；

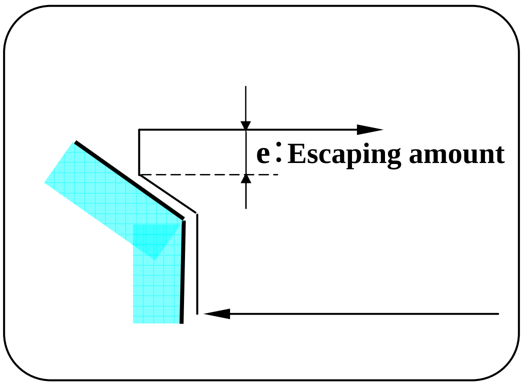
(5). Move Δd distance in X direction, continue next cycle；

(6). After finishing last cycle, tool will cut **A'→ B** once；

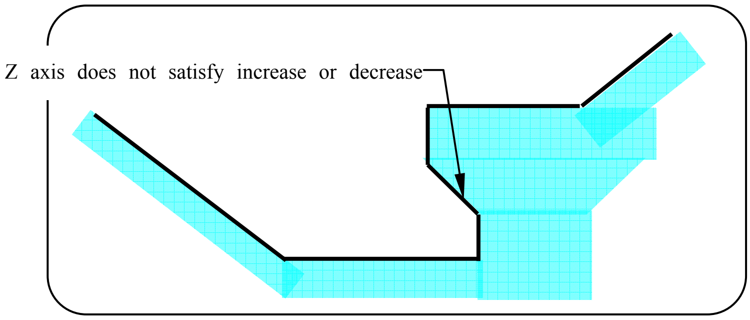
(7). After finishing cutting, tool will positioning to point A, wait for next cutting cycle start。

2. **TYPE II**：This is synchronous moving command (X axis and Z axis) in first block "**ns**", it usually performs in the middle of the workpiece。 At **TYPE II**, only Z axis need to satisfy increase or decrease condition。





← Escaping mode



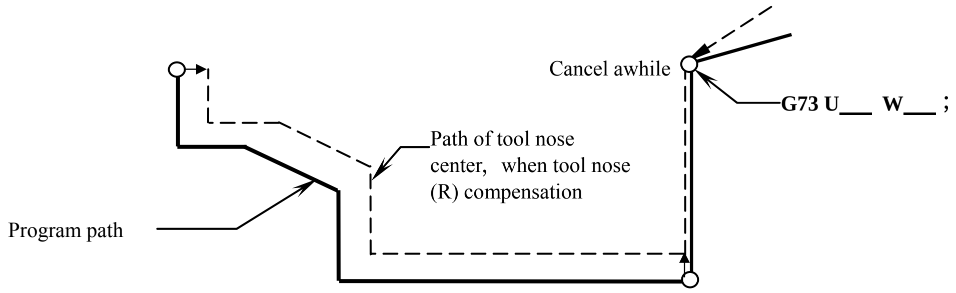
←Can not perform

※**Notice**：

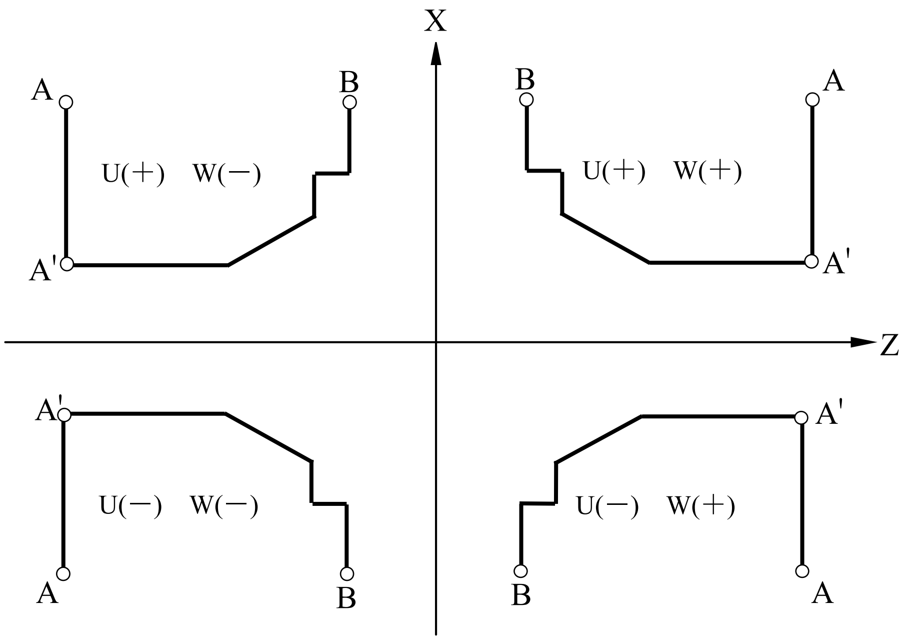
1. When **ns and nf** is not be specified, specified **U** in **G73** block in depth of cut Δ**d**, if it is specified then **U** is finishing allowance in X direction。
2. Outline path in descripted by the blocks from **ns and nf**, from point A to point A' and to point B。
3. F、S、T function is not effective in block of **ns→nf**, those command are effective in the block with G73。
4. G00/G01 command which use in each blocks will use to cut the workpiece along

this block。

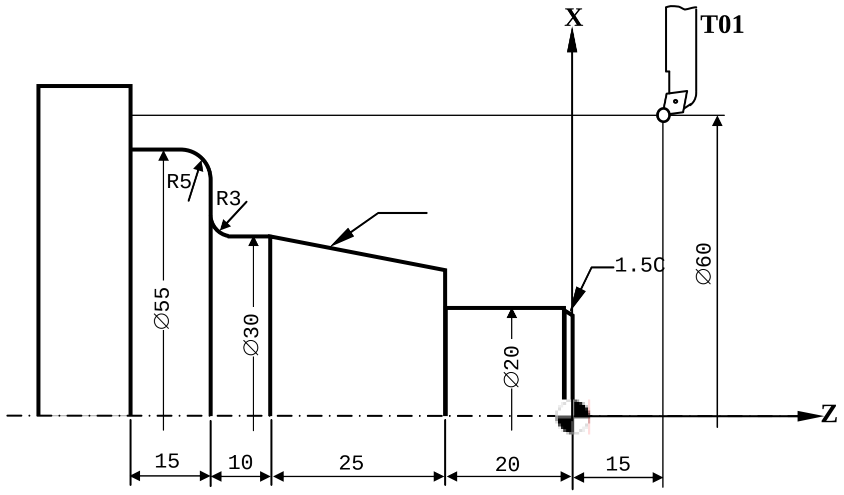
5. Sub-program can not be called during block **ns→nf**。
6. All tool nose compensation commands will be disable when G73 is in the block, but the compensation value will be added to the preparation size。



7. Direction of finishing allowance：it is depended as below figures。Path is A→A'→B。



Example one：TYPE I



Program Description：

T01；//use tool NO. 1

G92 S5000；//max. rotate speed 5000 rpm

G96 S130 M03；

//constant surface speed, surface speed 130 m/min, spindle rotate CW

G00 X60.0 Z15.0；//positioning to start point

M08；//cutting liquid ON

G73 U2.0 R1.0；

//depth of cutting in X direction is 2.0 mm, escaping amount 1.0 mm

G73 P01 Q02 U0.8 W0.1 F300；

//execute stock removal in turning, the sequence of block N01→N02, finishing allowance in X direction is 0.8 mm, finishing allowance in Z direction is 0.1mm, feedrate 300 μm/rev

```
N01 G00 X17.0；                      ← TYPE I
    G01 Z0.0；
        X20.0 Z-1.5；
        Z-20.0；
        X25.0；
        X30.0 Z-45.0；               shape of cutting
        Z-52.0；
    G02 X36.0 Z-55.0 R3.0；
    G01 X45.0；
    G03 X55.0 Z-60.0 R5.0；
N02 G01 Z-70.0；
```

M09；//cutting liquid OFF

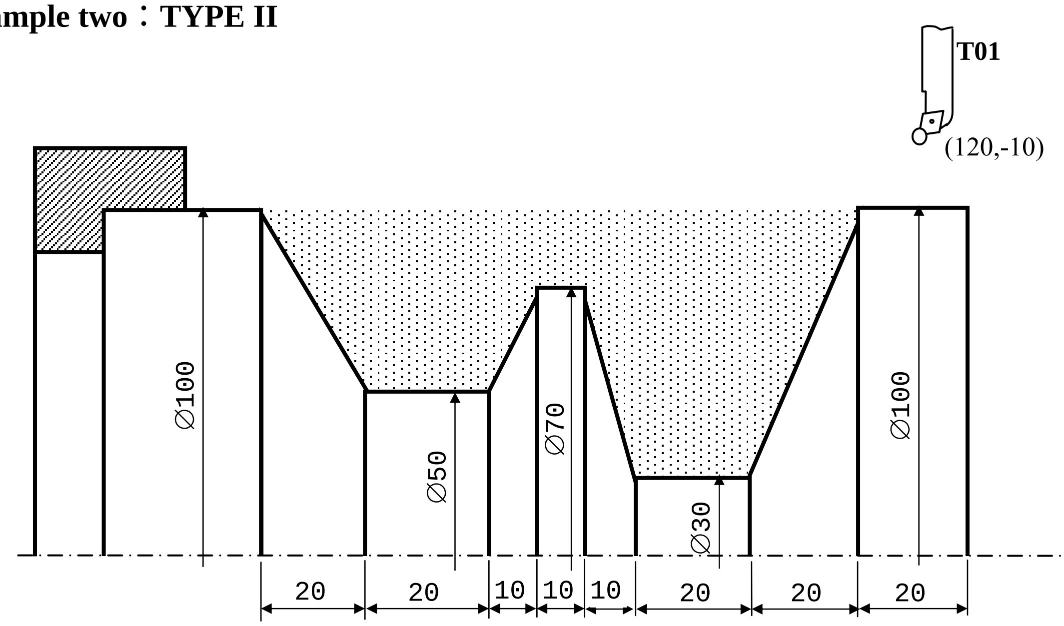
M28 X60.0 Z20.0；

//positioning to specified mid-point, then return to machine zero point

M05；//spindle stops

M30；//program ends

Example two：TYPE II



Program description：

T01；//use tool NO. 1

G92 S5000；//max. rotate speed 5000rpm

G96 S130 M03；//constant surface speed, surface speed 130 m/min

M08；//cutting liquid ON

G00 X120.0 Z-10.0；//positioning to start point

G73 U2.0 R1.0；

//depth of cutting in X direction is 2.0 mm, escaping amount is 1.0 mm

G73 P01 Q02 U0.8 W0.1 F300;

//execute stock removal in turning, the sequence of block N01→N02, finishing allowance in X direction is 0.8 mm, finishing allowance in Z direction is 0.1mm, feedrate 300 μm/rev

```
N01 G00 X101.0 Z-20.0；          ← TYPE II
G01 X100.0；
    X30.0 Z-40.0；
    Z-60.0；
    X70.0 Z-70.0；               shape of cutting
    Z-80.0；
    X50.0 Z-90.0；
    Z-110.0；
N02 X100.0 Z-130.0；
G28 X150.0 Z40.0；
     //positioning to specified mid-point,  then return to machine zero point
M09；//cutting liquid ON
M05；//spindle stops
M30；//program ends
```

1.2.29 G74：Stock Removal in Facing

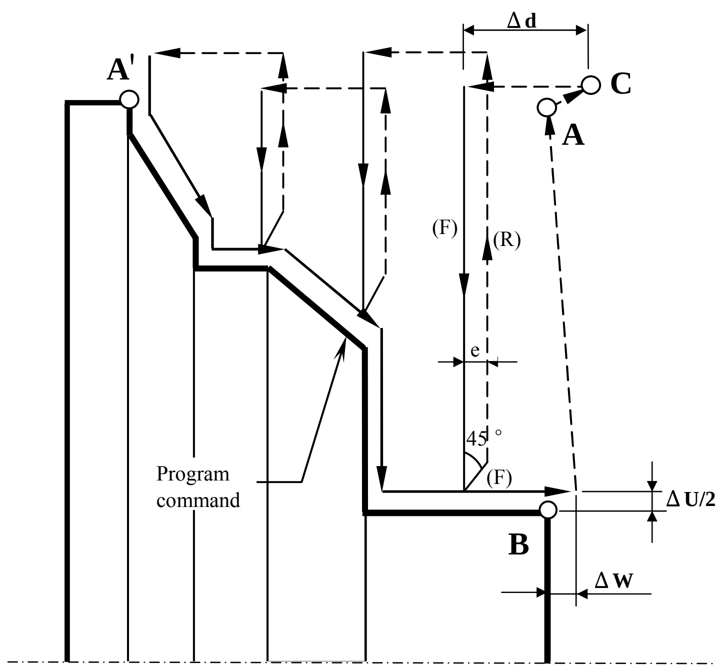
Format：

G74 W_d_ R_e_ ；
G74 P_(ns)_ Q_(nf)_ U_Δu_ W_Δw_ F___ S___ T___ ；

d：depth of cut in Z axis direction, it can be specified by the parameter#4013 and the parameter is changed by the program command
e：escaping amount, it can be specified by the parameter#4012
ns：sequence number of the first block for the program of finishing shape
nf：sequence number of the last block for the program of finishing shape
Δu：distance and direction of finishing allowance in X direction
Δw：distance and direction of finishing allowance in Z direction
F：feedrate **T**：number of the tools
S：spindle rotate speed

Description：G74 command is stock removal in facing, it uses when workpiece is big diameter and short length, when axle direction cutting value is bigger than diameter direction, then we use G74 to do it。

PIC：



Action description：

(1). Positioning to point A (start point) before cycle starts；

(2). After executing G74 command, tool offsets to C point according to specified **finishing allowance (ΔU/2 in X direction, ΔW in Z direction)**；

(3). After tool moves toward Z axis in Δd distance, feed to the outline face；

(4). Then escape e distance in Z axis direction by 45°, X axis feed in reverse direction and return to the start point that parallel to X axis；

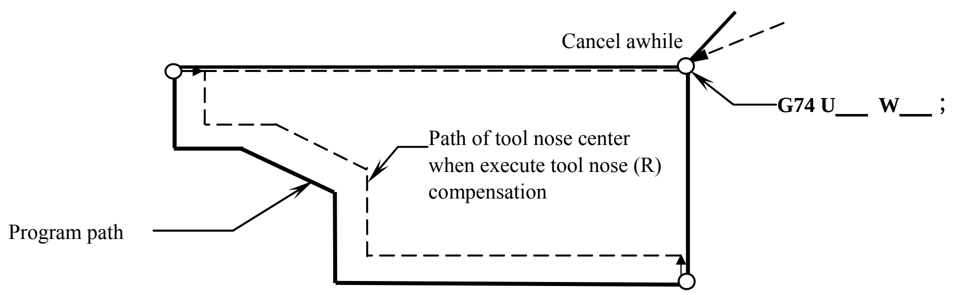
(5). Then move toward Z axis in Δd distance and continue next cycle；

(6). After finishing last cycle, tool will cut **A'→ B** once；

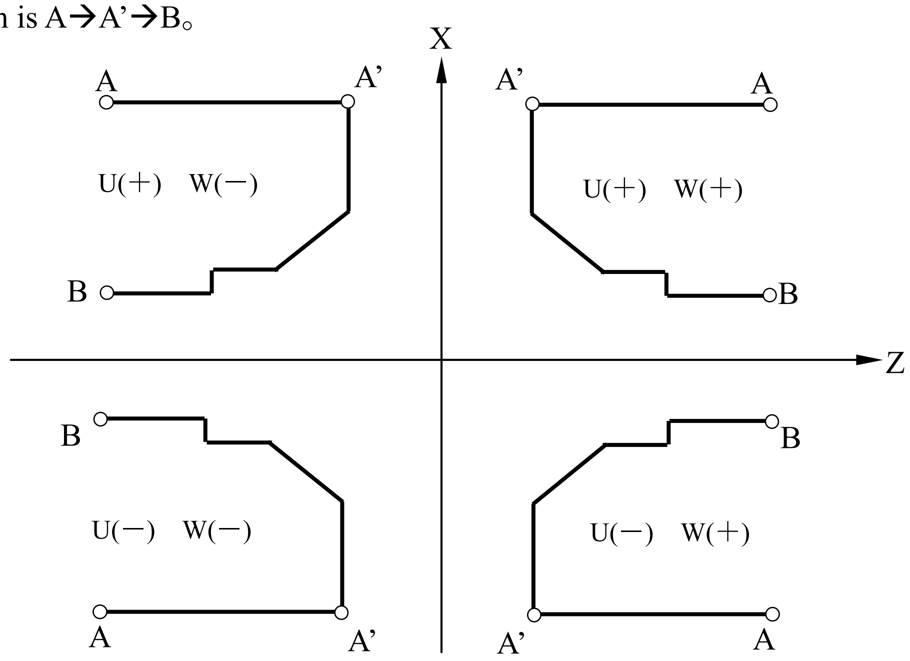
(7). After finishing cutting, tool will positioning to point A, wait for next cutting cycle start。

※**Notice：**

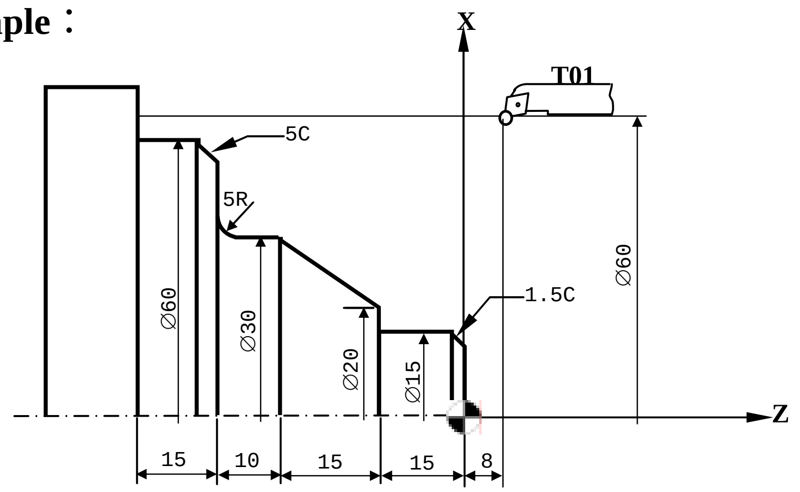
8. When **ns and nf** are not specified, specified **W** in **G74** block is depth **Δd** of cutting, if it is specified then W is prepalation values in Z direction。
9. Outline path in descripted by the blocks from **ns and nf**, from point A to point A' and to point B。
10. F、S、T function is not effective in block of **ns→nf**, those command are effective in the block with G73。
11. G00/G01 command which use in each blocks will use to cut the workpiece along this block。
12. Sub-program can not be called during block **ns→nf**。
13. We specify the path A→A’ in the block of sequence number **“ns”** and the block is included G00 or G01。We can not specify **the movement command in X direction**。Tool path between A’→B must increase or decrease in X and Z direction。
14. All tool nose compensation commands will be disable when G74 is in the block, but the compensation value will be added to the preparation size。



15. Direction of finishing allowance：the direction is depended on below figures。
    Path is A→A'→B。



Example：



Program description：

T01；//use tool NO. 1

G92 S5000；//max. rotate speed 5000 rpm

G96 S130 M03；

//constant surface speed,　surface speed 130 m/min,　spindle rotate CW

G00 X60.0 Z8.0；//positioning to start point

M08；//cutting liquid ON

G74 W3.0 R1.0；

//depth of cutting in Z direction is 3.0 mm,　escaping amount is 1.0 mm

G74 P01 Q02 U0.8 W0.2 F600；

// execute stock removal in turning, the sequence of block N01→N02, finishing allowance in X direction is 0.8 mm, finishing allowance in Z direction is 0.2mm, feedrate 600 μm/rev

```
N01 G00 Z-55.0；
    G01 X60.0；
        Z-45.0；
        X50.0 Z-40.0；
        X40.0；
    G03 X30.0 Z-35.0 R5.0；      shape of cutting
    G01 Z-30.0；
        X20.0 Z-15.0；
        X15.0；
        Z-1.5；
N02     X12.0 Z0.0；
    M09；//cutting liquid OFF
    G28 X60.0 Z10.0；
          //positioning to specified mid-point, then return to machine zero point
    M05；//spindle stops
    M32；//program ends
```

1.2.30 G75：Pattern Repeating

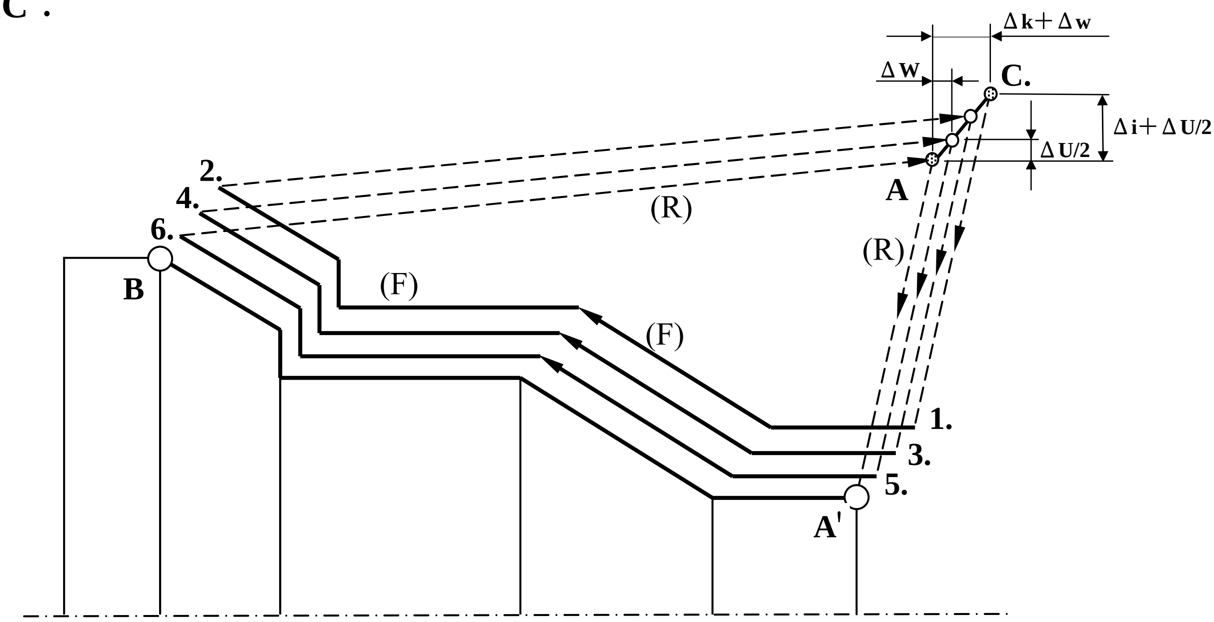
Format：

G75 U$\underline{\Delta i}$ W$\underline{\Delta k}$ R$\underline{d}$ ；
G75 P$\underline{(ns)}$ Q$\underline{(nf)}$ U$\underline{\Delta u}$ W$\underline{\Delta w}$ F___ S___ T___ ；

Δi：distance and direction of relief in the X axis direction, this value can be specified by the parameter #4015
ΔK：distance and direction of relief in the Z axis direction, this value can be specified by the parameter #4016
d：the number of division, it can be specified by parameter #4017
ns：sequence number of the first block for the program of finishing shape
nf：sequence number of the last block for the program of finishing shape
Δu：distance and direction of finishing allowance in X direction
Δw：distance and direction of finishing allowance in X direction
F：feedrate **T**：number of the tool
S：spindle speed

Description：G75 command is patten repeating, By this cutting cycle, it is possible to efficiently cut work whose rough shape has already been made by a rough machining, forging or casting method, etc。If we use G73、G74 to cut it, we will waste time to execute much needlessly cutting path。

PIC：



Action description：

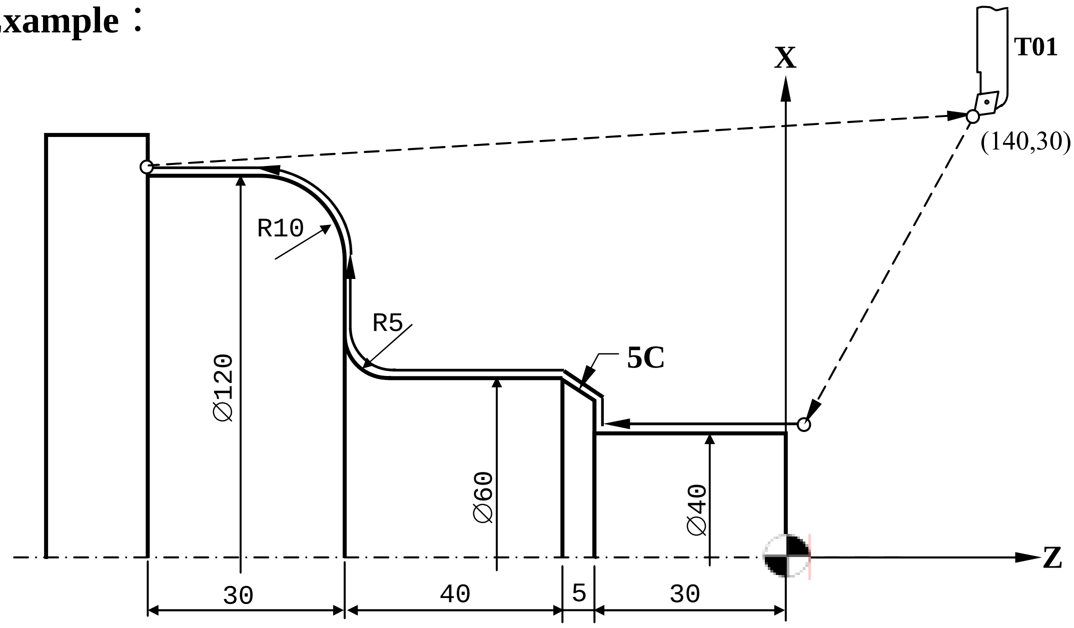
(1). Positioning to **point A** (start point) before cycle starts；

(2). After executing G75， tool offsets to point C by specified finishing allowance **(ΔU/2 for X axis, ΔW for Z axis) and add cutting value (Δi for X axis, ΔW for Z axis)**；

(3). Tool will be cutting by path A→A'→B， according feed value and times of cutting to finish the performance；

(4). After finishing cutting， tool will positioning to point A， wait for next cutting cycle start。

Example：



Program description：

T01 ; //use tool NO. 1

G92 S5000 ; //max. rotate speed 5000 rpm

G96 S130 M03 ;

//constant surface speed, surface speed 130 m/min, spindle rotate CW

G00 X140.0 Z30.0 ; //positioning to start point

M08 ; //cutting liquid ON

G75 U15.0 W3.0 R3.0 ;

//cutting value of X axis 15.0 mm, cutting value of Z axis 3.0 mm, cut 3 times

G75 P01 Q02 U0.8 W0.2 F300 ;

//execute Pattern Repeating, sequence of the block N01→N02, finishing allowance of X axis 0.8 mm, finishing allowance of Z axis 0.2 mm, feedrate 300 μm/rev

```
N01 G00 X40.0 Z5.0 ;
    G01 Z-30.0 ;
        X50.0 ;
        X60.0 Z-35.0 ;
        Z-70.0 ;                 shape of cutting
    G02 X70.0 Z-75.0 R5.0 ;
    G01 X100.0 ;
    G03 X120.0 Z-85.0 R10.0 ;
N02 G01 Z-105.0 ;
```

```
M09；//cutting liquid OFF
G28 X140.0 Z30.0；
        //positioning to specified mid-point,  then return to machine zero point
M05；//spindle stops
M30；//program ends
```

1.2.31 G76：End Face (Z axis) Peck Drilling Cycle

Format：

G76 R_e_ ；

G76 X(U)___ Z(W)___ P_Δi_ Q_Δk_ R_d_ F___ ；

e：return amount(return amount in Z direction when cut Δk distance) ← it can be setted by parameter #4011

X：X component of point B（diameter）

Z：Z component of point C

U：Incremental amount from A to B（diameter）

W：Incremental amount from A to C

Δi：Movement amount in X direction(display by radius, positive)

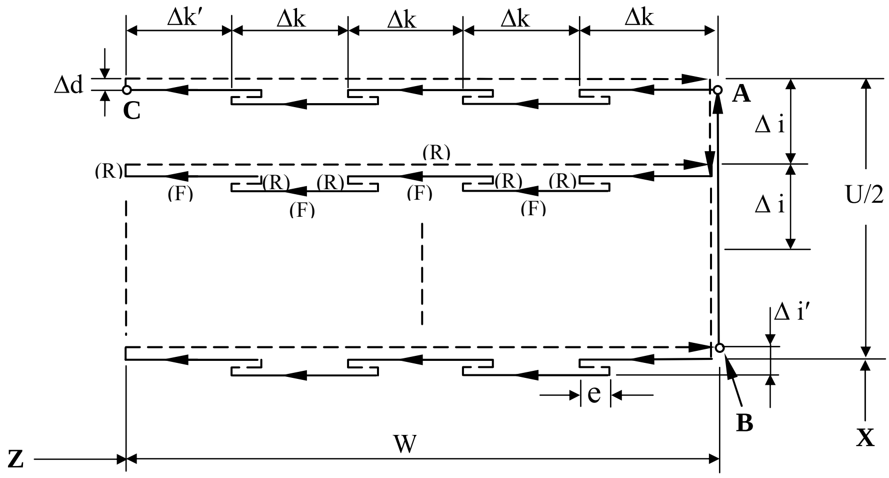
Δk：Movement amount in Z direction(positive)

Δd：Relief amount of the tool at the cutting bottom。(this value is 0 when it returns in origin path)

F：Feed rate

Description：G76 command is end face peck (Z axis) drilling cycle, it is used to slot cutting on the end face or peck drilling in Z direction；After this command is executed, tool will return e value every time when cut Δk distance in Z direction。So G76 can be used in workpiece end face slot or intermittent cutting or deep drilling of workpiece。

PIC：



Action description：

(1). Positioning to **point A (start point)** before cycle starts；

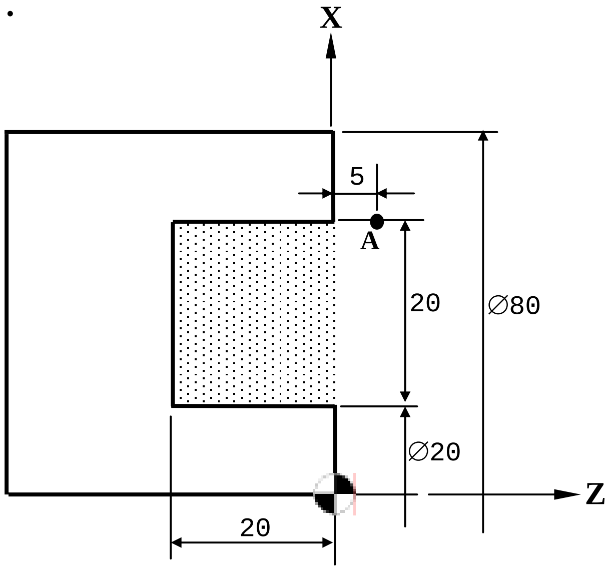
(2). After executing G76, tool will start peck drilling from point A to point C, and it will return e amount every time when tool cuts Δk distance, (and escape Δd distance in X direction), then escape to parallel start point；

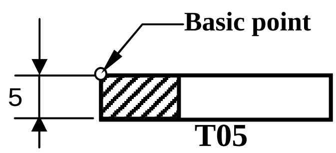
(3). And then tool move Δi distance toward X axis, and continue the same action of cycle。Finally it performs to point B, tool will return to point A from point B, and wait the next cutting cycle。

※**Notice**：

1. **e and Δd is** specified by parameter **R**, when **X__** or **Z__** are specified, **R__** is showed for escaping amount in X axis。
2. When there is only parameter **R** after G76 command, it is for escaping amount in **Z** axis direction, **G** code of this mode is always effective until changing to new program。
3. If **QΔk** is not be specified, then peck drilling cancels, tool cut to the end position of Z axis once。

Example：





Program description：

T05；//use tool NO. 5

G92 S1000；//max. rotate speed 1000 rpm

G96 S100 M03；

//constant surface speed, surface speed 100 m/min, spindle rotate CW

M08；//cutting liquid ON

G00 X60.0 Z5.0；//positioning to point A

G76 R1.0；

G76 X30.0 Z-20.0 P4.0 Q8.0 F100；

//execute end face peck drilling cycle, after cutting 8.0 mm, tool escape 1.0 mm distance, X axis moves 4.0 mm after cycle starts, feed rate 100 μm/rev

M09；//cutting liquid OFF

G28 X100.0 Z30.0；

//positioning to specified mid-point, then return to machine zero point

M05；//spindle stops

M30；//program ends

1.2.32 G77：Outer Diameter/Internal Diameter Drilling Cycle

Format：

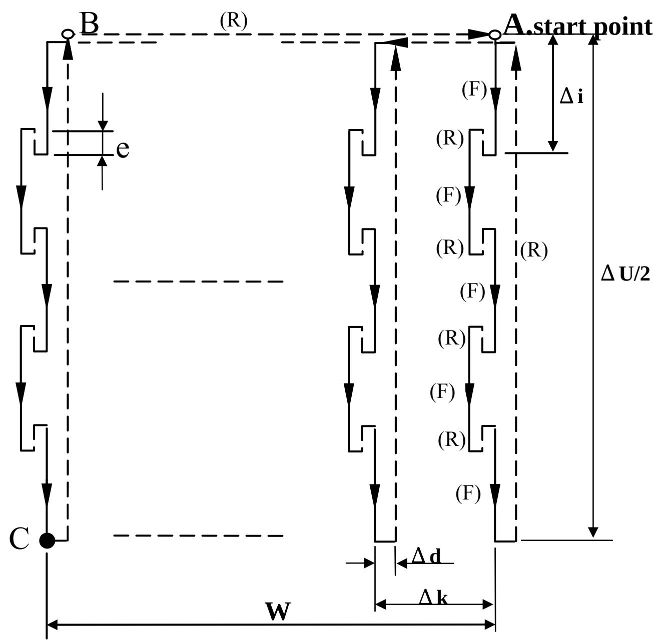
G77 R e ;
G77 X(U)___ Z(W)___ PΔi QΔk RΔd F___ ;

e：return amount(after cutting Δi distance in X axis direction) ←it can be setted by parameter #4011
X：X component of point C(diameter)
Z：Z component of point C
U：increment amount from B to C(diameter)
W：increment amount from A to B
Δi：movement amount in X direction(display by radius, positive)
Δk：depth of cut in Z direction(positive)
Δd：Relief amount of the tool at the cutting bottom。(this value is 0 when it returns in origin path)
F：feedrate

Description：

G77 is outer diameter/internal diameter drilling cycle, this command is used for grooving and peck drilling at X axis。If we want to cut a tank on the outer diameter in order to escape the tool and cut the the complete screw tooth。Lathe often need to use cutting holder to cut the workpiece。。

PIC：



Action description：

(1). Positioning to point **A(start point)** before cycle starts；

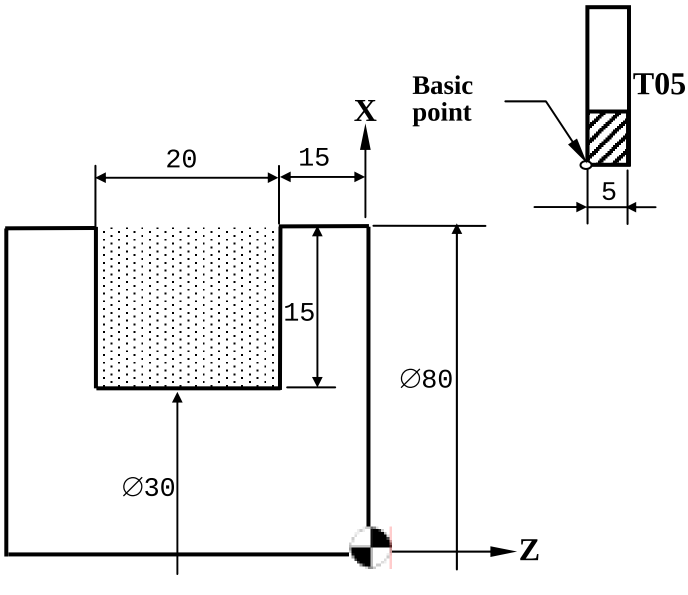
(2). After executing G77，it will start peck cutting from point A，when cutting distance Δi，then escaping distance e，it will cut to specified X，(escape distance Δd in Z direction)，then escape to parallel start point；

(3). Then tool moves Δk distance toward Z axis，continue the same cycle，finally it performs to end point B。Tool will return to point A from point B，and wait to next cutting cycle。

※**Notice**：

1. **e and Δd** is specified by parameter R，when **X__** or **Z__** are specified，R is escaping amount at Z axis
2. When there is only R parameter after G77，it is for escaping amount at X direction。It is modal **G code**，when it is specified once，it is effective during this program，it is not effective when changing the new program。
3. If value of **Q**<u>**Δk**</u> is not specified，then peck cutting cancels，and tool cut to the end position of X axis。

Example：



Program description：

T05；//use tool NO. 5

G92 S1000；//max. rotate speed 1000 rpm

G96 S100 M03；

//constant surface speed，surface speed 100 m/min，spindle rotate CW

M08；//cutting liquid ON

G00 X70.0 Z20.0；//positioning close to workpiece
　　Z-20.0；//positioning to start point of cutting
G77 R1.0；
G77 X30.0 Z-35.0 P8.0 Q4.0 D0.0 F150；
　//execute Outer Diameter/Internal Diameter Drilling Cycle, after cut 8.0 mm, then tool escapes 1.0 mm, Z axis moves 4.0mm after cycle starts, feed rate 100 μm/rev
M09；//cutting liquid OFF
G28 X80.0 Z50.0；
　　//positioning to specified mid-point, then return to machine zero point
M05；//spindle stops
M30；//program ends

1.2.33 G78：Multiple Thread Cutting Cycle

Format：

G78 P m r a Q___ R d ;
G78 X(U)___ Z(W)___ R i P k Q d H___ F___;

P：
m：repetitive count in finishing, specified by system parameter #4044。
r：chamfering amount, specified by system parameter #4043。
a：angle of tool tip, specified by system parameter #4042。

Q：minimum cutting depth$(\Delta d\sqrt{n}-\Delta d\sqrt{n-1})<Q$,specified by system parameter #4045

d：finishing allowance, specified by system parameter #4041

X(U)：X coordinate in end point(bottom of tooth)

Z(W)：Z coordinate in end point(bottom of tooth)

Δ**i**：difference of thread radius

Δ**k**：height of thread

Δ**d**：finishing allowance

F：Metric lead of thread(unit : mm/tooth)

E：English lead of thread(unit : tooth/inch)

H：numbers of thread(ex:H3 three thread type cutting, multiple thread F function is for neighbor thread)

Description：G78 command, multiple thread cutting cycle can product many thread cutting paths。Controller can help us computer numbers of thread cutting、depth of cutting and start point of cutting according to specified parameter。

※ There are three ways of thread cutting：

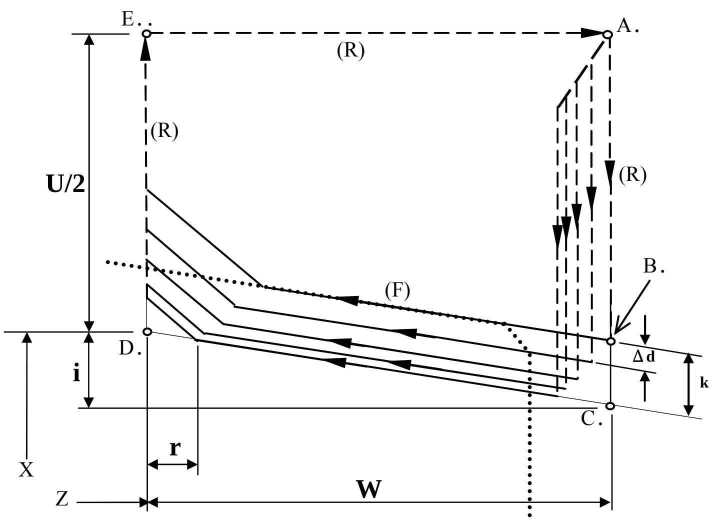
1. G33(thread cutting)：when doing thread cutting, we will use 4 blocks of

command to finish thread cutting, therefore it wastes much time。

2. G21(thread cutting cycle)：this is “single” cycle command of thread cutting, we can use one block of command to finish thread cutting, but it also need to repeating thread cutting many times so the program is also too long。
3. G78(multiple thread cutting cycle)：only use one command, we can finish all of the thread cutting, it can short the program。

PIC：

1. cutting path：



Action description：

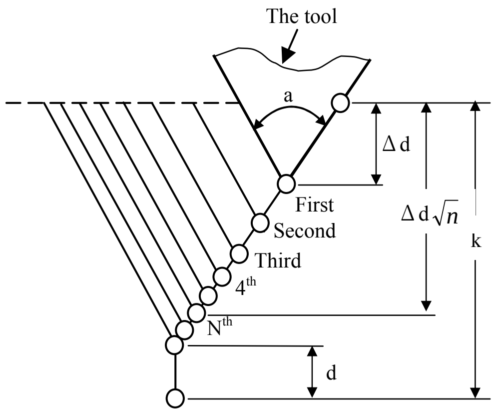
(1). Positioning to **point A(start point)** before cycle start；

(2). After executing G78, the tool will cut along path A→B→E→A, tool depends on the cutting feed to finish first time of threading；

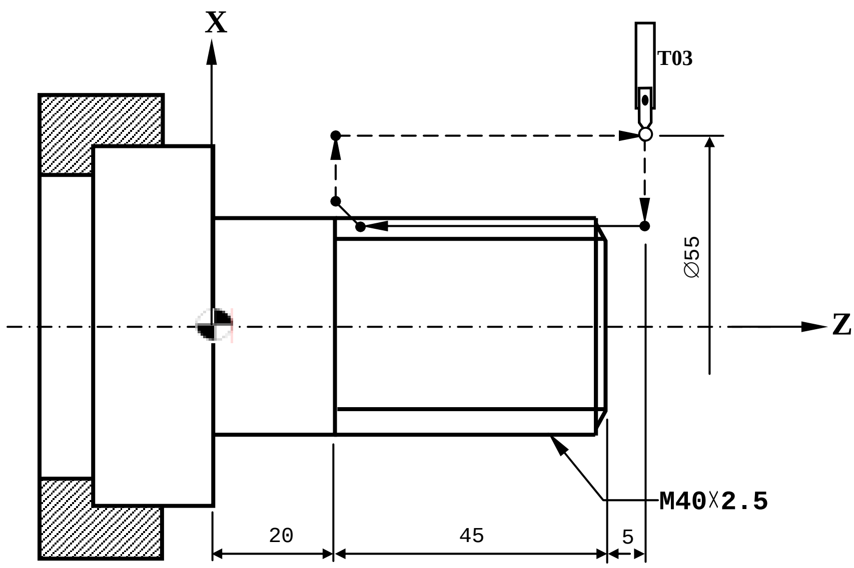
(3). After first time of threading it depends on **finishing allowance** and **repetivite count in finishing** to finish the threading；

(4). Final cuting (A→C→D→E→A), tool stops at point A, wait next cutting cycle。

2. how to cut when threading and the depth of cutting：



Example one：compare with example one of **G21**



Program description：

N001　T03；//use tool NO. 3

N002　G97 S600 M03；//constant rotate speed, 600 rpm CW

N003　G00 X50.0 Z70.0；//positioning to the start point of cycle

N004　M08；//cutting liquid ON

N005　G78 P011060 Q0.15 R0.02；

//execute multiple repetitive cycle, finishing cutting once, escaping amount ＝Lead, angle of tooth 60°, Min. depth of cutting 0.15 mm, finishing allowance 0.02 mm

N006　G78 X36.75 Z20.0 R0.0 P1.624 Q1.0 H3 F2.5；

//difference radius of multiple thread cutting cycle is 0 mm, depth of thread 1.624 mm, first cutting value is 1.0 mm, lead of thread 2.5 mm, three tooth thread cutting

N007 G28 X60.0 Z75.0；

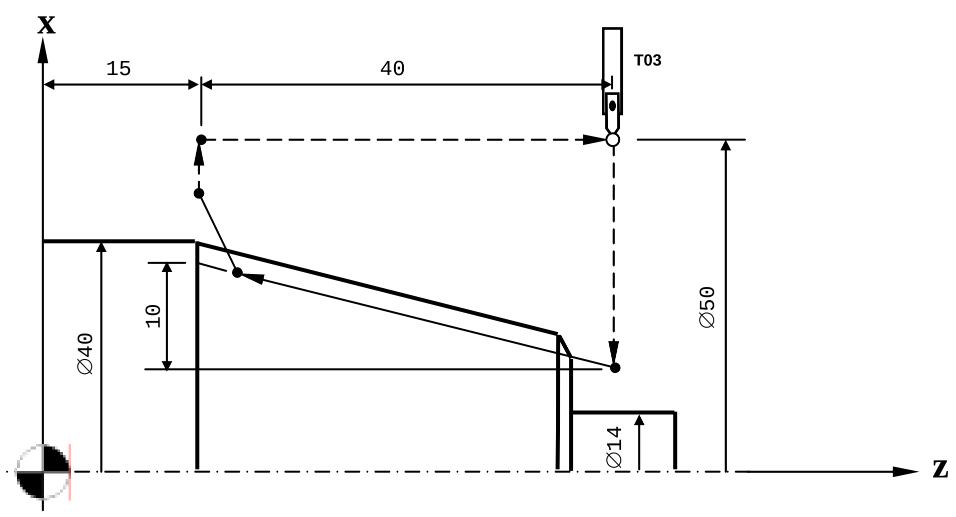
//positioning to specified mid-point and return to machine zero point

N008 M09；//cutting liquid OFF

N009 M05；//spindle stops

N0010 M30；//program ends

Example two：compare with example two of G21, single tooth type, Pitch ＝2.5 mm



Program description：

N001 T03；//use tool NO. 3

N002 G97 S600 M03；//constant rotate speed, 600 rpm CW

N003 G00 X50.0 Z55.0；//positioning to start point of cycle

N004 M08；//cutting liquid ON

N005　G78 P011060 Q0.15 R0.02；

// execute multiple repetitive cycle，finishing cutting once，escaping amount＝Lead，angle of tooth 60°，Min. depth of cutting 0.15 mm，finishing allowance 0.02 mm

N006　G78 X36.75 Z15.0 R-10.0 P1.624 Q1.0 F2.5；

// difference radius of multiple thread cutting cycle is 10.0 mm，depth of thread 1.624 mm，first cutting value is 1.0 mm，lead of thread 2.5 mm，single tooth thread cutting

N007　G28 X60.0 Z70.0；

//positioning to specified mid-point and then return to machine zero point

N008　M09；//cutting liquid OFF

N009　M05；//spindle stops

N0010 M30；//program ends

Canned Cycle For Drilling(G80　G89)

The canned cycle for drilling simplifies the program normally by directing the machining operation commanded with a few blocks，using one block including G function。

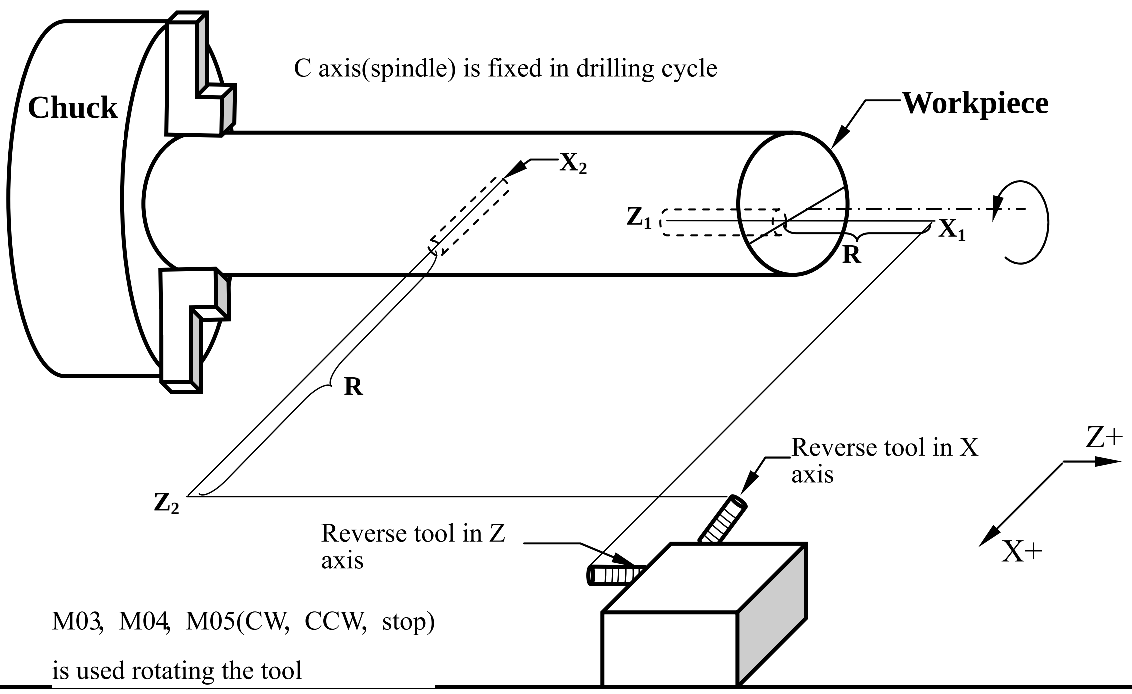
Table of Canned Cycle

G code	Drilling axis	Operation in the bottom hole position	Retraction operation	Applications
G80	----	----	----	Cancel
G83	Z	Dwell	Rapid traverse	Front drilling cycle
G84	Z	Spindle CCW	Cutting feed	Front tapping cycle
G85	Z	Dwell	Cutting feed	Front boring cycle
G87	X	Dwell	Rapid traverse	Front drilling cycle
G88	X	Spindle CCW	Cutting feed	Front tapping cycle
G89	X	Dwell	Cutting feed	Front boring cycle

Note 1：use M04 to reverse the spindle。

Note 2：G83、G87 is **cutting feed** or **intermitting feed**，that is decided by Q__ command。

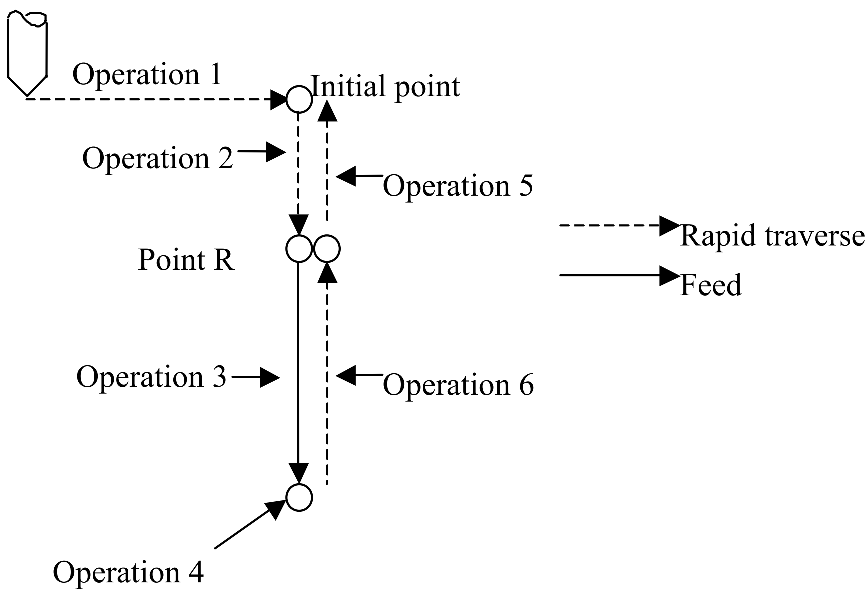
Drilling cycle figure：



※G83/G87、G84/G88、G85/G89 the front is for Z axis and the back is for X axis。

In general, the drilling cycle consists of the following six operation sequence：

Operation 1	positioning of X(Z) and C axis
Operation 2	Rapid traverse up to point R level
Operation 3	Hole machining
Operation 4	Operation at the bottom of a hole
Operation 5	Retraction to point R level
Operation 6	Rapid traverse up to the initial point



※ In G code system A, the tool returns to the initial level from the bottom of a hole。In G code system B or C, specifying G98 returns the tool to the initial level from the bottom of a hole and specifying G99 returns the hole of the point-R level from the bottom of a hole。

The following illustrates how the tool moves when G98 or G99 is specified。

Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation。

The initial level does not change even when drilling is performed in the G99 mode。

1.2.34 G83/G87：Front/Side Drilling Cycle

Format：

G83 X(U)__ C(H)__ Z(W)__ R__ Q__ P__ F__ K__ M__；
or
G87 Z(W)__ C(H)__ X(U)__ R__ Q__ P__ F__ K__ M__；

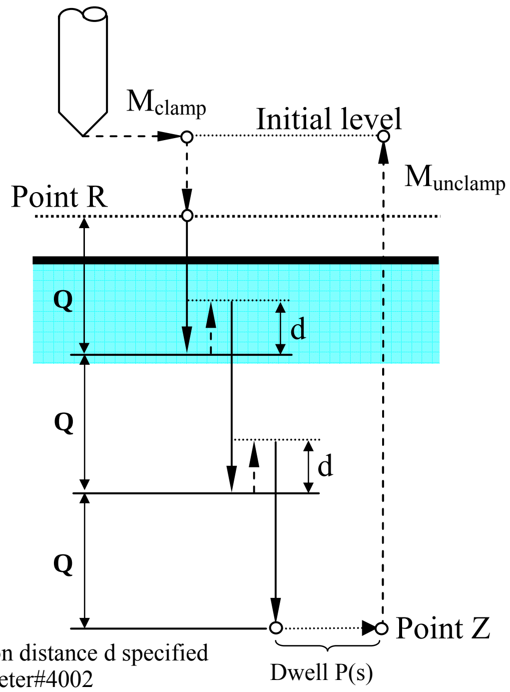
X(U)__C__or Z(W)__C__：Hole position data
Z(W)__C__or X(U)__C__：The distance from point R to the bottom of the hole
R__：The distance from the initial level to point R level
Q__：Depth of cut for each cutting feed
P__：Dwell time (s) at the bottom of a hole
F__：cutting federate
K__：Number of repeats
M__：M code for C axis clamp, C axis unclamp when Clamp Code add 1 (Unclamp Code)

Description：G83/G87 command is **front/side drilling cycle**, it is used in drilling of the lathe, it uses rotating tool to do front/side drilling cycle to clamped workpiece(fixed)。

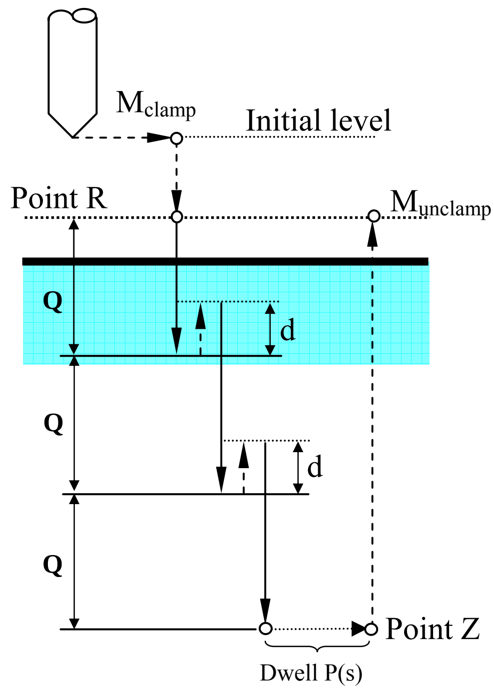
PIC：

TYPE I：High speed drilling cycle **(Custom Parameter No.4001= 1)**

G83/G87(G98)



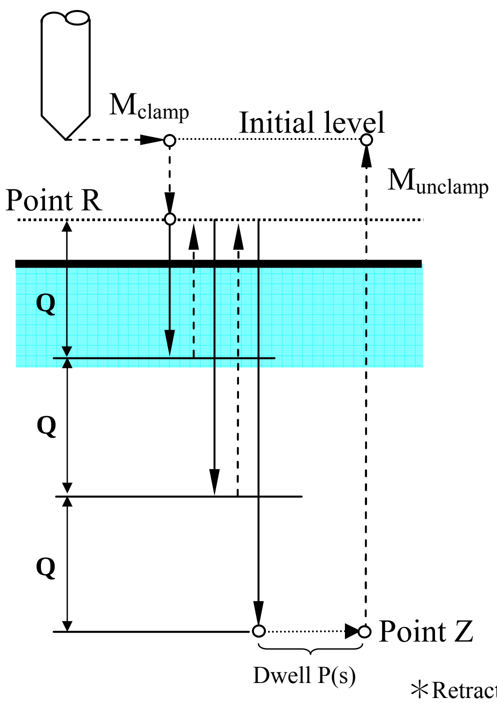
G83/G87(G99)



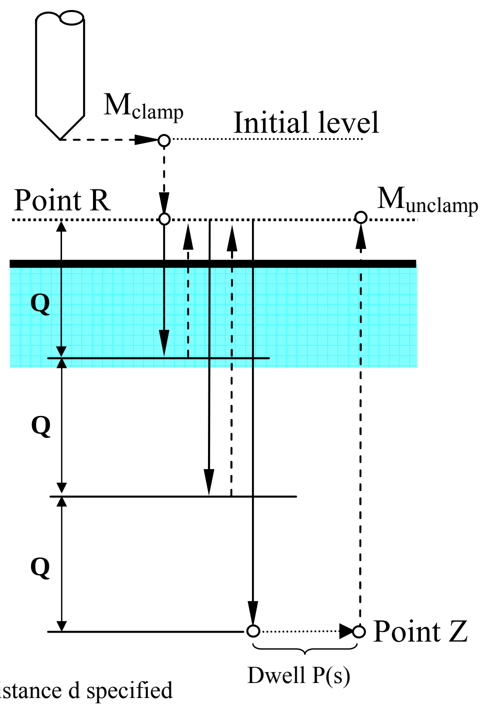
＊ Retraction distance d specified in parameter#4002

TYPE II：drilling cycle (**Custom Parameter No.4001=0**

G83/G87(G98)

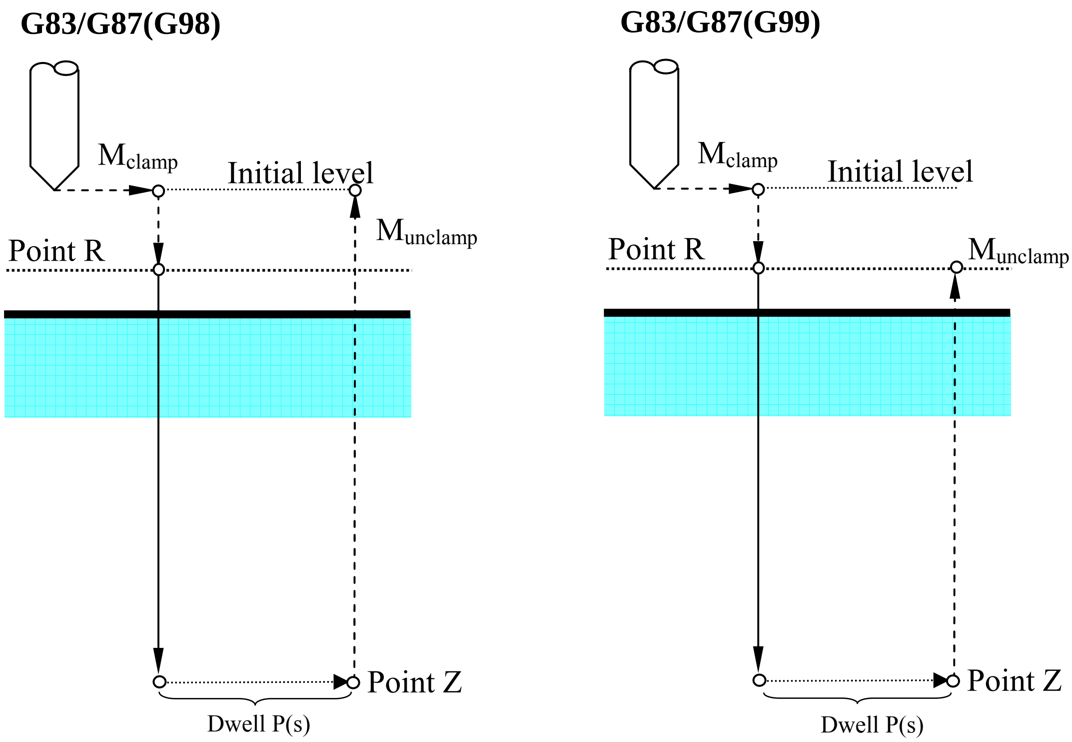


G83/G87(G99)



＊Retraction distance d specified in parameter#4002

TYPE III：Drilling without specified **Q__**



Program example：pretend M31 is Clamp command of C axis, M32 is Unclamp command of C axis

N001. S1000；//spindle speed 1000 rpm
N002. G00 X50.0；//positioning to start point
N003. G98 G83 Z-40.0 C0.0 R-5.0 P10.0 Q500 F500 M31；
// first hole drilling of C axis at 0°
N004. C90.0 M31；// second hole drilling of C axis at 90°
N005. C180.0 M31；// third hole drilling of C axis at 180°
N006. G80；//cycle cancels
N007. M02；//program ends

1.2.35 G84 / G88: Front/Side Tapping Cycle

Format：

G84 X(U)__ C(H)__ Z(W)__ R__ P__ F__ K__ M__；

or

G88 Z(W)__ C(H)__ X(U)__ R__ P__ F__ K__ M__；

X(U)__C__or Z(W)__C__：Hole positioning data

Z(W)__C__or X(U)__C__：The distance from point R to the bottom of the hole

R__：The distance from the initial level to point R level

P__：Dwell time (s) at the bottom of a hole

F__：Cutting feed rate(mm/rev), equal to the pitch of English system

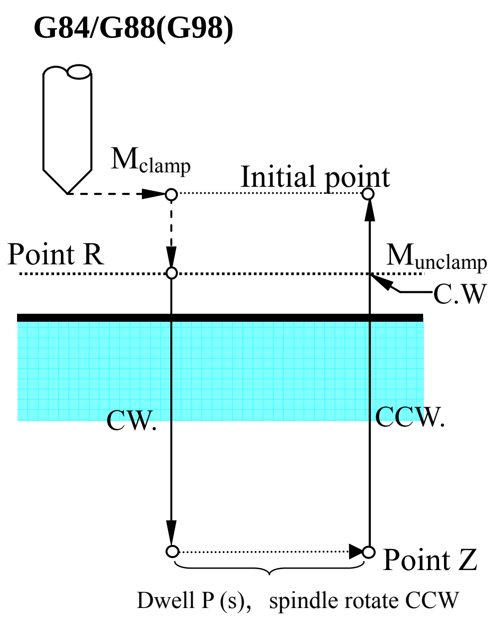
K__：Number of repeats

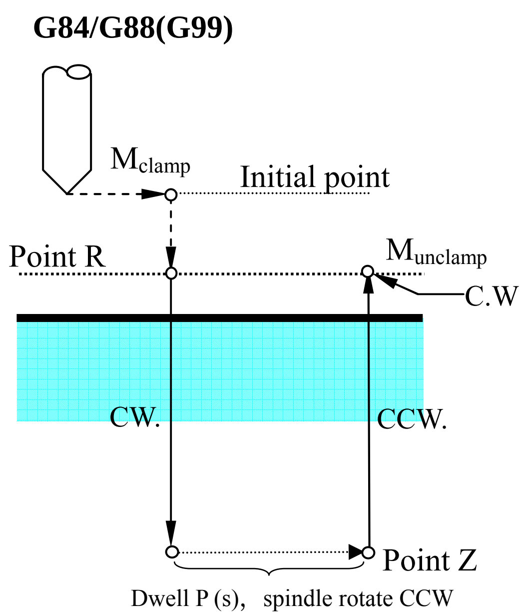
M__：M code for C axis clamp, C axis unclamp when Clamp Code add 1 (Unclamp Code)

Description：

G84 / G88 command is **Front/Side Tapping cycle**, it is used in tapping of the lathe, it uses rotating tool to do front/side tapping cycle to clamped workpiece(fixed)。

PIC：





Action description：

1. Action starts, Z axis uses G00 moving to point R(R only uses incremental)
2. Start tapping, pitch is specified F value
3. Until Z axis reach the Z depth of G84(Z absolute / W incremental)
4. Spindle stops
5. Dwell P(s) (floating point, unit : 1 s, no floating point, unit : 0.001 s)
6. Spindle rotates CCW (use M04 in CNC)
7. use the feedrate of tapping, return to point R
8. Spindle rotates CW (M03)
9. Return to initial point(G98) or stop at point R(G99)

※**Notice** ：

1. Spindle needs to rotate CW when first time tapping
2. If initial point is the same as point R, then we do not need to specify R
3. If there is no power tool seat on lathe, parameter X、C、K、M of G84 need not to specify
4. When G84/G88 ends, spindle return to rotate CW
5. G84/G88 is canceled by G80, or when G00/G01/G02/G03 executed G84/G88 will be canceled

Program example：//pretend M31 is Clamp command of C axis；M32 is Unclamp command of C axis

N001 M03 S500；//spindle starts to rotate CW 500rpm
N002 G00 X50.0；//positioning to start point
N003 G98 G84 Z-40.0 C0.0 R-5.0 P10.0 F500 M31；
// first hole drilling of C axis at 0°
N004 C90.0 M31；// second hole drilling of C axis at 90°
N005 C180.0 M31；// third hole drilling of C axis at 180°
N006 G80 M05；//cancel tapping mode, spindle stops
N007 M02；//program ends

1.2.36 G85/G89：Front/Side Boring Cycle

Format：

G84 X(U)__ C(H)__ Z(W)__ R__ P__ F__ K__ M__ ;

or

G88 Z(W)__ C(H)__ X(U)__ R__ P__ F__ K__ M__ ;

X(U)__C__or Z(W)__C__：Hole position data

Z(W)__C__or X(U)__C__：The distance from point R to the bottom of the hole

R__：The distance from the initial level to the point R level

P__：Dwell time(s) at the bottom of a hole

F__：Feed rate

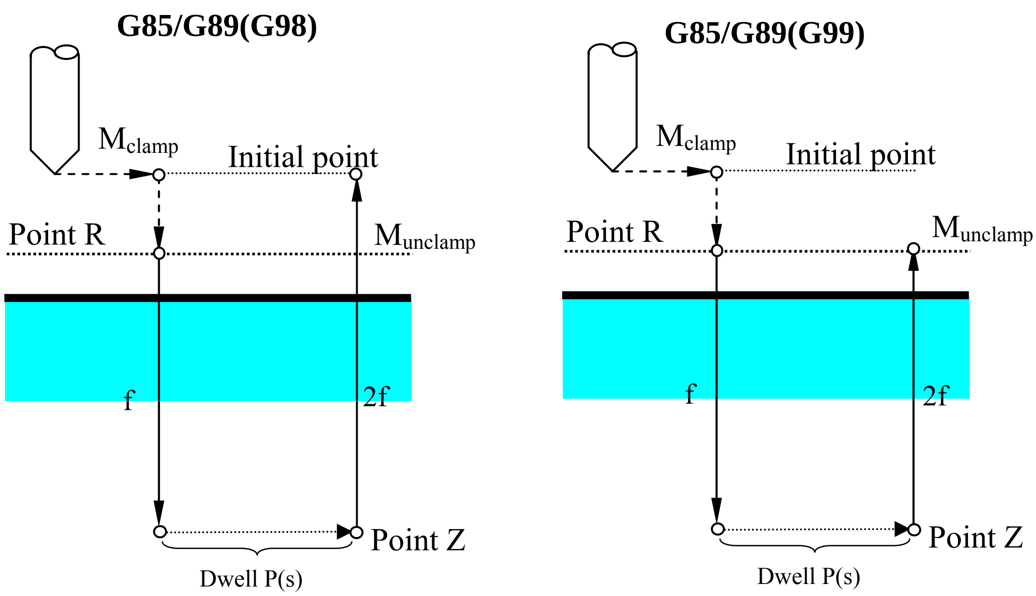
K__：Number of repeats

M__：M code for C axis clamp, C axis unclamp when Clamp Code add 1 (Unclamp Code)

Description：

G84 / G88 command is **Front/Side Boring cycle**, it is used in boring of the lathe, it uses rotating tool to do front/side tapping cycle to clamped workpiece(fixed)。

PIC：



Program example：pretend M31 is Clamp command of C axis；M32 is Unclamp command of C axis

N001 S1000 M03； //spindle rotates CW, rotate speed 1000 rpm
N002 G00 X50.0；//positioning to start point
N003 G98 G85 Z-40.0 C0.0 R-5.0 P100 F500 M31；
// first hole drilling of C axis at 0°
N004 C90.0 M31；// second hole drilling of C axis at 90°
N005 C180.0 M31；// third hole drilling of C axis at 180°
N006 G80；//cycle cancels
N007 M02；//program ends

1.2.37 G92：Coordinate System Setting/Max. Spindle Speed Setting

Format：

G92 X__ Z__ ;

or

G92 S___ ;

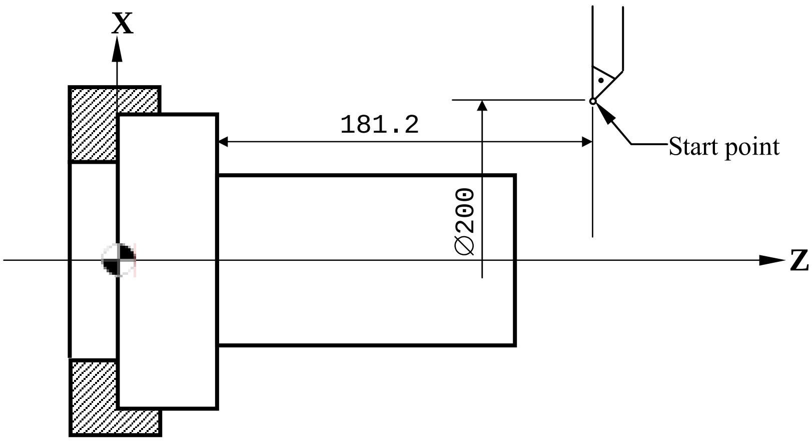
X、Z：basic coordinate system position setting (G92) in program coordinate system；

S：spindle speed；

Description：

There are two functions of G92 command 1. **coordinate system setting** or 2. Max. speed of spindle setting；It can define any fitting position to be zero point of workpiece coordinate。It uses the distance from the position of tool to the machine zero point and we use the distance to set a zero point of new coordinate。After setting, tool is machining from this point and absolute command is also reference this new coordinate。This command can use in the offset of coordinate system too。If the old coordinate is (X,Z), then the new coordinate will be (X＋ΔU,Z＋ΔW)。When we use G96 command, in order to avoid the effective diameter of workpiece is too small, and spindle speed is too high。We also use G92 to limit the max. speed of spindle。

Example one：coordinate system setting



Example： G92 X200.0 Z181.2 ;

//tool is starting from specified point

1.2.38 G94/G95：Unit Setting of Feed Amount

Format：

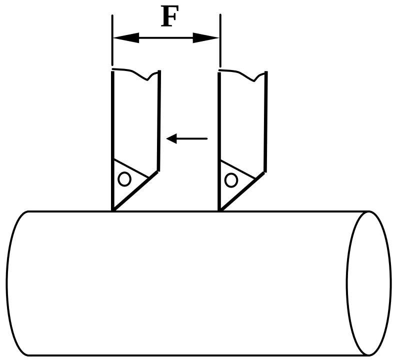
G94 F__；

G95 F__；

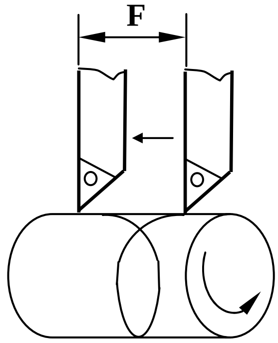
Description：

This command can set feed amount unit of F__ function(tool movement of per minute or per revolution)；G94 is for feedrate per minute(mm/min inch/min), G95 is for feedrate per minute (mm/rev, inch/rev)。

PIC：



G94. feed per minute(mm/min or inch/min)



G95. feed per revolution (mm/rev or inch/rev)

1.3.39 G96/G97：Constant Surface Speed Control

Format：

G96 S__；constant surface speed control ON

G97 S__；constant surface speed control OFF

Description：

G96 command can specify the surface speed of the contact point which is between tool and workpiece, G97 is the canceled command, G97 can set spindle speed too；If we need constant surface when cutting, we can use G96 S__ to control the surface speed；whether the diameter is big or small in machining, we can use G97 S__ to control spindle rotate speed, follow the formula：

$$\mathbf{V} = \frac{\pi \mathbf{D} \mathbf{N}}{1000}$$

V：surface speed, we can use G96 to specify a specified value, unit M/MIN or FEET/MIN。

D：valid diameter of workpiece, unit mm or inch

N：spindle rotate speed, it can be specified by G97, unit RPM。

Program example：

1.constant surface speed：G92 S2000；//limit max. rotate speed by G92

G96 S130 M03；

//cutting speed is 130m/min

※ **Notice**：G92 always use with G96, it can limit max. rotate speed of spindle。If we cut workpiece of 10mm follow upon example, then

$$N \frac{1000\times 130}{\pi \times 10} = 4140\text{rpm}$$

After G92, the max. rotate speed of spindle is 2000rpm, to avoid the rotate speed is too high, centrifugal too big, force of clip too low, so accident will happen；so we need to use G92 and G96 together

2.constant rotate speed：G97 S1300 M03；//spindle is 1300 rev/min

1.2.40 Chamfer, Corner Round, Angle Command (,C,R,A)

We could use the angle of straight line, chamfering, corner rounding in the mechanical drawing, and other dimensions that could directly use this function to input the drawing values。 And that, inserting the rounding and chamfering values in the straight lines under enough space。 This program would be effective on automatically operation only。

1.2.40-1 Chamfer (C), Corner Round (R) function：

In the continuous arbitrarily angles or arcs, and single block command formed by corner, it could execute the cutting of chamfering and R rounding by adding " **,C_** " or " **,R_** ", in the behind of the single block as mention above。 Chamfering C and rounding R are suitable in absolute value or incremental value command。

1.2.40-1.1 Chamfering (, C_)

In case of two continuous single blocks (including the arc), t the first single block "**,C_**" command could execute corner chamfering。 In case of arc, it bases on the length of arc。

Format：

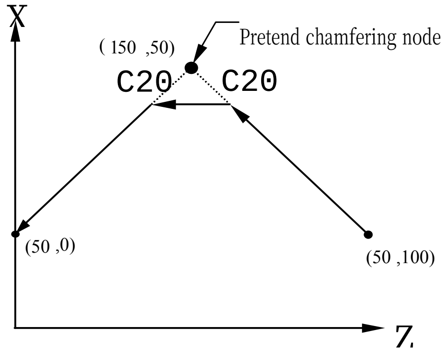
Standing for the length that supposed corner to the start or end of chamfering

N100 G03X _ Z _ I _ K _ , C_ ；

Chamfering in the intersection of N100 and N200

N200 G01X _ Z _ ；

Example：(the chamfer of straight line and arc)



Program description：

1. absolute command：

    G28 X0.0 Z0.0；
    G00 X50.0 Z100.0；
    G01 X150.0 Z50.0 F100.0 ,C20.0；
    G01 X50. Z0；

    Chamfering C20.0 between the movement of these two blocks

2. incremental command：

    G28 X0.0 Z0.0；
    G00 U50.0 W100.0；
    G01 U100.0 W-50.0 F100, C20.0；
    G01 U-100.0 W-50.0；

    Chamfering C20.0 between the movement of these two blocks

1.2.40-1.2 Corner Round R(, R_)

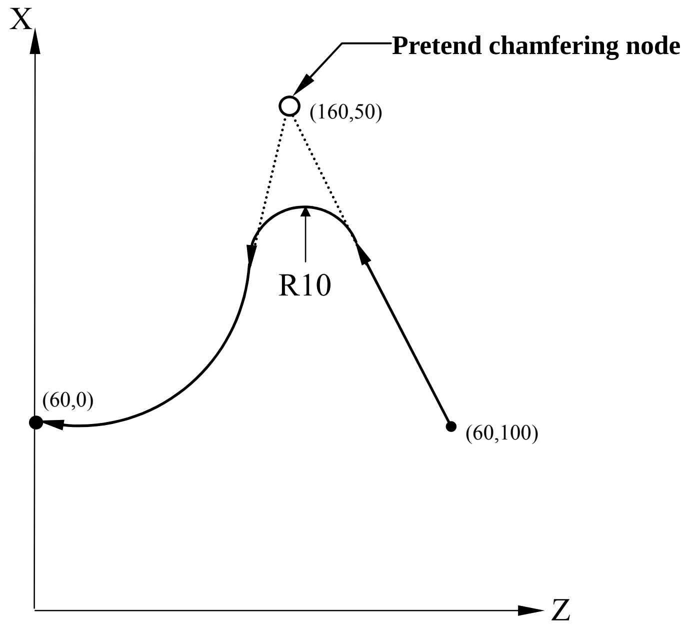
Responding to two continuous single blocks (including arc), at the first single block, it could use "**,R_**" command to indicate the corner R executing function。

Format：

, R__ ；

R：for radius of corner and arc。

Example：(corner between straight line and arc)



Program description：

1. absolute command

G28 X0.0 Z0.0；

G00 X60.0 Z100.0；

G01 X160.0 Z50.0 F100 ,R10.0；

G02 X60.0 Z0.0 I0.0 K-50.0;

Rounding R10.0 between the movement of these two blocks

2. incremental command

G28 X0.0 Z0.0；

G00 U60.0 Z100.0；

G01 U100.0 W-50.0 F100 , R10.0；

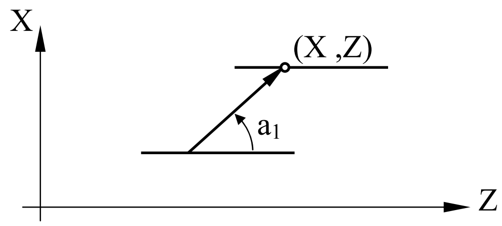
G02 U-100.0 W-50.0 I0.0 K-50.0；

Rounding R10.0 between the movement of these two blocks

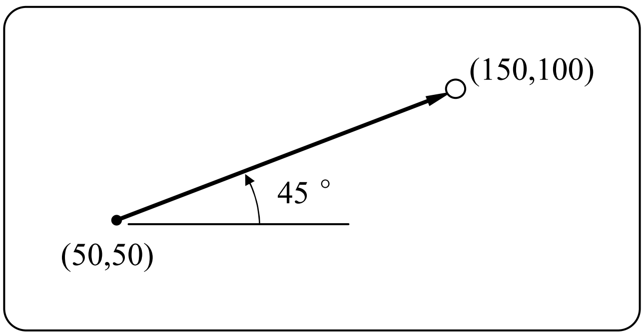
1.2.40-2 Angle Command (,A_)：

Format：

G01 Z__ (X__) ,A$\underline{a_1}$ ；//specify the angle and the coordinate of X or Z。



Example：



Program description：

N01 G00 X50.0 Z50.0 ；

//positioning to specified point

N02 G01 Z100.0,A45.0 ；

//the angle between tool path and horizontal axis is 45°

end point absolute coordinate value of Z is 100

＊after executing program→coordinate value of X is 150

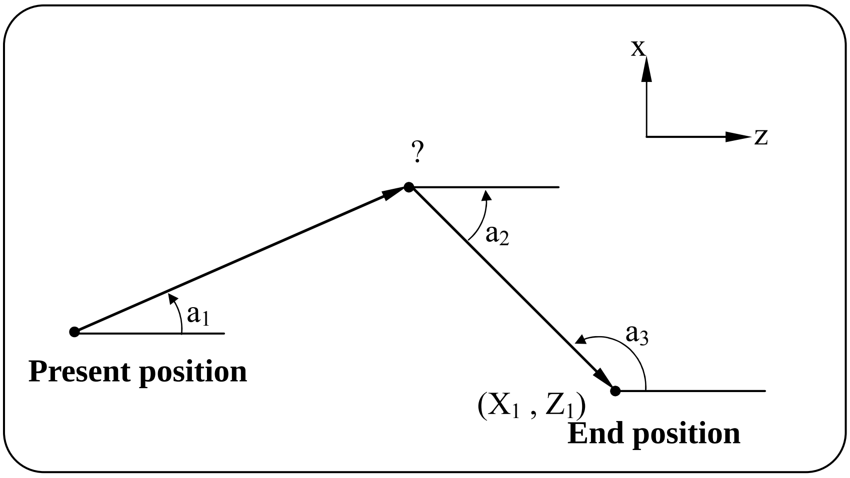
1.2.40-3 Geometric Function Command：

In continuous linear interpolation command, if it is hard to get the node of two lines。We can use the sloping angle of the first line, absolute coordinate value of second line and the sloping angle of the first line to be the command, then NC controller will compute the end of the first line, and it can do continuous straight line corner function。

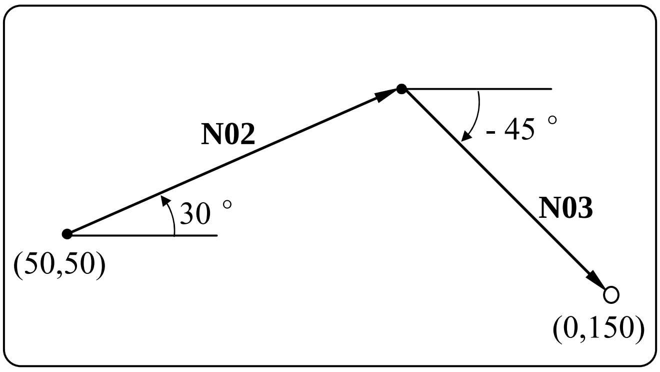
Format：

G01 ,A$\underline{a_1}$ **F__** ; //specified angle

X$\underline{x_1}$ **Z**$\underline{z_1}$ **,A**$\underline{-a_2}$; //specified the end coordinate value and the angle of the next block



Example：



Program description：

N01 G00 X50.0 Z50.0 ;

//positioning to specified point

N02 G01 ,A30.0 F300 ;

//angle (30°**) between the first path and horizontal axis**

N03 X0.0 Z150.0,A45.0 ;

// angle (-45°**) between the first path and horizontal axis, end point** (0,150)

＊after executing program→the node of path (104.904 ,97.548)

※Notice：

(1) This function is effective only under G01, it is not effective of other interpolation or positioning command。

(2) The angle is the horizontal axis adding the angle in + direction at specified plant, CCW is for positive, CW is for negative。

(3) The sloping angle can be specified in start point or end point of start side or end side。 NC in side can specified the start side or end side of sloping angle automatically。

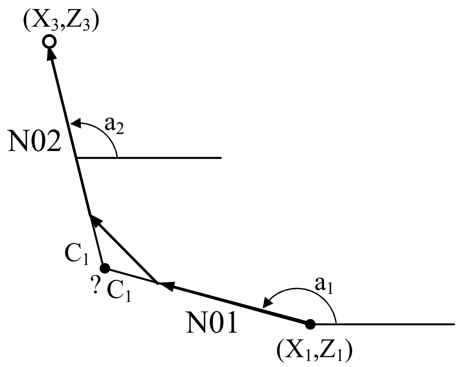
(4) If we use the second way to specify, we need to specify the end point to be absolute coordinate。

Relative usage：

TYPE I： In the first angle command, we can specify **Chamfer command** or **Angle Round command**

(1). Format：

N01 ,Aa_1 ,Cc_1 ;
N02 Xx_3 Zz_3, Aa_2 ;

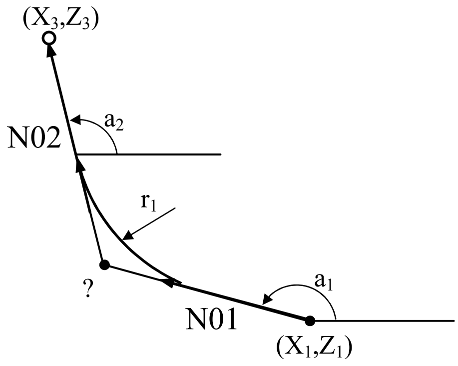


Description：Tool reaches to **specified position(X_3,Z_3)** according to the command, and there are specified **angle『a_1』、『a_2』** between the twice movement path and horizontal axis, and there is a **chamfer angle 『C_1』** of the corner of two path。 Contorller use specified value to computer the unknow intersection **"?"** of two path, and tool do cutting to specified position**(X_3,Z_3)** along the two path。

(2). Command Format：

N01 ,Aa$_1$,Rr$_1$；

N02 Xx$_3$ Zz$_3$ Aa$_2$；

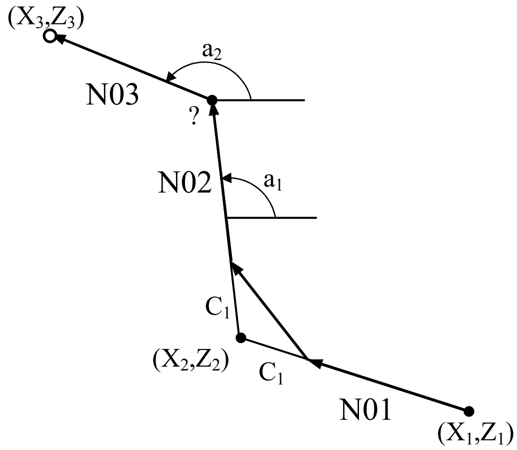


Description：Tool reaches to **specified position(X$_3$,Z$_3$)** according to the command, and there are specified **angle**『**a$_1$**』,『**a$_2$**』 between the twice movement path and horizontal axis, and there is a **round angle**『**r$_1$**』of the corner of two path。Contorller use specified value to computer the unknow intersection **"?"** of two path, and tool do cutting to specified position**(X$_3$,Z$_3$)** along the two path。

TYPE Ⅱ：After Chamfering command、Angle round command (R), we can continue to do linear angle command

Format：

N01 Xx_2 Zz_2, Cc_1；
N02 ,Aa_1；
N03 Xx_3 Zz_3, Aa_2；

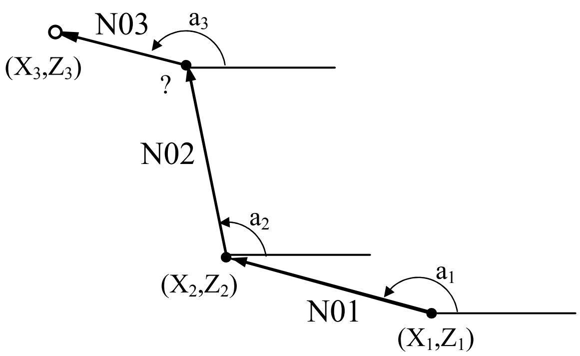


Description：Tool reaches to specified position **(X_2,Z_2)→ (X_3,Z_3)** according to the command, and there is a **chamfering angle**『**C_1**』between the corner of the front two path, and there are **specified angle**『**a_1**』、『**a_2**』between the back two path and horizontal axis。Contorller use specified value to computer the unknow intersection **"?"** of two path, and tool cuts to end point **(X_3,Z_3)** along the three pathes

TYPE Ⅲ：After linear angle command, we can continue to do linear angle command

Format：

N01 Xx_2 Aa_1；
N02 ,Aa_2；
N03 Xx_3 Zz_3, Aa_3；



Desctiption：Set the the X axis coordinate value **“X_2”** of the first movement path according to the command, and the **angle**『a_1』 to horizontal axis, and the end point value**(X_3,Z_3)** of the third movement path, and the **angle**『a_2』、『a_3』between the front path, **the angle** between horizontal axis and the axis of the front path； Contorller use specified value to computer the unknow intersection **”?”** of two path, and tool cuts to end point **(X_3,Z_3)** along the three pathes。

※**Notice**：

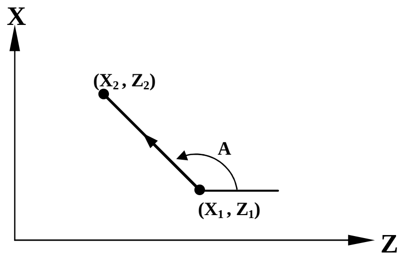
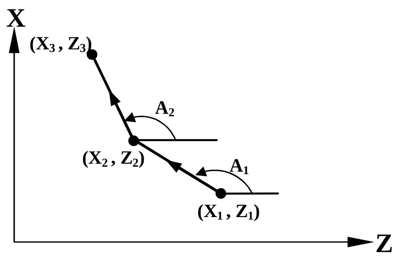
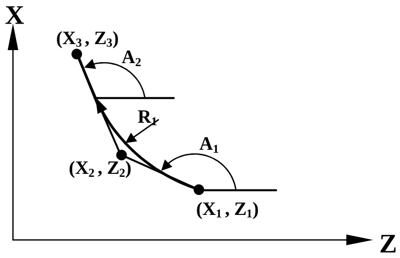
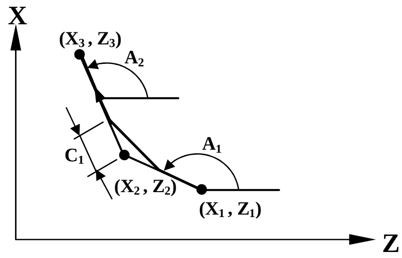
1. Round angle value can not be inserted in threading area.
2. Entering the continuous command in next area by drawing size。 Than the end point is already be specified in the front area。 Stop can not be executed in single area, but dwell can be executed in the front area.
3. allowance range of angle computing is $\pm 1°$.

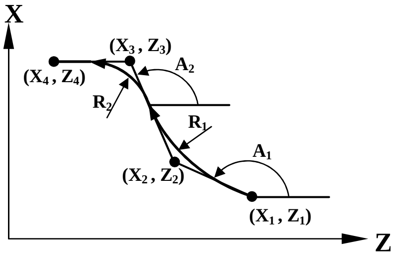
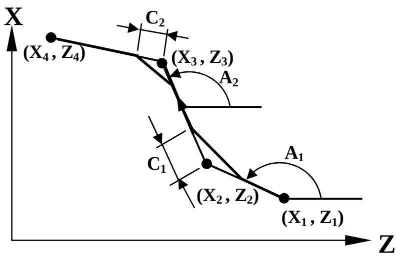
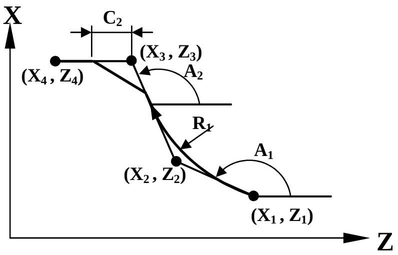
(0). X_ , A_ ; (when the angle is 0 ° ±1 , 180 ° ±1 , it will be alarming)

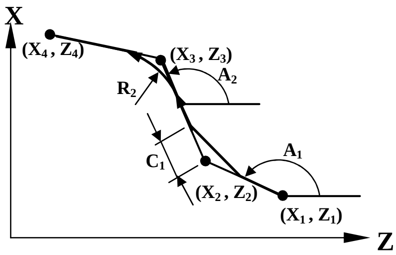
(1). Z_ , A_ ; (when the angle is 90 ° ±1 , 270 ° ±1 , it will be alarming)

4. If the angle between two lines is under ±1 ° , it will be alarming when we computer the intersection.
5. If the angle between two lines is under ±1 ° , we can ignore chamfer angle and round angle.

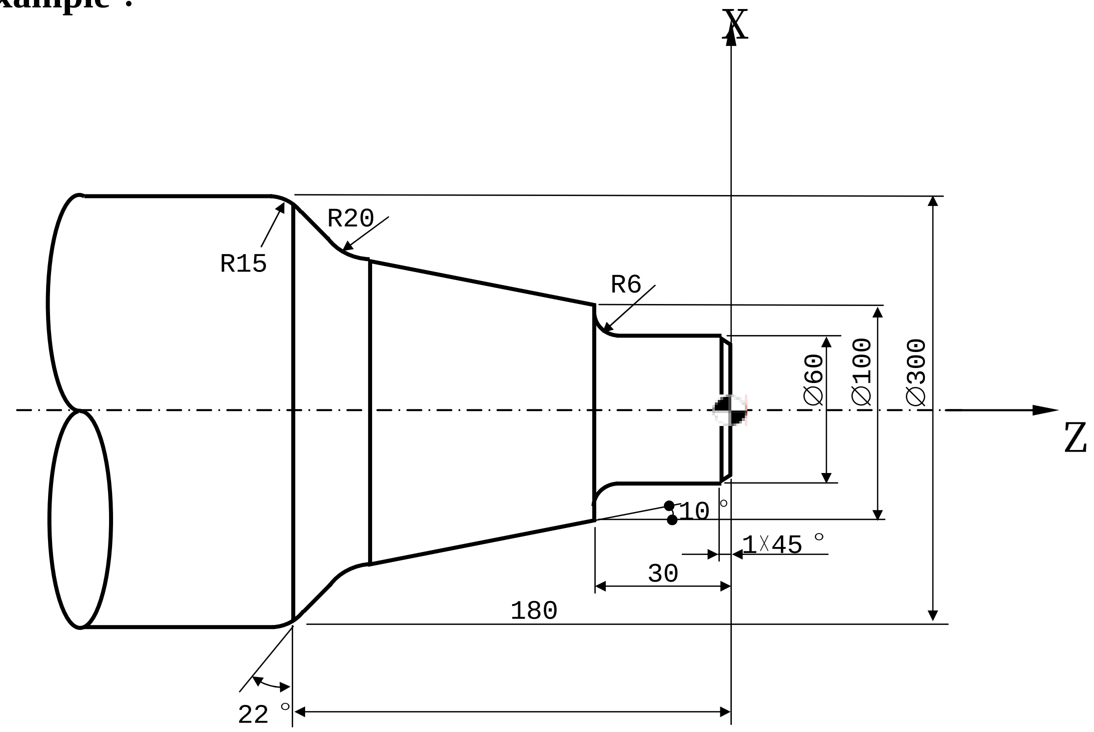
Geometric Function Usage Table

	Command	Movement	Description
1.	**X_2__ (Z_2)__, A__;**		According to the any coordinate value of **X_2(or Z_2)** and the **angle** 『**A**』 which is between path and horizontal axis。Use controller to computer the other unknow Z_2(or X_2), and tool can cut to specified position (X_2,Z_2) along this path
2.	**,A_1__ ;** **X_3__ Z_3__, A_2__ ;**		According to the setting command to reach specified point **(X_3,Z_3)**, and the specified **angle** 『**A_1**』、『**A_2**』 which are between each path and horizontal axis。Use controller to computer the unknoew intersection**(X_2 , Z_2)**, and tool will cut to specified point**(X_3,Z_3)** along the two path
3.	**X_2__ Z_2__,R_1__ ;** **X_3__ Z_3__ ;** **Or** **,A_1__, R_1__ ;** **X_3__ Z_3__, A_2__ ;**		According to the setting command to reach specified point **(X_3,Z_3)**, and the specified **angle** 『**A_1**』、『**A_2**』 which are between each path and horizontal axis, and the corner is the **round angle** 『**R_1**』。Use controller to computer the unknoew intersection**(X_2 , Z_2)**, and tool will cut to specified point**(X_3,Z_3)** along the two path
4.	**X_2__ Z_2__,C_1__ ;** **X_3__ Z_3__ ;** **Or** **,A_1__, C_1__ ;** **X_3__ Z_3__, A_2__ ;**		According to the setting command to reach specified point **(X_3,Z_3)**, and the specified **angle** 『**A_1**』、『**A_2**』 which are between each path and horizontal axis, and the corner is the **chamfer angle** 『**R_1**』。Use controller to computer the unknoew intersection**(X_2 , Z_2)**, and tool will cut to specified point**(X_3,Z_3)** along the two path

	Command	Movement	Description
5.	X_2__ Z_2__,R_1__ ; X_3__ Z_3__, R_2__; X_4__ Z_4__ ; **Or** ,A_1__, R_1__ ; X_3_ Z_3__, A_2__,R_2__ ; X_4__ Z_4__ ;		According to the command to reach to the **specified position** **(X_2,Z_2)→ (X_3,Z_3) → (X_4,Z_4)**, the corner of the front two path is a **round angle** 『**R_1**』, the corner of the back two path is a **round angle** 『**R_2**』, (or we do not specify **(X_2,Z_2)** but we add 『**A_1**』、『**A_2**』)。Controller will computer『**A_1**』、『**A_2**』or unknow intersection**(X_2 , Z_2)** by the specified value。Tool will cut to end point **(X_4,Z_4)** along these pathes
6.	X_2__ Z_2__,C_1__ ; X_3__ Z_3__, C_2__; X_4__ Z_4__ ; **Or** ,A_1__, C_1__ ; X_3__ Z_3__, A_2__,C_2__ ; X_4__ Z_4__ ;		According to the command to reach to the **specified position** **(X_2,Z_2)→ (X_3,Z_3) → (X_4,Z_4)**, the corner of the front two path is a **chamfer angle** 『**C_1**』, the corner of the back two path is a **chamfer angle**『**C_2**』, (or we do not specify **(X_2,Z_2)** but we add 『**A_1**』、『**A_2**』)。Controller will computer『**A_1**』、『**A_2**』or unknow intersection**(X_2 , Z_2)** by the specified value。Tool will cut to end point **(X_4,Z_4)** along these pathes
7.	X_2__ Z_2__,R_1__ ; X_3__ Z_3__, C_2__; X_4__ Z_4__ ; **Or** ,A_1__, R_1__ ; X_3__ Z_3__, A_2__,C_2__ ; X_4__ Z_4__ ;		According to the command to reach to the **specified position** **(X_2,Z_2)→ (X_3,Z_3) → (X_4,Z_4)**, the corner of the front two path is a **round angle** 『**R_1**』, the corner of the back two path is a **chamfer angle**『**C_2**』, (or we do not specify **(X_2,Z_2)** but we add 『**A_1**』、『**A_2**』)。Controller will computer『**A_1**』、『**A_2**』or unknow intersection**(X_2 , Z_2)** by the specified value。Tool will cut to end point **(X_4,Z_4)** along these pathes

8.	**X_2___ Z_2___,C_1___** ; **X_3___ Z_3___, R_2___** ; **X_4___ Z_4___** ; Or **,A_1___, C_1___** ; **X_3___ Z_3___,** **A_2___,R_2___** ; **X_4___ Z_4___** ;		According to the command to reach to the **specified position (X_2,Z_2)→ (X_3,Z_3) → (X_4,Z_4)**, the corner of the front two path is a **chamfer angle**『**C_1**』, the corner of the back two path is a **round angle**『**R_2**』, (or we do not specify **(X_2,Z_2)** but we add『**A_1**』、『**A_2**』)。Controller will computer『**A_1**』、『**A_2**』or unknow intersection**(X_2 , Z_2)** by the specified value。Tool will cut to end point **(X_4,Z_4)** along these pathes

Example：



Program description： (input diameter by Metric system)

N002 G01 X60.0 A90.0, C1.0 F80；

//linear interpolation, the angle between the straight line and horizontal axis is "+90°", and chamfering C1.0 angle at the next block, feed rate 80 μm/rev

N003 Z-30.0, A180.0 R6.0；

// linear interpolation, the angle between the straight line and horizontal axis is "+180°", and rounding R6.0 angle at the next block

N004 X100.0, A90.0；

// linear interpolation, cutting to specified point, the angle between the straight line and horizontal axis is "+90°"

N005 ,A170.0, R20.0；

// linear interpolation, the angle between the straight line and horizontal axis is "+170°", and rounding R20.0 angle at the next block, the end point is specified in the next block

N006 X300.0 Z-180.0, A112.0, R15.0；

// linear interpolation, the angle between the straight line and horizontal axis is "+112°", and rounding R15.0 angle at the next block

N007 Z-230.0, A180.0；

// linear interpolation, the angle between the straight line and horizontal axis is "+180°", cutting to specified position

1.2.41 Tool Function : T code command

Format：

T__

Description：

Tool function is also called T function。It's main function is tool exchange。It will use with （M０６） normally。We can do auto tool exchange according the number of tool。

Example：

T03 M06；//it is for changing to tool NO.3

1.2.42 Spindle Rotate Speed Function : S code command

Format：

S__

Description：

S function is spindle speed command, it specifies revolution per minute or surface speed of spindle by G96/G97。

Example：

G96 S150 M03；//constant surface speed of spindle, 150 m/min

G97 S500 M03；//spindle keeps 500 rev/min

1.2.43 Feed Function : F code command

Format：

F__

Description：

In cutting mode, the specified movement speed of tool in the program is called feed。There are two way to set feed mode, G94/G95。G94 F300 is for 300 mm/min；G95 F0.5 is for 0.5 mm/rev。

Example：

G94 G01 X100.0 Y100.0 F300；//linear interpolation, feed rate 300mm/min

G95 G01 X100.0 Y100.0 F0.5；//linear interpolation, feed rate 0.5mm/rev

1.2.44 PROGRRAMBLE MIRROR IMAGE

Format:

G68; Start X axis progrramble mirror image

G69; Cancel progrramble mirror image

Description: With double turrets in lathe we can mirror the location in X-axis with XO by G code. It is more convenient with double turrets because it is not necessary to consider the moving direction of the turret.

1. The direction of the circular interpolation and tool nose radius compensation or the coordinate reversal is opposite.

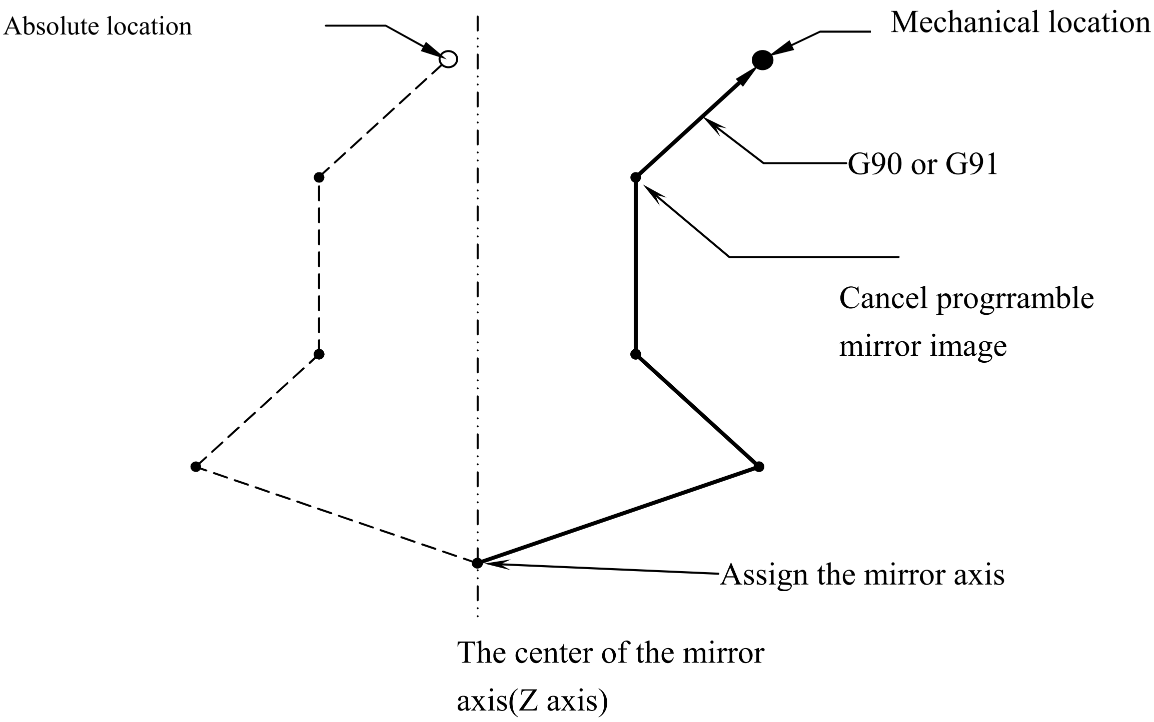
2. Because this instruction is used in local coordinate, the center of the mirror still moves when reset the counter or the working coordinate changes.

3. When execute the instructions (G28,G30) within the one of the progrramble mirror image it works until middle point not from it to the origin.

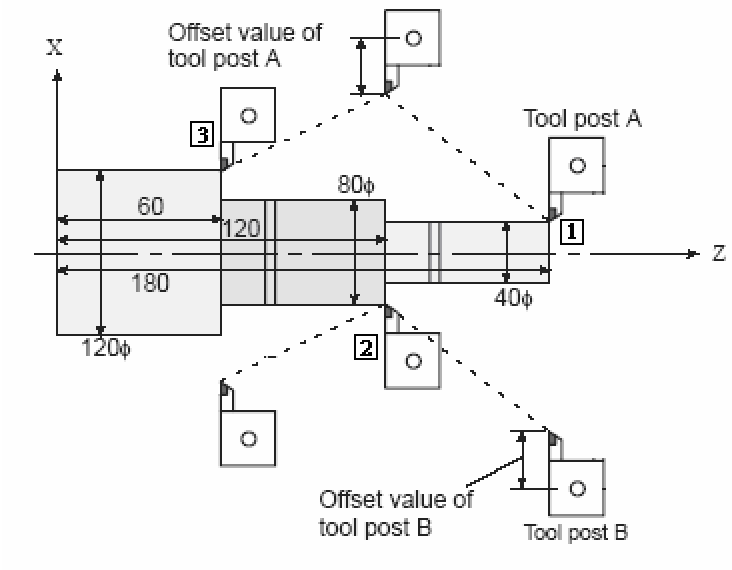
4. When execute the instruction (G29) within the one of the progrramble mirror image it works in the middle point.

Attention:

Cancel progrramble mirror image except the center of the mirror and it can not match between absolute location and mechanical location. The situation is below.it lasts until set the instructions G90 G28 G30. Under motionless setting in the center of the mirror if we re-assign it , the location is unexpected. So we should use G90 after cancelling progrramble mirror image.



Ex:



Program illustration：

```
N001   T0101                 //turret 1
N002   G01   Z180.   X40.    //position-1
N003   Z120.
N004   T0202                 //turret 2
```

```
N005  G68                    //enable X-axis mirror image
N006  G01  Z120.  X80.   //position 2
N007  Z60.
N008  T0101                   //turret 1
N009  G69                    //disable X-axis mirror image
N009  G01  Z60.  X120.   //position 3
N010  M99
```

B、 M Code Command Description：

auxiliary function is used on controlling the On and OFF of the machine function. There are two numbers behind the code to be the format。We introduce the the application number and function list as below：

M Function Table

M code	Function
M00	Dwell
M01	Optional dwell
M02	End of program
M03	Spindle rotates (CW)
M04	Spindle rotates (CCW)
M05	Spindle stops
M06	Tool exchange
M08	Cutting liquid ON
M09	Cutting liquid OFF
M10	Tight the clamp
M11	Loose the clamp
M19	Spindle location，let spindle stops at a specified position
M30	Program ends，return to start point
M98	Calling of subprogram
M99	End of subprogram

1、M00：Dwell

when CNC executes M00 command，the spindle will stop，the feed will dwell, and the cutting liquid will close for dimension inspection and calibrating compensation tasks of operator。We could decide the program will dwell or not from the M00 signal button on the panel when we operate it。

2、M01：Optional dwell

M01 function is similar with M00；but M01 is controlled by “optional stop”；when the switch is in ON status，and M01 is effective，then the program will dwell。If the switch is in OFF status，then M01 is invalid。

3、M02：Program ends

If there is M02 command on the end of program, CNC execute this command, the machine will stop all action at the same time。If you want to restart the program, would be effective only by pressing the “RESET” button, then the “STOP”。

4、M03：Spindle rotates CW

M03 command can let spindle rotate CW。Spindle can rotate in specified speed and rotate CW when M03 is used with S function。

5、M04：Spindle rotates CCW

M04 command can let spindle rotate CCW

6、M05：Spindle stops

M05 command can let spindle stop, when we want to change the gear or the direction of rotating, we need to use M05 to stop the spindle first。

7、M06：Tool exchange

M06 command can execute tool exchange, this command is not include tool choosing, it must use with T__function。

8、M08/M09：Cutting liquid ON/OFF

M08 command is for cutting liquid ON, M09 command is for OFF

9、M19：Spindle locates and stops

This command can locate the spindle at specified corner

10、M30：Program ends

M30 command is the end of the program。when program executes M30, all action will stop, and the memory will return to the beginning of the program。

＊11、M98/M99：Subprogram Control

Format：

(1). **M98 P__ H__ L__**；Calling of subprogram

P：the number of calling subprogram(when P is ignored, it is for program itself, and it is only for memory running or MDI mode)

H：call the starting executing sequence number in subprogram (when it is ignored, it will execute from the front)

L：times of program repeat executing。

(2). **M99 P__ L__**；subprogram ends

P：the sequence number of returning to program after subprogram ends。

Description：

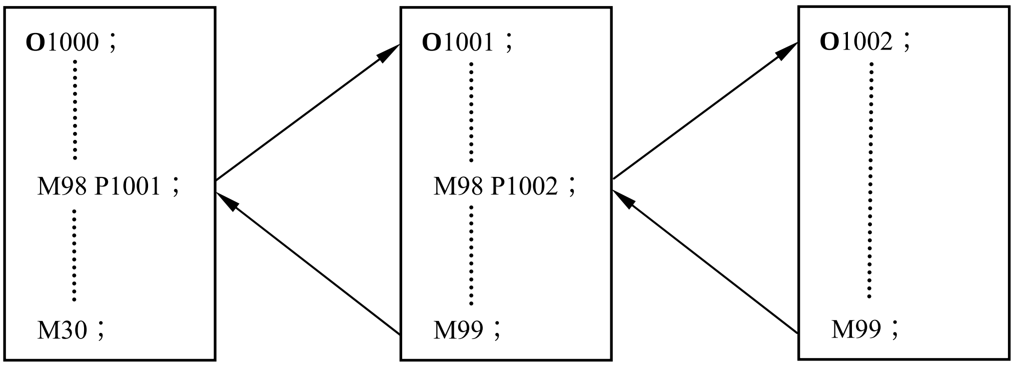
1. Subprogram is the parameter that including fixed cutting procedures or repeat using frequently。 We should prepare it in advance and put it into the memory。 We call from the main progra when we need to use。 Calling subprogram is executed by M98, and it would stop by executing M99。
2. When run M02 and M30 in the subprogram we regard it as the end of the subprograms and return to the main program to run.

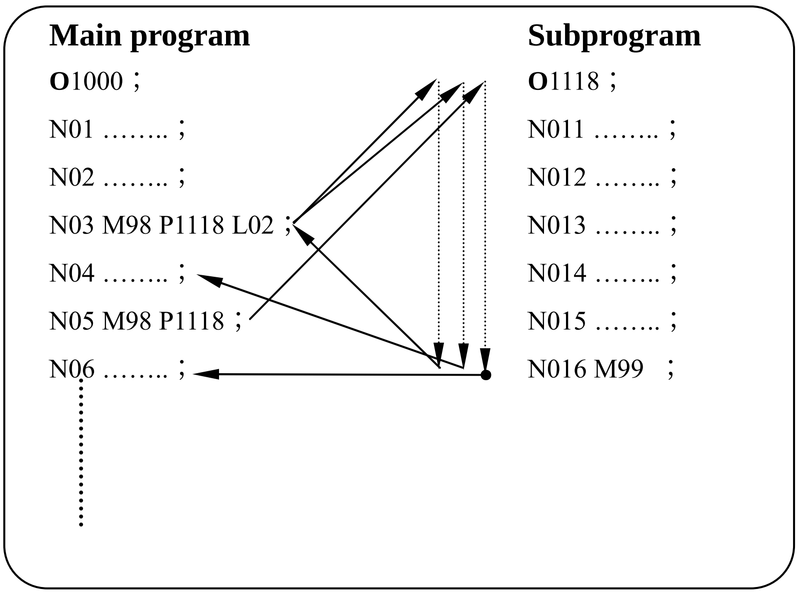
※ Making and Executing of Subprogram：

The normal format as below：

```
O××××；------------------------------------------------- number of subprogram
   G01 ……….. ；
   G02 ……….. ；        Content of program
   G01 ……….. ；
        ⋮
   M99；------------------------------------------------------ subprogram ends
```

Main program use with calling of subprogram, and sequence of executing：





Special usage of subprogram：

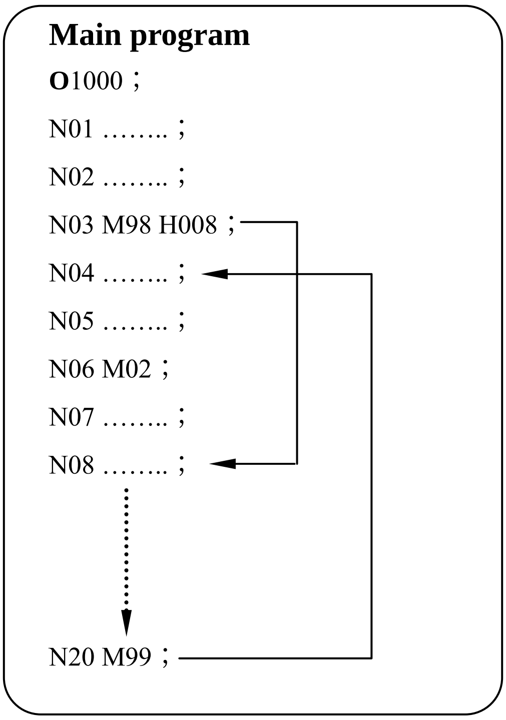
(1). We can execute subprogram by adding **H__** function after M99 in the ending of the final block。 After finishing this program, it will return to main program, and execute the block of the sequence number which is specified by H_ function。

Main program	**Subprogram**
O1000；	**O**1118；
N01 ……..；	N011 ……..；
N02 ……..；	N012 ……..；
N03 M98 P1118；	N013 ……..；
N04 ……..；	N014 ……..；
N05 ……..；	N015 ……..；
N06 ……..；	N016 M99 H006；

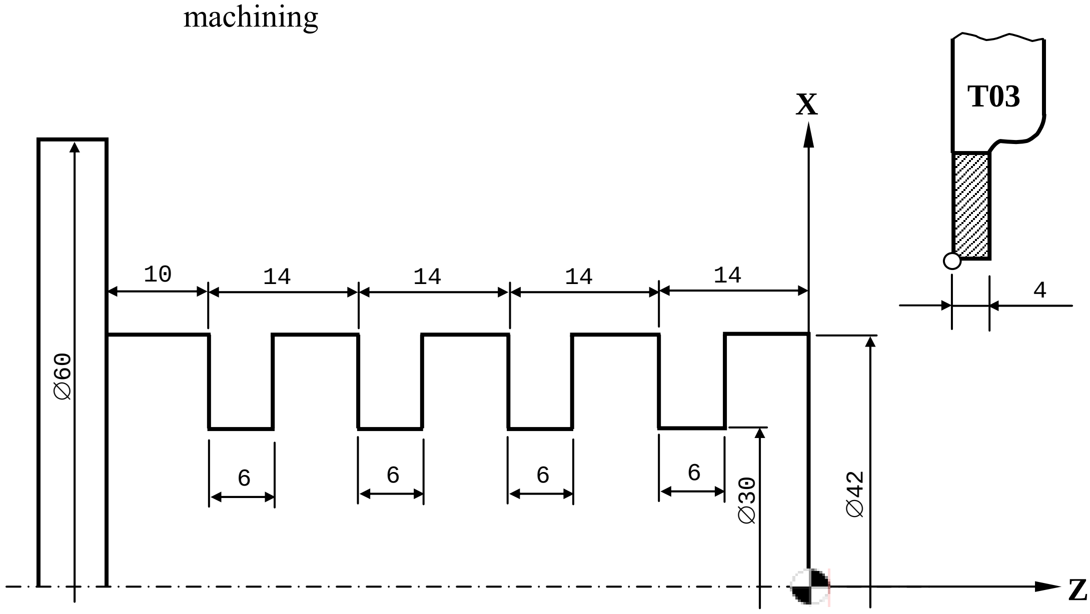
(2). Subprogram also can execute P_ command and H_ command in M98, it will execute the program from the sequence number (specified by H_) of subprogram (specified by P_)。 This kind of subprogram is multipurpose, it can save more memory space。

Main program	**Subprogram**
O1000；	**O**1118；
N01 ……..；	N011 ……..；
N02 ……..；	N012 ……..；
N03 M98 P1118 H013；	N013 ……..；
N04 ……..；	N014 ……..；
N05 ……..；	N015 ……..；
N06 ……..；	N016 M99；

(3). If there is no P_ command and only has H_ command in the subprogram。 The result after calling, it will execute sequence number of main program that specified by H_ command。 After executing M99 it will return to the next block of M98 and continue to execute the program。



Example： cutting a tank, use "calling of subprogram" to execute repeating machining



Program description：

(1). First way：P command in block of M98

＊Main program.

T03；//use tool NO.3
G97 S710 M03；//constant rotate speed of spindle, 710 rpm CW
M08；//cutting liquid ON
G00 X45.0 Z-12.0；//positioning to the above of first tank
M98 P1234 H102 L4 ；
//call the subprogram of sequence number “O1234”, machining from the block of N102, and repeating 4 times
G28 X80.0 Z80.0；
//positioning to specified mid-point and return to machine zero point
M09；//cutting liquid OFF
M05；//spindle stops
M30

＊Subprogram.

O1234

N101 G00 X45.0 Z-12.0；
N102 G01 X30.0 F200； **←Start from this block**
//linear interpolation to the bottom of the tank, feedrate 200 μ m/rev
N103 G00 X45.0；//escaping to start position
N104 W-2.0；//move 2mm toward negative direction of Z
N105 G01 X30.0；//linear interpolation to the bottom of the tank
N106 G00 X45.0；// escaping to start position
N107 W-12.0；// move 12mm toward negative direction of Z, and wait for cutting next tank
N108 M99；//return to main program

(2). Second way：without executing P_ command in block of M98

＊Main program.

N001 T03；//use tool NO.3
N002 G97 S710 M03；//constant rotate speed of spindle, 710 rpm CW
N003 M08；//cutting liquid ON
N004 G00 X45.0 Z-12.0；//positioning above the first tank
N005 M98 H0010 L4 ；
//execute from the block of main program sequence number N0010, and repeat 4 times

```
N006   G28 X80.0 Z80.0；
              //positioning to specified mid-point and return to machine zero point
N007   M09；//cutting liquid OFF
N008   M05；//spindle stops
N009   M02；//program ends
N0010 G01 X30.0 F200；     ←start with this block after executing M98
       //linear interpolation to the bottom of the tank， feedrate 200μm/rev
N0011 G00 X45.0；//escaping to start point
N0012       W-2.0；// move 2mm toward negative direction of Z
N0013 G01 X30.0；//linear interpolation to the bottom of the tank
N0014 G00 X45.0；//escaping to start point
N0015       W-12.0；// move 12mm toward negative direction of Z， and wait for
                    cutting next tank
N0016 M99；//return the next block N006 of M98
```

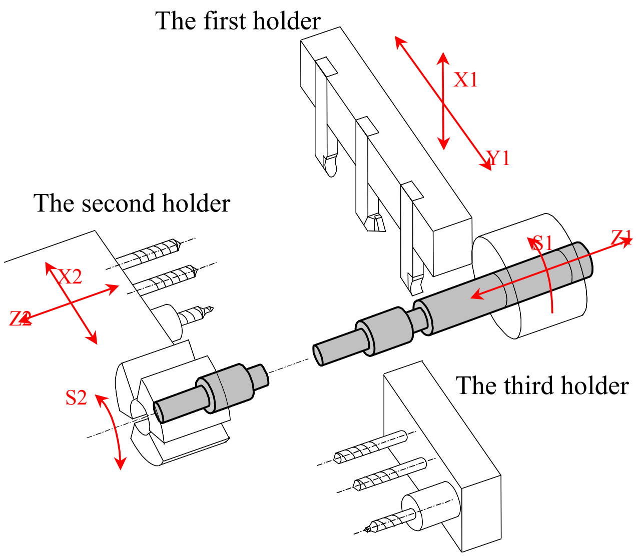
Postscript 1：Description of lathe parameter

NO	Explain	Input range	Unit	Description
4001	Drilling mode	[0,1]		0:high speed;1:normal
4002	Escaping amount of drilling cycle	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4011	Escaping amount of peck drilling cycle	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4012	Escaping amount of cutting cycle	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4013	Cutting value of cutting cycle	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4015	Cutting value of pattern repeating in X direction	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4016	Cutting value of pattern repeating in Z direction	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4017	Number of repeats of pattern repeating	[1,999]	Number of times	
4018	Camfer angle of thread cutting G21	[0,89]	degree	
4041	Finishing allowance of threading	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4042	Thread angle of threading	{0,29,30,55,60,80}	Degree	
4043	Chamfering value of threading	[0,99]	0.1 pitch	
4044	Times of finishing allowance in threading	[0,99]	Number of times	
4045	Min. cutting value in threading	[0,999999999]	LIU	LIU is min. input unit，and it will be effective by Metric or English system。
4050	C axis motor is used on spindle or not	[0,1]		This function is used with Marco command M19 C_，when we use this function，we need entry M18 /M50 /M51 to system parameter 360X M code Marco registry table

NO	Explain	Input range	Unit	Description
4051	*start the setting screen of workpiece coordinate	[0,1]		Start the setting screen of workpiece coordinate,0 for disable；1 for enable

Postscript 2：Description of lathe double program

To save the time of the processing, the SYNTEC lathe's controllers can drive two programs simultaneously. They can drive two pairs of turret to program linear interpolation and circular interpolation. We can lathe workpieces in external diameter and internal diameter effectively when programming.



X1 Y1 Z1 are the first group axes
X2 Z2 are the secnd group axes
S1 is the first main axis
S2 is the second main axis

1、The description of the related instructions:

$1->the contents after the instructions in the program is the first group
$2->the contents after the instructions in the program is the second group
The second group in the program must end with M99.
G04.1 P_->synchronous instructions,G04.1 P1 in the first group and one in the second group would wait for each other until synchronous and go to next section. In the same way, it would happen with G04.1 P1.

2、The related M code:

M_code	The specification
M03	The first main axis rotates positively
M04	The first main axis rotates negatively
M05	The first main axis stops
M63	The second main axis rotates positively
M64	The second main axis rotates negatively
M65	The second main axis stops
M70	Assign the first main axis to be the first group of the main axis
M71	Assign the second main axis to be the second group of the main axis

3、A matter needing attention when compiling program:

1. The first group of the program must start with $1 and the second one must do that with $2.
2. The quantites of G04.1 P_ must be the same in the first and second group and the number after P need using in order from small to big.
3. Put M30 or M02 in the first group when promgram ends and M99 must be set in the end of the second group absolutely.
4. With repeating to process several workpieces automatically put M99 in the end of the first group program. But notice that for make the first and second groups to process synchronously, we must compile the same G04.1 P_ code before the M99 of the first and second group.

```
$1
----------
----------
G04.1 P20
M99

$2              The same
-----------
-----------
G04.1 P20
M99
```

5. With the axis set belong to the second group we only can start G code in the second group. With the one set belong to the first group if we start G code in the second group Iit can not work.
6. The first and second group all can support M code、 S code and T code. Therefore we can run M code、 S code and T code in the first and second group simultaneously.

4、Compiling programs:

Start a new file and imitate the example below to compile processing programs

The first group

```
$1
G00 X50.        //move X axis in the first group
G04.1 P1
      Z100.     // move Z axis in the first group
------------
------------
G04 P30
M30
```

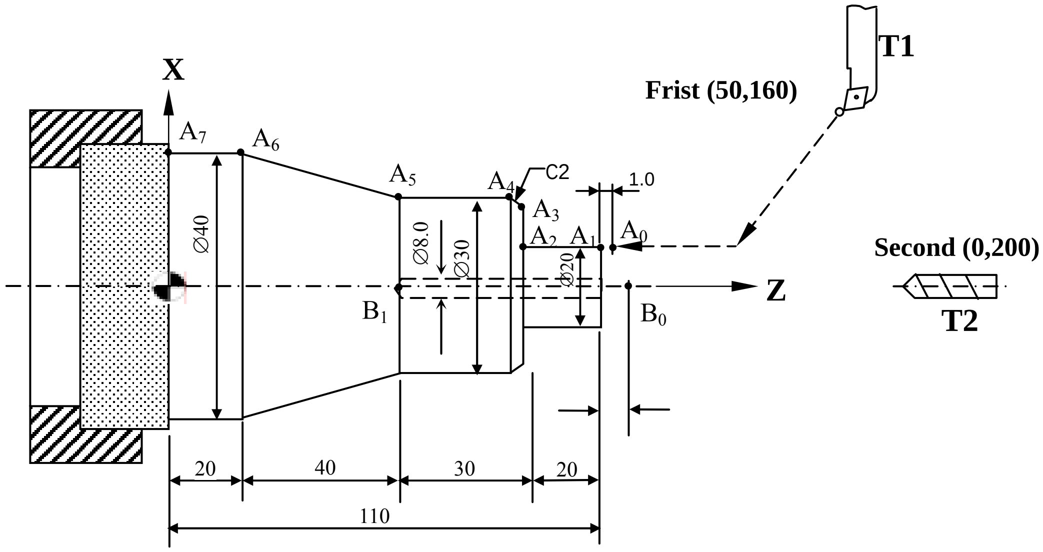
Simultaneous

The second group

```
$2
G00 X250        // move X axis in the second group
      Z500..    // move Z axis in the second group
G04.1 P1
------------
------------
------------
------------
G04.1 P30
M99
```

Simultaneous

5、Examples for processing program:




```
$1                          //the first group
G92 X50.0 Z160.0 S10000；//set origin, the highest speed 10000 rpm
T01；                       //use the No.1 knife
G96 S130 M03；   //face speed130m/min, main axis rotates positively
M08；                       //turn on cutting liquid
G04.1 P1;
G00 X20.0 Z111.0；         //positon to A0 rapidly
G01 Z90.0 F0.6；           //linear cutting A0→A2
    X26.0；                //A2→A3
    X30.0 Z88.0；          //A3→A4
    Z60.0；                //A4→A5
G04.1 P2;
    X40.0 Z20.0；          //A5→A6
    Z0.0；                 //A6→A7
G00 X50.0；                //back knife rapidly
    Z160.0；               //return to origin
G04.1 P3
M05 M09；                  //stop the main axis, turn off cutting liquid
G04.1 P4;
M30；                      //end program

$2                         //the second group
G04.1 P1;
T02;                       // use the No.2 knife
G04.1 P2;
G00 X0 Z120.;              //postion to B0 rapidly
G01 Z60. F0.5;             //move knife B0→B1
G00 Z120.                  //back knife B1→B0
G04.1 P3;
G00 Z200.                  //back the knife
G04.1 P4;
M99;
```