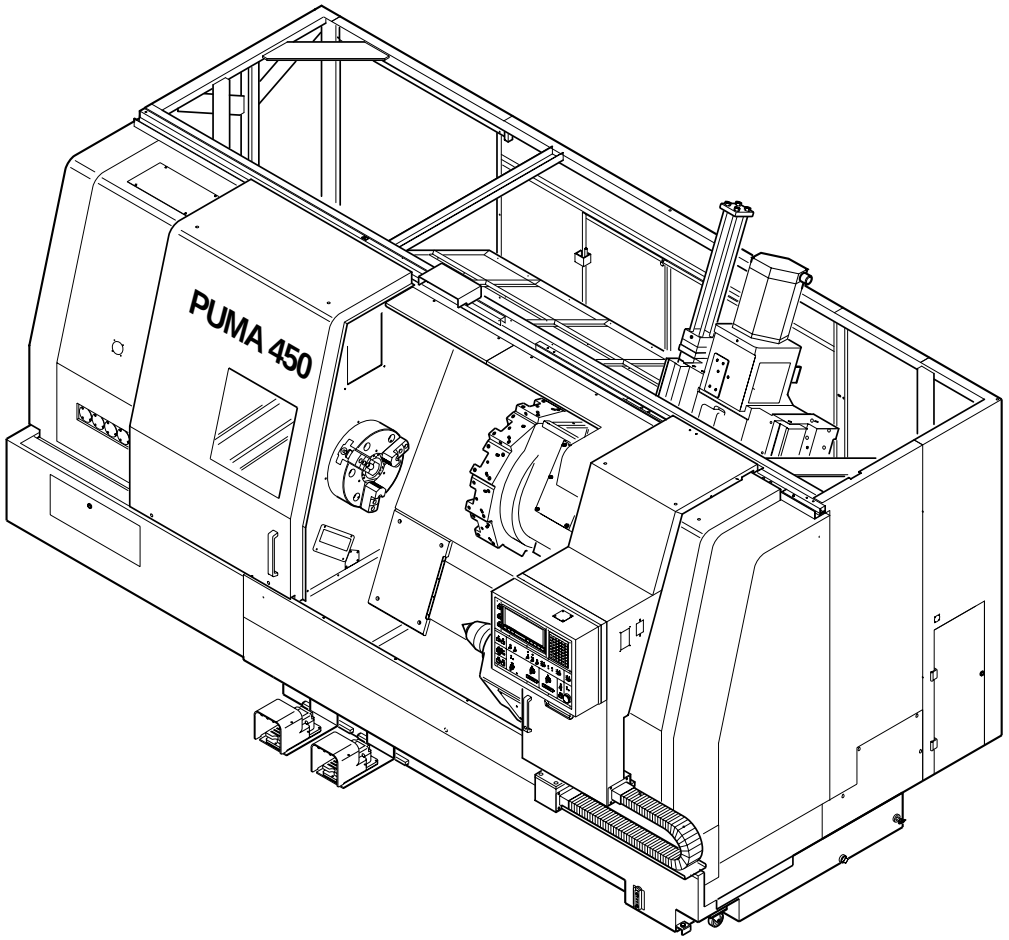
CNC
PROGRAM MANUAL



DAEWOO
DAEWOO HEAVY INDUSTRIES &
MACHINERY LTD.

Forward

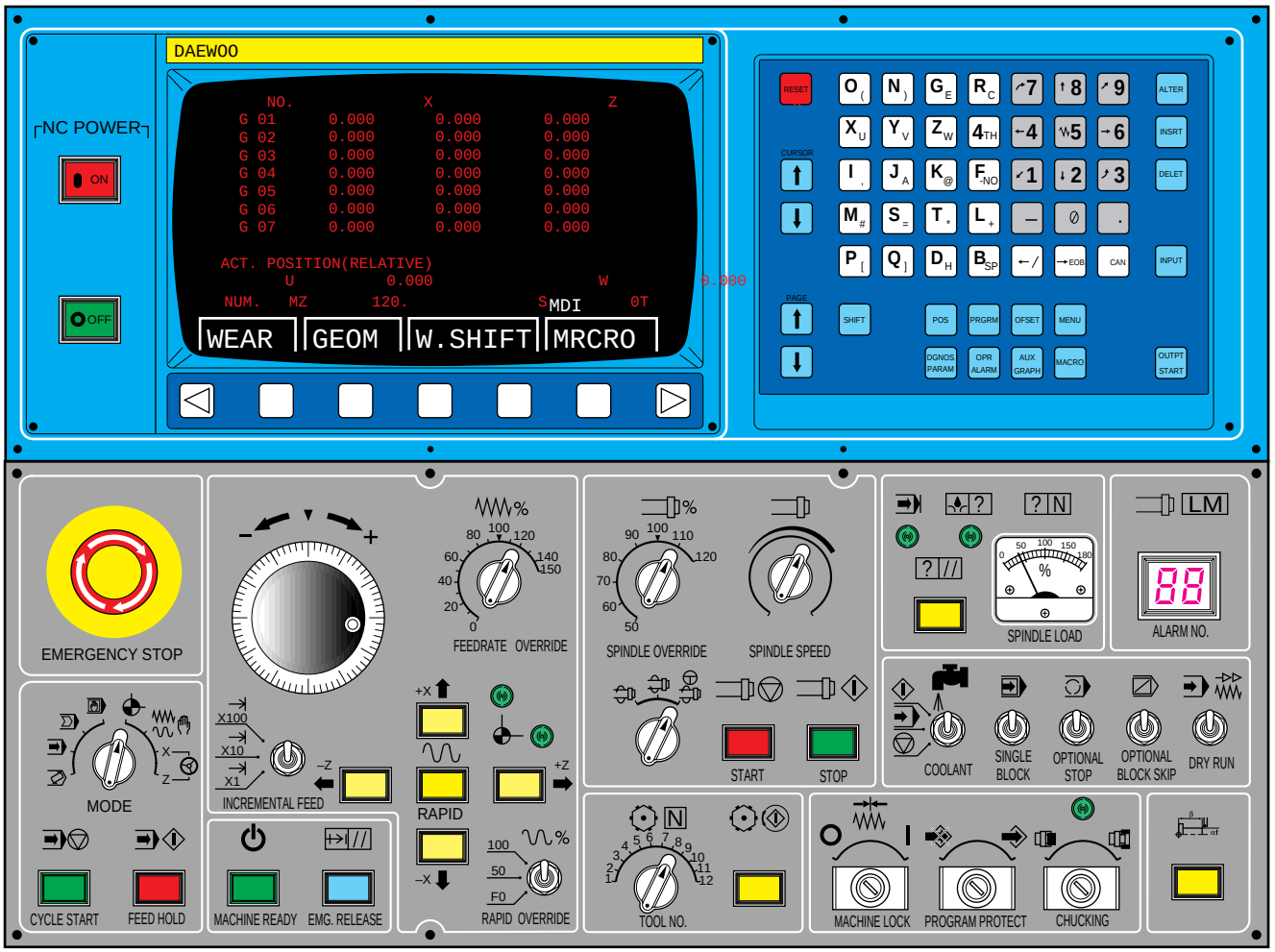
Thank you very much for participating in our education.

DAEWOO constantly makes an effort to research and develop to satisfy the
requirements of customers positively.
DAEWOO does its utmost to accept and practice the Quality Confirmation of DAEWOO and Customers' requirements through the Dealer-net-work of about 350 as practicing the World Quality Management.
DAEWOO provides with the technical data and support the technical coaching, therefore, if you contact us when you need of them , we will immediately help you.

We will do our best during your education period.

Thank you.

O-T


DAEWOO
NC POWER
ON
OFF
NO. X Z
G 01 0.000 0.000 0.000
G 02 0.000 0.000 0.000
G 03 0.000 0.000 0.000
G 04 0.000 0.000 0.000
G 05 0.000 0.000 0.000
G 06 0.000 0.000 0.000
G 07 0.000 0.000 0.000
ACT. POSITION(RELATIVE)
U 0.000 W
NUM. MZ 120. S MDI 0T
WEAR GEOM W.SHIFT MRCRO
EMERGENCY STOP
FEEDRATE OVERRIDE
SPINDLE OVERRIDE
SPINDLE SPEED
SPINDLE LOAD
ALARM NO.
MODE
INCREMENTAL FEED
RAPID
RAPID OVERRIDE
COOLANT
SINGLE BLOCK
OPTIONAL STOP
OPTIONAL BLOCK SKIP
DRY RUN
START
STOP
CYCLE START
FEED HOLD
MACHINE READY
EMG. RELEASE
TOOL NO.
MACHINE LOCK
PROGRAM PROTECT
CHUCKING

G-FUNCTION

STANDARD G CODE	SPECIAL G CODE	GROUP	FUNCTION
#G00 G01 G02 G03	G00 G01 G02 G03	01	Positioning (Rapid feed) Straight interpolation Circular interpolation (CW) Circular interpolation (CCW)
G04	G04	00	Dwell
G20 #G21	G20 G21	06	Data input (inch) Data input (mm)
#G22 G23	G22 G23	04	Stored distance limit is effective (Spindle interference check ON) Stored distance limit is ineffective (Spindle interference check OFF)
G27 G28 G29 G30	G27 G28 G29 G30	00	Machine reference return check Automatic reference return Return from reference Tte 2nd rererence return
#G32	G33	01	Thread process
G40 G41 G42	G40 G41 G42	07	Cancel of compensation Compensation of the left Compensation of right
G50 G70 G71 G72 G73 G74 G75 G76	G92 G70 G71 G72 G73 G74 G75 G76	00	Creation of virtual coordinate/Setting the rotating time of principal spindle Compound repeat cycle(Finishing cycle) Compound repeat cycle(Stock removal in turning) Compound repeat cycle(Stock removal in facing) Compound repeat cycle(Pattern repeating cycle) Compound repeat cycle(Peck drilling in Z direction) Compound repeat cycle(Grooving in X direction) Compound repeat cycle(Thread process cycle)
G90 G92 G94	G77 G78 G79	01	Fixed cycle(Process cycle in turning) Fixed cycle(Thread process cycle) Fixed cycle(Facing process cycle)
G96 #G97	G96 #G97	02	Control the circumference speed uniformly(mm/min) Cancel the uniform control of circumference speed. Designate r.p.m
G98 #G99	G94 #G95	05	Designate the feedrate per minute(mm/min) Designate the feedrate per the rotation of principal spindle(mm/rev.)
- -	G90 G91	03	Absolute programming Incremental programming

Note) 1. # mark instruction is he modal indication of initial condition which is immediately available when power is supplied.

2. In general, the standard G code is used in lathe, and it is possible to select the special G code according to setting of parameters.

NC LATHE M-CODE LIST

M-CODE	DESCRIPTION	REMARK	M-CODE	DESCRIPTION	REMARK
M00	PROGRAM STOP		M39	STEADY REST 1 UNCLAMP	OPTION
M01	OPTIONAL STOP		M40	GEAR CHANGE NETURAL	
M02	PROGRAM END		M41	GEAR CHANGE LOW	
M03	MAIN-SPINDLE FORWARD		M42	GEAR CHANGE MIDDLE	
M04	MAIN-SPINDLE REVERSE		M43	GEAR CHANGE HIGH	
M05	MAIN-SPINDLE STOP		M46	PTS BODY UNCL & TRACT-BAR ADV.	OPTION
M07	HIGH PRESSURE COOLANT ON	OPTION	M47	PTS BODY CL & TRACT-BAR RET.	OPTION
M08	COOLANT ON		M50	BAR FEEDER COMMAND 1	OPTION
M09	COOLANT OFF		M51	BAR FEEDER COMMAND 2	OPTION
M10	PARTS CATCHER ADVANCE	OPTION	M52	SPLASH GUARD DOOR OPEN	OPTION
M11	PARTS CATCHER RETRACT	OPTION	M53	SPLASH GUARD DOOR CLOSE	OPTION
M13	TURRET AIR BLOW	OPTION	M54	PARTS COUNT	OPTION
M14	MAIN-SPINDLE AIR BLOW	OPTION	M58	STEADY REST 2 CLAMP	OPTION
M15	AIR BLOW OFF	OPTION	M59	STEADY REST 2 UNCLAMP	OPTION
M17	MACHINE LOCK ACT	(ONLY) MDI	M61	SWITCHING LOW SPEED (N.J)	α P60
M18	MACHINE LOCK CANCEL	(ONLY) MDI	M62	SWITCHING HIGH SPEED (N.J)	α P60
M19	MAIN-SPINDLE ORIENTAION	OPTION	M63	MAIN-SPDL CW & COOLANT ON	
M24	CHIP CONVEYOR RUN	OPTION	M64	MAIN-SPDL CCW & COOLANT OFF	
M25	CHIP CONVEYOR STOP	OPTION	M65	MAIN-SPDL & COOLANT OFF	
M30	PROGRAM END & REWIND		M66	DUAL CHUCKING LOW CLAMP	OPTION
M31	INTERLOCK BY-PASS(SPDL &T/S)		M67	DUAL CHUCK HIGH CLAMP	OPTION
M32	INTERLOCK BY-PASS(SPDL &S/R)	3 AXIS	M68	MAIN-CHUCK CLAMP	
M33	REV.-TOOL-SPINDLE FORWARD	3 AXIS	M69	MAIN-CHUCK UNCLAMP	
M34	REV.-TOOL-SPINDLE REVERSE		M70	DUAL TAILSTOCK LOW ADVANCE	OPTION
M35	REV.-TOOL-SPINDLE STOP		M74	ERROR DETECT ON	
M38		OPTION	M75	ERR0R DETECT OFF	

NC LATHE M-CODE LIST

M-CODE	DESCRIPTION	REMARK	M-CODE	DESCRIPTION	REMARK
M76	CLAMFERING ON		M131	INTERLOCK BY-PASS (SUB-SPDL)	
M77	CLAMFERING OFF		M163	SUB-SPDL CW & COOLANT ON	
M78	TAILSTOCK QUILL ADVANCE		M164	SUB-SPDL CCW & COOLANT OFF	
M79	TAILSTOCK QUILL RETRACT		M165	SUB-SPDL & COOLANT STOP	
M80	Q-SETTER SWING ARM DOWN	OPTION	M168	SUB-CHUCK CLAMP	
M81	Q-SETTER SWING ARM UP	OPTION	M169	SUB-CHUCK UNCLAMP	
M84	TURRET CW ROTATION		M203	FORWARD SYNCHRONOUS COM.	
M85	TURRET CCW ROTATION		M204	REVERSE SYNCHRONOUS COM.	
M86	TORQUE SKIP ACT	B AXIS	M205	SYNCHRONOUS STOP	
M87	TORQUE SKIP CANCEL	B AXIS	M206	SPINDLE ROTATION RELEASE	
M88	SPINDLE LOW CLAMP				
M89	SPINDLE HIGH CLAMP				
M90	SPINDLE UNCLAMP				
M91	EXTERNAL M91 COMMAND	3 AXIS			
M92	EXTERNAL M92 COMMAND	3 AXIS			
M93	EXTERNAL M93 COMMAND				
M94	EXTERNAL M94 COMMAND	OPTION			
M98	SUB-PROGRAM CALL	OPTION			
M99	END OF SUB-PROGRAM	OPTION			
M103	SUB-SPINDLE FORWARD				
M104	SUB-SPINDLE REVERSE				
M105	SUB-SPINDLE STOP				
M110	PARTS CATCHER ADVANCE(SUB)	OPTION			
M111	PARTS CATCHER RETRACT(SUB)	OPTION			
M114	SUB-SPINDLE AIR BLOW	OPTION			
M119	SUB-SPINDLE ORIENTATION	OPTION			

Note) 1. M00 : For this command, main spindle stop, cutting oil, motor stop, tape reading stop are carriedout.

M01 : While this function is the same as M00, it is effective when the optional stop switch of console is ON.
This command shall be overrided if the optional stop switch is OFF.

M02 : Indicates the end of main program.

M30 : This is the same as M02 and it returns to the starting position of the programme when the memory and the tape are running.

2. M code should not be programmed in the command paragraph containing S code or T code. It is favorable for M code to programe in a command paragraph independently.

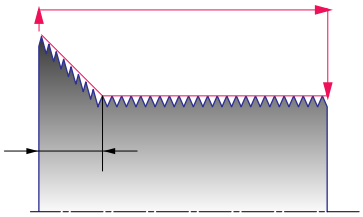
3. The edges of processed material become round due to the effect of characteristics of AC servo motor. To avoid it, M74 and M75 functions are used.

When command of M75
(Error detection is OFF)

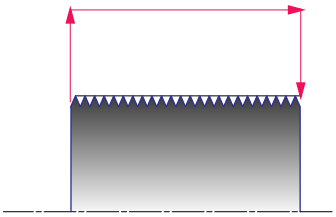
When command of M74
(Error detection is ON)

4. M76, M77

These codes are effective when thread process is programmed by G92, and they are used for ON and OFF of thread beveling. Thread chamferingis set as much as one pitch by setting of parameters and it is possible to set double.



(Thread chamferingON)



(Thread chamferingOFF)

Function	Address	Meaning of address
Program number	O(EIA)/(ISO)	Program number
Block sequence number	N	Sequence number
Preparatory function	G	Sercifies a motion mode (Linear, arc, etc)
Dimension word	X, Z	Command of moving position(absolute type) of each axis
	U, W	Instruction of moving distance and direction(incremental type)
	I, K	Ingredient of each axis and chamfering volume of circulat center
	R	Radius of circle, corner R, edge R
Feed function	F, E	Designation of feedrate and thread lead
Auxiliary function	M	Command of ON/OFF for operating parts of machine
Spindle speed function	S	Designation of speed of main spindle or rotation time of main spindle
Function (Tool)	T	Designation of tool number and tool compensation number
Dwell	P, U, X	Designation of dwell time
Dewignation of program number	P	Designation of calling number of auxiliary program
Designation of sequence No	P, Q	Callling of compound repeat cycle, end number
Number of repetitions	L	Repeat time of auxiliary program
Parameters	A, D, I, K	Parameter at fixed cycle

One block is composed as follows

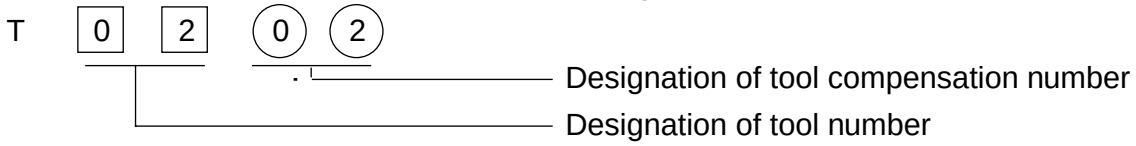
One block

N	G	X Y	F	S	T	M	:
Sequence Auxiliary No.	Preparation function	Dimension word	Feed function	Spindle speed function	Tool function	Function auxiliary	EOB

Meaning of Address

T function is used for designation of tool numbers and tool compensation.

T function is a tool selection code made of **4** digits.



Example) If it is designated as(T 0 2 ⓪ ②)

0 2 calls the tool number and calls the tool compensation value of number , and the tool is compensation as much as momoried volume in the storage.

The cancel of tool compensation is commanded as T □ □ ⓪ ⓪

If you want to call the next tool and compensation, you should cancel the tool compensation. For convenient operation, it is recommended to used the same number of tool and compensation.

It is not allowed to use the same tool compensation number for 2 different tools.

Minimum compensation value : + 0.001mm

Maximum compensation value : + 999.999mm

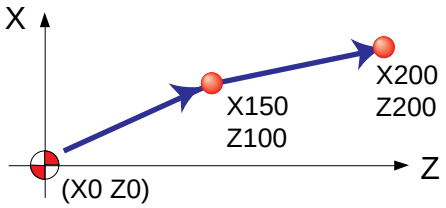
Tool compensation of X spindle is designated as diameter value.

G00

G00(Positioning)

Each axes moves as much as commanded data in rapid feedrate.

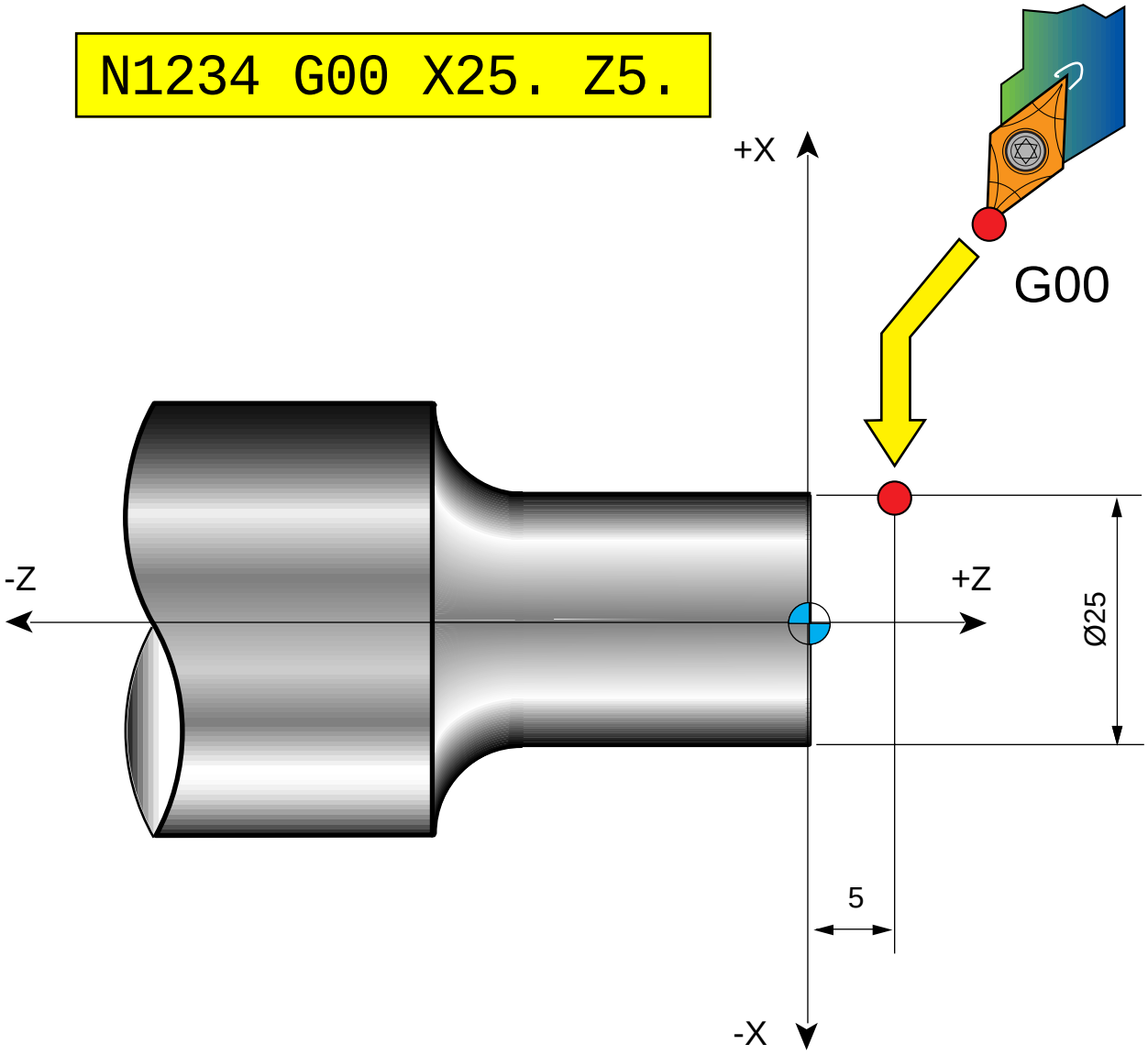
```
G00    X(U)    Z(W);
```



G00 X150.0 Z100.0
X200.0 Z200.0

G00 U150.0 W100.0
U50.0 W100.0

```
N1234 G00 X25. Z5.
```

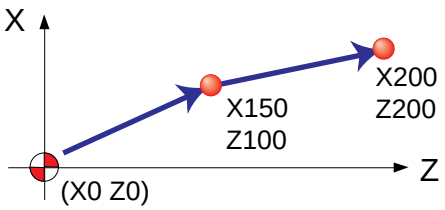


G01

G01(Linear interpolation)

Each axes moves straigrtly as much as commanded data in commanded rate.

G01 X(U)_ Z(W)_ F_



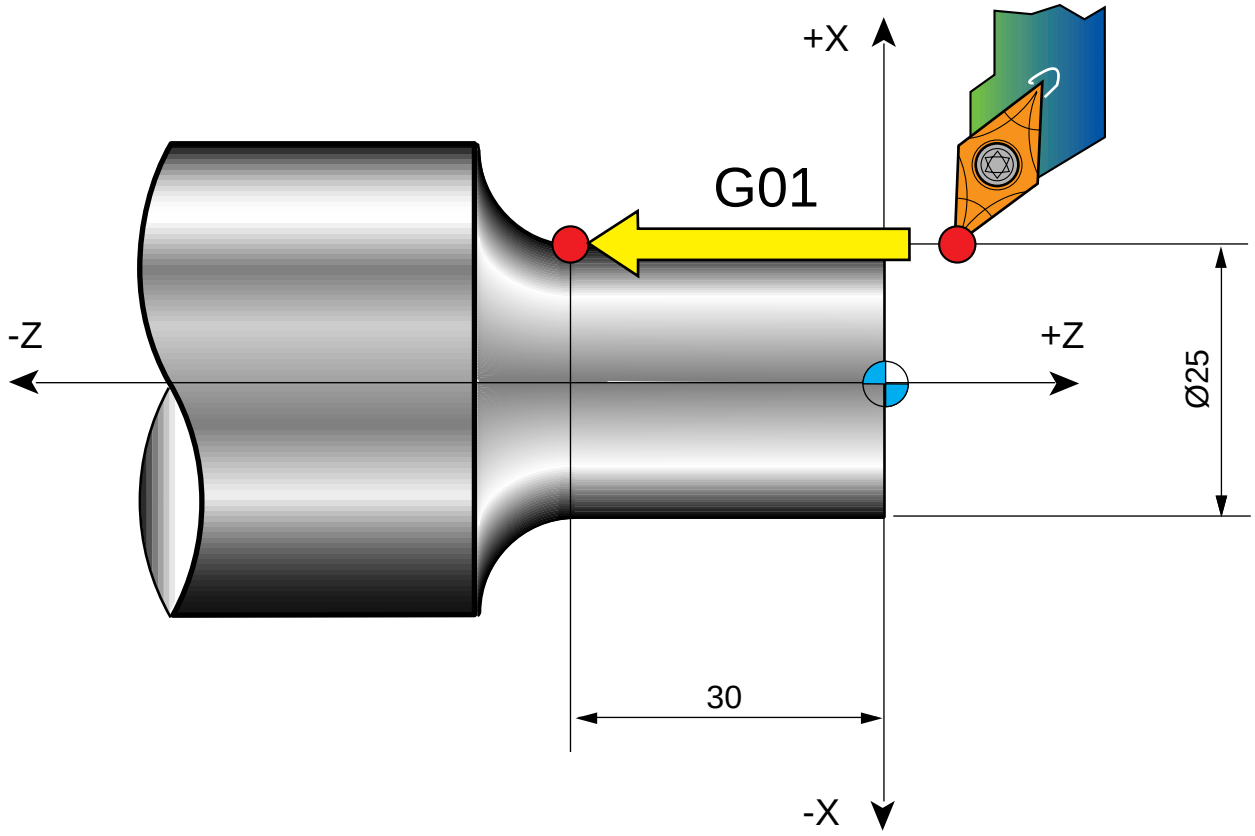
G01 X150.0 Z100.0 F0.2 :

X200.0 Z200.0 :

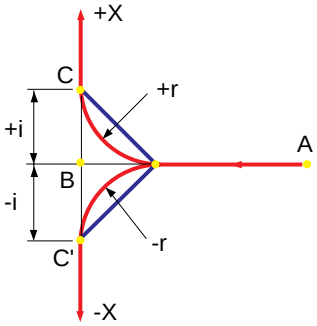
G01 U150.0 W100.0 F0.2 :

U50.0 W100.0 :

```
N1234 G01 X25. Z-30. F0.2
```



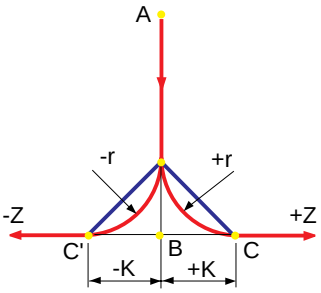
AUTO CHAMFERING “C” AND CORNER “R” (Option)



Command path Z→X : A : Start point of instuction

G01 Z(w) B C (¡ i) : B : End point of instruction

G01 Z(w) B C (¡ r) :CC’ : Running point of command



Command path X→Z :

G01 X(u) B C (¡ k)

G01 X(u) B R (¡ r)

Note) (1) After instructing from G01 to one axis, the next command paragraph should be fed in vertical direction.

(2) If the next command paragraph is incremental type, designate the incremental volume baed on B point.

(3) In following cases, errors occur. (G01 Mode)

– When instruction one of I, K, R and X and Z at the same time.

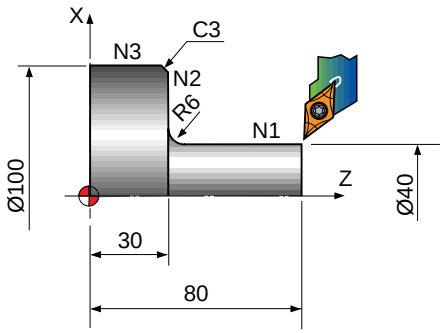
– When instructing two of I, K, R in the same block.

– When instructing Xand I or Z and K.

– When the moving distance is less than the next command are not right angled.

(4) During the operation of single command paragraph, the operation at C point stops.

Example)



N1 G01 Z30.0 R6.0 F0.2 :

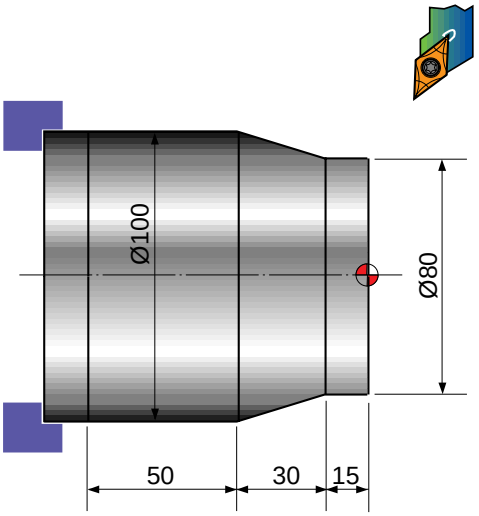
N2 X100.0 K-3.0 :

N3 Z0 :

(N2 X100.0 C3.0 :)Normal

G01 PROGRAM

Example1)



O0001 :

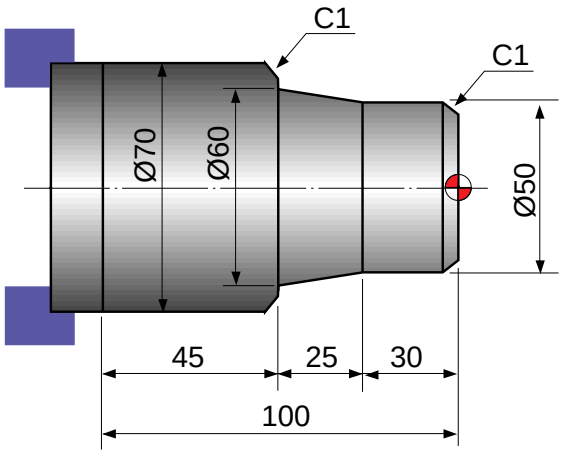
N10 G50 S1500 T0100 M42 :
G96 S180 M03 :
G00 X100.5 Z5.0 T0101 M08 :
G01 Z-95.0 F0.25 :
G00 U2.0 Z0.5 :
G01 X-1.6 F0.2 :
G00 X95.0 W1.0 :
G01 Z-37.3 F0.25 :
X100.0 Z-45.5 :
G00 Z1.0 :
X90.0 :
G01 Z-29.8 :
X95.0 Z-37.3 :
G00 Z1.0 :
X85.0 :
G01 Z-22.3 :
X90.0 Z-29.8 :
G00 Z1.0 :
X80.5 :
G01 Z-15.55 :
X85.0 Z-22.3 :
G00 X200.0 Z200.0 M09 T0100 :
M01 :

N20 G50 S2000 T0300 :
G96 S200 M03 :
G00 X85.0 Z5.0 T0303 M08 :
Z0 :
G01 X-1.6 F0.2 :
G00 X80.0 Z3.0 :
G42 Z1.0 :
G01 Z-15.0 F0.18 :
X100.0 Z-45.0 :
Z-95.0 :
G40 U2.0 W1.0
G00 X200.0 Z200.0 M09 T0300 :
M30 :

G50 : Setting the rotating time of max. speed of main spindle
G96 : Constant surface speed control command
G40 : Compensation cancel
G42 : Right hand compensation

G01 PROGRAM

Example2)



O0002 :

N10 G50 S2000 T0100 :

G96 S180 M03 :

G00 X70.5 Z5.0 T0101 M08 :

G01 Z-100.0 F0.25 :

G00 U2.0 Z0.5 :

G01 X-1.6 F0.23 :

G00 X65.0 W1.0 :

G01 Z-54.5 F0.25 :

G00 U2.0 Z1.0 :

X60.0 :

G01 Z-54.5 :

G00 U2.0 Z1.0 :

X55.0 :

G01 Z-30.0 :

X60.0 Z-54.5 :

G00 U2.0 Z1.0 :

X50.5 :

G01 Z-30.0 :

X60.3 Z-54.7 :

X72.0

G00 X150.0 Z200.0 T0100 :

M01 :

N20 G50 S2300 T0300 :

G96 S200 M03 :

G00 X55.0 Z5.0 T0303 M08 :

Z0 :

G01 X-1.6 F0.2 :

G00 X46.0 Z3.0 :

G42 Z1.0 :

G01 X50.0 Z-1.0 F0.15 :

Z-30.0 :

X60.0 Z-55.0 :

X68.0 :

X70.0 W-1.0 :

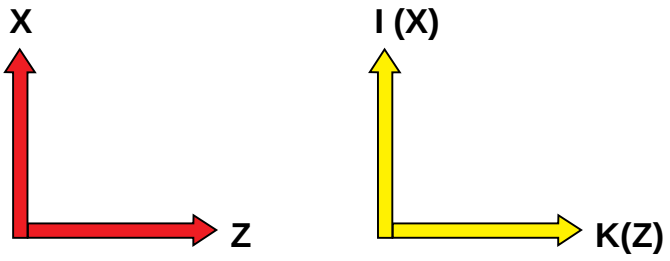
Z-100.0 :

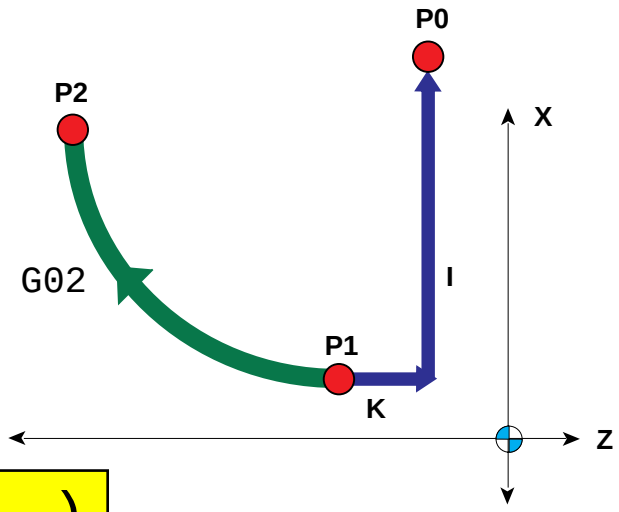
G40 U2.0 W1.0

G00 X150.0 Z200.0 M09 T0300 :

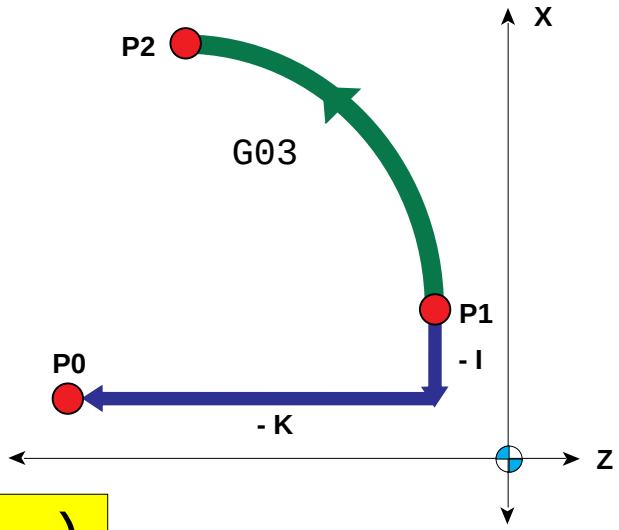
M30 :

G02 G03





N1234 G02 X.. Z.. (R..)



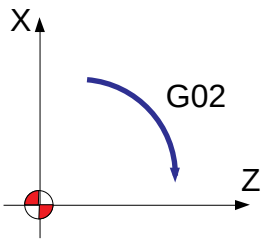
N1234 G03 X.. Z.. (R..)

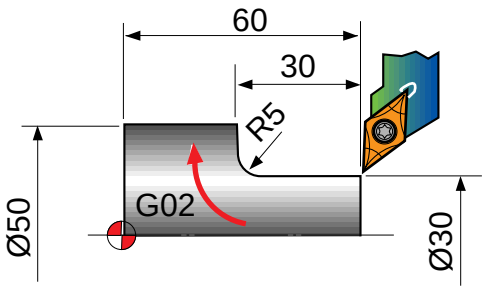
G02, G03(Circular interpolation)

Each axis interpolates circularly to the commanded coordinate in instructed speed.

Conditions		Instruction	Meaning	
			Right hand coodinate	Left hand coodinate
1	Rotation direction	G02	CW	CCW
		G03	CCW	CW
2	Location of end point	X,Z	Location X,Z of commanded point from coordinate	
	Distance to the end point	U,W	Distance from start point to commanded point	
3	Distance between start point and the center point	I,K	Distance from start point to the center of and arc with sign, radius value (I always designates the radius)	
	Arc radius with no sign radius of circumference	R	Radius of circumference	

G02 X(u) Z(w) R_ F_ :





G01 X30.0 Z60.0 F0.3 :

Z35.0 :

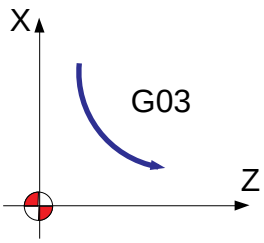
G02 X40.0 Z30.0 I5.0 :

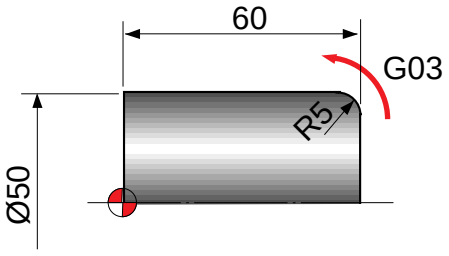
(G02 U10.0 W-5.0 I5.0)

G01 X50.0 :

Z0 :

G03 X(u) Z(w) R_ F_ :





G01 X40.0 Z60.0 F0.3 :

G03 X50.0 Z55.0 K-5.0 :

Note) (1) If I or K is 0 it is omissible.

(2) G02 I_: Make a round of circle.

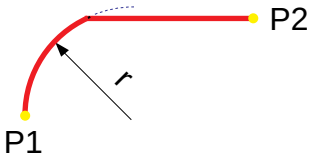
(3) It is recommended to use R as + value, and designates the circumferences less than 180.

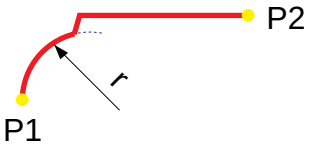
G03 R_: No moving

(4) When designating R which is less than the half of moving distance, override R and make half circle.

(5) When designating I, K and R at the same time, R is effective.

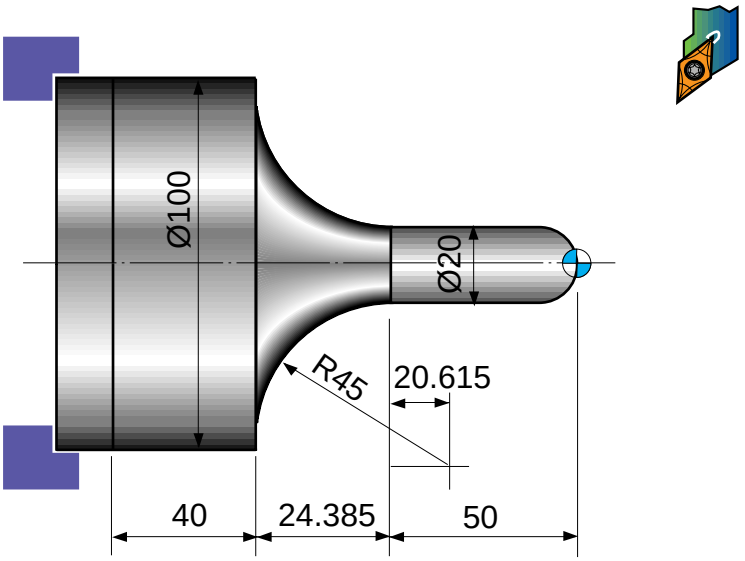
(6) When the moving end point is not on the circumference as a result of wrong designation of and K :





G03) PROGRAM
G02

Example 1)



N10 :

N20 G50 S2000 T0300 :

G96 S200 M03 :

G00 X0 Z3.0 T0303 M08 :

G42 G01 Z0 F0.2 :

G03 X20.0 Z-10.0 R10.0 :

G01 Z-50.0 :

G02 X100.0 Z-74.385 I40.0 K20.615 : (G02 X100.0 Z-74.385 R45.0)

G01 Z-125.0 :

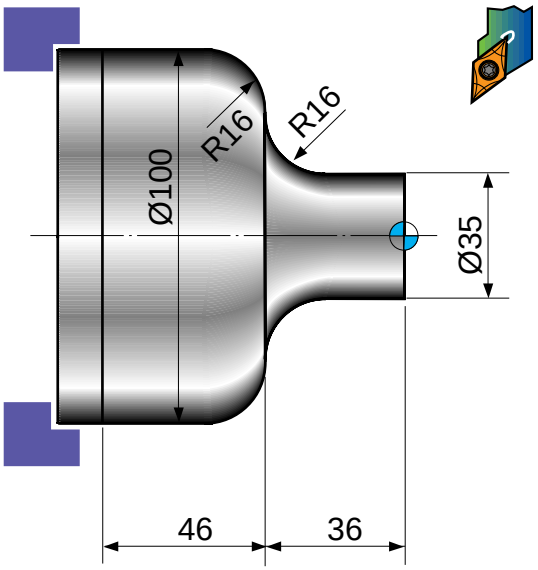
G40 U2.0 W1.0

G00 X200.0 Z200.0 M09 T0300 :

M30 :

G02) PROGRAM
G03

Example 2)



N10 :

N20 G50 S2000 T0300 :

G96 S200 M03 :

G42 G00 X35.0 Z5.0 T0303 M08 :

G01 Z-20.0 F0.2 :

G02 X67.0 Z-36.0 R16.0 : (G02 X67.0 Z-36.0 I16.0 K0)

G01 X68.0 :

G03 X100.0 Z-52.0 R16.0 : (G02 X100.0 Z-52.0 I0 K-16.0)

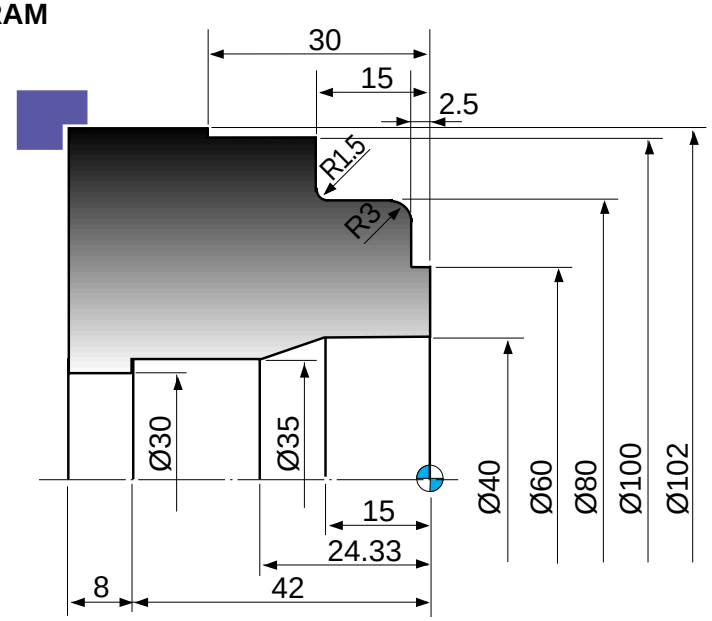
G01 Z-82.0 :

G40 G00 X200.0 Z200.0 M09 T0300 :

M30 :

When I and K instruction, if the data value is “0” it can be omitted.

G01
G02) **PROGRAM**
G03



O0000 :

N10 (ø30 DRILL)

G50 T0200 :

G97 S250 M03 :

G00 X0 Z5.0 T0202 M08 :

G01 Z-5.0 F0.07 :

W1.0 :

Z-40.0 F0.25 :

G00 Z5.0 :

Z-39.0 :

G01 Z-60.0 :

G00 Z10.0 :

X200.0 Z200.0 T0200 :

M01 :

N20 (Outside diameter stock removal)

G50 S1500 T0100 :

G96 S180 M03 :

G00 X94.0 Z5.0 T0101 M08 :

G01 Z-14.8 F0.27 :

G00 U2.0 Z0.5 :

G01 X28.0 F0.23 :

G00 X87.0 W1.0 :

G01 Z-14.8 F0.27 :

G00 U2.0 Z1.0 :

X80.5 :

G01 Z-14.1 :

G02 X81.9 Z-14.8 R0.7 :

G00 X100.5 W1.0

G01 Z-29.8

G00 U2.0 Z-1.0 :

G01 X60.5 F0.23 :

G00 X82.0 W1.0 :

Z-2.4 :

G01 X60.5 :

X72.9 :

G03 X80.5 Z-6.2 R3.8 :

G00 U2.0 Z5.0 :

X200.0 Z200.0 T0100 :

M01 :

```
N30 (Inside diameter stock removal)
    G50 S1500 T0400 :
    G96 S180 M03 :
    G00 X34.5 Z3.0 T0404 M08 :
    G01 Z-41.8 F0.27 :
    G00 U-0.5 Z1.0 :
        X39.5 :
    G01 Z-15.0 :
        X34.5 Z-24.3 :
    G00 Z10.0 :
        X200.0 Z200.0 T0400  :
    M01 :
N40 (Out diameter finishing)
    G50 S1800 T0500 :
    G96 S200 M03 :
    G00 X63.0 Z5.0 T0505 M08 :
        Z0 :
    G01 X38.0 F0.2 :
    G00 X60.0 Z3.0 :
     G42  Z1.0 :
    G01 Z-2.5 F0.2 :
        X74.0 :
    G03 X80.0 Z-5.5 R3.0 :
    G01 Z-13.5 :
    G02 X83.0 Z-15.0 R1.5 :
    G01 X100.0 :
        Z-30.0 :
        X103.0 :
    G40 G00 U2.0 W1.0 :
    G00 Z10.0 :
        X200.0 Z200.0 T0500  :
    M01 :
```

```
N50 (Inside diameter finishing)
    G50 S1800 T0600 :
    G96 S200 M03 :
    G00 X40.0 Z5.0 T0606 M08 :
    G41 Z1.0 :
    G01 Z-15.0 F0.2 :
        X35.0 Z-24.33 :
        Z-42.0 :
        X29.0 :
    G40 G00 Z10.0 :
        X200.0 Z200.0 T0600 M09 :
    M30 :
```

1G04 (Dwell)

After passing as much time as commanded by X(u) or P code in the same block, carry out the next block.
In case of 10 seconds' dwell
G04 X10.0 : (G04 X10000 :)
G04 U10.0 : (G04 U10000 :)
G04 P10000.0 : (G04 P1000 :)

Automatic reference return

Reference means certain point fixed in the machine, and coordinate value of reference is set in NC parameter.

	OT-C/F	FS16/18T
Parameter NO	N708(X) N709(Z)	N1240(X, Z)

1) G27(Reference return check)

Position is decided through rapid feed to the position of value set in NC PARAMETER by command.

Example) When PARAMETER N708(X) is 330000
N709(Z) is 529000

G00 X100.0 Z100.0 :
G27 X330.0 Z529.0 :

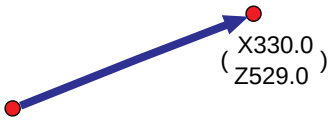
If arrived position is the reference, reference Lamp is ON.

Note) When instructing G27, you should cancel the OFFSET volume

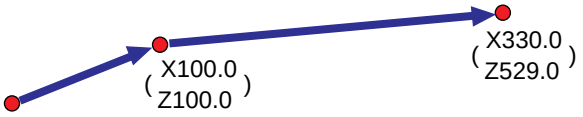
2) G28(Reference automatic return)

By command, commanded axis automatically returns to the reference.
G28 X(u) Z(w) :
Example) When PARAMETER N708(X) is 330000
N709(Z) is 529000

G28 U0 W0 :



G27 X100.0 Z100.0



Action of G28 block presents that the commanded axis goes via the center in rapid feedrate and returns to the reference.

Note) When instructing G28 block, tool, tool compensation, tool location offset should be canceled principlly.

3) G29(Automatic return in reference)

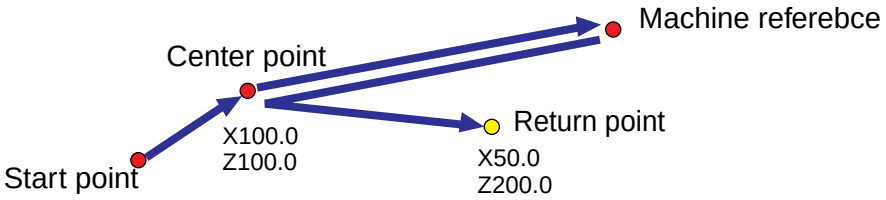
Commanded spindle goes via the remoried center point and decides the position as commanded point.

G29 X(u) Z(w) :

∴Generally, it is used right after G28 or G30 command.

G28 X100.0 Z100.0 :

G29 X50.0 Z200.0 :



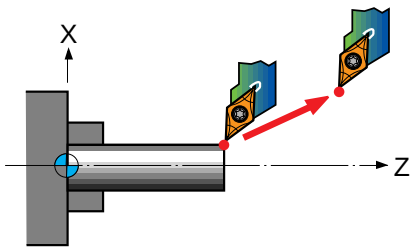
4) G30(The 2nd reference return)

Commanded spindle automatically returns to the 2nd reference
(coordinate point set in parameter)

G30 X(u) Z(w)) :

∴You should input appropriate distance between works and tool exchangeposition in the relative parameter.

PARAMETER NO N735(X) = 200000 FS16/18T
N736(Z) = 300000 N1241(X,Z)



The 2nd reference
X200.0 G30 U0 W0 :
Z300.0

Reference) Generally, the 2nd reference is used for the start point of program.

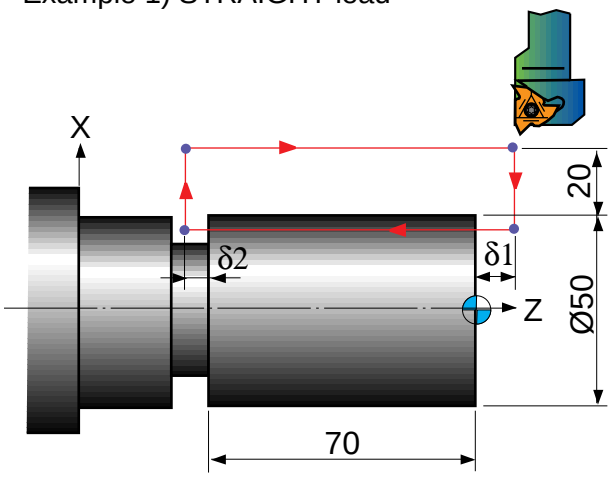
G32(THREAD CYCLE)

According to G32 command, straight thread and taper thread of certain lead are cut.

G32 Z(w) F : (G32 is applied to only single block)

X(u) F :

Example 1) STRAIGHT lead



Lead of screw : 3mm

δ1 : 5mm

δ2 : 1.5mm

Depth of cut : 1mm(2cut two times)

(ABSOLUTE)

G50 T0100 :

G97 S800 M03 :

G00 X90.0 Z5.0 T0101 M8 :

X48.0 :

G32 Z-71.5 F3.0 :

G00 X90.0 :

Z5.0 :

X46.0 :

G32 Z-71.5 :

G00 X90.0 :

Z5.0

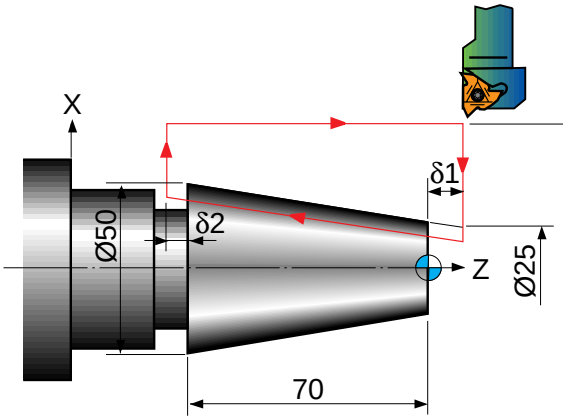
X150.0 Z150.0 T0100 :

M30 :

∗ When processing G32 thread, feed(pitch) is modal.

Example 1) STRAIGHT lead

G32 X(u) Z(w) F : Because it is taper, it is applied to both axis at the same time.



Lead of screw : 3mm

δ1 : 5mm

δ2 : 1.5mm

Depth of cut : 1mm(2cut two times)

(ABSOLUTE)

G50 S800 T0100 :

G97 S800 M03 :

G00 X90.0 Z5.0 T0101 :

X22.026 :

G32 X49.562 Z-71.5 F3.0 :

G00 X90.0 :

Z5.0 :

X21.052 :

G32 X48.588 Z-71.5 :

G00 X90.0 :

Z5.0 :

X150.0 Z150.0 T0100 :

M30 :

(INCREMENTAL)

G50 S800 T0100 :

G97 S800 M03 :

G00 X90.0 Z5.0 T0101 :

U-67.974 :

G32 U27.321 W-76.5 F3.0 :

G00 U40.438 :

W76.5 :

U-68.948 :

G32 U27.321 W-76.5 :

G00 X90.0 :

W76.5 :

X150.0 Z150.0 T0100 :

M30 :

Reference)

Values of incomplete thread δ1 and δ2.

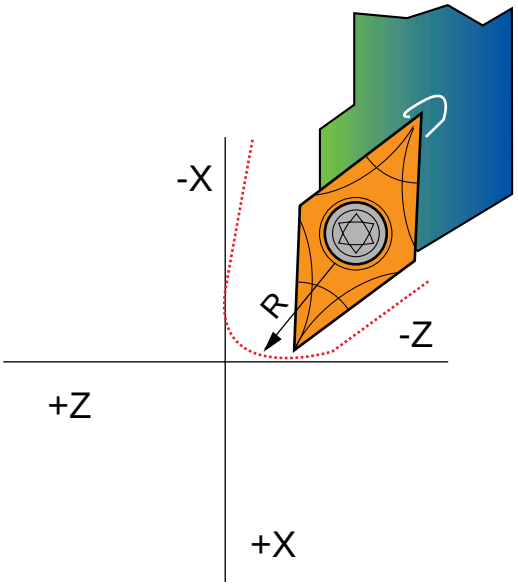
$$\delta 1 = \frac{3.6 \times L \times n}{1800}$$

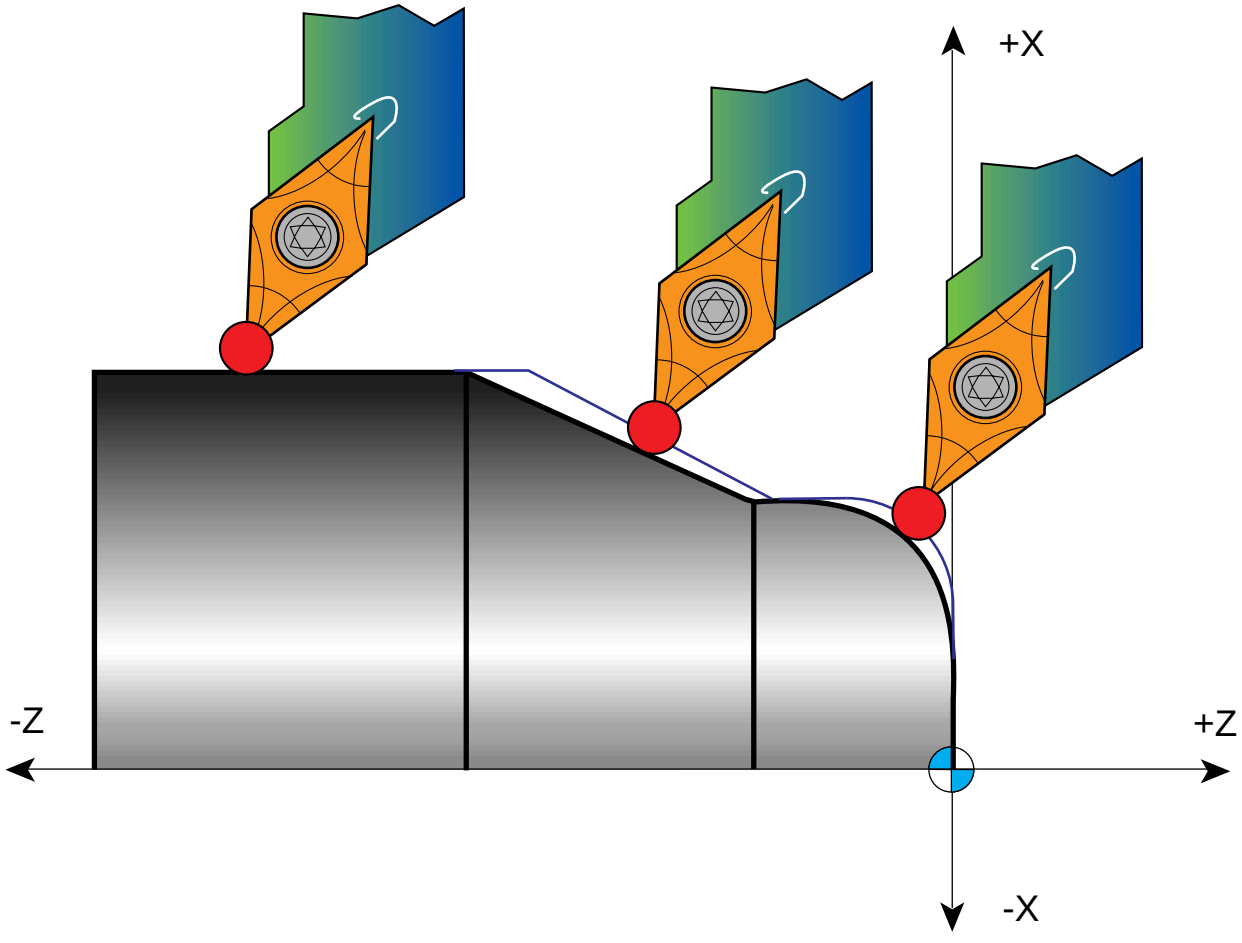
L = Lead of thread

n = Rotating time of main spindle

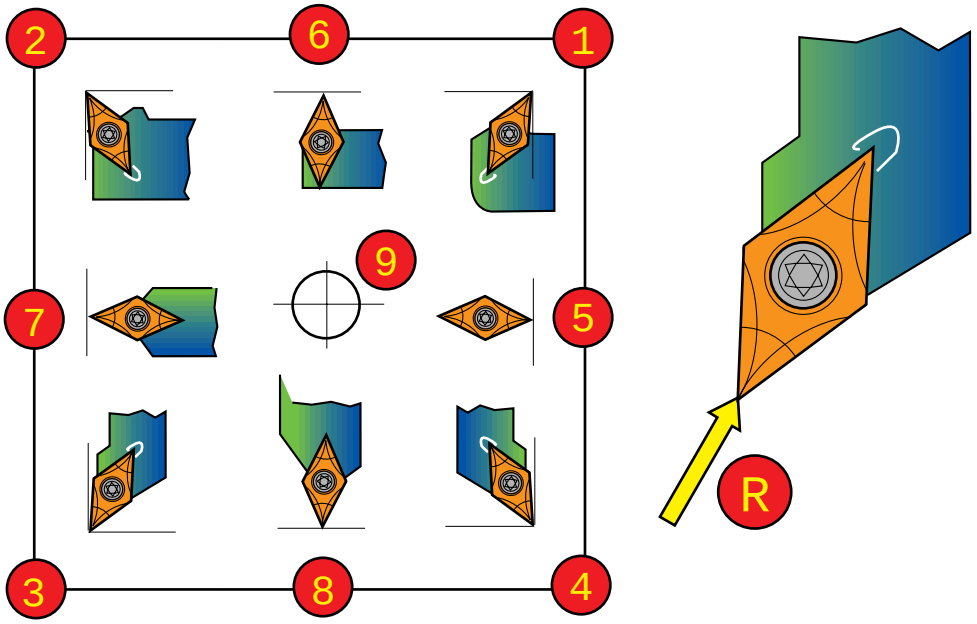
$$\delta 2 = \frac{L \times n}{1800}$$

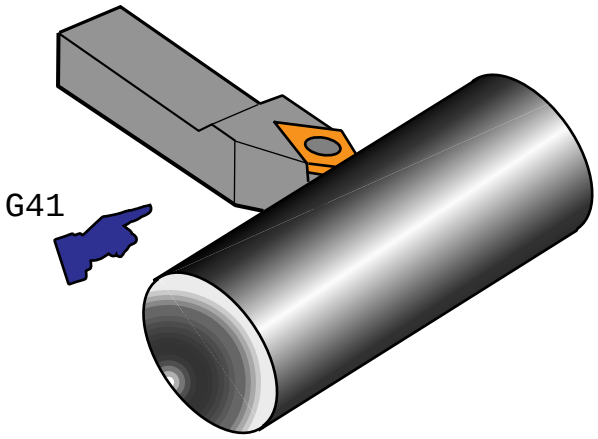
G42

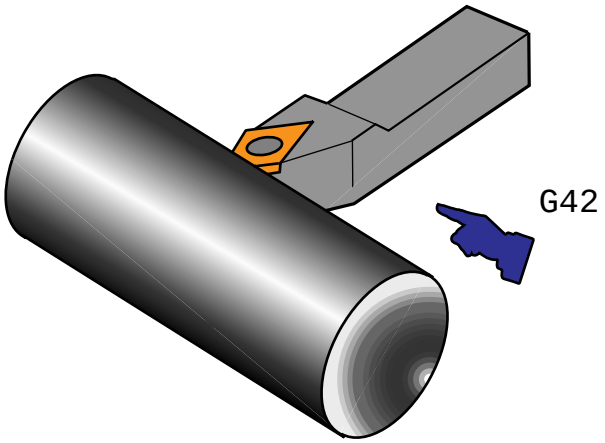




G41 G42



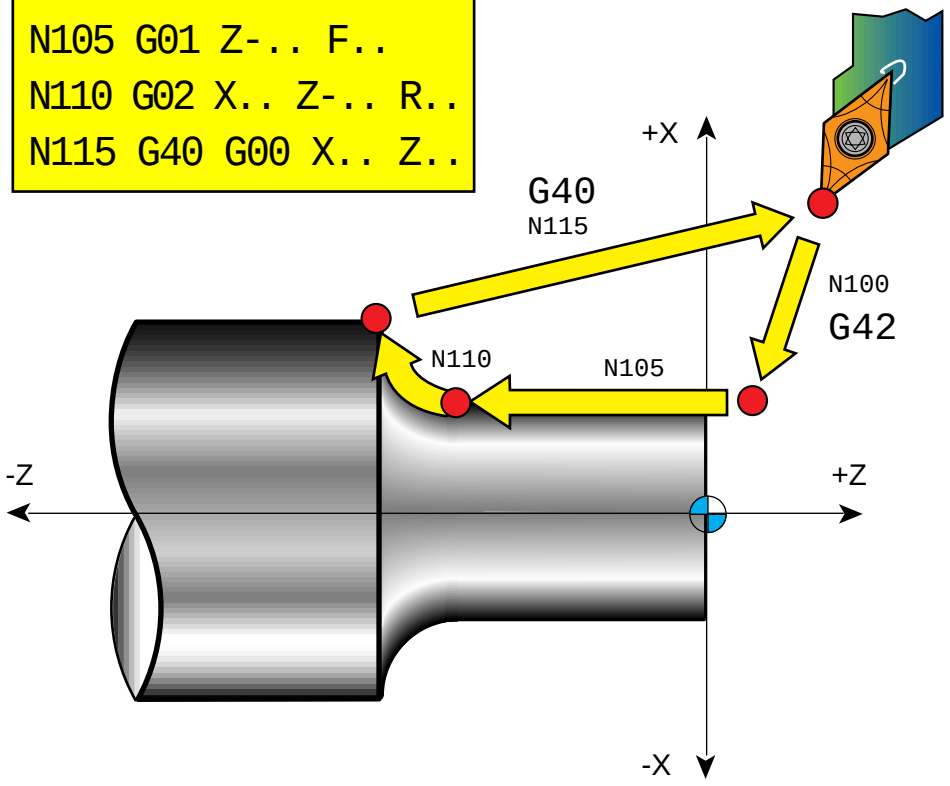




G40

G42

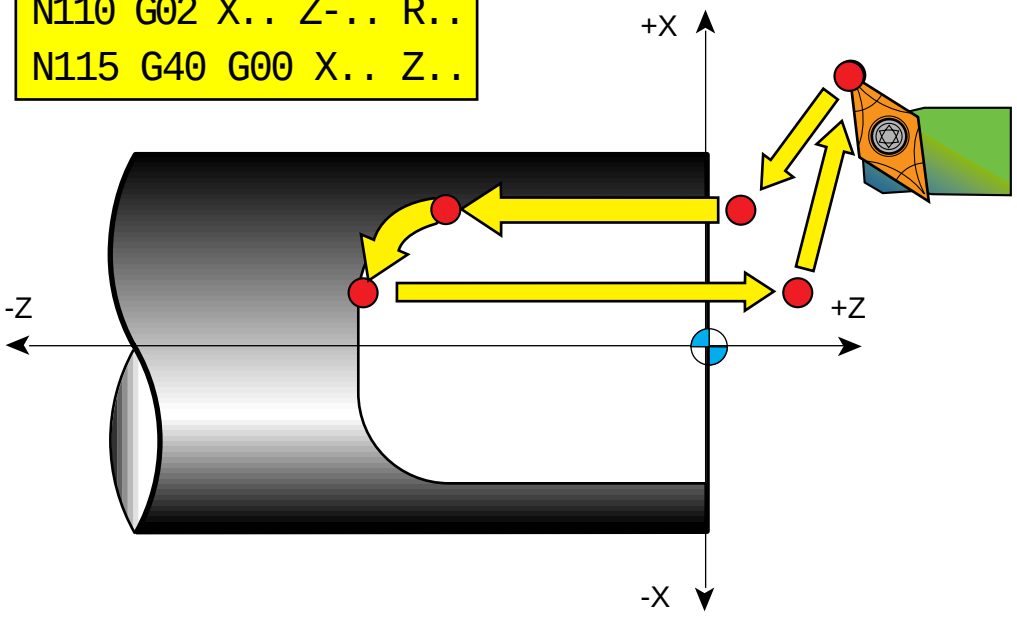
```
N100 G42 G00 X.. Z..
N105 G01 Z-.. F..
N110 G02 X.. Z-.. R..
N115 G40 G00 X.. Z..
```



G41

G40

```
N100 G41 G00 X.. Z..
N105 G01 Z-.. F..
N110 G02 X.. Z-.. R..
N115 G40 G00 X.. Z..
```

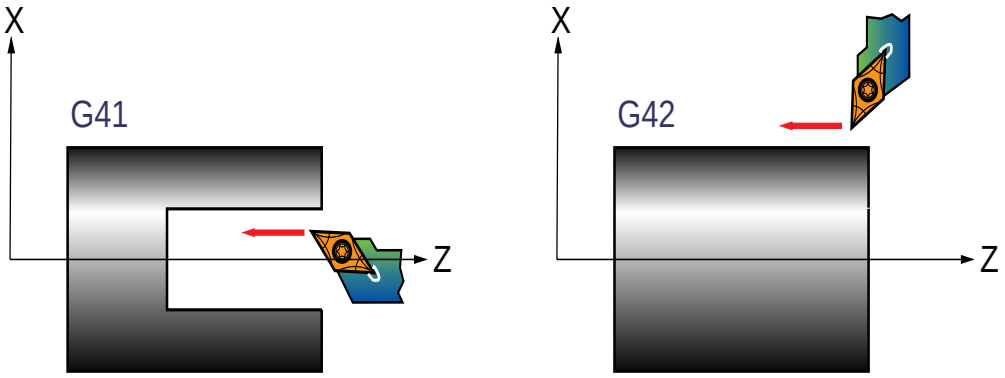


Tool diameter compensation

G40 : R compensation cancel

G41 : When located on the left side of material based on the progressing direction,

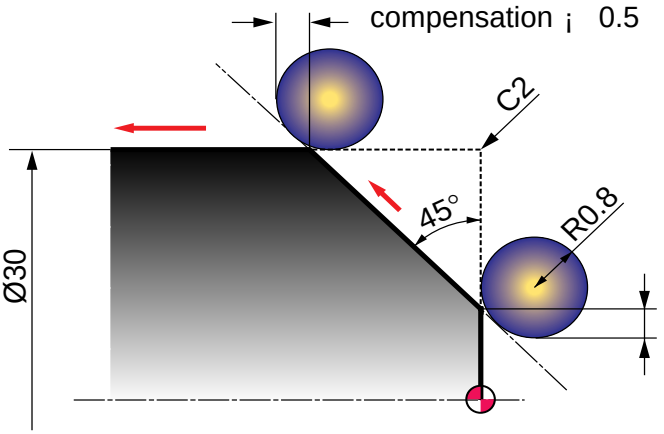
G42 : When located on the right side of material based on the progressing direction,



What is Tool diameter compensation?

If R is on the end of the tool edge, parts which are not impensated only by tool position OFFSET are occured during the taper cutting or circlar cutting. Therefor, impensating this error automatically is namelyR compensation.(During the tool diameter compensation, add theR and T-direction in the R compensation column of OFFSET PAGE.

Example 1) When not using tool diameter compensation(R compensation ⓐ and ⓑ should be calculated)



PROGRAM

G01 X25.0 Z0 F0.2 :

X30.0 Z-2.5 :

G00 U1.0 Z1.0 :

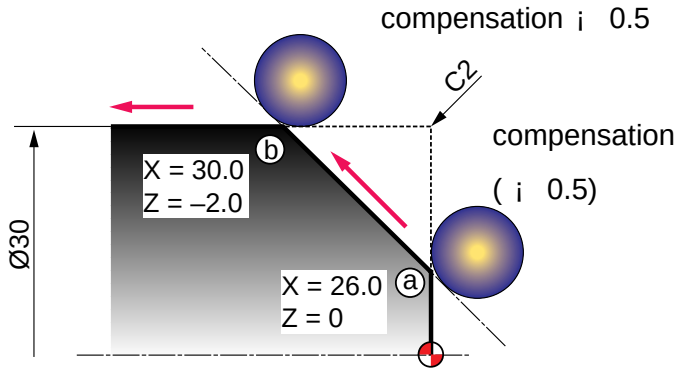
G28 UO WO :

M30 :

*

Example 2) When using tool diameter compensation

* You do not have to calculate R compensation ⓐ and ⓑ
* If ⓐ position and ⓑ position are given on the program, the tool performs automatically R compensation and moves to the next progressing direction.



PROGRAM

G42 X26.0 Z0 F0.2 :

G01 X30.0 Z-2.0 :

Z-30.0 :

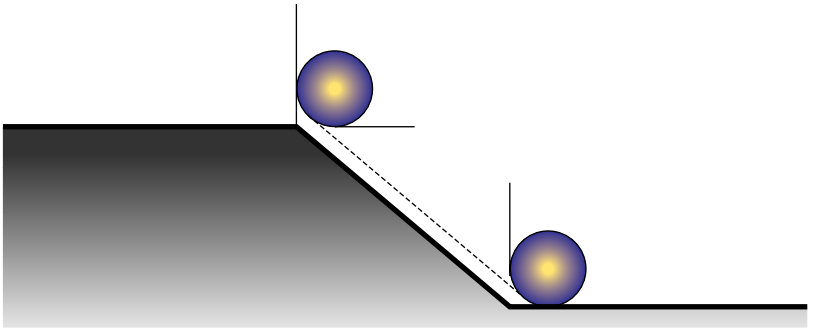
G00 U1.0 Z1.0 :

G28 UO WO :

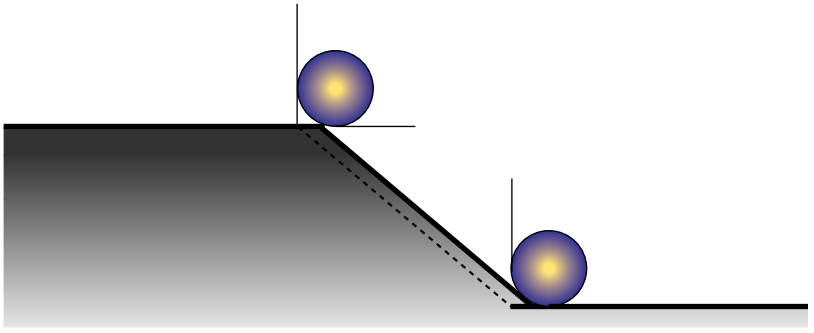
M30 :

*

Presentation 1) In case of no compensation



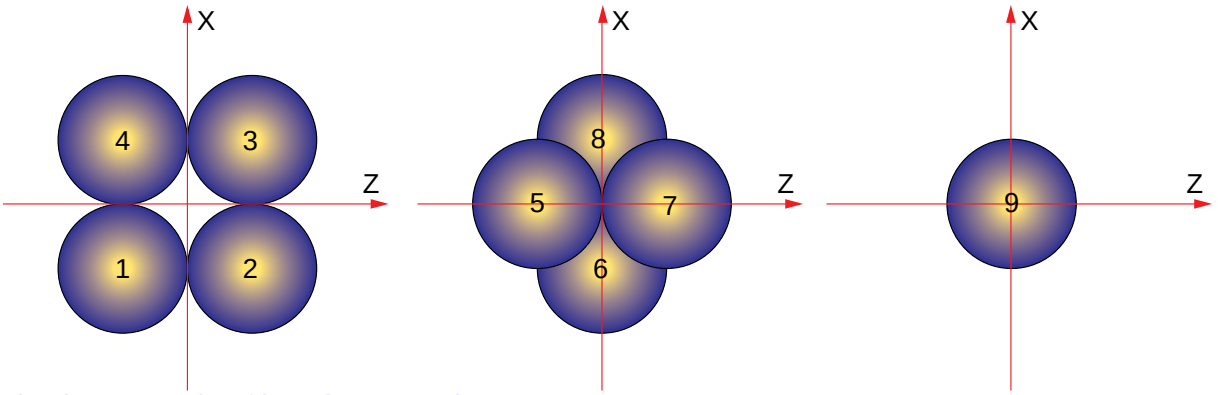
Presentation 2) In case of compensation



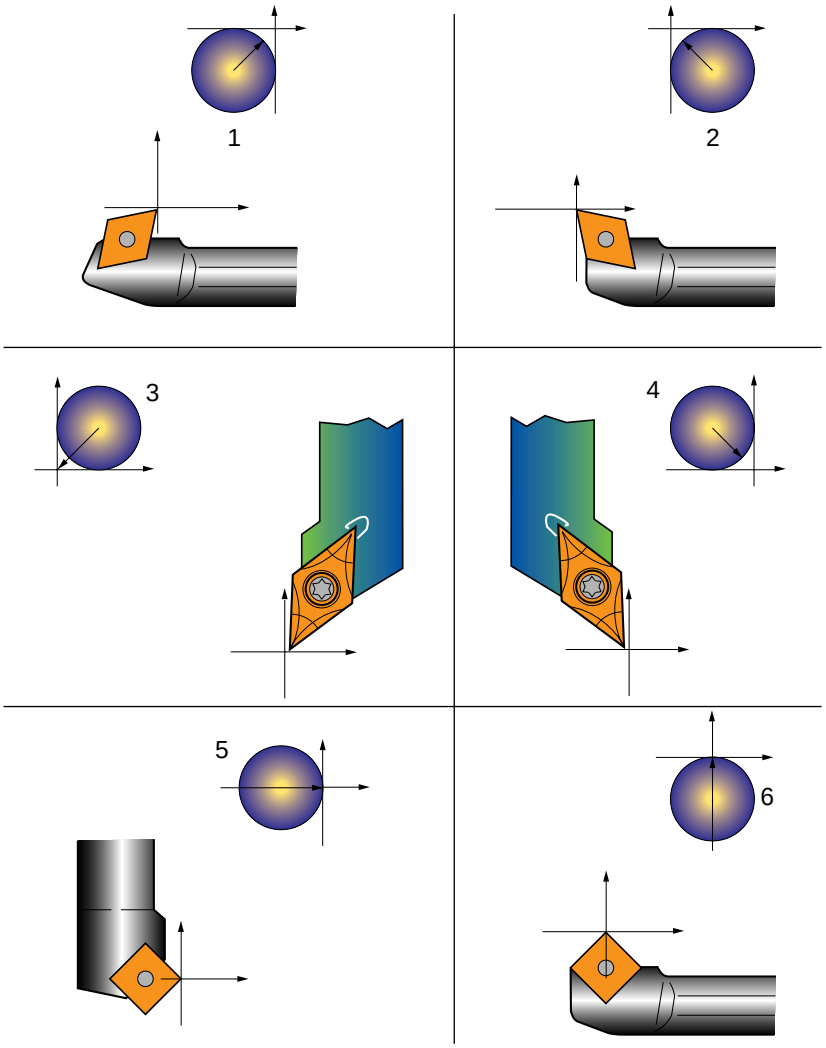
1) Direction of imaginary (In case of right hand coordinate)

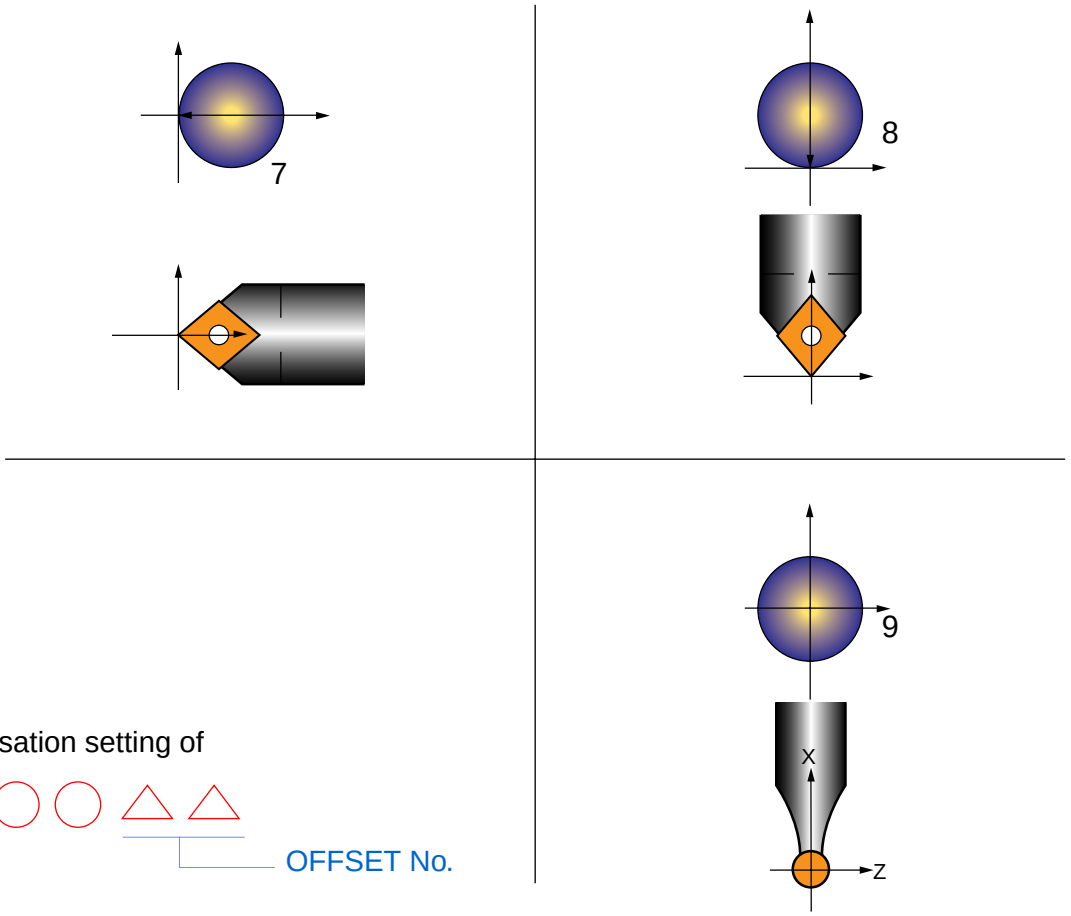
Direction of imaginary seen from the center of radius is decided by the cutting direction of tool during the cutting. Therefor, it should be set as much as compensation volume.

Direction and number of imaginary are decided among the following eight types.



<Selecting example of imaginary number>





2) Compensation setting of

T ○ ○ △ △

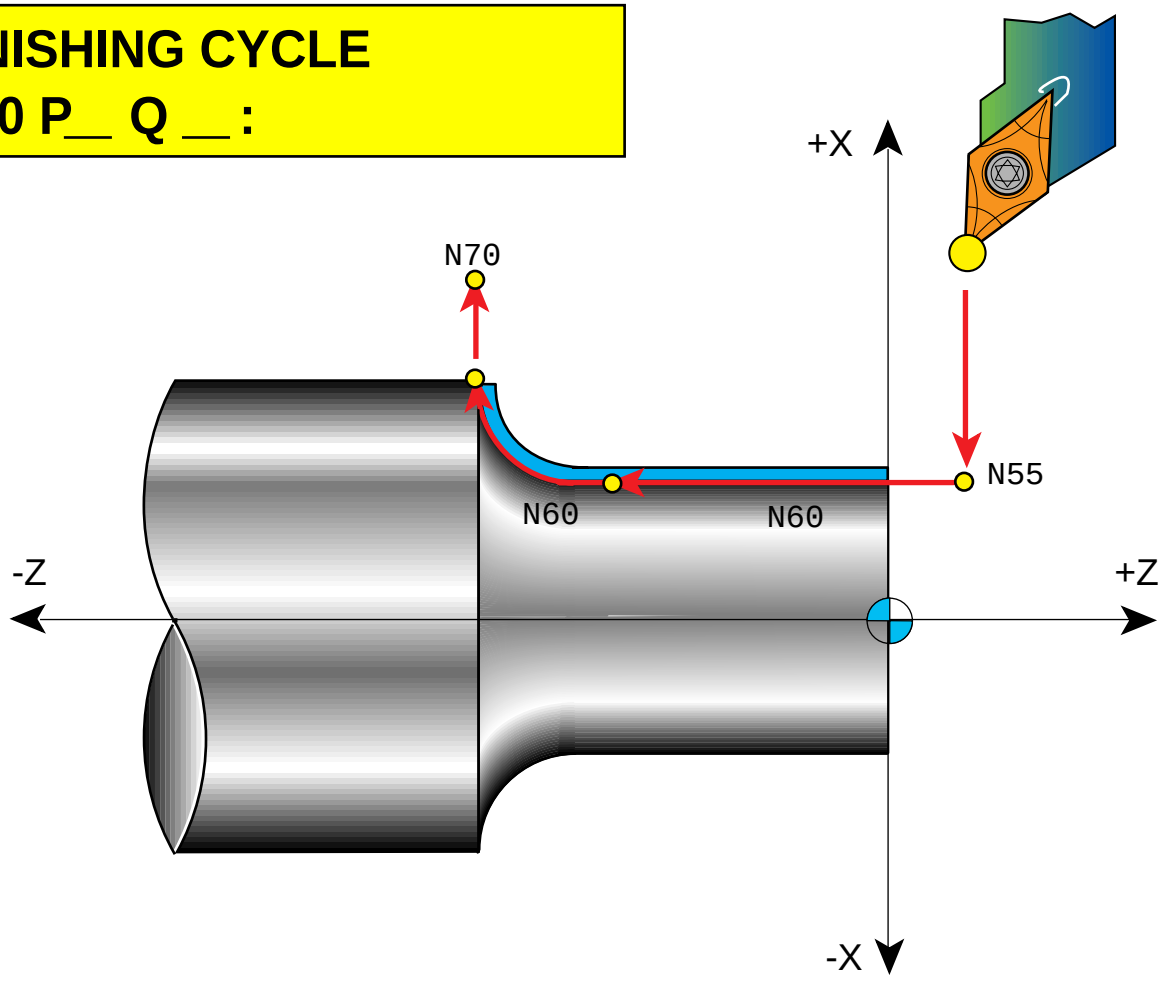
OFFSET No.

OFFSETNO.	X	Z		TOOL DIRECTION
01	0.75	-0.93	0.4	3
0.2	-1.234	10.987	0.8	2
.	.	.	.	.
.	.	.	.	.
16	.	.	.	.

Command scope of OFFSET volume0– + 999.999mm

G70

FINISHING CYCLE
G70 P__ Q __ :

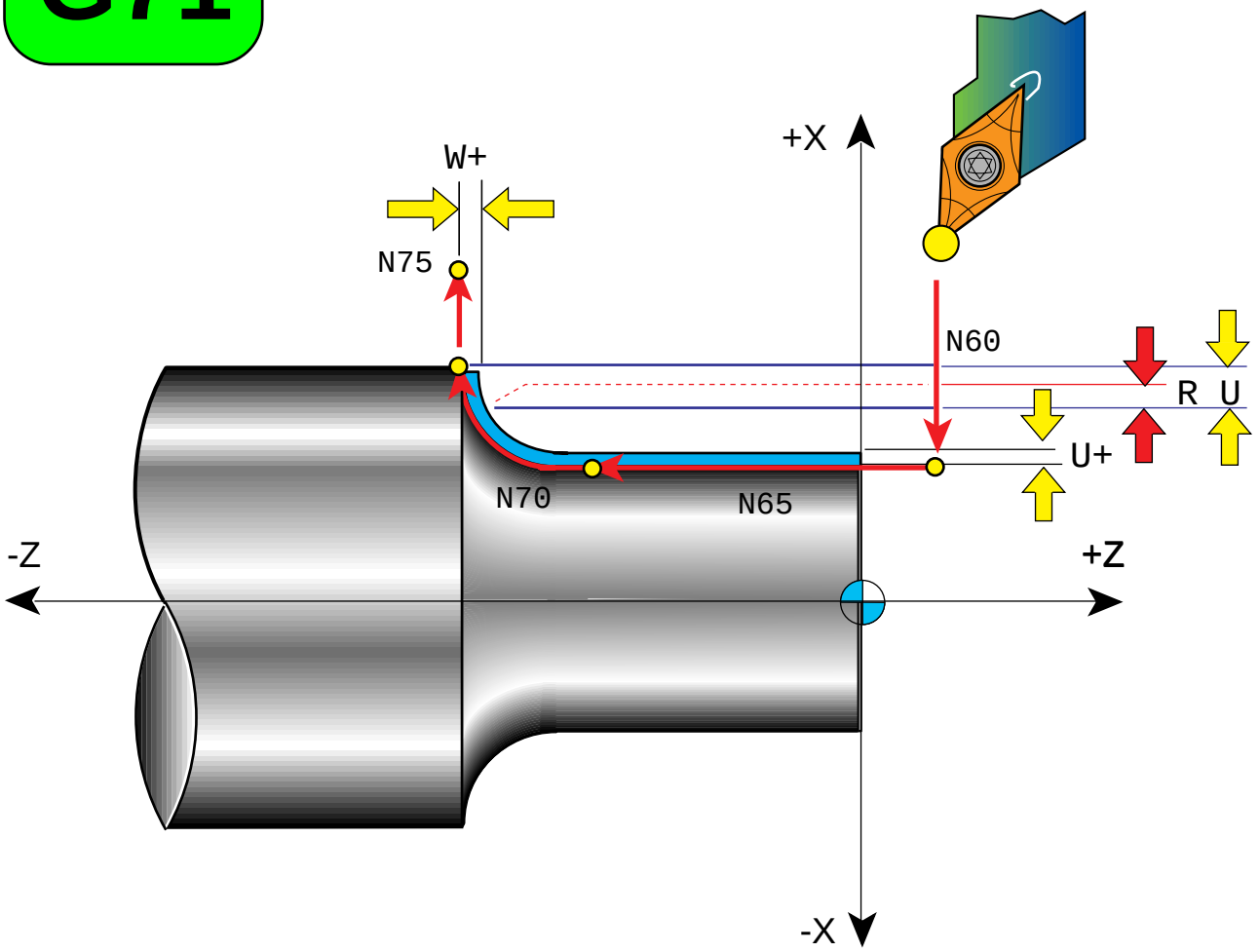



```
N..
N50  G70  P55  Q70
```

P →
```
N55   G0   G42   X..
N60   G1   Z-..
N65   G2   X..   Z..   R..
```

Q →
```
N70   G1   G40   X..
N..
```

G71



N..

N50	G71	U..	R..		
N55	G71	P60	Q75	U+..	W+..

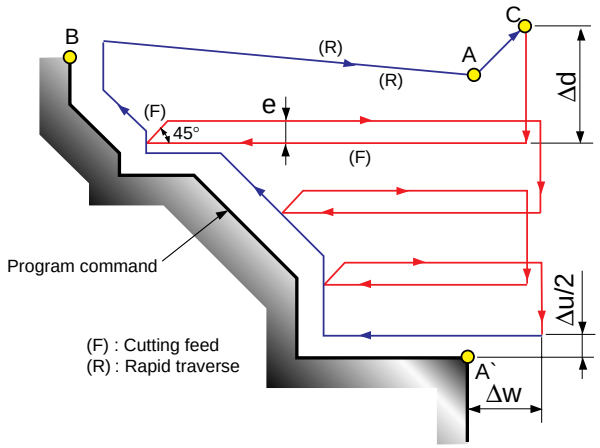
P →
```
N60   G0   G42   X..
N65   G1   Z-..
N70   G2   X..   Z-..   R..
```

Q →
```
N75   G1   G40   X..
N..
```

G71(STOCK REMOVAL IN TURNING)

G71 U(¡ d) R(e) :

G71 P__ Q__ U(¡ u) W(¡ w) F :



P : Start sequence no.

Q : Final sequence no.

U(¡ d) : Cut volume of one time(Designate the radius.

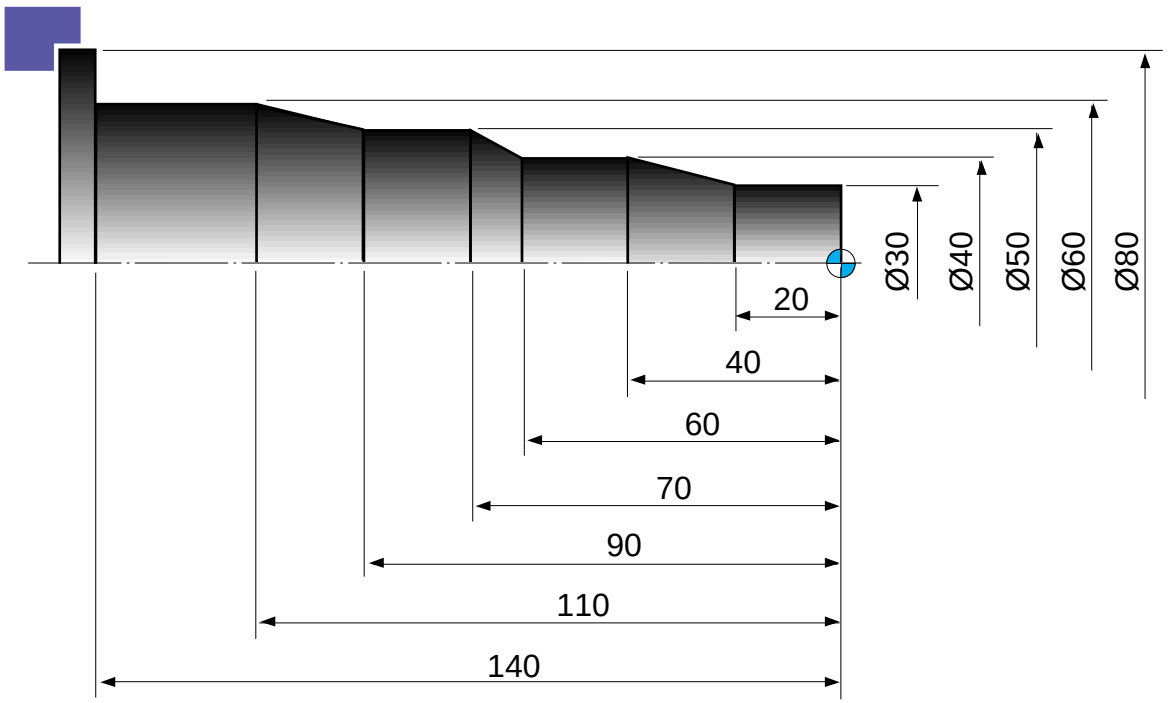
R(e) : Escape volume(Always 45) escape

U(¡ u) : Finishing tolerance in X axis

W(¡ w) : Finishing tolerance in Z axis

F(f) : Cutting feedrate

Example of program



(G70, G71)

N10 G50 S1500 T0101 :

G96 S180 M03 :

G00 X85.0 Z5.0 M08 :

Z0 :

G01 X-1.6 F0.25 :

G00 X83.0 Z2.0 :

G71 U3.0 R1.0 :

G71 P20 Q30 U0.5 W0.1 F0.27 :

N20 G42 G00 X30.0 : ← G71 CYCLE CUTTING FEED

G01 Z-20.0 F0.17 :

← G70 CYCLE CUTTING FEED

X40.0 Z-40.0 :

Z-60.0 :

X50.0 Z-70.0 :

Z-90.0 :

X60.0 Z-110.0 :

Z-140.0 :

X80.0 :

N30 G40 :

G70 P20 Q30 : (When using the same bite)

G00 X200.0 Z200.0 T0100 :

M30 :

¡¯When finishing, if a different bite is used

G00 X200.0 Z200.0 T0100 :

M01 :

N40 G50 S2000 T0303 :

G96 S200 M03 :

G00 X83.0 Z2.0 M08 :

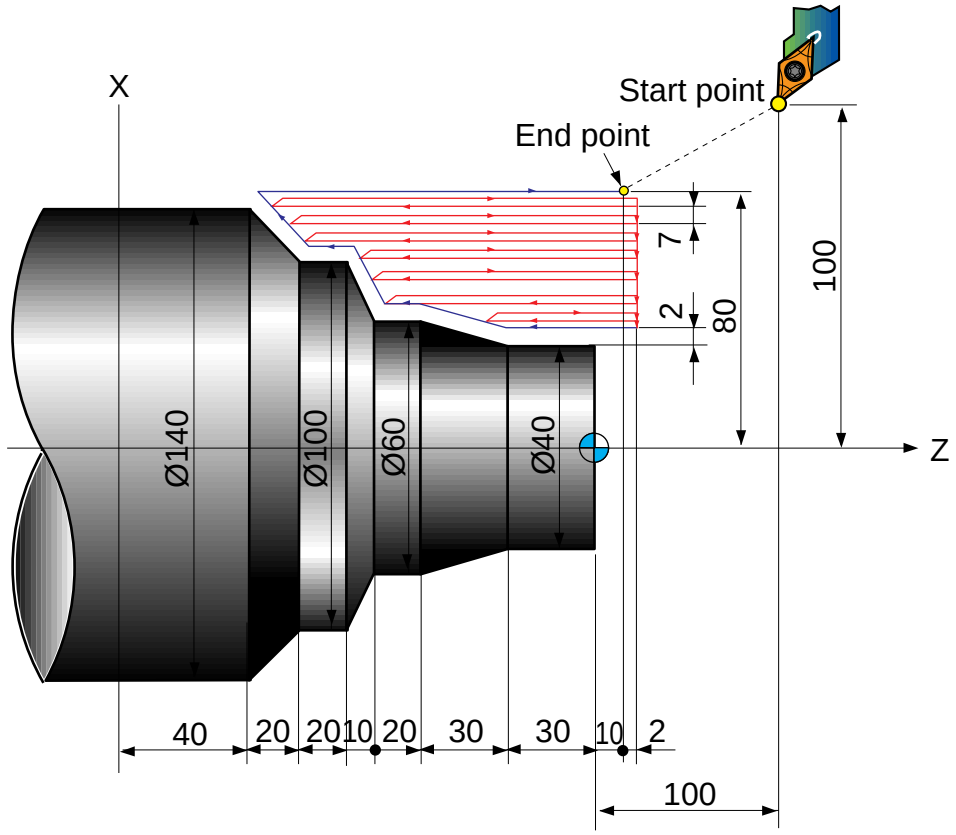
G70 P20 Q30 :

G00 X200.0 Z200.0 T0300 :

M30 :

Examples of program

Stock Removal in Turning(G71) (Type I)



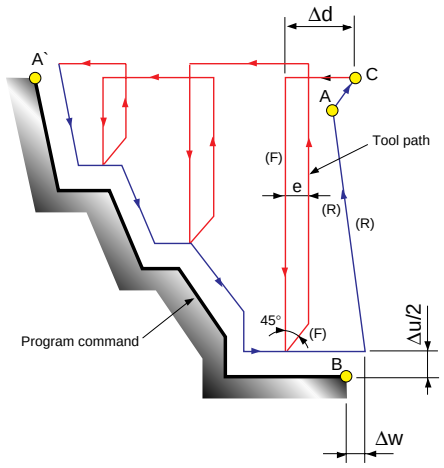
(Diameter designation, metric input)

```
N010 G00 X200.0 Z100.0 :
N011 G00 X160.0 Z10.0 :
N012 G71 U7.0 R1.0 :
N013 G71 P014 Q021 U4.0 W2.0 F0.3 S550 :
N014 G00 G42 X40.0 S700 :
N015 G01 W-40.0 F0.15 :
N016      X60.0 W-30.0 :
N017      W-20.0 :
N018      X100.0 W-10.0 :
N019      W-20.0 :
N020      X140.0 W-20.0 :
N021      G40 U2.0 :
N022 G70 P014 Q021 :
N023 G00 X200.0 Z100.0 :
M30 :
```

G72(STOCK REMOVAL IN FACING)

G72 W(¡ d) R(e) :

G72 P_ Q_ U(¡ u) W(¡ w) F :



U(¡ d) : Cut volume of one time

R(e) : Escape volume

P : Start sequence No.

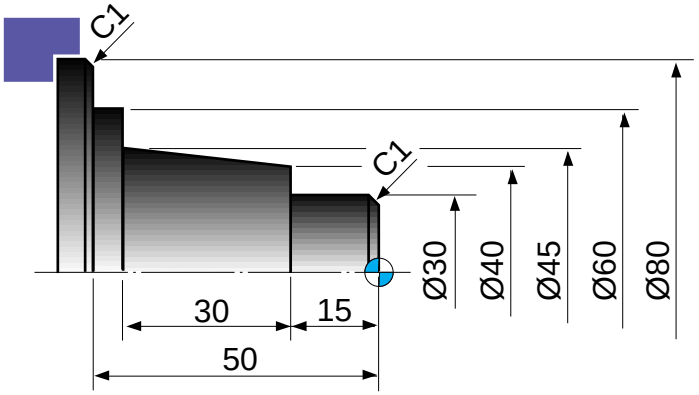
Q : Final sequence No.

U(¡ u) : Finishing in clearance X axis(Diameter command)

W(¡ w) : Finishing in clearance Z axis

F(f) : Cutting feedrate

Example of program



N10 G50 S2000 T0100 :

G96 S180 M03 :

G00 X85.0 Z5.0 T0101 :

Z0 :

G01 X-1.6 F0.2 :

G00 X85.0 Z1.0 :

G72 W2.0 R1.0 :

G72 P12 Q14 U0.5 W0.2 F0.25 :

N12 G00 G41 Z-51.0 :

G01 X80.0 F0.2 :

X78.0 W1.0 :

X60.0 :

Z-45.0 :

X40.0 Z-15.0 :

X30.0 :

Z-1.0 :

X26.0 Z1.0 :

N14 G40 :

G70 P12 Q14 :

G00 X200.0 Z200.0 T0100 :

M30 :

¡ ¯(When finishing with a different tool)

G00 X200.0 Z200.0 T0100 :

M01 :

N16 G50 S2500 T0300 :

G96 S200 M03 :

G00 X85.0 Z5.0 T0303 :

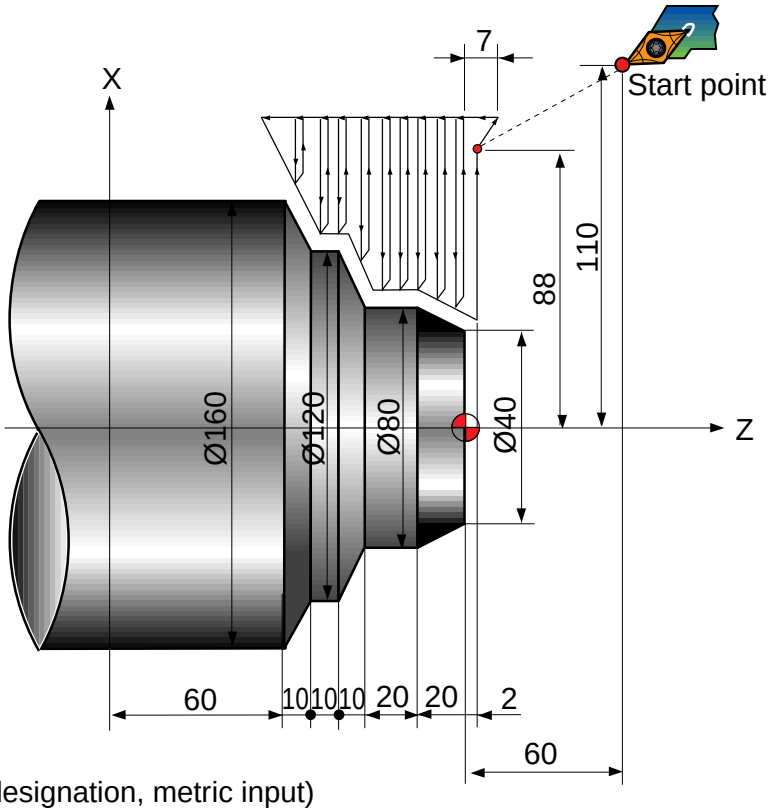
G70 P12 Q14 :

G00 X200.0 Z200.0 T0300 :

M30 :

Examples of program

Stock Removal in Pacing(G72)



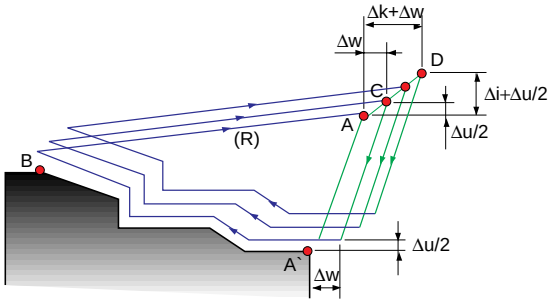
(Diameter designation, metric input)

```
N010 G00 X220.0 Z60.0 :
N011 G00 X176.0 Z2.0 :
N012 G72 W7.0 R1.0 :
N013 G72 P014 Q021 U4.0 W2.0 F0.3 S550 :
N014 G00 G41 Z-70.0 S700 :
N015 X160.0 :
N016 G01 X120.0 Z-60.0 F0.15 :
N017      W10.0 :
N018      X80.0 W10.0 :
N019      W20.0 :
N020      X36.0 W22.0 :
N021      G40 :
N022 G70 P014 Q021 :
N023 G00 X220.0 Z60.0 :
N024 M30 :
```

G73(PATTEN REPEATING)

G73 U(¡ i) R(d) W(¡ k) :

G73 P Q U(¡ u) W(¡ w) F :



U(¡ i) : Excape distance and direction in X axis
(Designated the radius)

W(¡ k) : Escape distance and direction in Z axis

R(d) : Repeating time
(It is conneeted with the cut volume of each time)

P : Start sequence No.

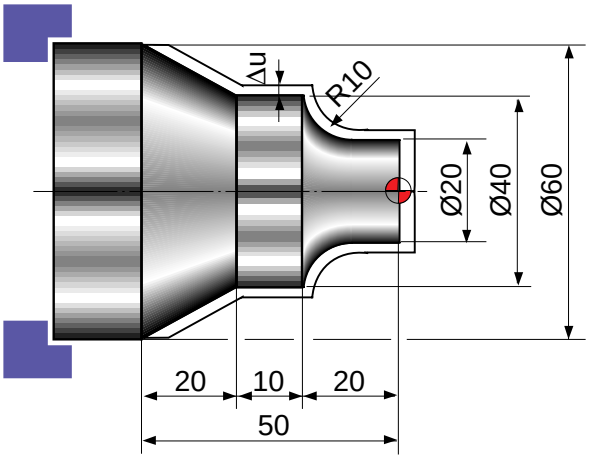
Q : Final sequence No.

U(¡ u) : Finishing in clearance X axis(Radius designated)

W(¡ w) : Finishing in clearance Z axis

F(f) : Cutting feedrate

Example of program



N10 G50 S2000 T0300 :
G96 S200 M03 :

G00 X35.0 Z5.0 T0303 :

Z0 :

G01 X-1.6 F0.2 :

G00 X70.0 Z10.0 :

G73 U3.0 W2.0 R2 :

G73 P12 Q16 U0.5 W0.1 F0.25 :

N12 G00 G42 X20.0 Z2.0 :

G01 Z-10.0 F0.15 :

G02 X40.0 Z-20.0 R10.0 :

G01 Z-30.0 :

X60.0 Z-50.0 :

N16 G40 U1.0 :

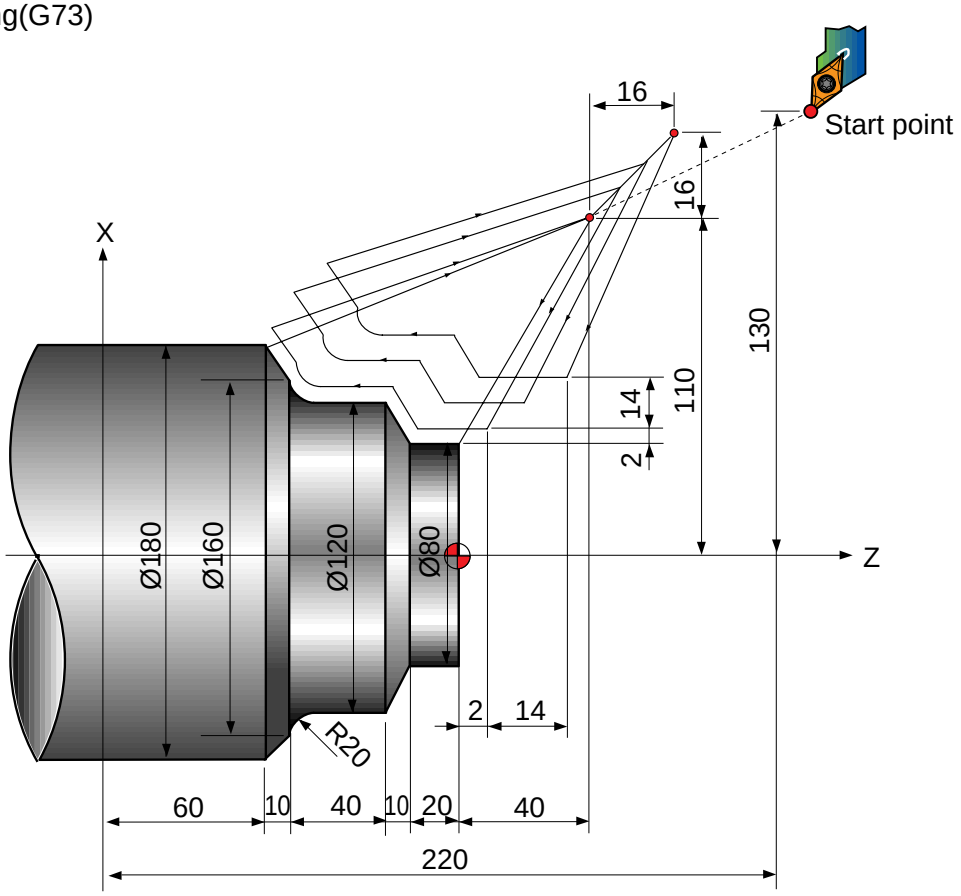
G70 P12 Q16 :

G00 X200.0 Z200.0 T0300 :

M30 :

Examples of program

Pattern Repeating(G73)



(Diameter designation, metric input)

N010 G00 X260.0 Z80.0 :

N011 G00 X220.0 Z40.0 :

N012 G73 U14.0 W14.0 R3 :

N013 G73 P014 Q020 U4.0 W2.0 F0.3 S0180 :

N014 G00 G42 X80.0 Z2.0 :

N015 G01 W-20.0 F0.15 S0600 :

N016 X120.0 W-10.0 :

N017 W-20.0 S0400 :

N018 G02 X160.0 W-20.0 R20.0 :

N019 G01 X180.0 W-10.0 S0280 :

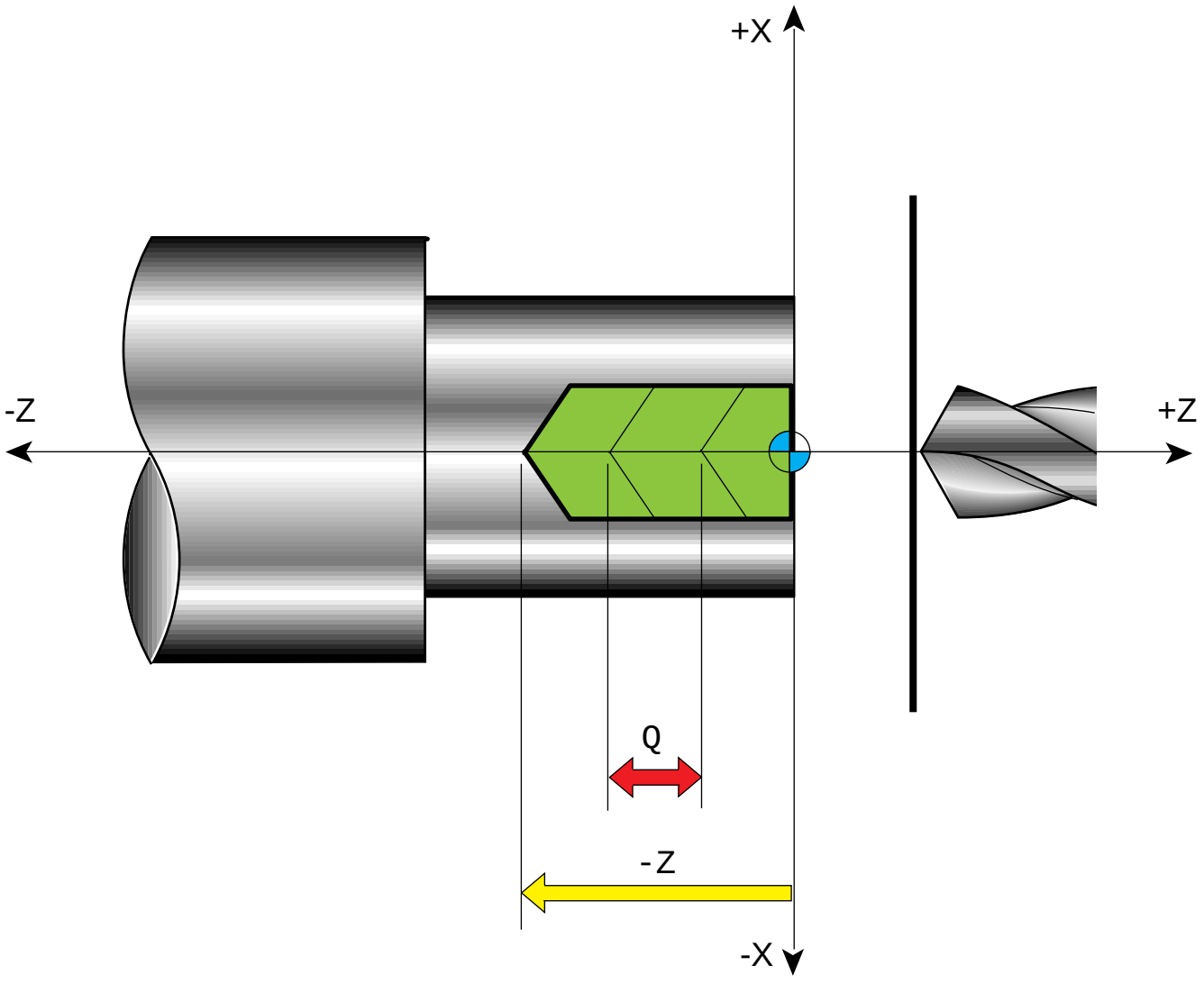
N020 G40 :

N021 G70 P014 Q020 :

N022 G00 X260.0 Z80.0 :

N023 M30 :

G74



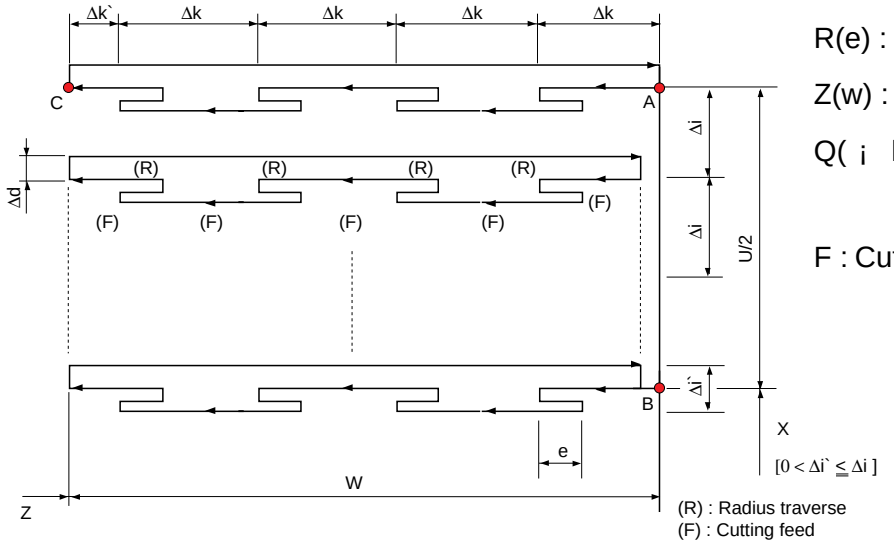

```
N40   G74   R..
N50   G74   Z-..   Q..   F..
```

G74(Peck drilling in Z axis divection)

1) Drill cutting cycle

G74 R(e) :

G74 Z(w) Q(¡ k) F :



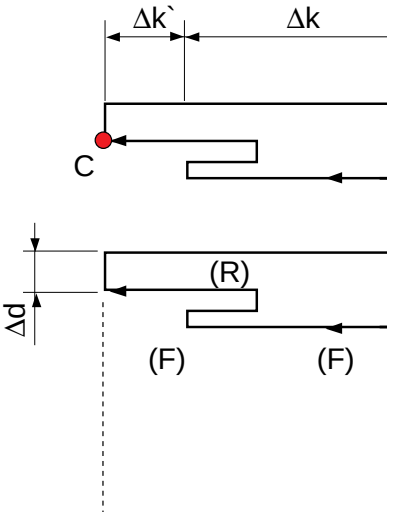
R(e) : Retreat volume

Z(w) : Final cutting depth

Q(¡ k) : One time cutting depth
(1000=1mm)

F : Cutting feedrate

Examples of program



N10 G50 S500 T0200 :

G97 S280 M03 :

G00 X0 Z5.0 T0202 M08 :

→ Start point of drilling

G74 R1.0 :

G74 Z-90.0 Q5000 F0.23 :

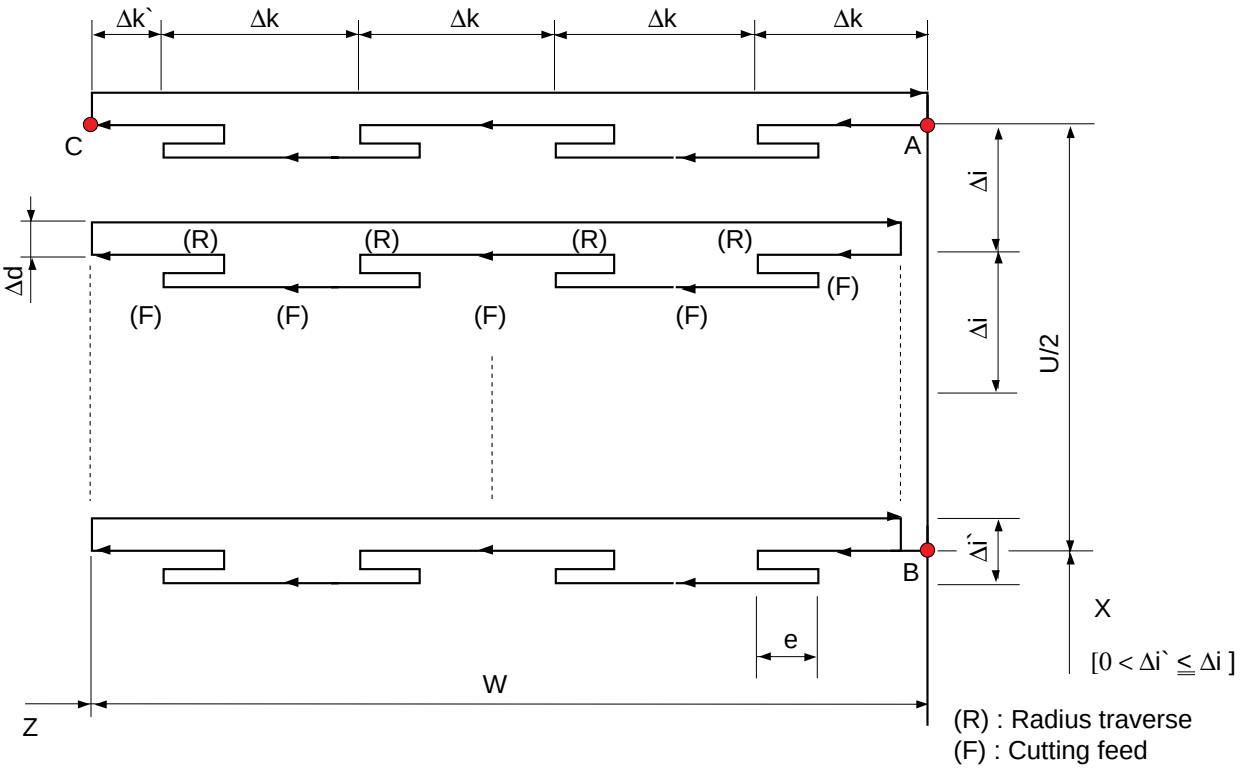
G00 X200.0 Z150.0 T0200 :

M01 :

2) Stock removal cycle in side

G74 R(e) :

G74 X(u) Z(w) P(¡ i) Q(¡ k) R(¡ d) F :



R(e) : Retreat volume(Modal command)

P(¡ i) : Moving volume of X axis

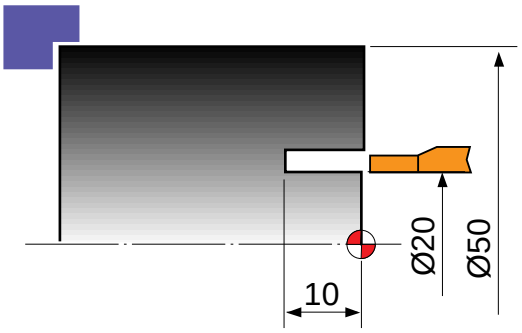
Q(¡ k) : Cut volume in Z axis(Q5000=5mm)

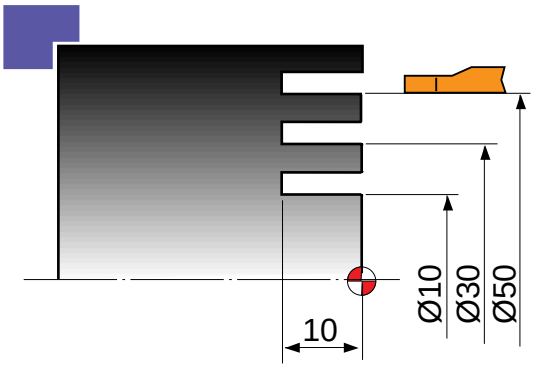
X(u) : Composition of X axis

Z(w) : Final cutting depth

R(¡ d) : Escape wlume at the end point of Z axis proess(Designate the symbol and radius according to the direction of escape)

F : Cutting feedrate





¡¯ If there is one groove, X(u), P(¡ i) can be omitted.

(In case of omitting, it shall be done at the same time)

N10

G00 X20.0 Z1.0 :

G74 R1.0 :

G74 Z-10.0 Q3000 F0.1 :

G00 X200.0 Z200.0 :

M30 :

Attention

FANUC 0TC
Q3000=3mm
P10000=10MM

N10 G50 S2000 T0100 :

G96 S80 M03 :

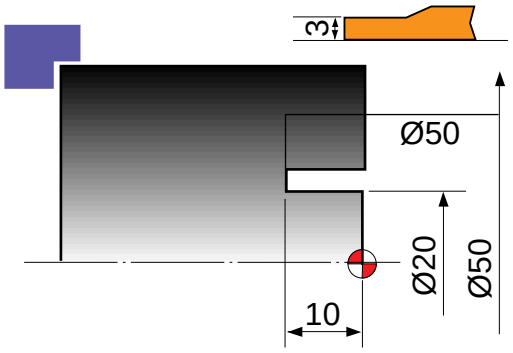
G00 X50.0 Z1.0 T0101 :

G74 R1.0 :

G74 X10.0 Z-10.0 P10000 Q3000 F0.1 :

G00 X200.0 Z200.0 T0100 :

M30 :



N1 G50 S2000 T0100 :

G96 S80 M3 :

G0 X47.0 Z1.0 T0101M8 :

G74 R1.0 :

G74 Z-10.0 Q3000 F0.1 :

G0 U-5.0 :

G74 X20.0 Z-10.0 P2500 Q3000 F0.1 :

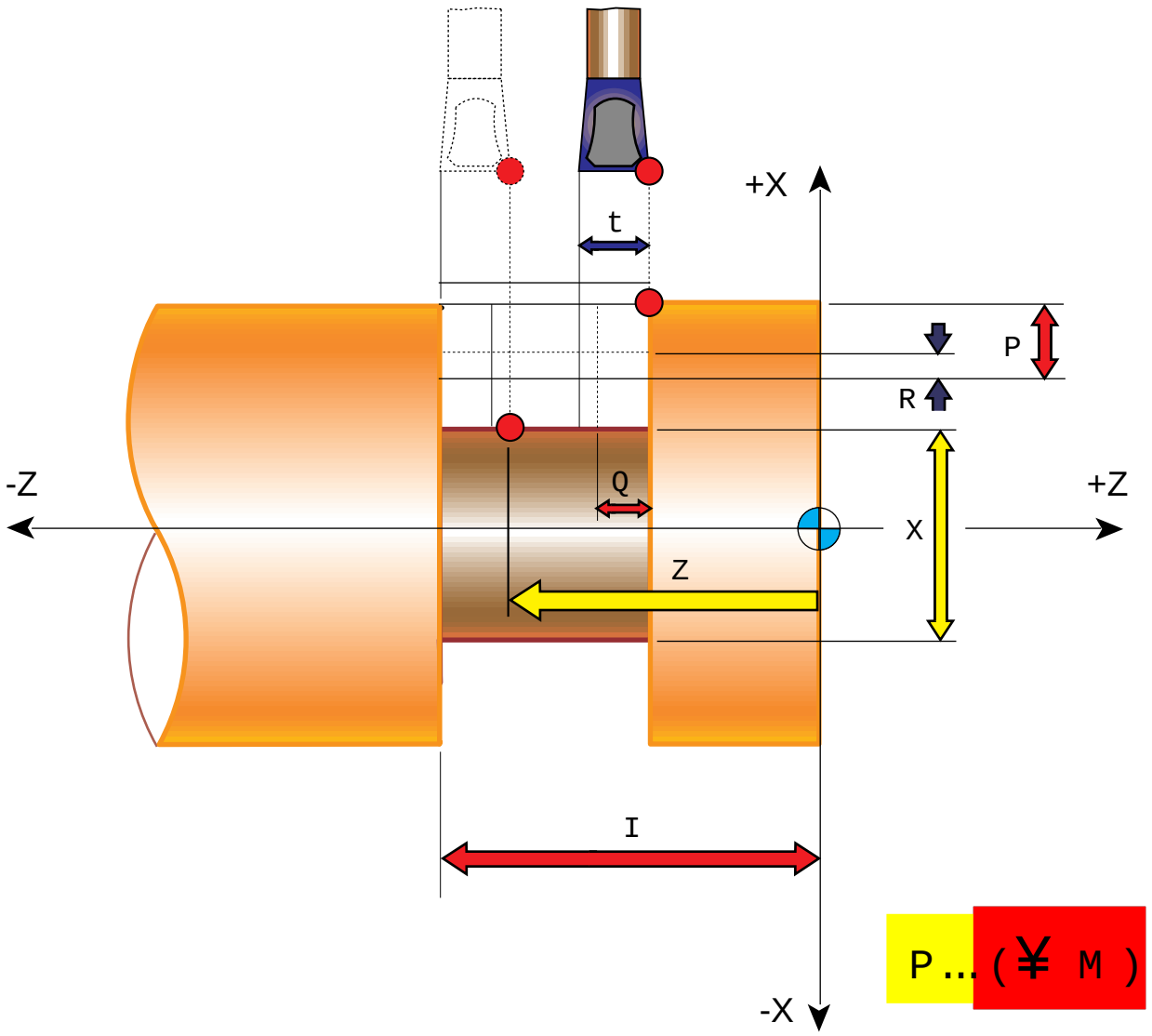
G0 X200.0 Z200.0 T0100 :

M30 :

G75

Q<T!

Z = I - T!



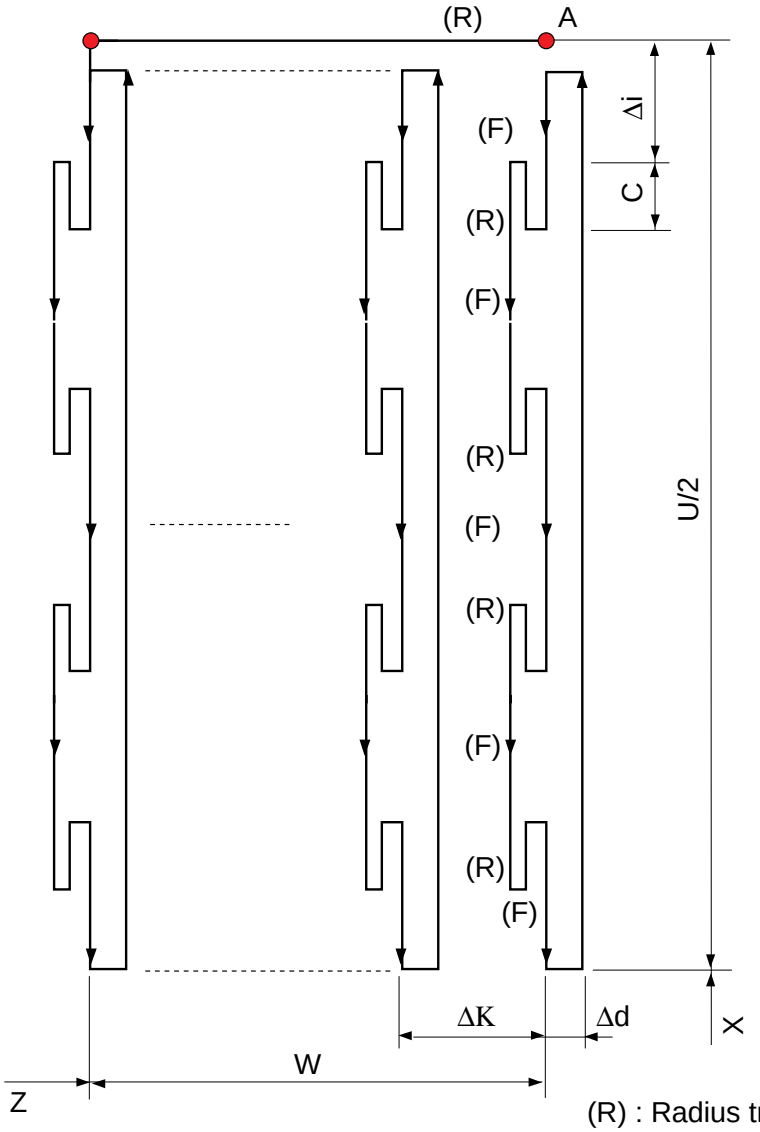
P... (¥ M)

```
N50 G75 R
N55 G75 X... Z-... P... Q...
```

G75(X directiion grooving : Peck drill cycle in turining)

G75 R(e) :

G75 X(u) Z(w) P(¡ i) Q(¡ k) R(¡ d) F :



(R) : Radius traverse
(F) : Cutting feed

R(e) : Retreat volume(Modal command)

X(u) : Compostion of X axis

Z(w) : Composition of Z axis

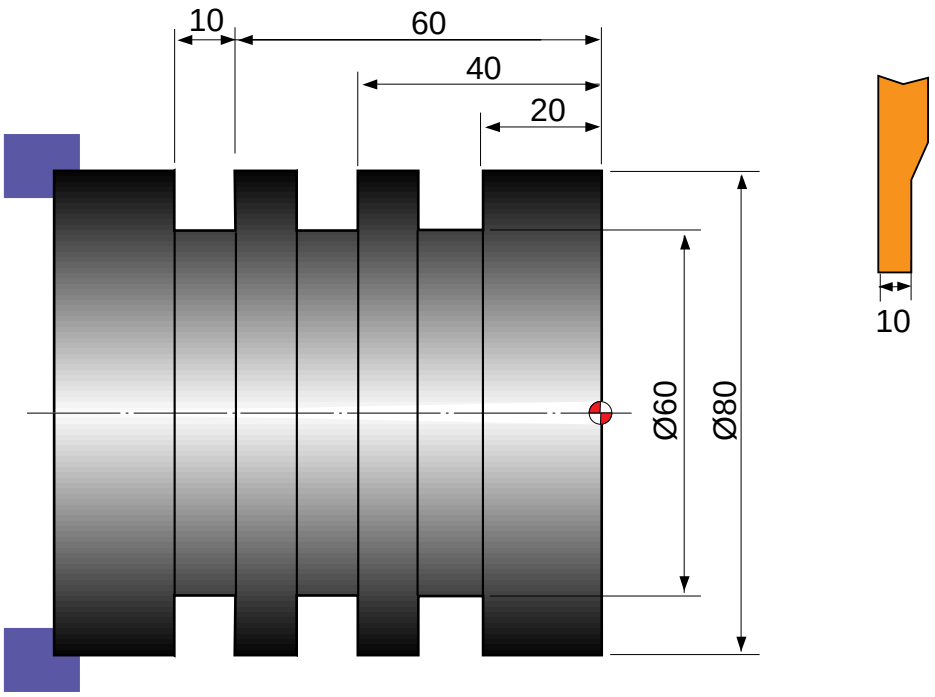
Q(k) : Moving volume in Z axis(Designate with out symblo)

P(i) : Cut volume or X axis(Designate the radius)

R(d) : Escape volume at the end point of X axis process

(Designate the symble according to escape dinetion)

F : Cutting feedrate



N10 G50 S500 T0100 :

G97 S_ M03 :

G00 X90.0 Z1.0 T0101 :

X82.0 Z-60.0 :

G75 R1.0 :

G75 X60.0 Z-20.0 P3000 Q20000 F0.1 : ¡¸ ¡£

G00 X90.0

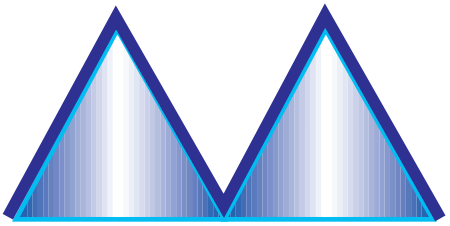
X200.0 Z200.0 T0100 :

M30 :

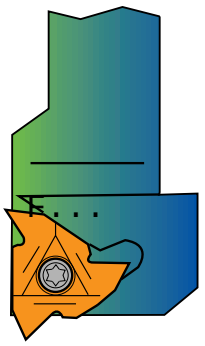
¡¯While it has the same function with G74, X and Z are exchanged.
If there is one groove, volues of Z and P can be omitted at the same time.

G76

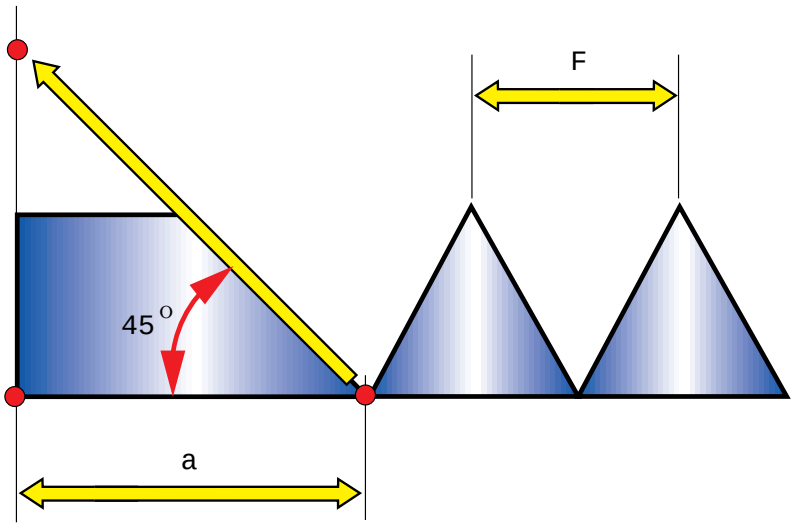
```
N50   G76   Pxx xx xx   Q...   R...
N55   G76   X...   Z... R0   P...   Q...   F...
```



Pxx (0 - 99)



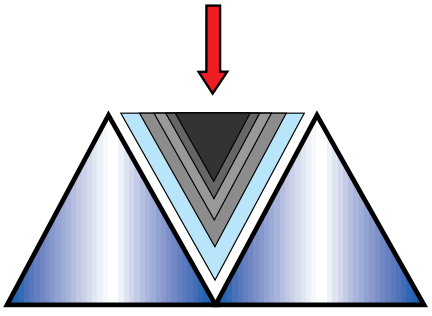
```
N50   G76   Pxx xx xx   Q...   R...
N55   G76   X...   Z... R0   P...   Q...   F...
```



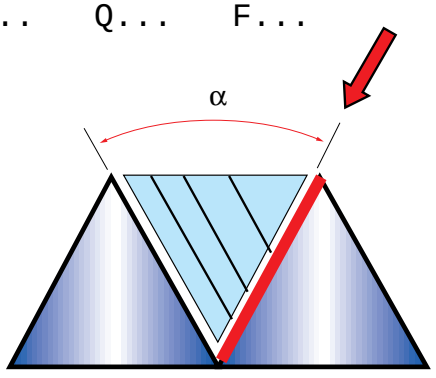
$$a = F*(\frac{Pxx}{10})$$

Pxx (0 - 99)

```
N50   G76   Pxx xx xx   Q...   R...
N55   G76   X...   Z... R0   P...   Q...   F...
```



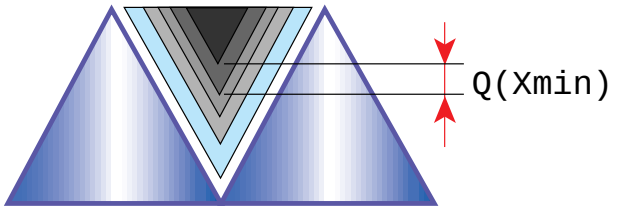
Pxx = 0

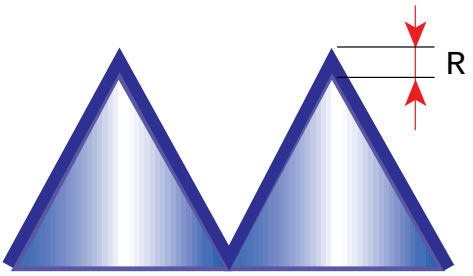


Pxx = α (80° , 60° , 55° , 30° , 29)

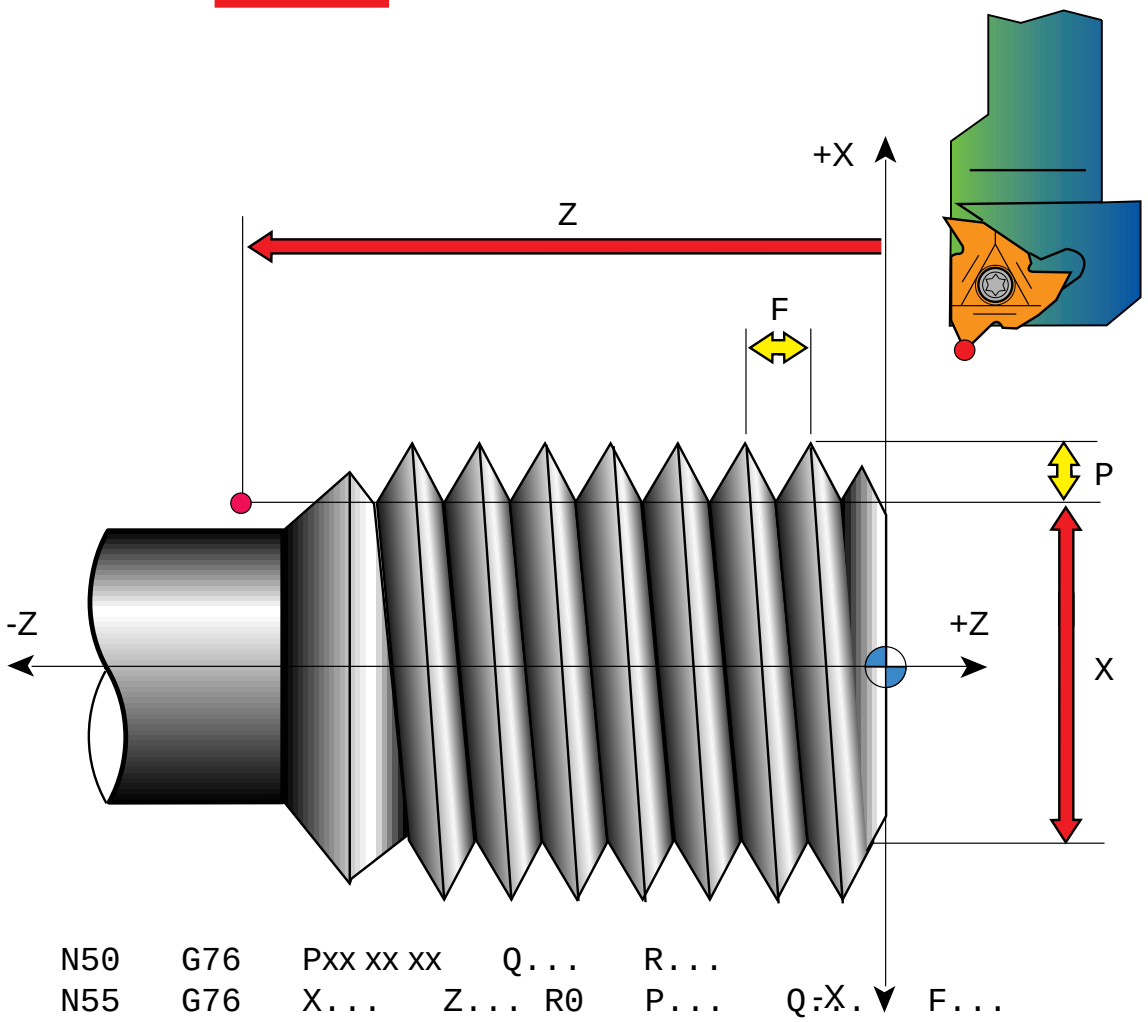
G76

```
N50   G76   Pxx xx xx   Q...   R...
N55   G76   X...   Z... R0   P...   Q...   F...
```





Q .. (µm)




```
N50   G76   Pxx xx xx   Q...   R...
N55   G76   X...   Z... R0   P...   Q...   F...
```

G73(Compound type thread cutting cycle)

By G76 command, thread cutting cycle is possible.

FORMAT	G76 P(m) (r) (a) Q(Δdmin) R(d) G76 X(u) Z(w) R(i) P(k) Q(Δd) F(f)

P(m) : Repeating time before the final thread
(r) : Chamfering at the end part of thread
(a) : Angle between threads

ex) P 0 2 1 0 6 0

- 02 : Repeating time
- 10 : Chanfering volume 1.0 lead
- 60 : Angle of thread face
- 10, 60 : omissible

Q(§Edmin) : Min. cut volume(Example : Calculate as Q100=NC and process at least more than 0.1 for processing of one time)-0.1(Decimal point is vot allowed)

R(§Ed) : Finishing clearance(Final finishing clearance)

X(u) : Core diameter of thread
(Command the value of Outer diameter of thread-<height of threadx2>)

Z(w) : Z spindle coordinate at the end point of thread process

R(i) : For omitting, straight thread and
- R– : X+ and Taper thread
- R+ : X– and Taper thread

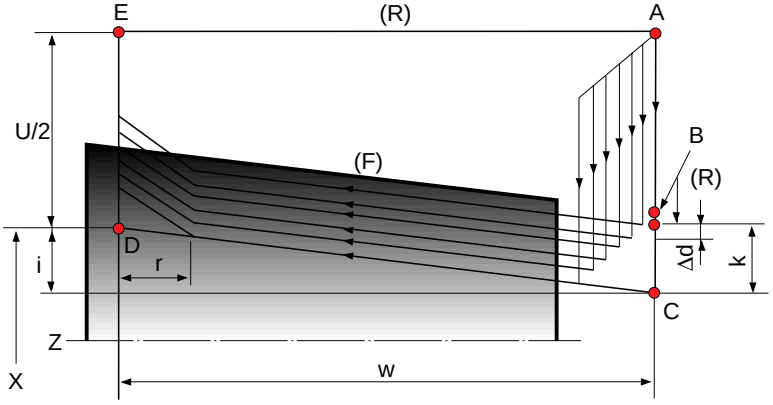
P(k) : Height of thread(Omit the decimal point <Example>P900=0.9mm)

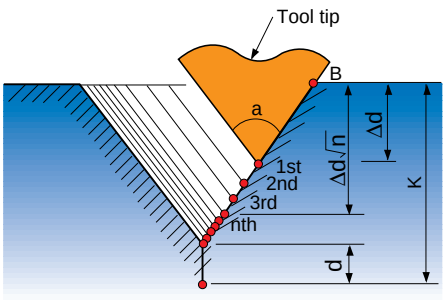
Q(d) : Initial cut volume (Omit the decimal point <Example>Q500=Designate) the radius value

F(f) : Cutting feedrate(Lead)

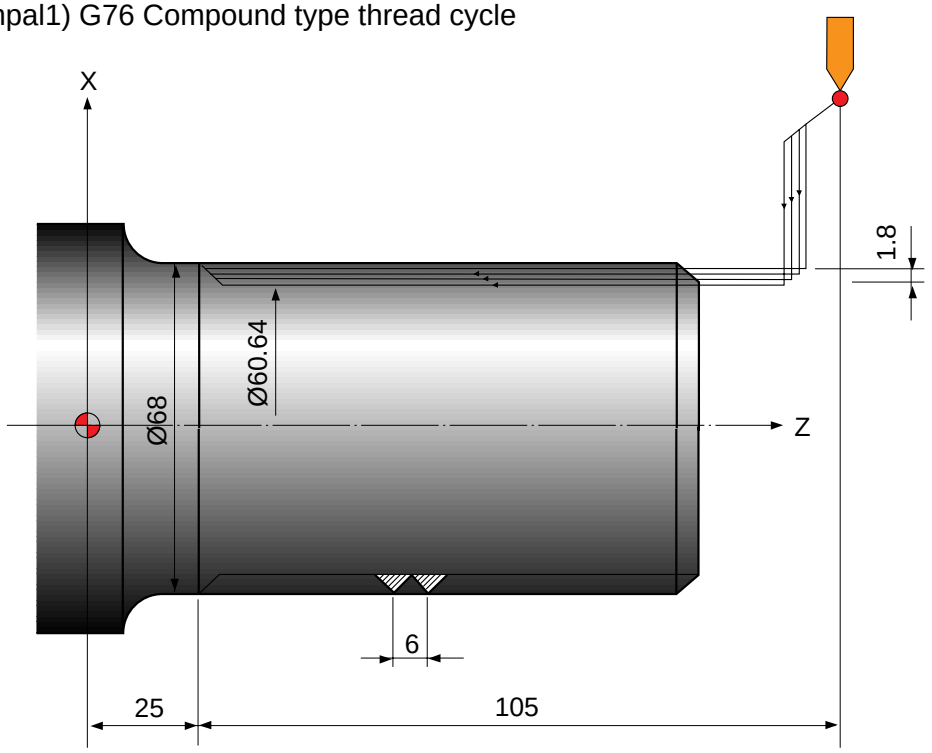
*P(k) : 0.6 x Pitch = Core diameter of thread
- Hikgh value
- Midium value = 0.6
- Low value

(Exampal1) G76 Compound type thread cycle





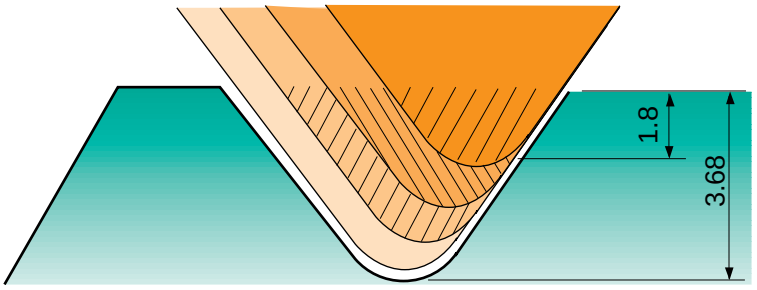
(Exampal1) G76 Compound type thread cycle



G00 X80.0 Z130.0 :

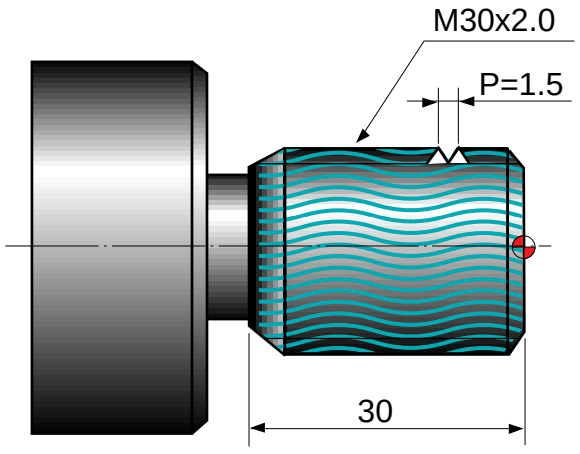
G76 P011060 Q100 R200 :

G76 X60.64 Z25.0 P3680 Q1800 F6.0 ;



PROGRAM

N10 G97 S1000 M03



T0100

G00 X50.0 Z5.0 T0101

G76 P021060 Q100 R100

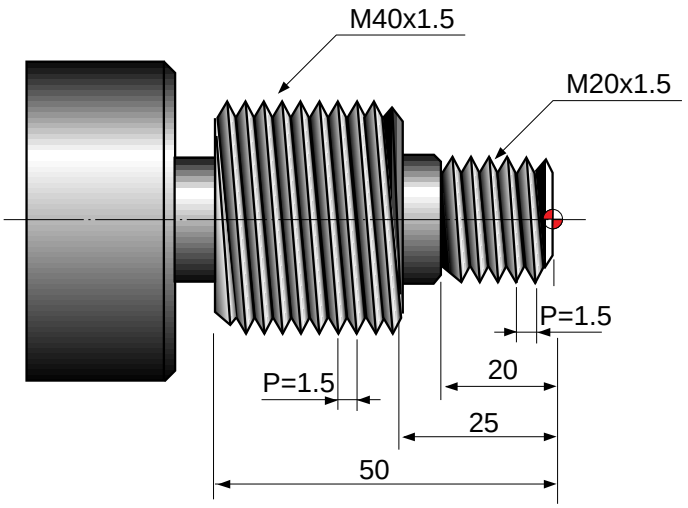
G76 X28.2 Z-32.0 P900 Q500 F1.5

G00 X200.0 Z200.0 T0100

M30

*

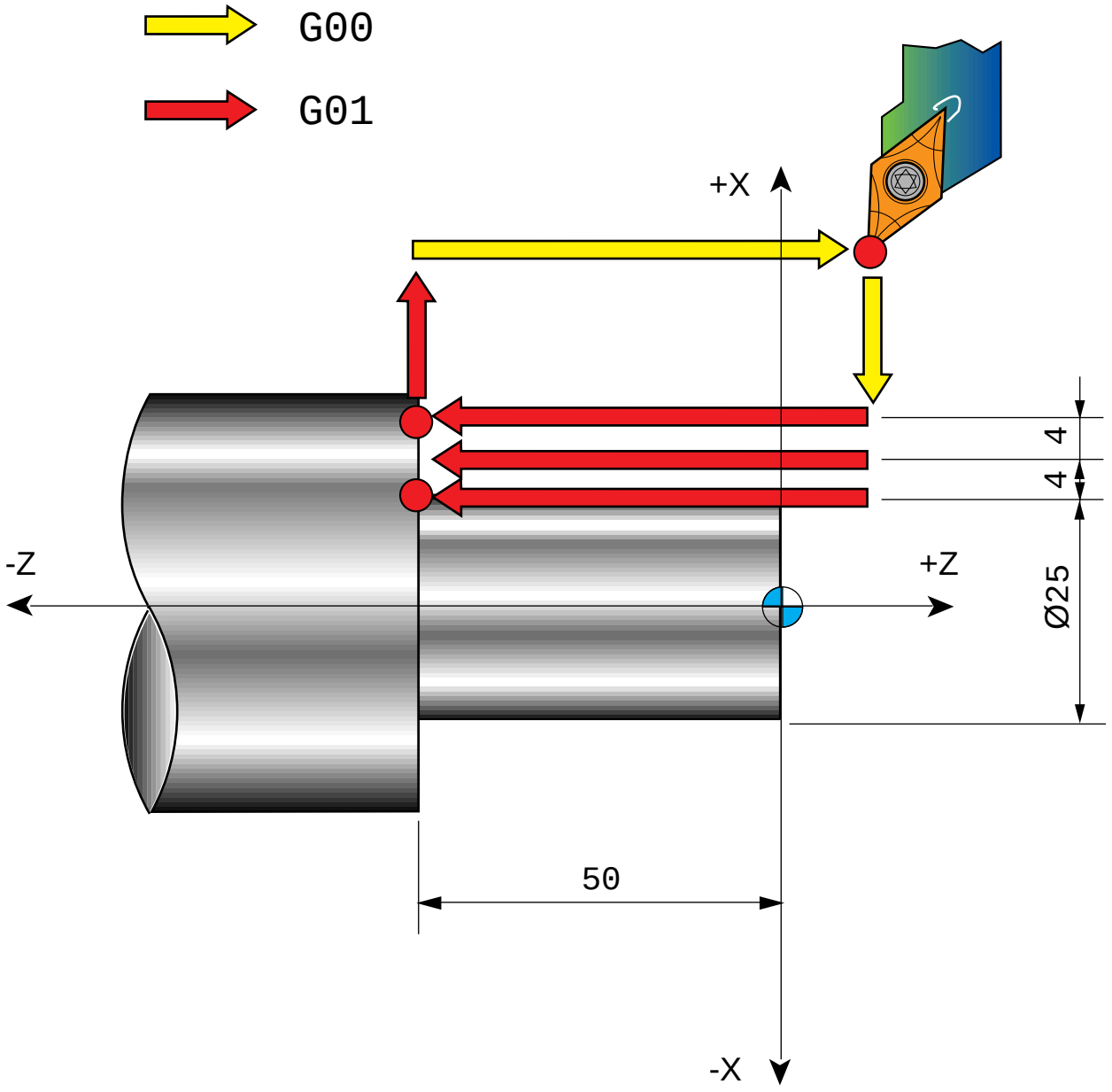
(Exampal1) G76 Compound type thread cycle



PROGRAM

```
N10 G97 S800 M03
    T0300
    G00 X30.0 Z5.0 T0303
    G76 P021060 Q100 R100
    G76 X18.2 Z-20.0 P900 Q500 F1.5
    G00 X50.0 Z-20.0
    G76 P021060 Q100 R100
                          Omissible
    G76 X38.2 Z-52.0 P900 Q500 F1.5
    G00 X200.0 Z200.0 T0300
    M30
    *
```

G90




```
N1234  G90
N1235  G90  X41  Z-50
N1236  U-8
N1237  U-8
```

G90 Fixed cycle

1) Single fixed cycle for cutting

FORMAT G90 X(U) Z(W) _R _F_	Taper cutting

X(U) : X coordinate at the tnd point of Z

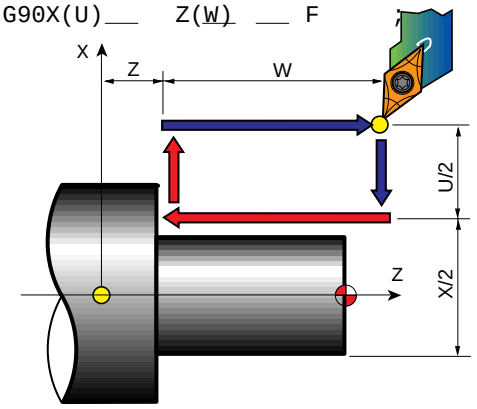
Z(W) : End point

R- : When cutting from the start point to X+ direction

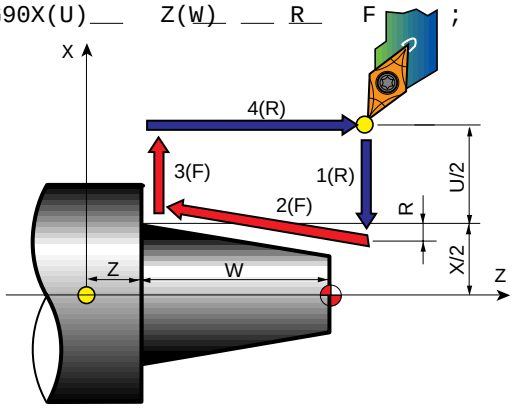
R+ : When cutting from the start point to X- direction

I/R : Inclination(Designate the radius value)

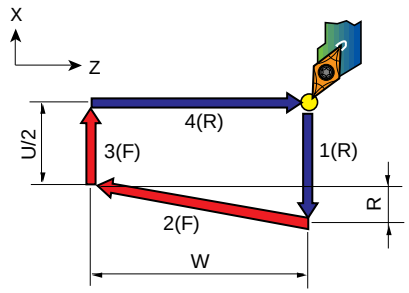
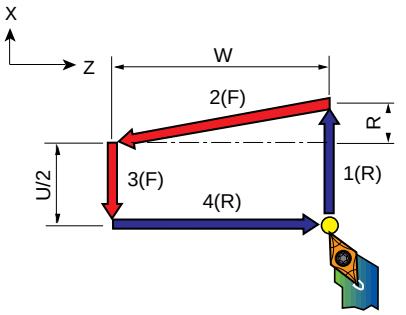
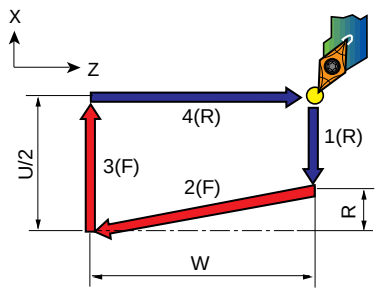
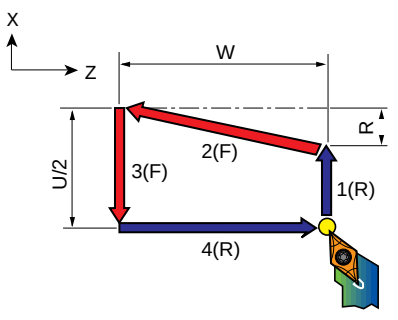
G90X(U)__ Z(W) __ F ;



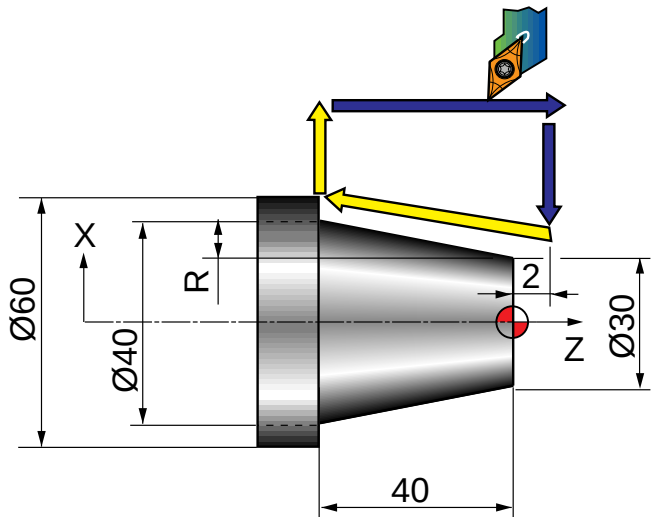
G90X(U)__ Z(W) __ R __ F ;



R... Rapid traverse
F... Cutting traverse specified by F code

1. U<0, W<0, R<0	2. U>0, W<0, R>0
	
3. U<0, W<0, R>0 at $\lvert R \rvert \leqq \frac{\lvert U \rvert}{2}$	4. U>0, W<0, R<0 at $\lvert R \rvert \leqq \frac{\lvert U \rvert}{2}$
	

Exampal1) When the taper is R



PROGRAM

G30 U0 W0 :

G50 S2000 T0100 :

G96 S200 M03 :

G00 X61.0 Z2.0 T0101 M8 :

G90 X55.0 W–42.0 F0.25 :

X50.0 :

X45.0 :

X40.0 :

Z-12.0 R-1.75 :

Z-26.0 R-3.5 :

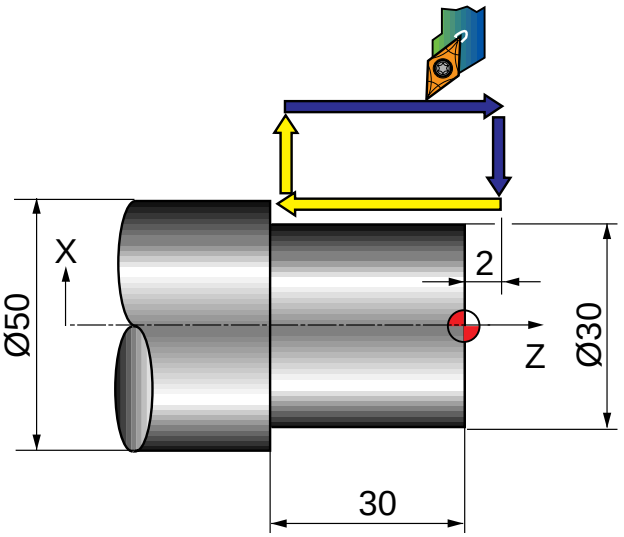
Z-40 R-5.25 :

G30 U0 W0 :

M30 :

*f*T

Example)



PROGRAM

G30 U0 W0 :

G50 S2000 T0100 :

G96 S200 M03 :

G00 X56.0 Z2.0 T0101 M08 :

G90 X51.0 W-32.0 F0.25 :

X46.0 :

X41.0 :

X36.0 :

X31.0 :

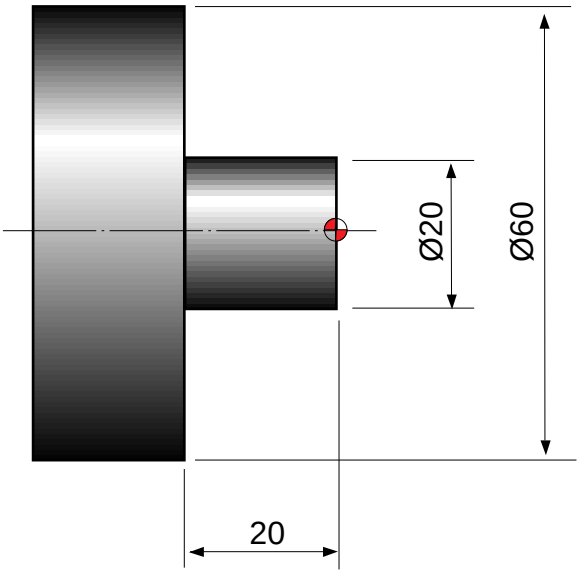
X30.0 :

G30 U0 W0 :

M30 :

When cutting of inside diameter,above format can be used.

(Exampal1) G90 Fixed cycle

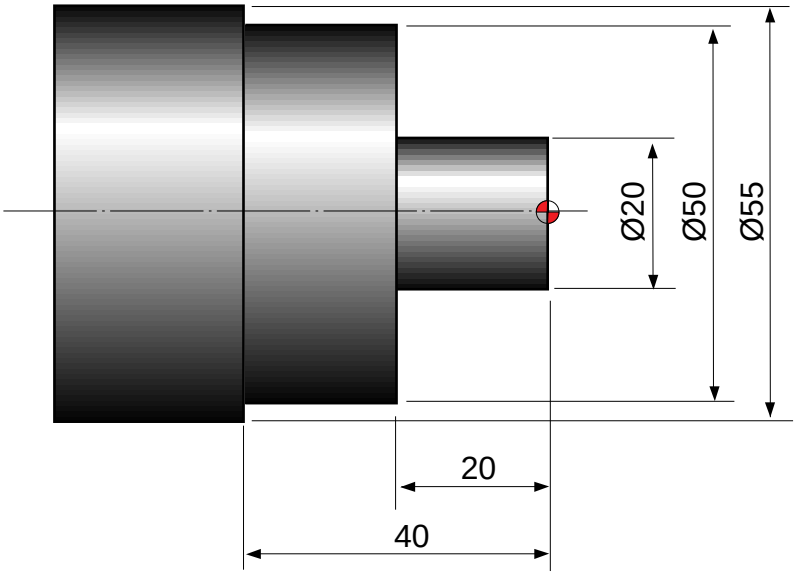


PROGRAM

```
N10 G50 S2000
    G96 S180 M03
    T0100
    G00 X65.0 Z3.0 T0101
    G90 X55.0 Z-20.0 F0.25
        X50.0
        X45.0
        X40.0
        X35.0
        X30.0
        X25.0
        X20.5
        X20.0
    G00 X200.0 Z200.0 T0100
    M30
```

*f*T

(Exampal2) G90 Fixed cycle



PROGRAM

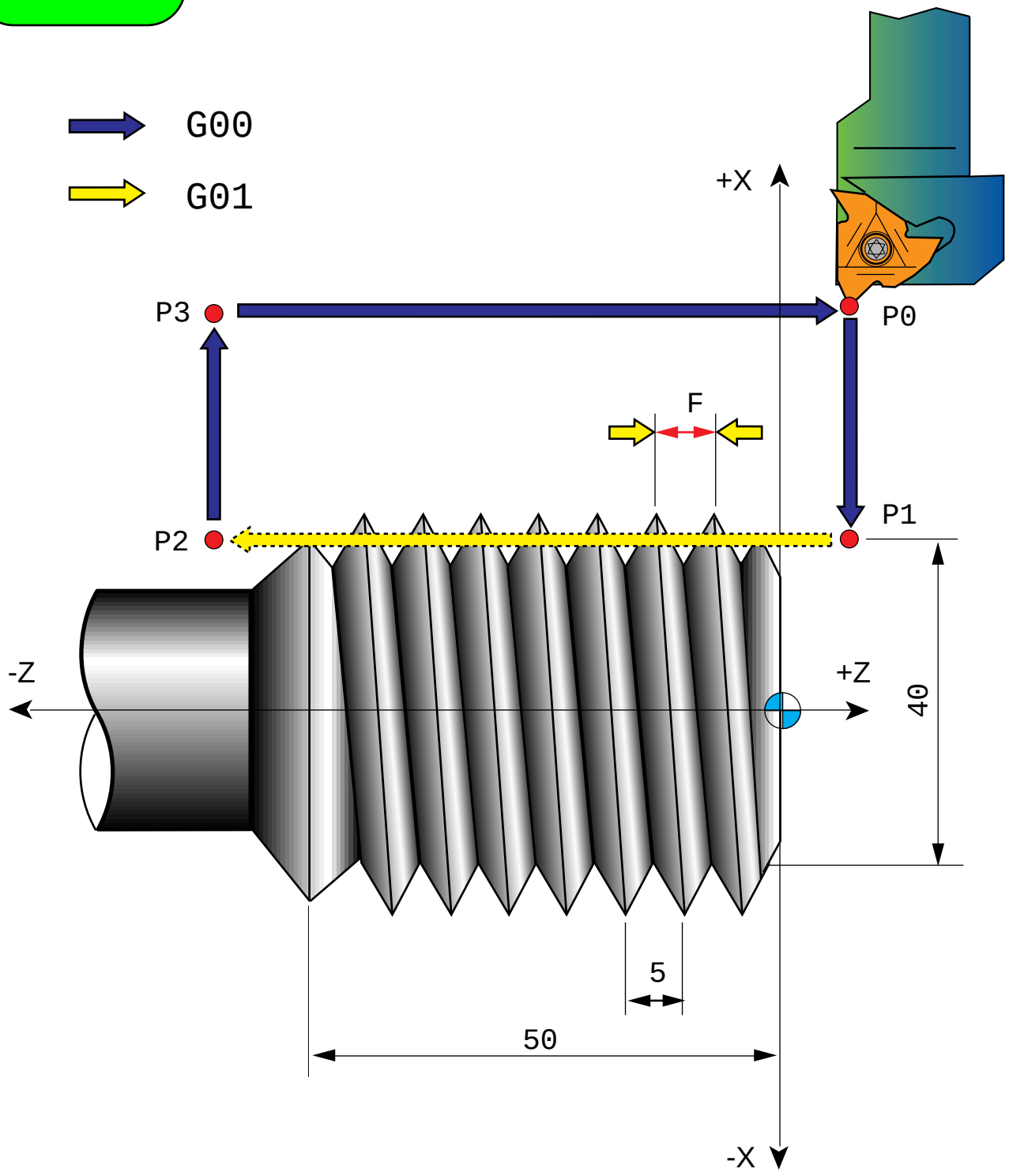
ex1)

```
N10 G50 S2000
    G96 S180 M03
    T0100
    G00 X60.0 Z0 T0101
    G01 X-1.6 F0.2
    G00 X50.0 Z1.0
    G01 Z-40.0 F0.25
    G00 U1.0 Z1.0
    G90 X45.0 Z-20.0 F0.25
        X40.0
        X35.0
        X30.0
        X25.0
        X20.5
        X20.0
    G00 X200.0 Z200.0 T0100
    M30
    fT
```

ex2)

```
N10 G50 S2000
    G96 S180 M3
    T0100
    G0 X60.0 Z5.0 T0101 M8
    G90 X50.0 Z-40.0 F0.25
        X45.0 Z-20.0
        X40.0
        X35.0
        X30.0
        X25.0
        X20.0
    G00 X200.0 Z200.0 T0100
    M30
```

G92




```
N1234  G92  X40.  Z-55.  F5.
```

G92 Fixed cycle

1) Single fixed cycle for cutting

FORMAT	G92 X(U) Z(W) _R_F_

X(U) : X axis coordinate of thread process position of each time

Z(W) : End point

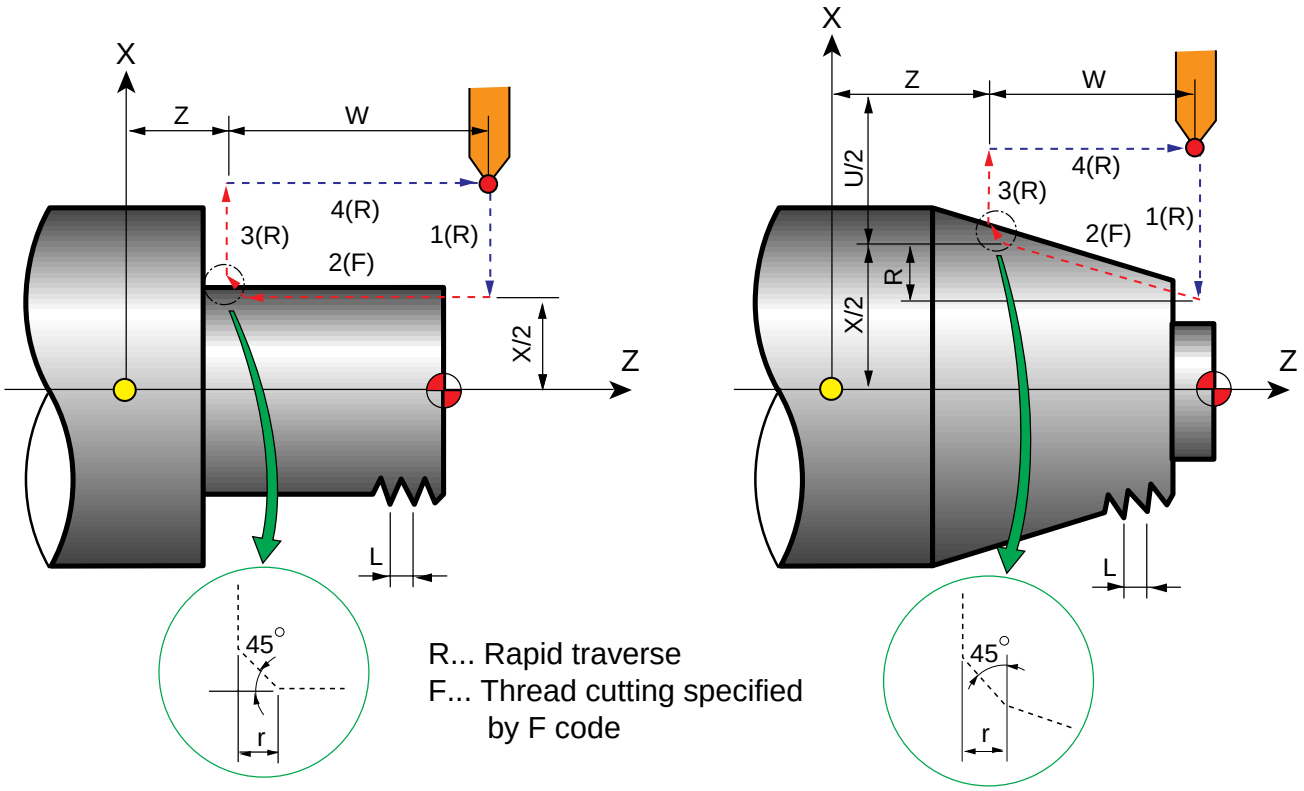
R- : When cutting form the start point to X+ direction.

R+ : When cutting from the start point to X- direction.

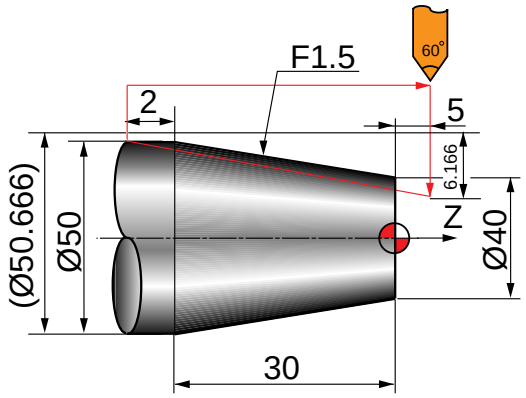
I/R : Lead(pitch)

Note) Spindle override and feedrate override of cycle distance are disregarded.

```
G92x(U)___  Z(W) ___ F__   ; Lead(L) is specified      G92x(U)_ Z(W)_ F_    ;
```



Exampal1) When the taper is R



PROGRAM

G30 U0 W0 :

G50 S1000 T0100 :

G97 S1000 M03 :

G00 X70.0 Z5.0 T0101 M08 :

G92 X49.4 Z–32.0 R–6.166 F1.5 :

X49.0 :

X48.7 :

X48.5 :

-

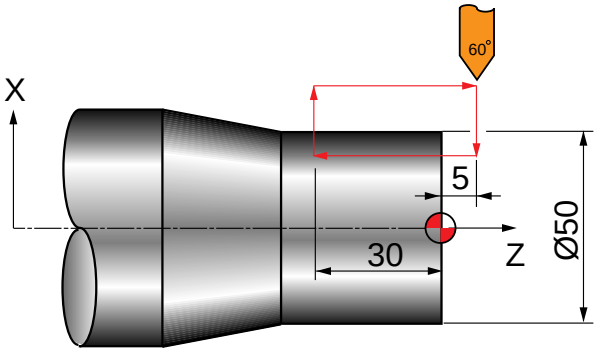
-

G30 U0 W0 :

M30 :

ƒT

Example) M50 x 1.5



PROGRAM

G30 U0 W0 :

G50 S1000 T0100 :

G97 S1000 M03 :

G00 X60.0 Z5.0 T0101 M08 :

G92 X49.5 Z–30.0 F1.5 :

X49.2 :

X48.9 :

X48.7 :

-

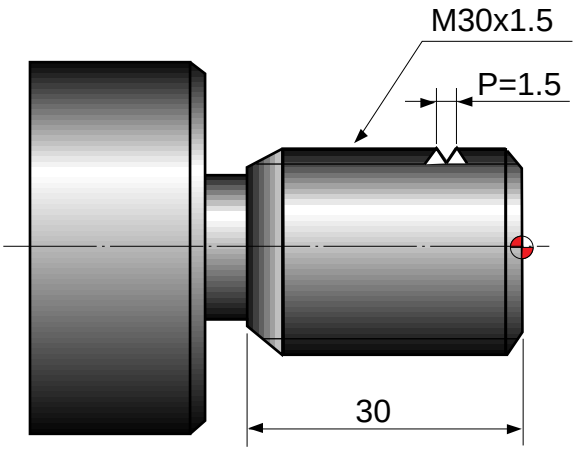
-

G30 U0 W0 :

M30 :

ƒT

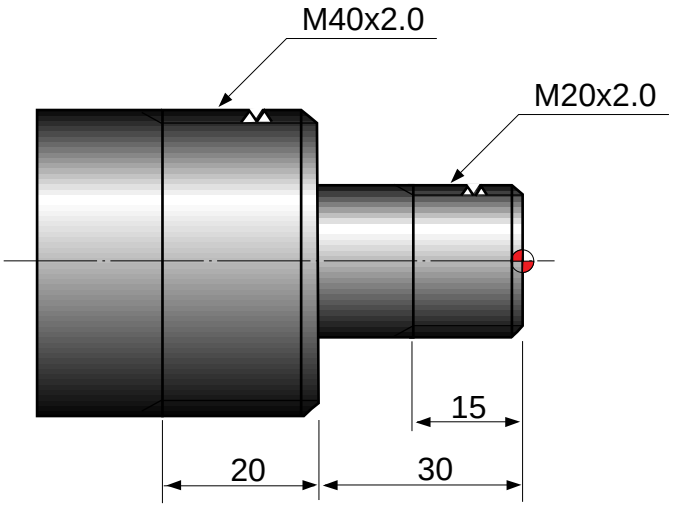
(Exampal1) G90 Fixed cycle



PROGRAM

```
N10 G97 S1000 M03
    T0300
    G00 X35.0 Z5.0 T0303
    G92 X29.5 Z-32.0 F1.5
        X29.2
        X28.9
        X28.7
        :
    G00 X200.0 Z200.0 T0300
    M30
    ƒT
```

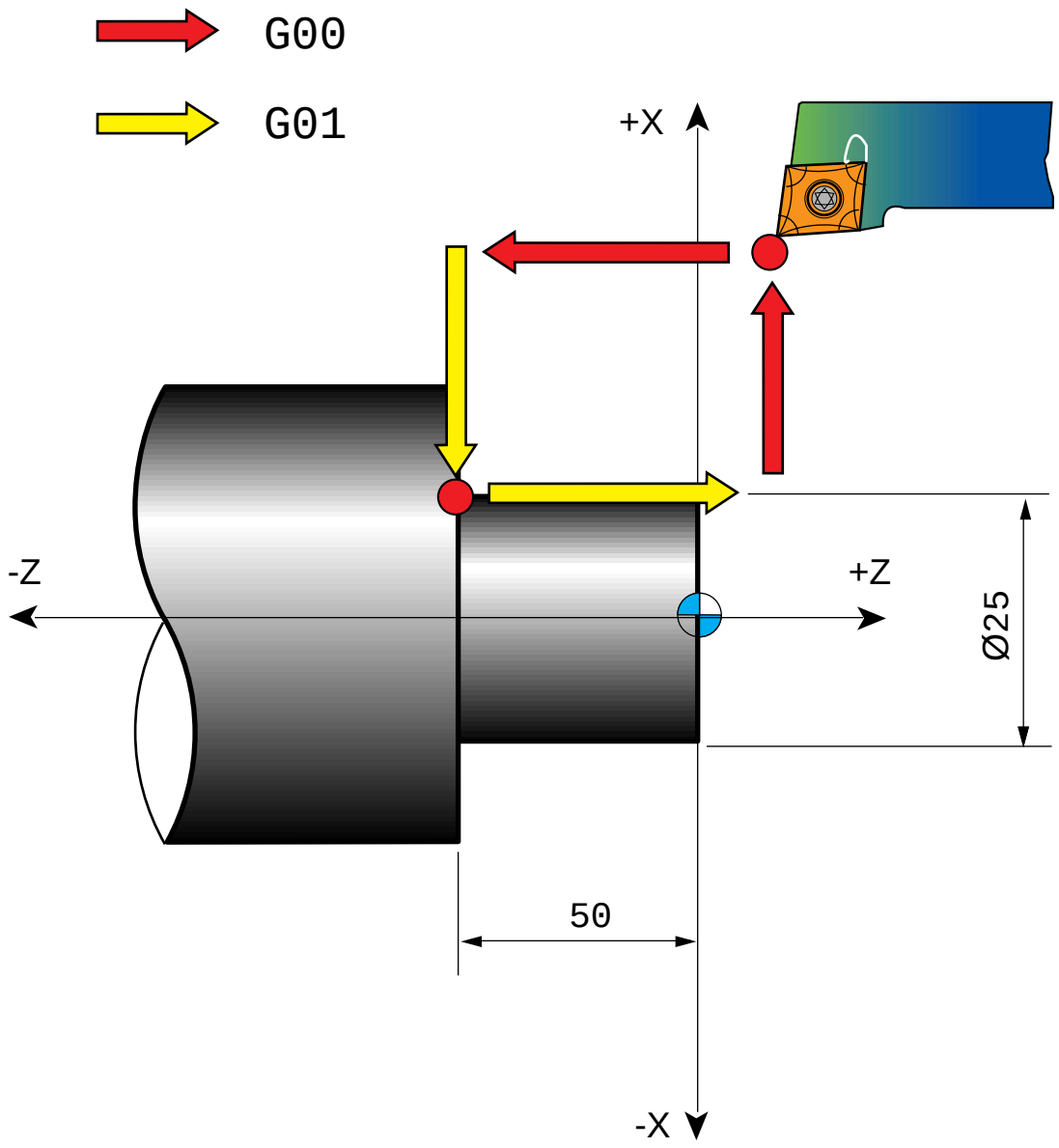
(Exampal2) G92 thread cycle



PROGRAM

```
N10 G97 S1500 M03
    T0300
    G00 X30.0 Z5.0 T0303
    G92 X19.5 Z-15.0 F2.0
        X19.2
        X18.9
        X18.6
        X18.4
        :
    G00 X50.0
        Z-25.0 S1000
    G92 X39.5 Z-50.0 F2.0
        X39.2
        X38.9
        X38.6
        X38.4
    G00 X200.0 Z200.0 T0300
    M30
    *
```

G94




```
N1234  G94  X25.  Z-50.
```

G94 (Stock vemoval cycle in facing)

FORMAT	G92 X(U) Z(W)_R_F_

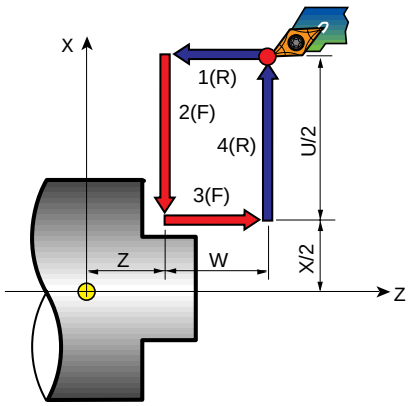
X(U) : End point

Z(W) : (End point of inclination)= ⓐ point of cycle distance

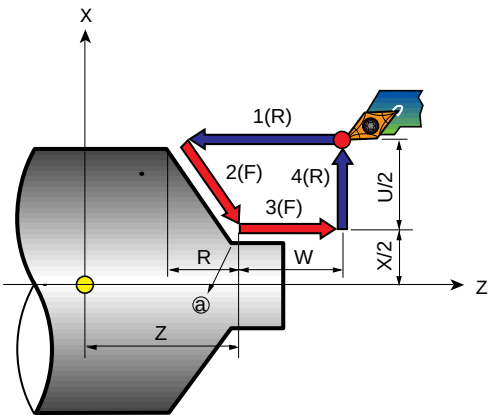
R- : program the veal inclined value.

F : Cutting feedrate

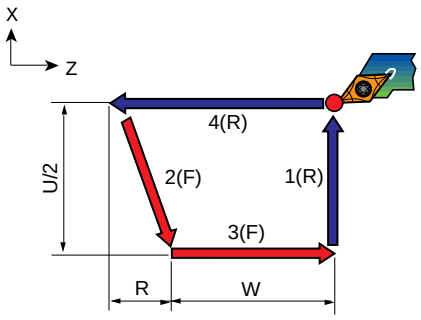
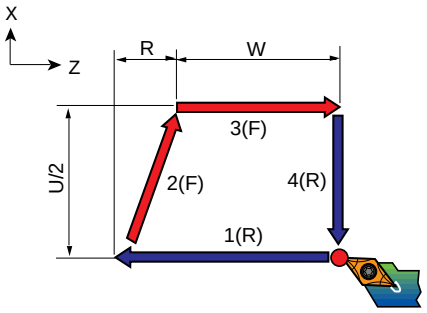
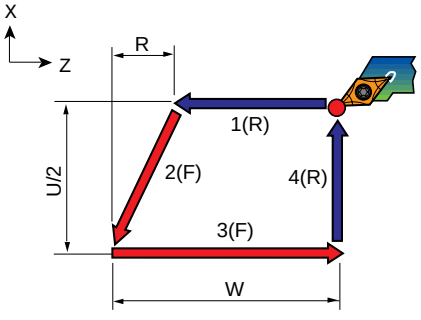
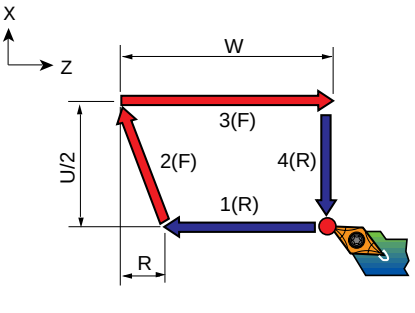
G94X(U)__ Z(W)__ __ F ;



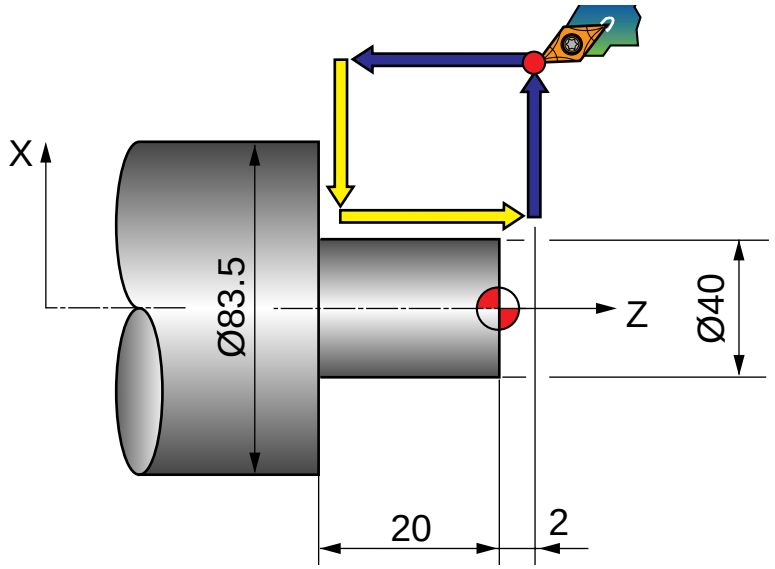
G90X(U)__ Z(W)__ __ R__ F ;



R... Rapid traverse
F... Cutting traverse specified by F code

1. U<0, W<0, R<0	2. U>0, W<0, R<0
	
3. U<0, W<0, R>0 at |R|≦|W|	4. U>0, W<0, R<0 at |R|≦|W|
	

Exampal)



PROGRAM

G30 U0 W0 :

G50 S2000 T0100 :

G96 S200 M03 :

G00 X85.0 Z2.0 T0101 M08 :

G94 X40.0 Z–2.0 F0.2

Z–4.0 :

Z–6.0 :

Z–8.0 :

Z–10.0 :

Z–12.0 :

Z–14.0 :

Z–16.0 :

Z–18.0 :

Z-19.7 :

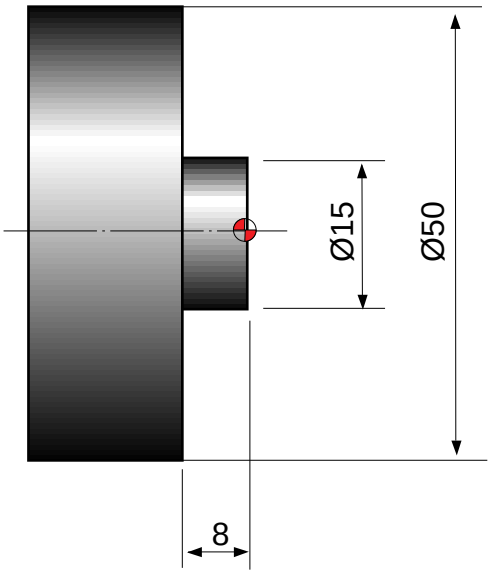
Z–20.0 :

G30 U0 W0 :

M30 :

*

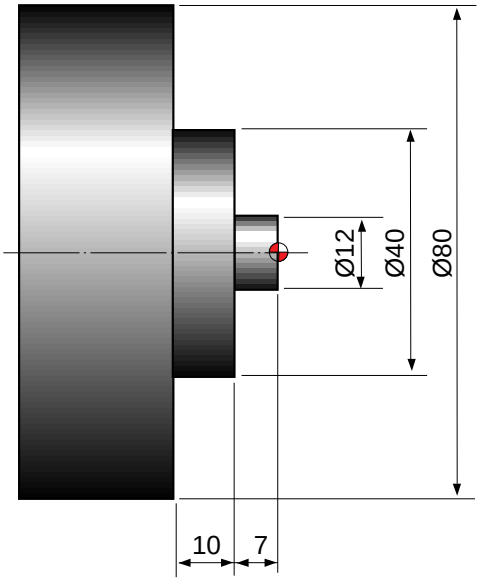
(Exampal 1) G94 Stock removal cycle in facing



PROGRAM

```
N10 G50 S2500
    G96 S180 M03
    T0100
    G00 X55.0 Z2.0 T0101
    G94 X15.0 Z-2.0 F0.2
        Z-4.0
        Z-6.0
        Z-8.0
    G00 X200.0 Z200.0 T0100
    M30
    *
```

(Exampal 2) G94 Stock removal cycle in facing



PROGRAM

ex1)

```
N10 G50 S2500 :
    G96 S180 M03 :
    T0300 :
    G00 X85.0 Z2.0 T0303 :
    G94 X12.0 Z-2.0 F0.2 :
        Z-4.0 :
        Z-6.0 :
        Z-7.0 :
    G00 X85.0 Z-5.0 :
    G94 X40.0 Z-9.0 F0.2 :
        Z-11.0 :
        Z-13.0 :
        Z-15.0 :
        Z-17.0 :
    G00 X200.0 Z200.0 T0300 :
    M30 :
    *
```

ex2)

```
N10 G50 S2500 :
    G96 S180 M3 :
    T0300 :
    G0 X85.0 Z2.0 T0303 :
    G94 X12.0 Z-2.0 F0.2 :
        Z-4.0 :
        Z-6.0 :
        Z-7.0 :
      X 40.0 Z-9.0 :
        Z-11.0 :
        Z-13.0 :
        Z-15.0 :
        Z-17.0 :
    G0 X200.0 Z200.0 T0300 :
    M30 :
    *
```

G96, G97(Constant travelling speed control ON, OFF)

G Code	Constant travelling speed control	Meaning	Unit
G 96	ON	To control the travelling speed constantly	m/min
G 97	OFF	Designate the rotating time of main spindle	rpm

Example) G96 S100 :

Cutting speed is 100m/min

G97 S100 :

Rotating time of main spindle is 100rpm

G98, G99(Feedrate selection)

G GODE	Meaing	Unit
G 98	Feedrate per minute	mm/min
G 97	Feedrate per rotation	mm/rev

Example) G98 G01 Z100.0 F50.0 :

Feedrate of tool is 50mm per minute.

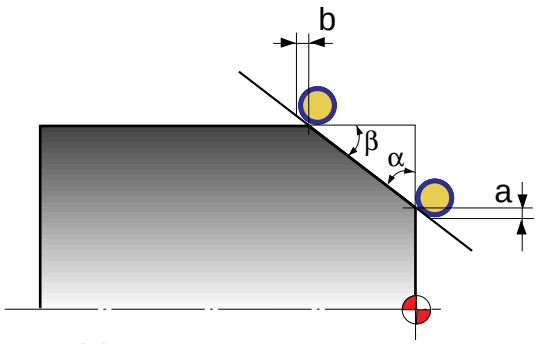
G97 G01 Z10.0 F0.3 :

Feedrate of tool is 0.3mm per rotation of main spindle.

However, unless there is the G98 command, N.C unit is always in G99 condition.

Therefor it is not necessary to command G99 seperately.

<Calculation formular of bite noser>



∗ Calculation formular of compensation volume

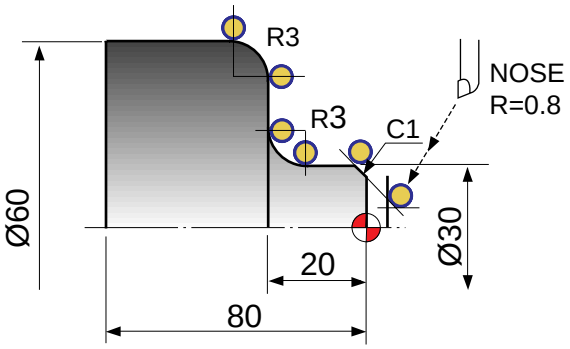
$a = r(1-\tan\frac{\alpha}{2})$

$b = r(1-\tan\frac{\beta}{2})$

r = Rvalue of bite

Bite Nose	a	b
0.4	0.468	0.234
0.8	0.937	0.468

Example)



O0035 :

N10 G50 S1500 T0100 :

N20 G50 S2000 T0303 :

G96 S180 M03 :

G00 X35.0 Z5.0 M08 :

Z0:

G01 X-1.6 F0.2 :

G00 X25.063 Z1.0 :

G01 X30.0 Z-1.468 F0.17 :

Z-17.8 :

G02 X34.4 Z-20.0 R2.2 :

G01 X52.4 :

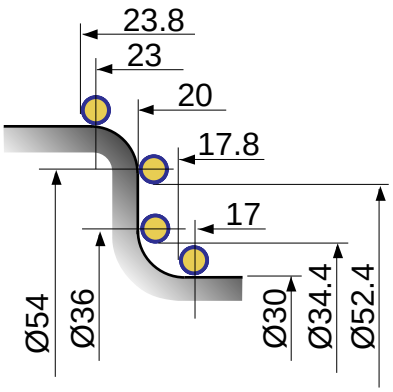
G03 X60.0 Z-23.8 R3.8 :

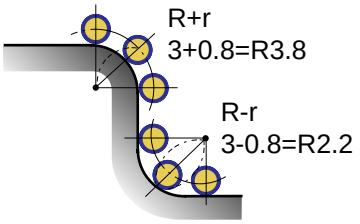
G01 Z-80.0 :

G00 X150.0 Z150.0 :

T0300 :

M30 :





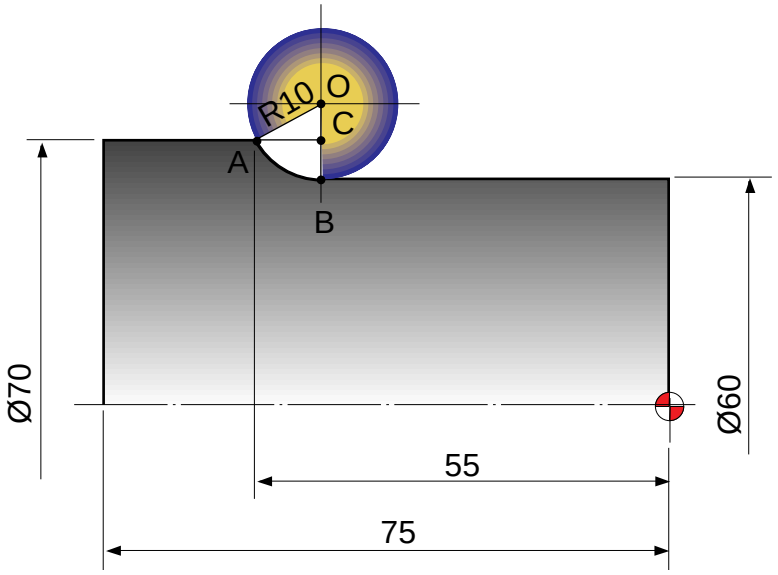
Concave R = R–r

Convex R = R+r

R : Circumference R

r : Bite r

Example) PROGRAM



$\overline{CB} = (70 - 60) \div 2 = 5$

$\overline{OC} = R10 - 5 = 5$

$\overline{AO} = 10$

$\overline{AC} = \sqrt{(\overline{AO})^2 - (\overline{OC})^2}\ 28.66$

$55 - 8.66 = 46.34$

G00 X60.0 Z3.0 :

G42 Z1.0 :

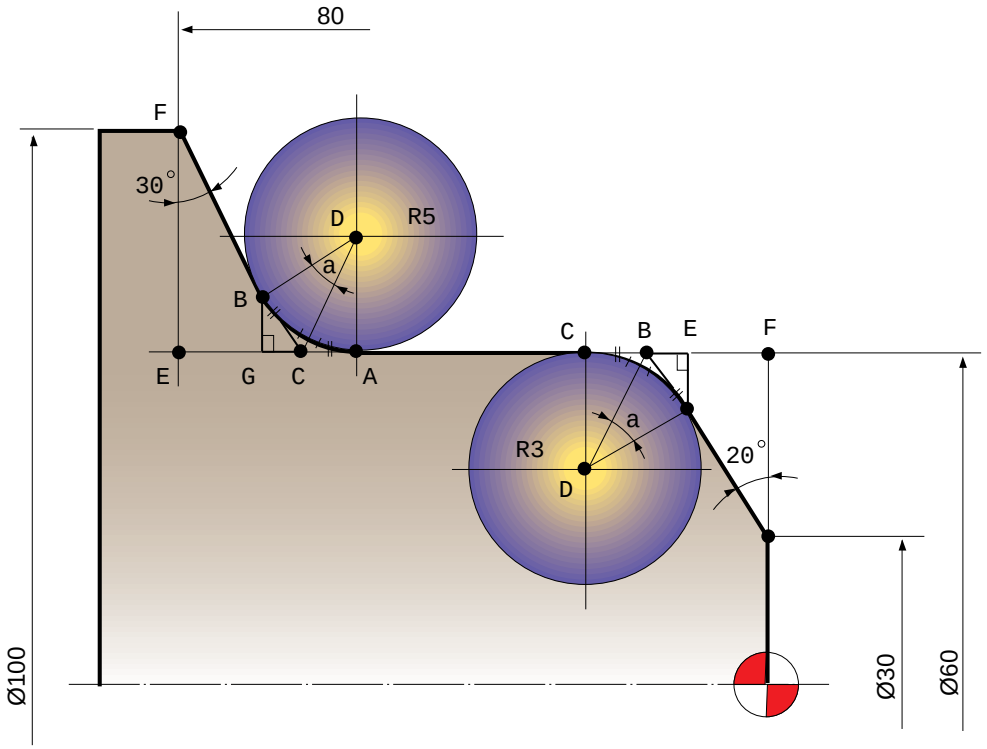
G01 Z-46.34 F0.23 :

G02 X70.0 Z-55.0 R10.0 :

I10.0

G01 Z-75.0

Example) PROGRAM



$\overline{EF}$ = (100 – 60) ÷ 2 = 20

$\overline{OC}$ = 20 x 30 tan = 11.547

α = (180 – 60) ÷ 2 = 60°

$\overline{AC}$ = BC

$\overline{AC}$ = 2.887 x 60° sin = 2.5

2.887 x 30° cos = 2.5

∗ X ¡ 2.5 x 2 = 5

$\overline{CG}$ = 2.887 x 30° sin = 1.444

2.887 x 60° cos = 1.444

♠ Coordinate value

A ¡ X = 60

Z = 80 – ($\overline{CE}$ – $\overline{AC}$) = 65.566

B ¡ X = 60 + $\overline{BG}$ = 65

Z = 68.453 + 1.444 = 69.897

A ¡ X = R5 = 5

Z = 0

$\overline{BF}$ = 20° tan x 15 = 5.45955

α = (180 – 70) ÷ 2 = 55°

$\overline{BC}$ = 3 x 35° tan = 2.1

$\overline{AC}$ = $\overline{AB}$

$\overline{AE}$ = 2.1 x 70° sin = 1.973

∗ X ¡ 1.973 x 2 = 3.947

♠ Coordinate value

A ¡ X = 60 – 3.947 = 56.053

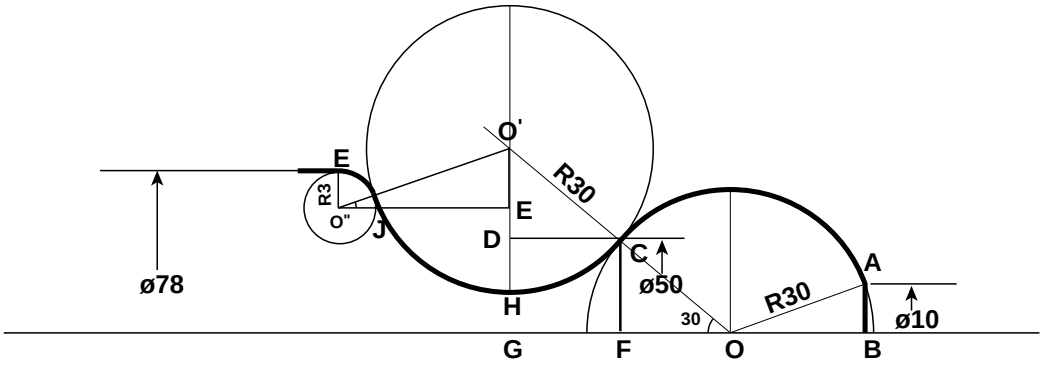
Z = 5.459 – 0.718 = 4.741

C ¡ X = 60

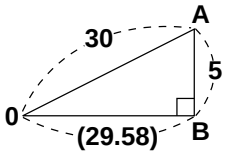
Z = 5.459 + 2.1 = 7.559

D ¡ X = R3 – $\overline{AE}$ ¡ 3 – 1.973 = 2.054

Z = $\overline{BE}$ + $\overline{BC}$ ¡ 2.1 + 0.718 = 2.816

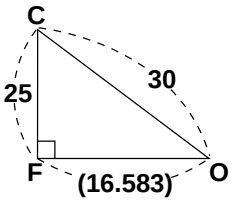


1)



$(\overline{OB})^2 = (\overline{OA})^2 - (\overline{AB})^2 = (30)^2 - (5)^2 = \sqrt{875} = 29.58$

2)

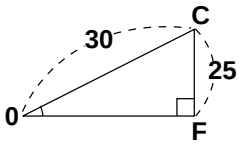


$\overline{OC} = 30,\ \overline{CF} = 25 \quad \therefore \sqrt{\overline{OF}} = (50)^2 - (25)^2 = 16.583$

$\angle COF = SIN \angle COF = \frac{25}{50} = 56.442^{\circ}$

$\therefore \angle COF = \angle O'CD$

$\therefore \triangle COF = \triangle O'CD$



$\overline{CF} = \overline{O'D} \qquad \therefore \overline{O'D} = 25$

$\overline{DH} = \overline{O'H} - \overline{O'D} = 30 - 25 = 5 \qquad 6D = 25$

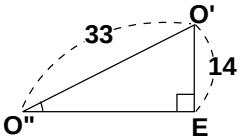
O' of X 50 + 25 + 25 = 100

O' of Z $\overline{OB} + \overline{OF} + \overline{CD} = 29.58 + 16.383 + 16.583 = 62.746$

O" of X 78 - 6 = 72

$\therefore \overline{O'E} = \frac{(100-72)}{2} = 14$ $\qquad \overline{O''O'} = 3 + 30 = 33$

$O'E = 14$



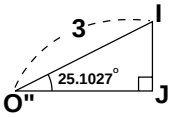
$\sqrt{\overline{O''E}} = 33^2 - 14^2 = 29.883$

$\therefore$ E of Z 62.746 + 29.883 = 92.629

$\therefore$ I of X 72 + 1.2727 + 1.2727 = 74.5454

$\therefore$ I of Z 92.629 - 2.7166 = 89.9124

$SIN \angle O'O''E = \frac{14}{33} = 25.1027^{\circ}$

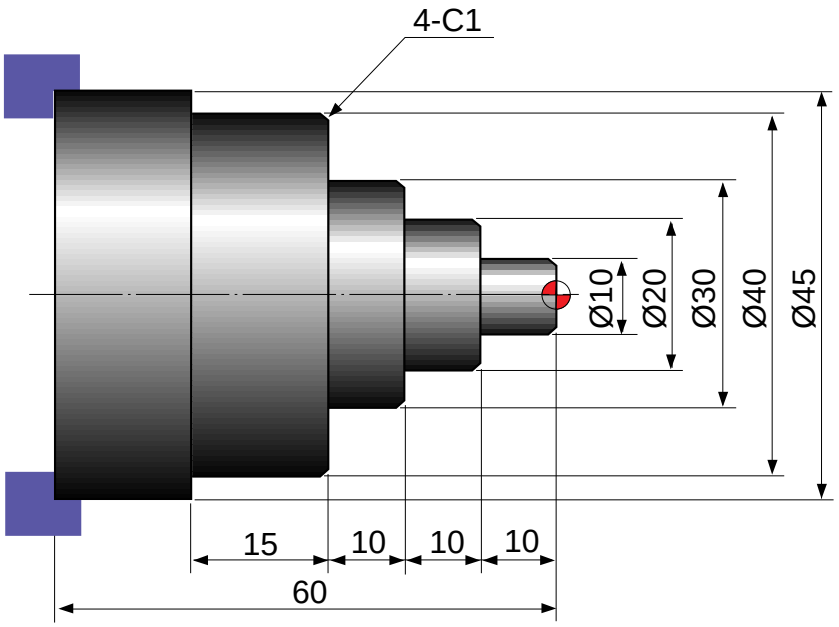


$\overline{IJ} = SIN\ 25.1027 \times 3 = 1.2727$

$\overline{O''J} = COS\ 25.1027 \times 3 = 2.7166$

(Example 1)

Process	Facing process, Outside diameter process
Dimension	ø 45 x 60L
Material	S45C



Condition of using tool

Facing process

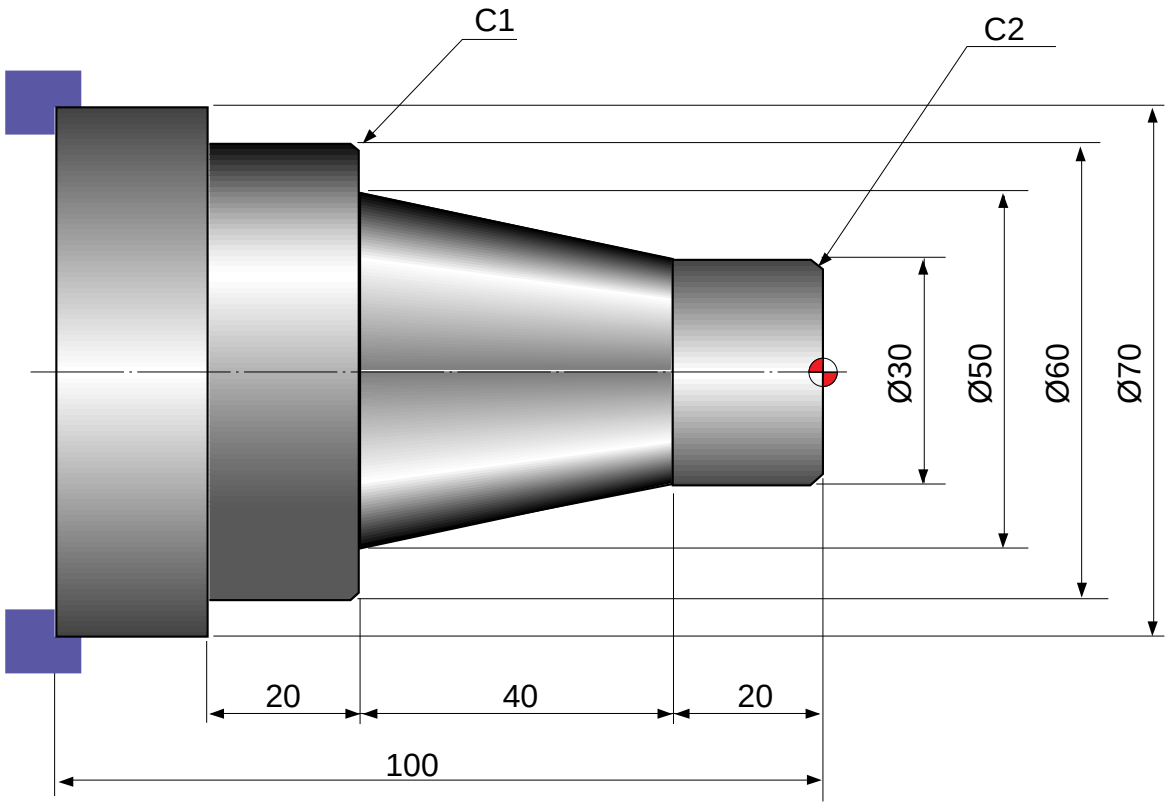
TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

(Example 2)

Process	Facing process, Outside diameter taperprocess
Dimension	ø 70 x 100L
Material	S45C



Condition of using tool

Facing process

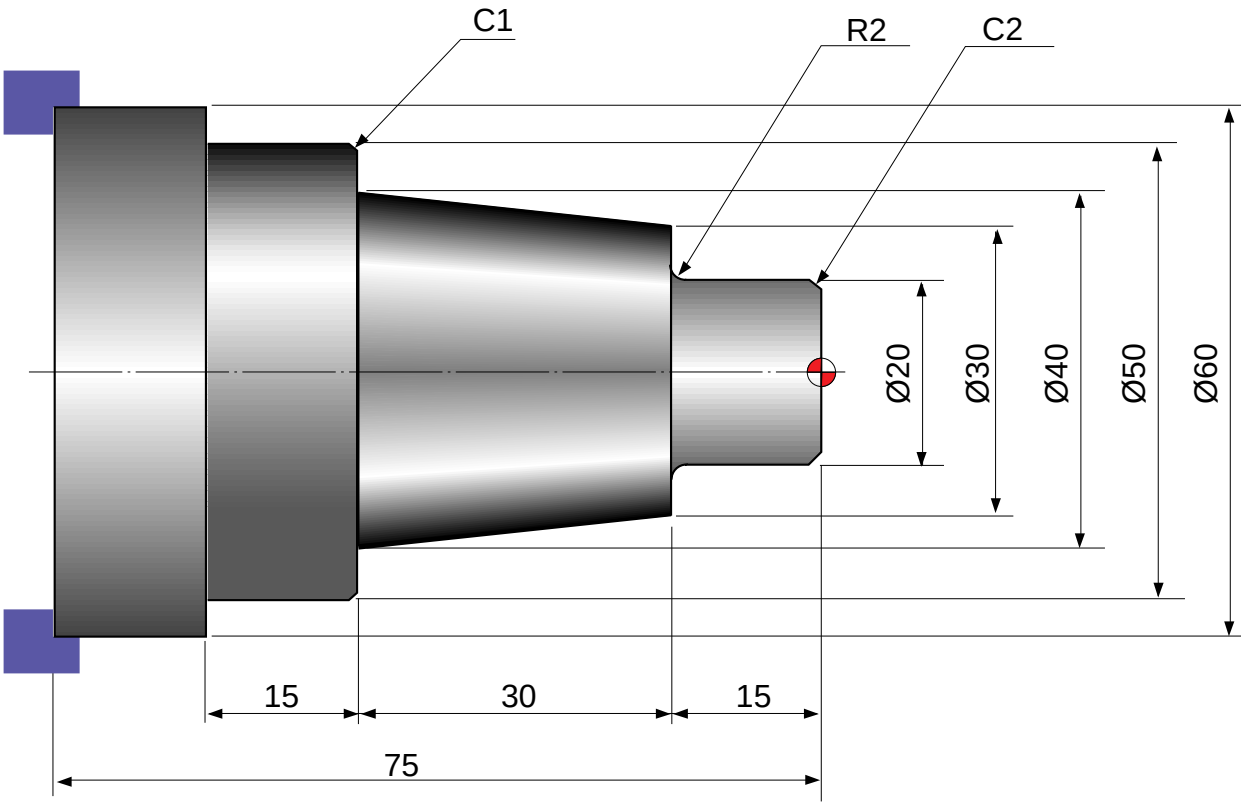
TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

(Example3)

Process	Facing process, Outside diameter taper process(Chamfering, R process)
Dimension	ø 60 x 75L
Material	S45C



Condition of using tool

Facing process

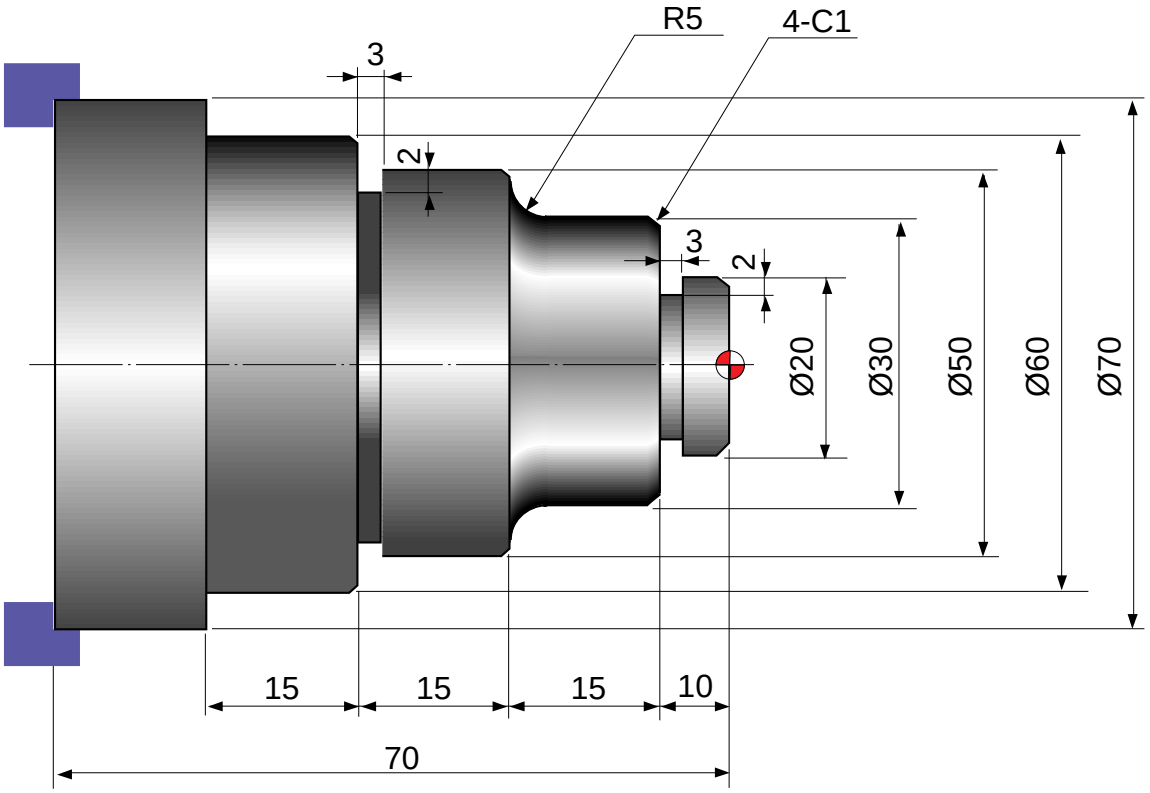
TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

(Example4)

Process	Facing process, Outside diameter(Groove process, Chamfering R process)
Dimension	ø 70 x 70L
Material	S45C



Condition of using tool

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

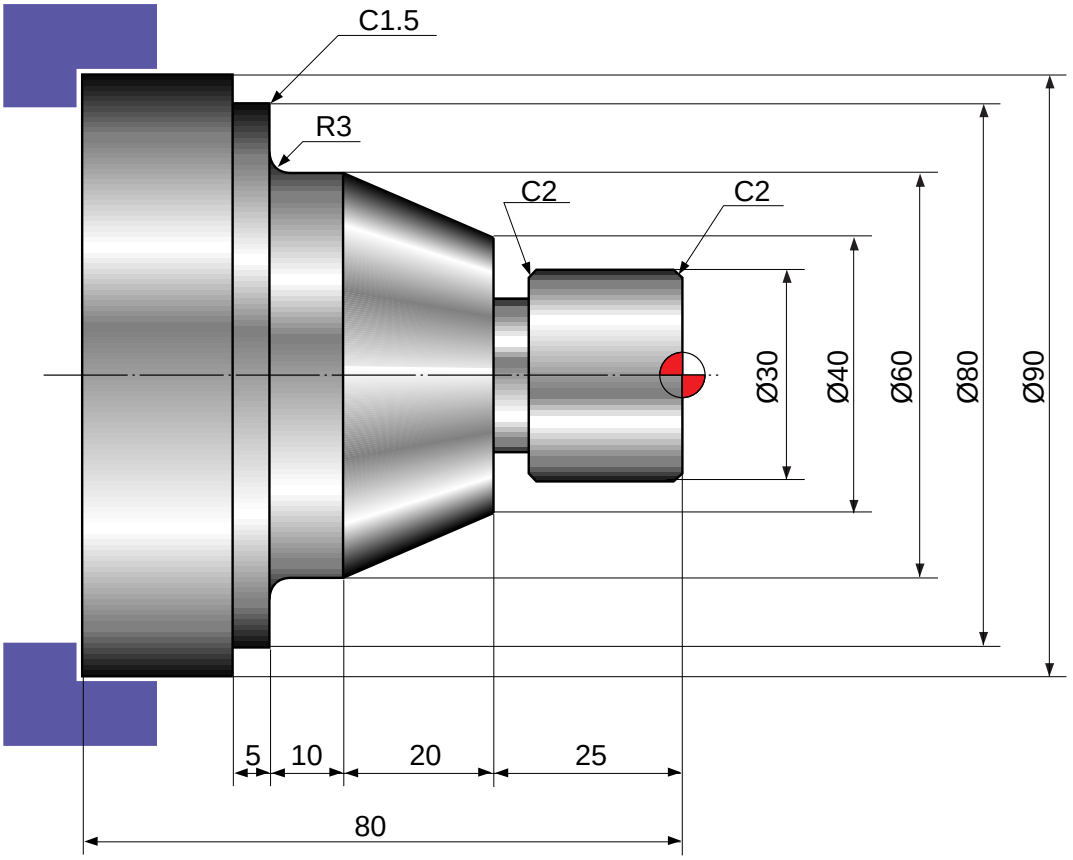
TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Groove process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal + Finishing
PCLNR/L–1	

(Example5)

Process	Facing process, Outside diameter(Groove process, Chamfering R process, Thread process)
Dimension	ø 90 x 80L
Material	S45C



Condition of using tool

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Groove process

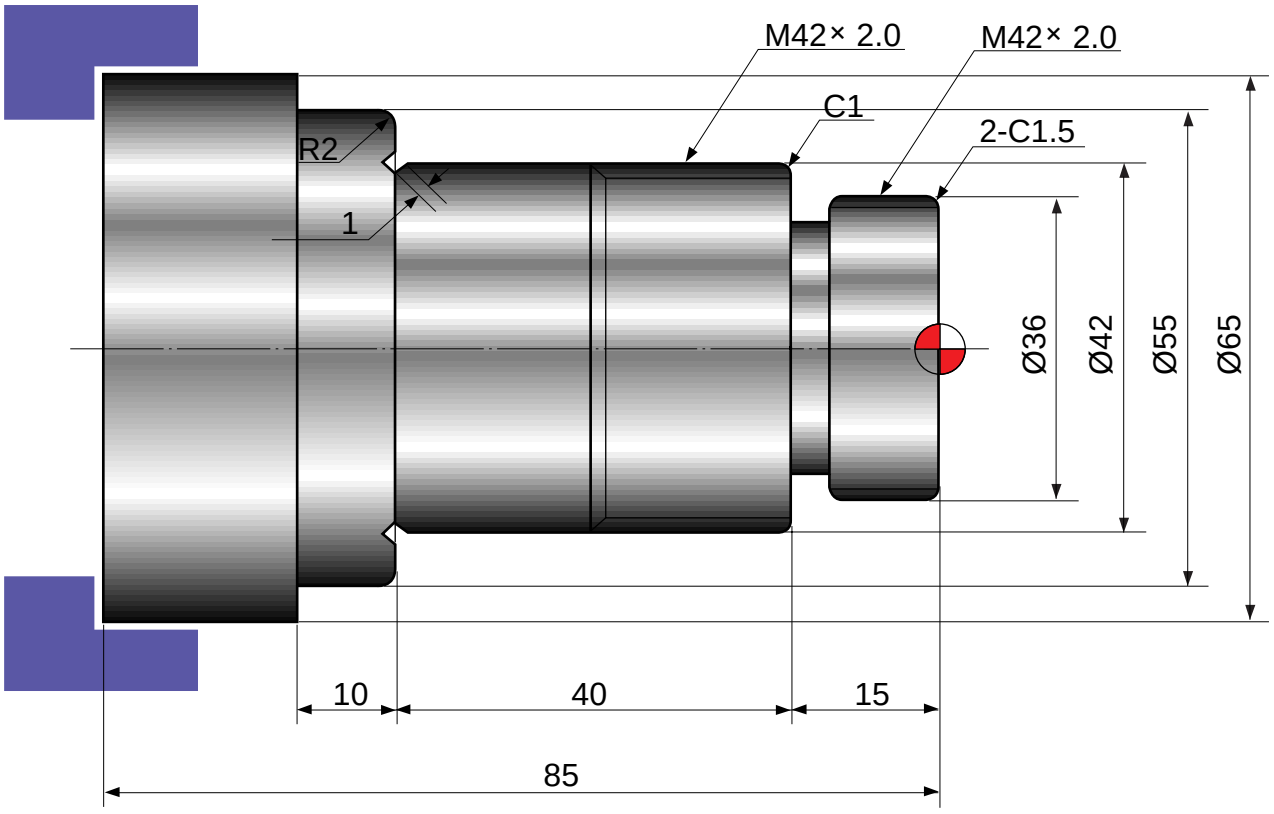
TOOL	PROCESS TYPE
R/L 154.91	Stock remova + Finishing

Thread process

TOOL	PROCESS TYPE
R/L 166.0	Stock remova + Finishing

(Example6)

Process	Facing process, Outside diameter(Groove process, Thread process, Relief)
Dimension	ø 65 x 88L
Material	S45C



Condition of using tool

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Groove process

TOOL	PROCESS TYPE
R/L 154.91	Stock removal + Finishing

Facing process

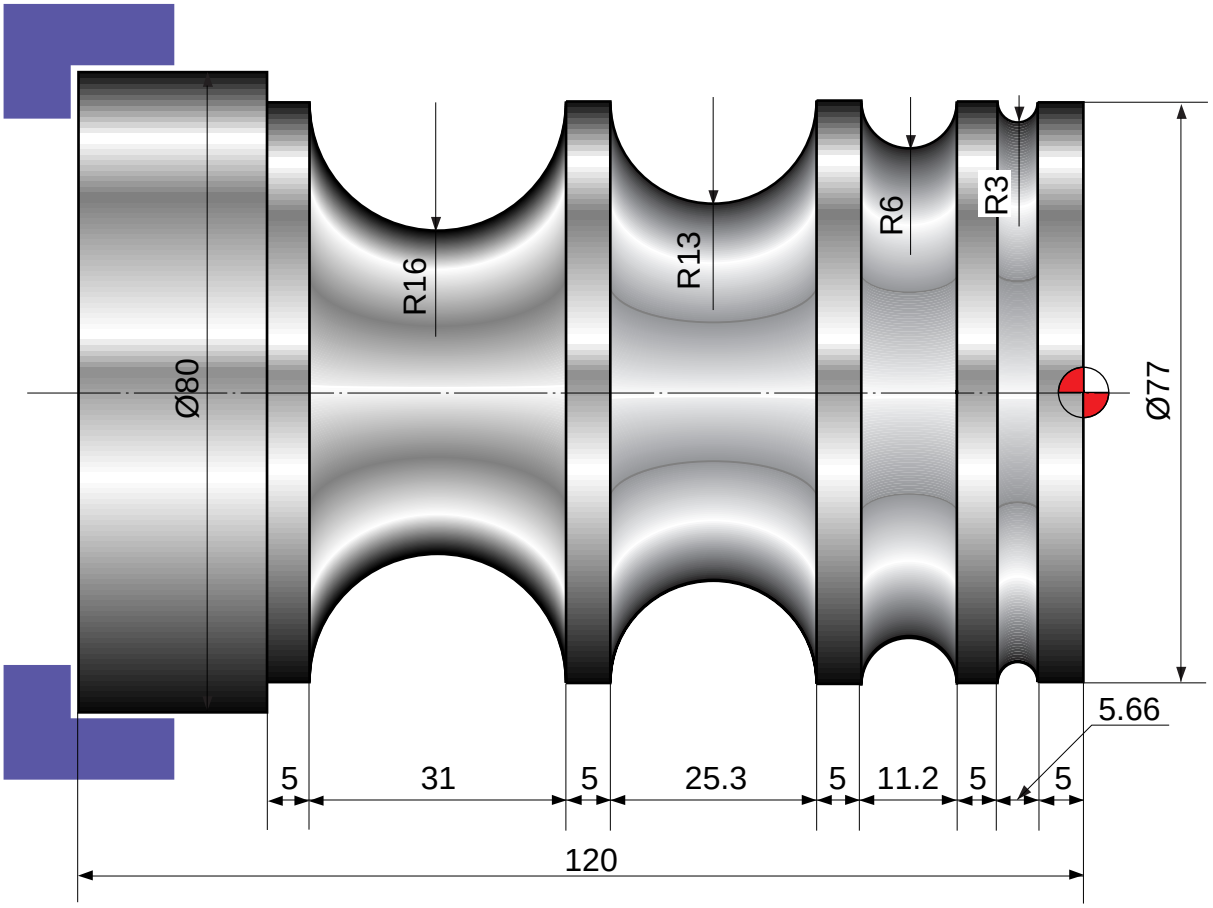
TOOL	PROCESS TYPE
Relief	Stock remova + Finishing

Thread process

TOOL	PROCESS TYPE
R/L 166.0	Stock remova + Finishing

(Example7)

Process	Outside diameter R process
Dimension	ø 80 x 120L
Material	S45C



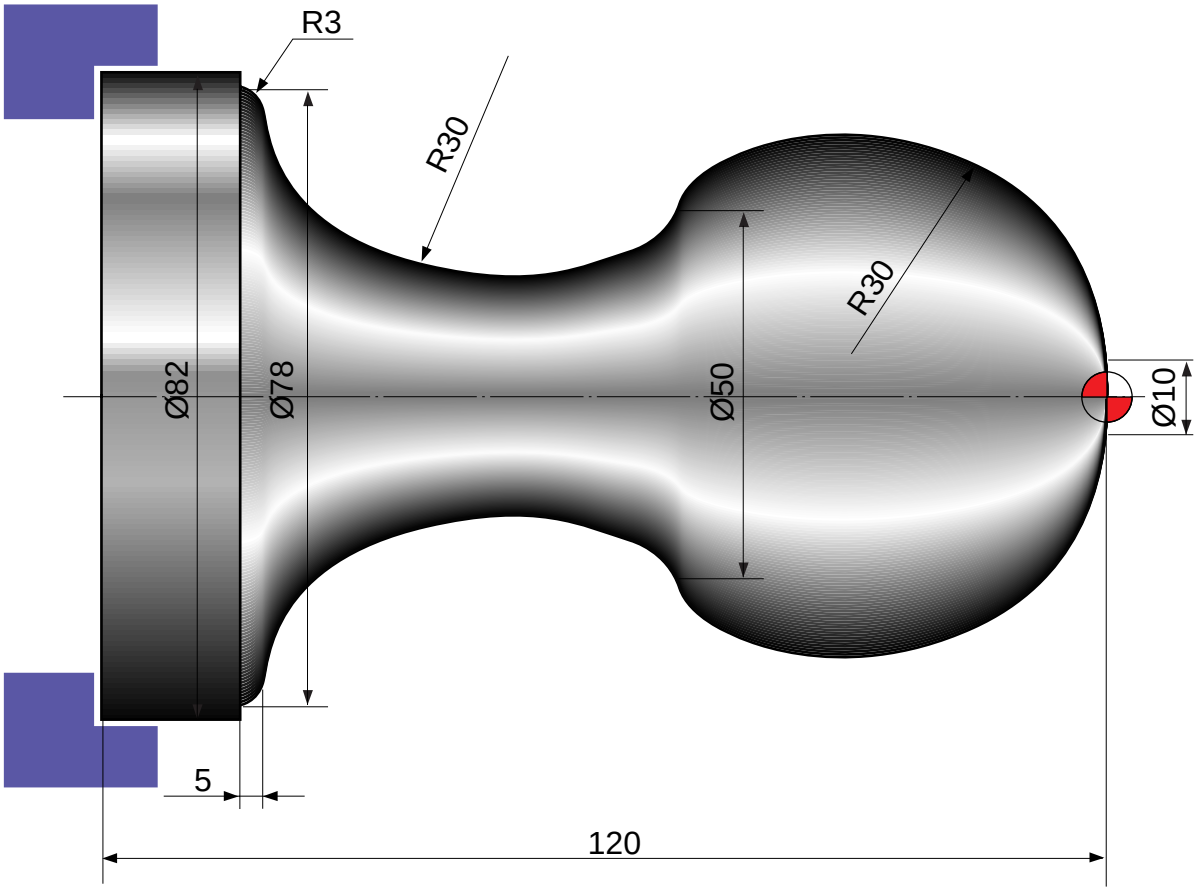
Condition of using tool

Outside diameter process

TOOL	PROCESS TYPE
SVVBN	Stock removal + Finishing

(Example8)

Process	Outside diameter circumference process
Dimension	ø 82 x 120L
Material	S45C



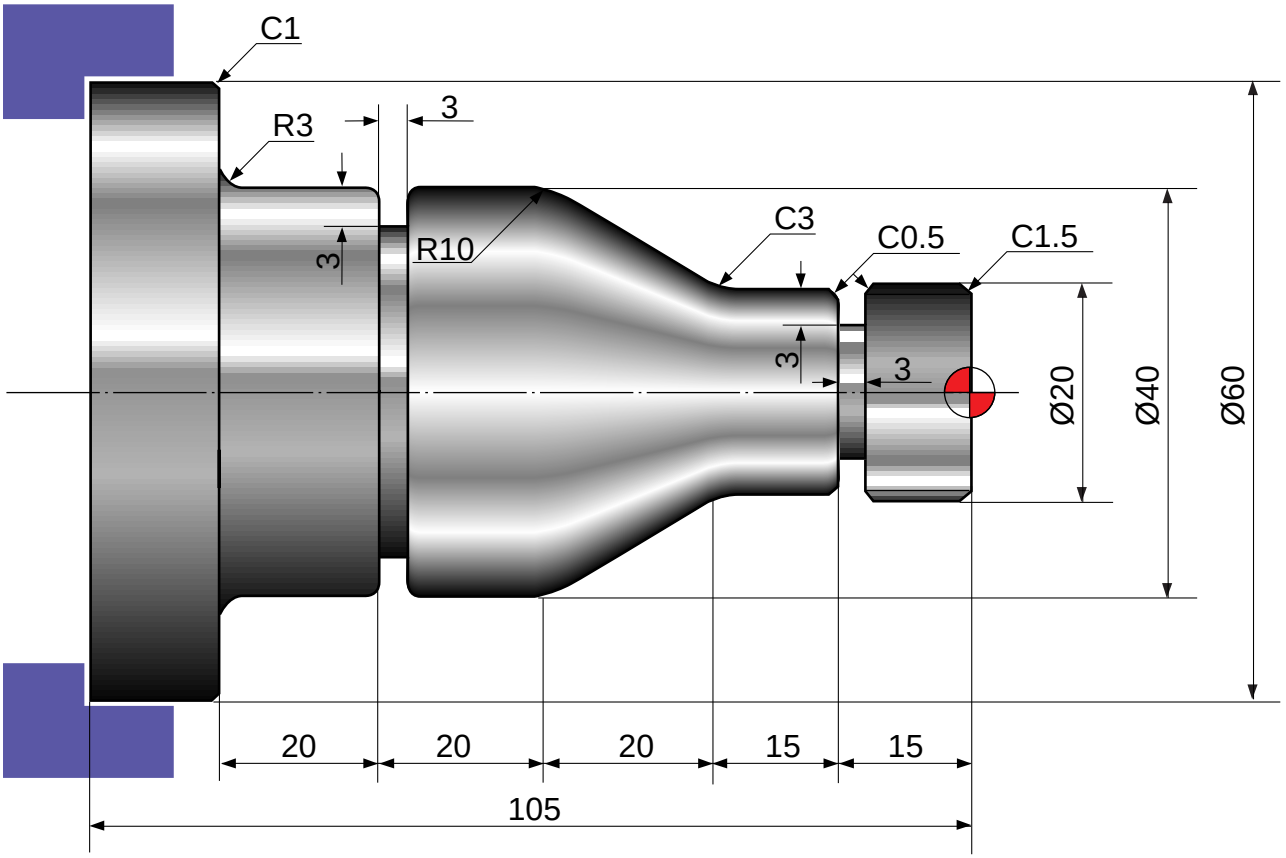
Condition of using tool

Outside diameter circumference process

TOOL	PROCESS TYPE
SVVBN	Stock removal + Finishing

(Example9)

Process	Outside diameter(Groove process, Thread process, Chamfering R process)
Dimension	ø 60 x 110L
Material	S45C



Condition of using tool

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Groove process

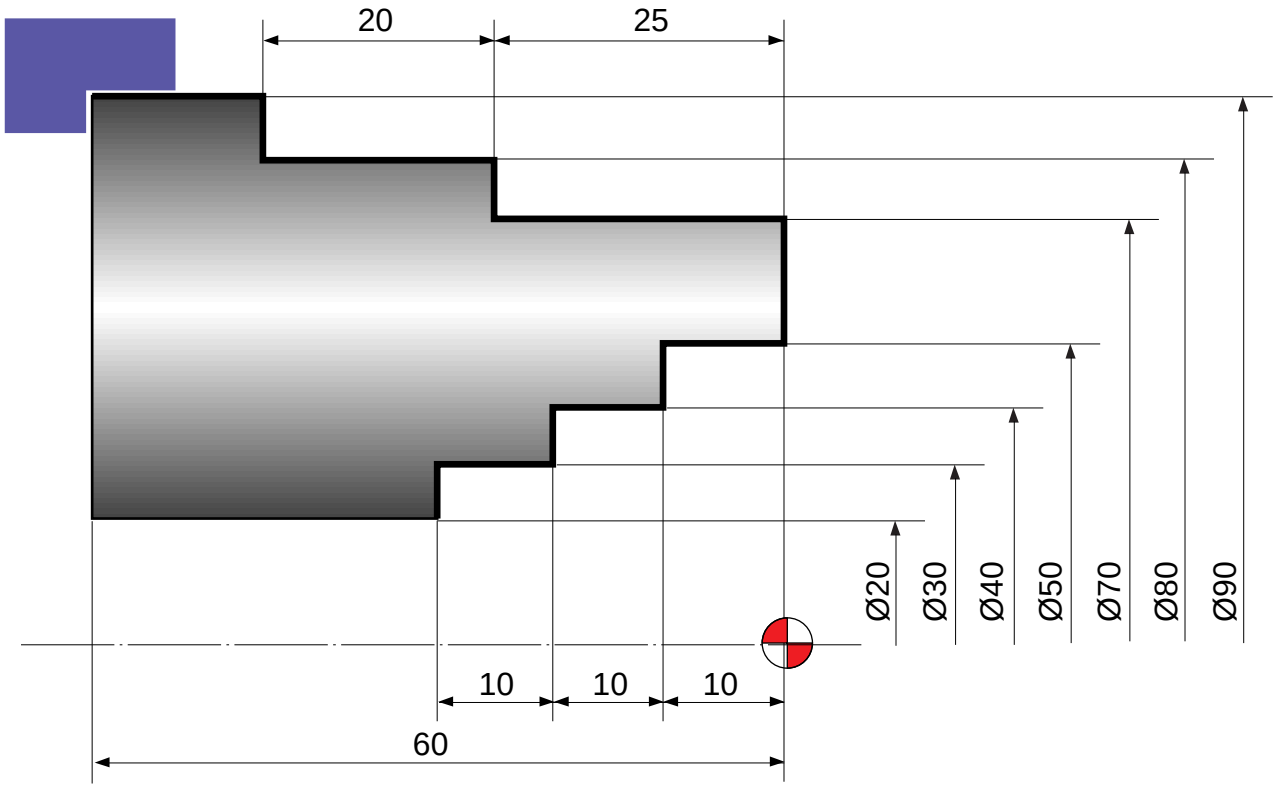
TOOL	PROCESS TYPE
R/L 154.91	Stock remova + Finishing

Thread process

TOOL	PROCESS TYPE
R/L 166.0	Stock remova + Finishing

(Example10)

Process	Outside diameter process, Inside diameter process
Dimension	ø60 x 110L
Material	S45C



Condition of using tools

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

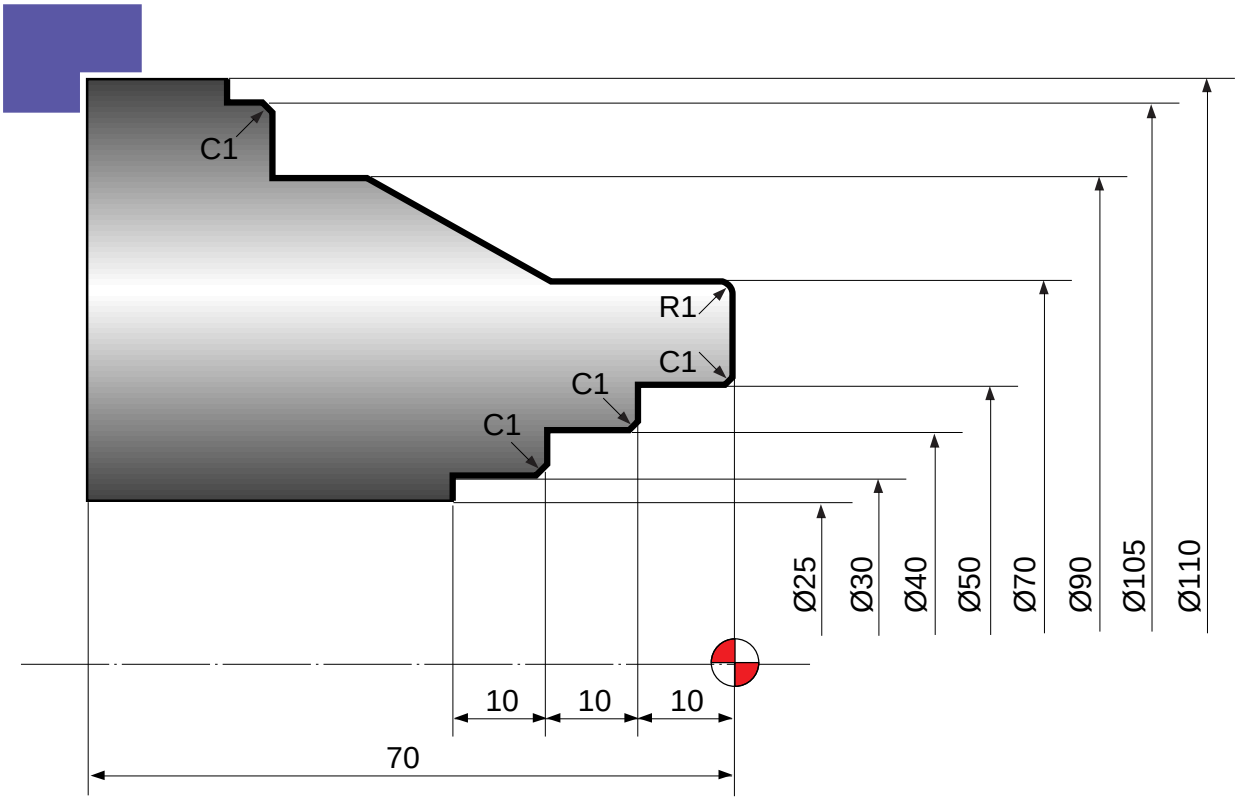
TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Inside diameter process

TOOL	PROCESS TYPE
S-20S PCLNR/L	Stock removal
S-20S PCNR/L-1	Finishing

(Example11)

Process	Outside diameter process, Inside diameter process
Dimension	ø110 x 75L x ø25(Pipe)
Material	S45C



Problem 1) Program when the material is pipe

Problem 2) Program when the material is a round bar

Condition of using tools

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

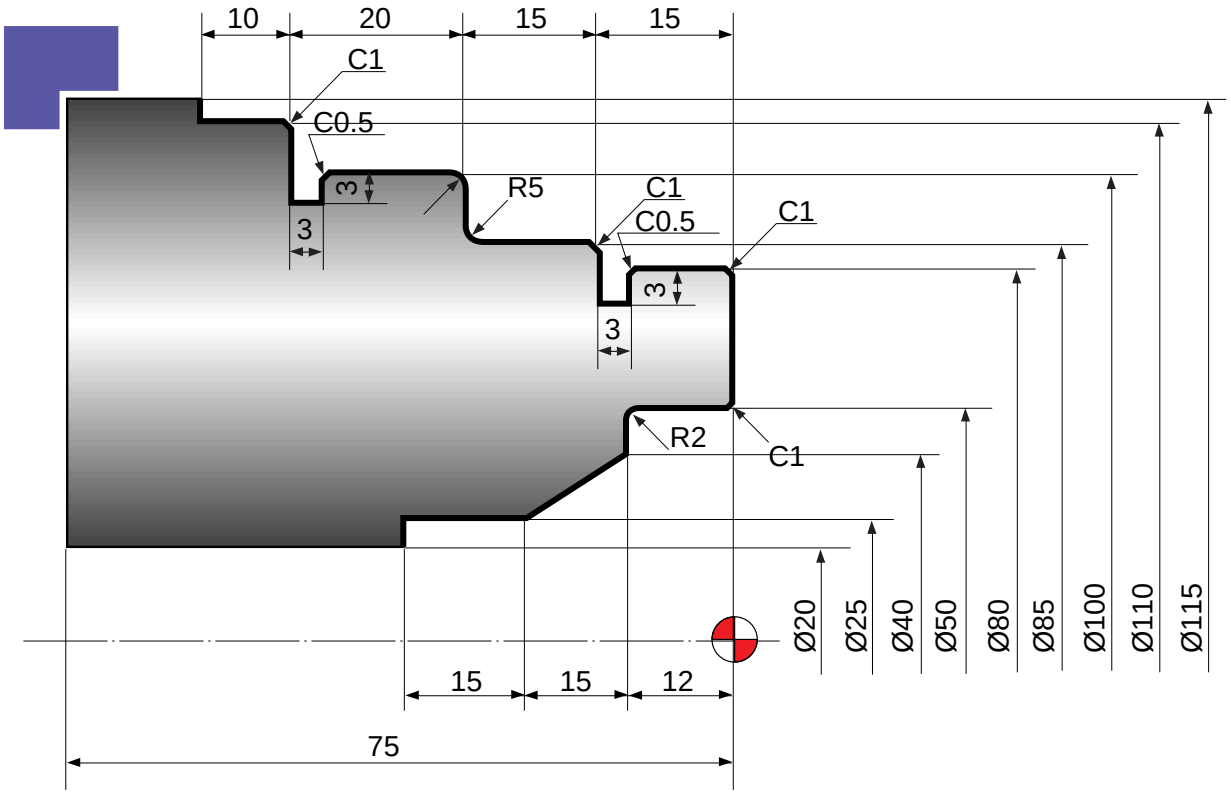
TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Inside diameter process

TOOL	PROCESS TYPE
S-20S PCLNR/L	Stock removal
S-20S PCNR/L-1	Finishing

(Example12)

Process	Outside diameter process, Inside diameter process
Dimension	ø 110 x 75L x ø 25(Pipe)
Material	S45C



Condition of using tool

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Groove process

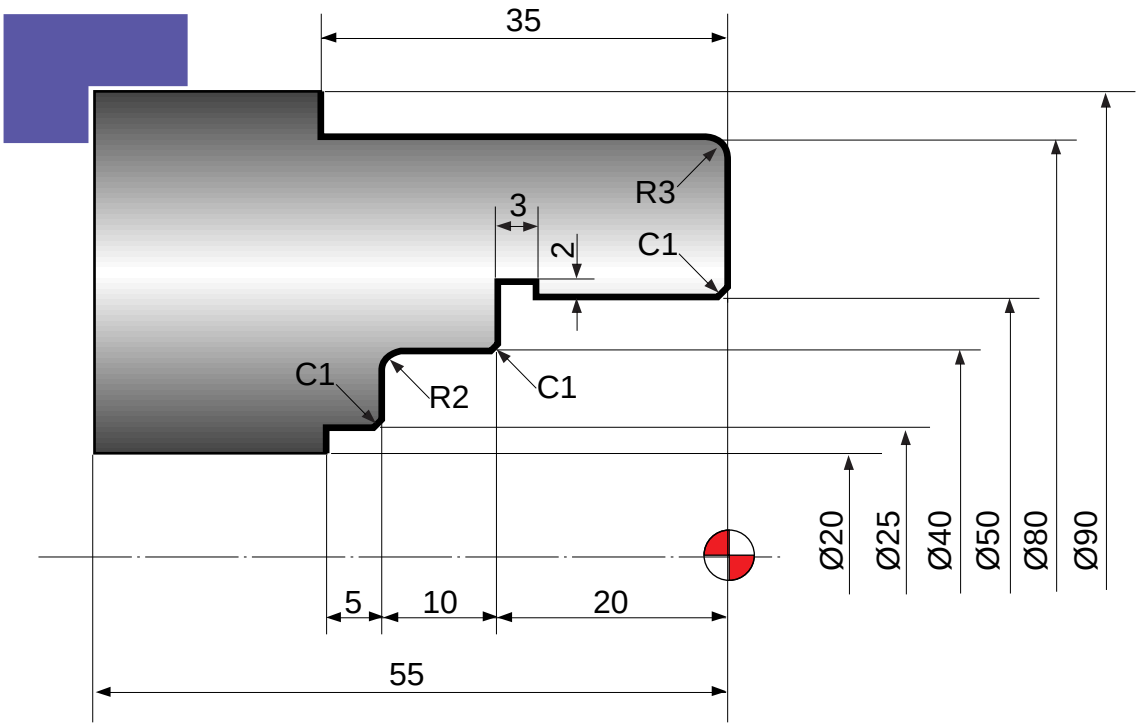
TOOL	PROCESS TYPE
PCLNR/L	Stock remova + Finishing
PCLNR/L–1	

Inside diameter process

TOOL	PROCESS TYPE
S-20S PCLNR/L	Stock remova
S-20S PCLNRL-1	Finishing

(Example13)

Process	Outside diameter process, Inside diameter process(Chamfering, R, Groove)
Dimension	ø90 x 60L x ø20(Pipe)
Material	S45C



Problem 1) Program when the material is pipe

Problem 2) Program when the material is a round bar

Condition of using tool

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Inside diameter Groove process

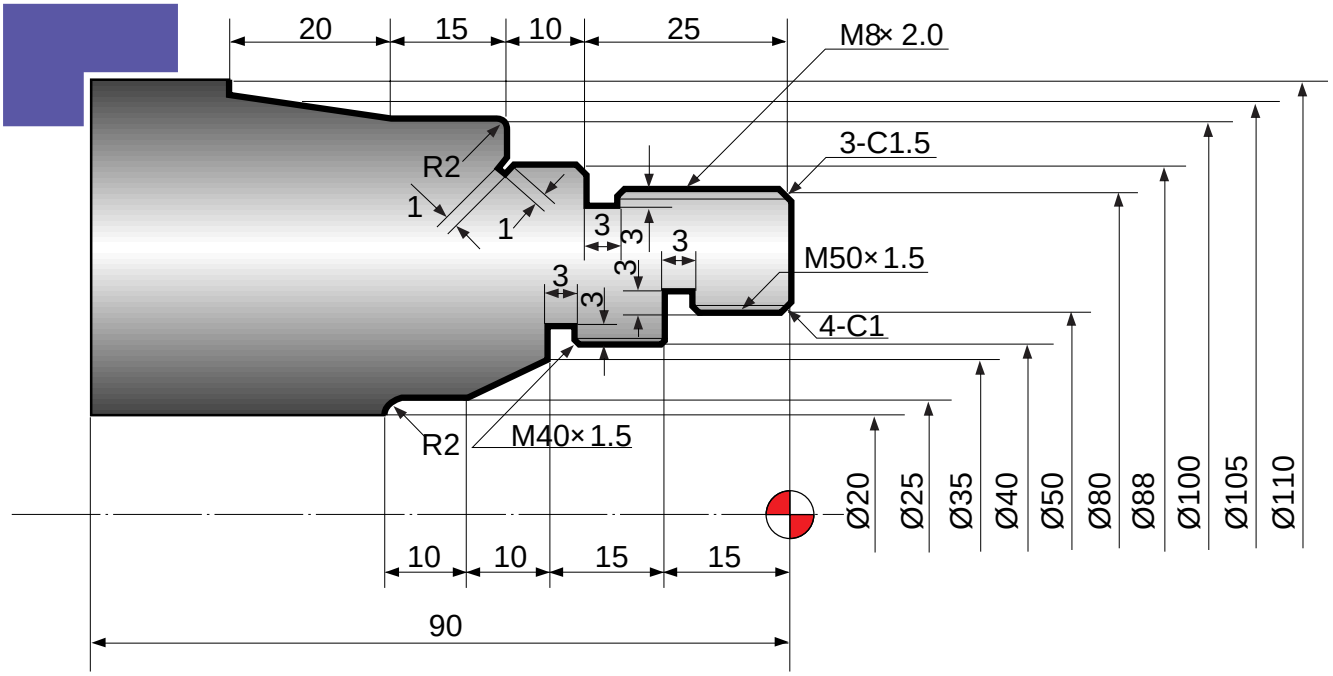
TOOL	PROCESS TYPE
PCLNR/L	Stock remova + Finishing
PCLNR/L–1	

Inside diameter process

TOOL	PROCESS TYPE
S-20S PCLNR/L	Stock remova
S-20S PCLNRL-1	Finishing

(Example14)

Process	Outside diameter process(Chamfering, R, Groove, Thread, Relief process)
Dimension	ø110 x 90L x ø20(Pipe)
Material	S45C



Problem 1) Program when the material is pipe

Problem 2) Program when the material is a round bar

Condition of using tools

Facing process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Inside diameter process

TOOL	PROCESS TYPE
S-20S PCLNR/L	Stock removal
S-20S PCNR/L-1	Finishing

Inside diameter Groove process

TOOL	PROCESS TYPE
R/L 154.3	Stock removal + Finishing

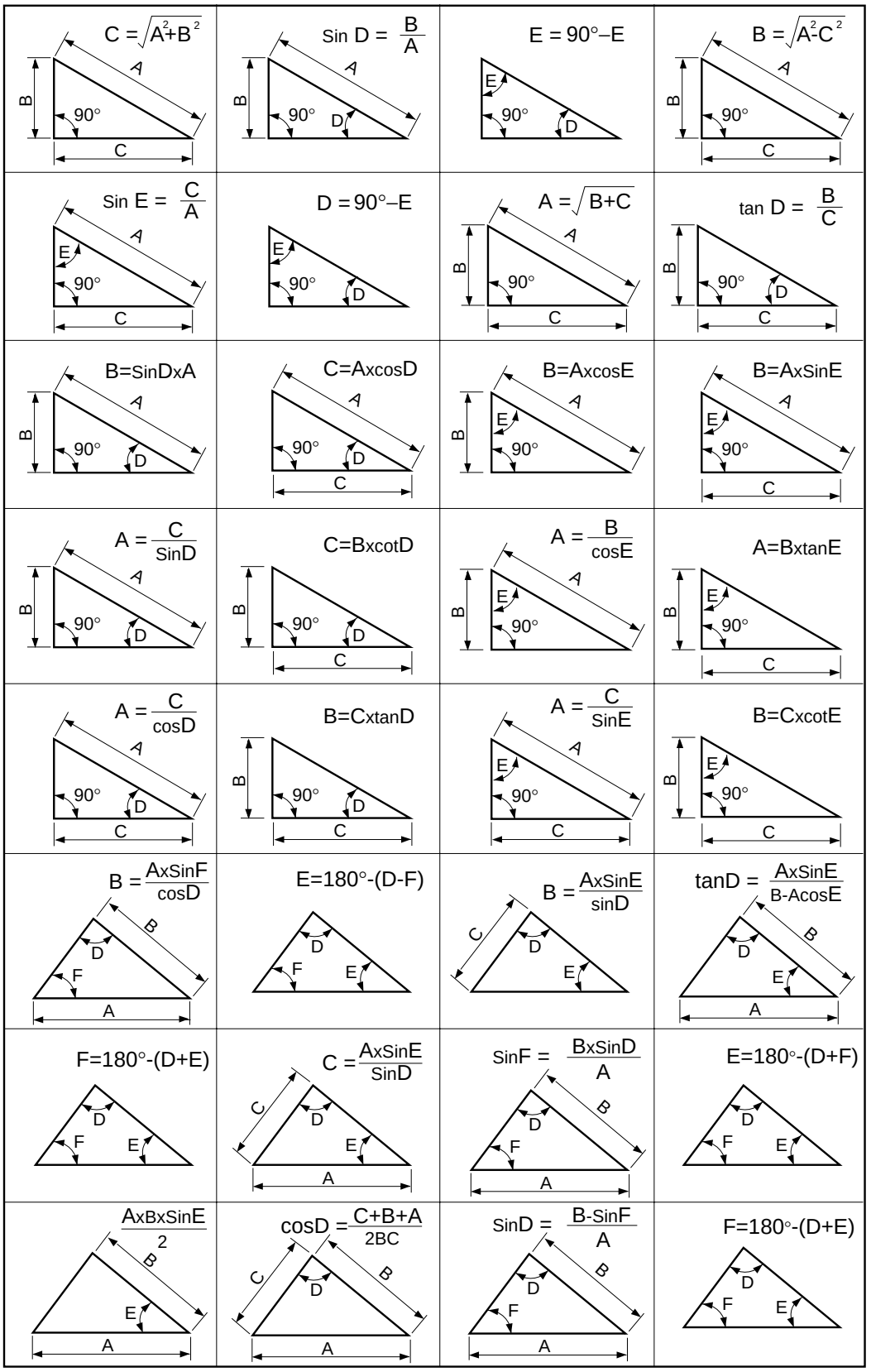
Vutsude diameter relief process

TOOL	PROCESS TYPE
PCLNR/L	Stock removal
PCLNR/L–1	Finishing

Outside diameter Groove process

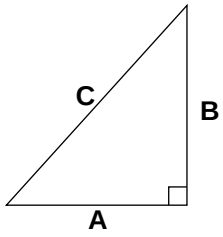
TOOL	PROCESS TYPE
R/L 154.91	Stock removal + Finishing

Calesslating table of trigonometric function



FORMULA

1. The puthagorean theorem

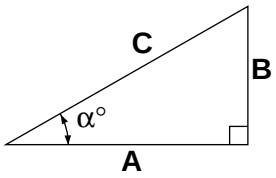


$C^2 = A^2 + B^2$ $\quad C = \sqrt{A^2 + B^2}$

$A^2 = C^2 - B^2$ $\quad A = \sqrt{C^2 - B^2}$

$B^2 = C^2 - A^2$ $\quad B = \sqrt{C^2 - A^2}$

2. Trigonometric function



$\text{SIN}\alpha^\circ = \frac{B}{C}$, $\text{COS}\alpha^\circ = \frac{A}{C}$, $\text{TAN}\alpha^\circ = \frac{B}{A}$

$A = C \times \text{COS}\alpha^\circ$ $\quad A = \frac{B}{\text{TAN}\alpha^\circ}$

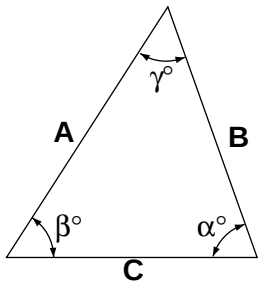
$B = C \times \text{SIN}\alpha^\circ$ $\quad B = A \times \text{TAN}\alpha^\circ$

$C = \frac{B}{\text{SIN}\alpha^\circ}$ $\quad C = \frac{A}{\text{COS}\alpha^\circ}$

3. SIN law

When finding the length of the two sides(Oneside and two angles are known)

When finding the other angle(Two sides and one angle are know)

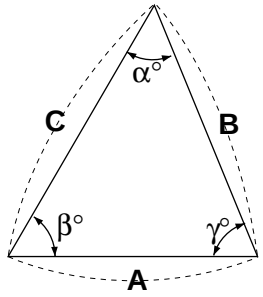


$$\frac{A}{\text{SIN}\alpha^\circ} = \frac{B}{\text{SIN}\beta^\circ} = \frac{C}{\text{SIN}\gamma^\circ}$$

4. COS law

When finding the other side(Two sides and one angle are known)

When finding the other angle(Lengthsof three sides are known)



$A^2 = B^2 + C^2 - 2B.C\ \text{COS}\alpha^\circ$ $\quad \text{COS}\alpha^\circ = \frac{B^2 + C^2 - A^2}{2BC}$

$B^2 = C^2 + A^2 - 2C.A\ \text{COS}\beta^\circ$ $\quad \text{COS}\beta^\circ = \frac{C^2 + A^2 - B^2}{2CA}$

$C^2 = A^2 + B^2 - 2A.B\ \text{COS}\gamma^\circ$ $\quad \text{COS}\gamma^\circ = \frac{A^2 + B^2 - C^2}{2AB}$

♠ **DECHNICAL GUIDE**

CALCULATING FORMULA

♠ Drocess time(sec/ea) = $\frac{¥\ D.\ L \times 60}{100V \times F} = \frac{\text{Cutting length} \times 60}{\text{Arerage of rotating time}}$ = sec

♠ Output(8Hrs/day) = 8Hrs x 60 x 60 = ea

Required time per unit

♠ Required day for process = $\frac{\text{Object time} \times \text{Quantity to be processed}}{\frac{8 \times 60}{60}}$ =Day

♠ Surface roughress = $\frac{\text{Feed volume}^2}{8 \times \text{NOSER}}$ x 1000 = R.t μm

♠ Cutting volume = cm^3/min

V. F.D = LT

$\frac{ft \times W \times D}{1000}$ = ML

V = Cutting speed

F = Feed volume(mm/rev)

D = Depth of cutting

ft = Feedrate(mm/min)

W= Width of cutting

♠ Cutting condition(Material : AL)

∗ EXTREME – FINISHING V = 870

F = 0.05~0.15

t = 0.025~2.0

∠ FINISHING V = 720

F = 0.1~0.3

t = 0.5~2.0

∠ LIGHT V = 600

ROUGHING F = 0.2~0.5

t = 2.20~4.0

Cutting condition

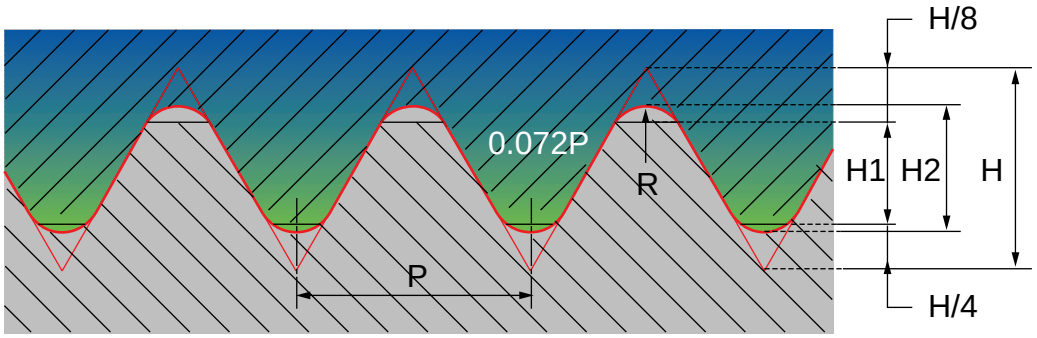
1. Cutting condition

Material	Classification	Depth of cutting d(mm)	Cutting speed v (m/min)	Feedrate F (mm/rev.)	Material of tool
Carbon steel 60kg/mm (Tensile strength)	Stock vemoval	3 ~ 5	180 ~ 200	0.3 ~ 0.4	P 10 ~ 20
		2 ~ 3	200 ~ 250	0.3 ~ 0.4	P 10 ~ 20
	Finishing	0.2 ~ 0.5	250 ~ 280	0.1 ~ 0.2	P 01 ~ 10
	Thread		124 ~ 125		P 10 ~ 20
	Grooving		90 ~ 110	0.08 ~ 0.2	P 10 ~ 20
	Center drill		1000 ~ 1600 rpm	0.08 ~ 0.15	SKH 2
	Drill		~ 25	0.08 ~ 0.2	SKH9
Alloy steel 140kg/mm^2	Stock removal	3 ~ 4	150 ~ 180	0.3 ~ 0.4	P10 ~ 20
	Finishing	0.2 ~ 0.5	200 ~ 250	0.1 ~ 0.2	P 10 ~ 20
	Grooving		70 ~ 100	0.08 ~ 0.2	P 10 ~ 20
Castiron HB 150	Stock removal	3 ~ 4	200 ~ 250	0.3 ~ 0.5	K 10 ~ 20
	Finishing	0.2 ~ 0.5	250 ~ 280	0.1 ~ 0.2	K 10 ~ 20
	Grooving		100 ~ 125	0.08 ~ 0.2	K 10 ~ 20
Aluminum	Stock removal	2 ~ 4	400 ~ 1000	0.3 ~ 0.5	K 10
	Finishing	0.2 ~ 0.5	700 ~ 1600	0.1 ~ 0.2	K 10
	Grooving		350 ~ 1000	0.1 ~ 0.2	K 10
Bronge Brass	Stock removal	3 ~ 5	150 ~ 300	0.2 ~ 0.4	K 10
	Finishing	0.2 ~ 0.5	200 ~ 500	0.1 ~ 0.2	K 10
	Grooving		150 ~ 200	0.1 ~ 0.2	K 10
Staialess steel	Stock removal	2 ~ 3	150 ~ 180	0.2 ~ 0.35	P 10 ~ 20
	Finishing	0.2 ~ 0.5	180 ~ 200	0.1 ~ 0.2	P 01 ~ 10
	Grooving		60 ~ 90	~ 0.15	P 10 ~ 20

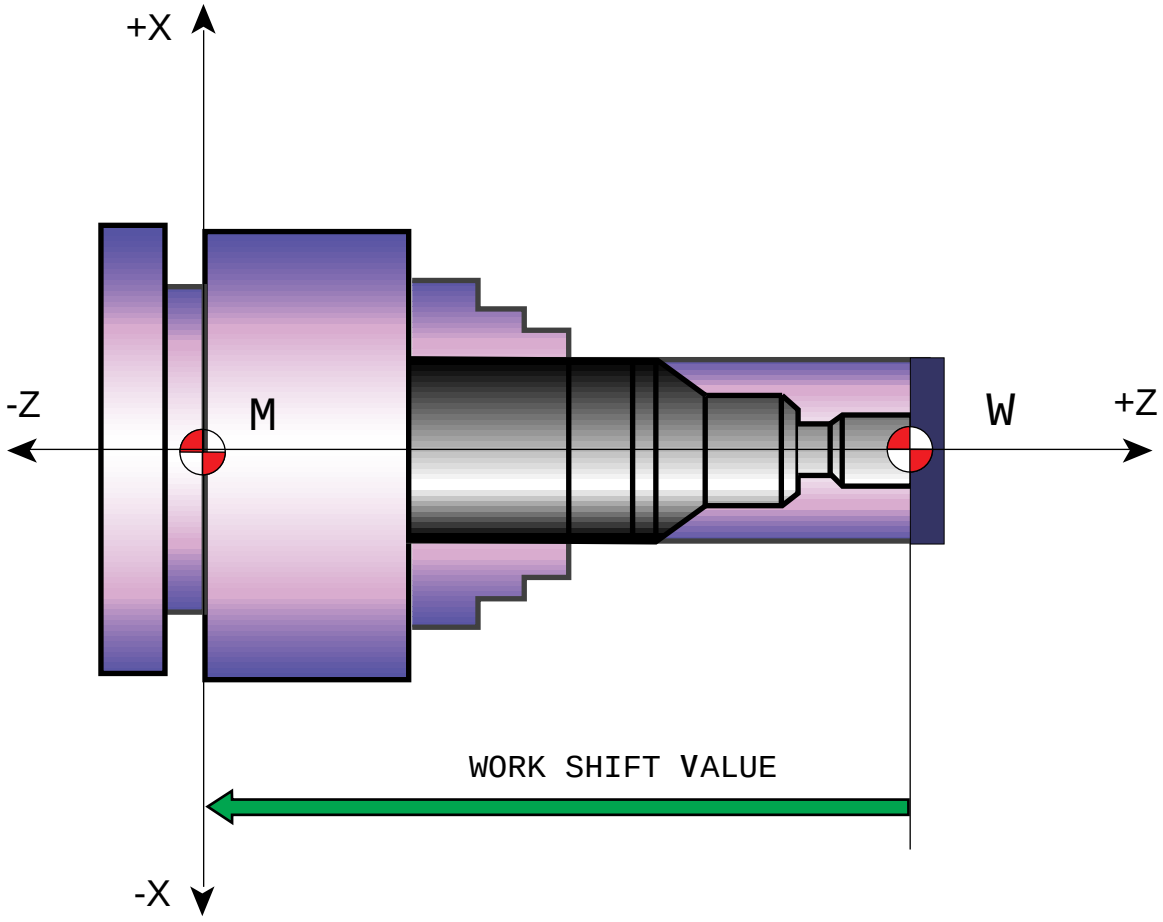
(Note) 1) Conditions for tools coated

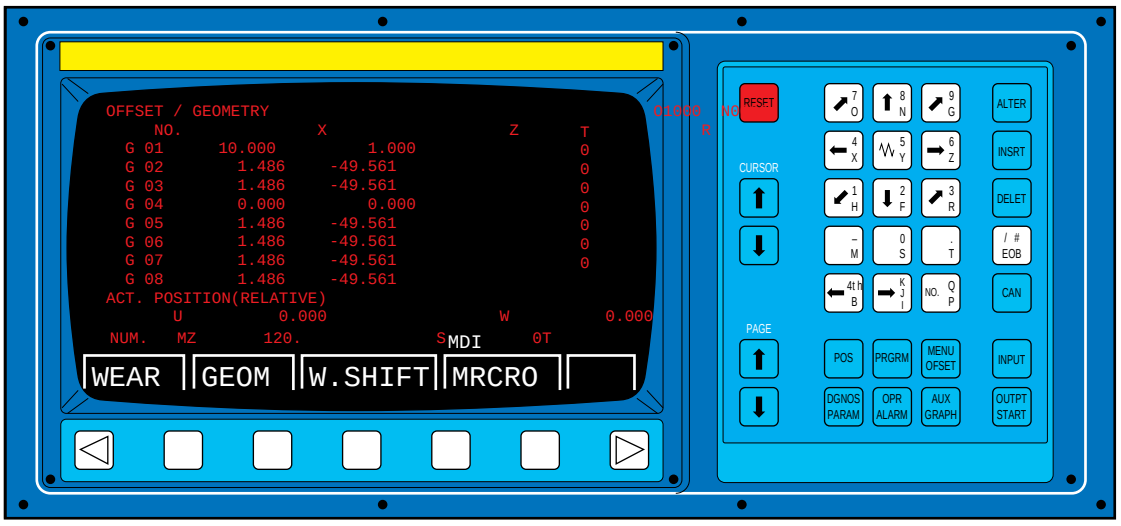
2) Cutting condition shall be changed by the shape and angle of tools

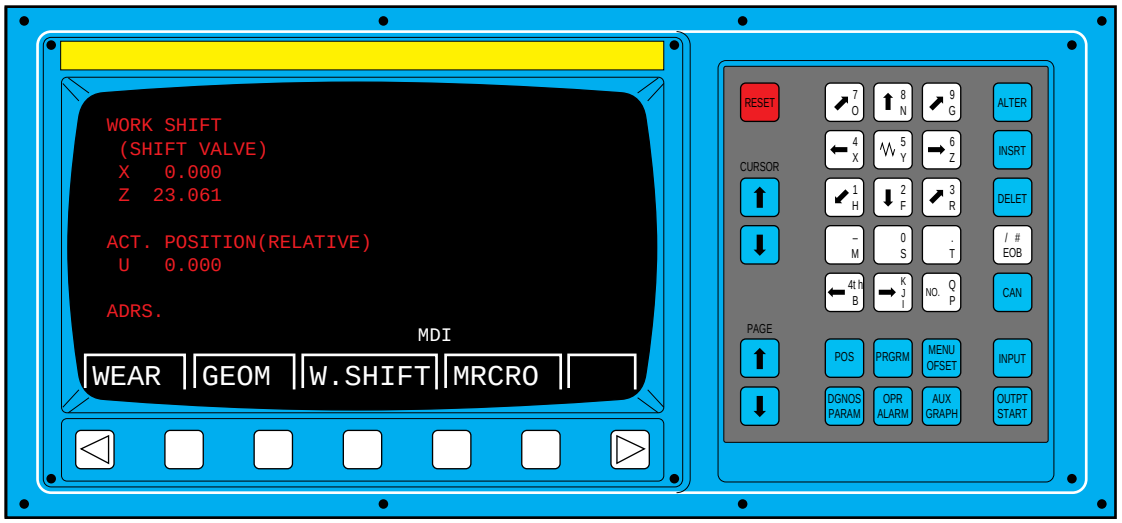
2. Cutting time of thread process(For thread precessing with the S 45 C)



PITCH	P1.0	1.0	1.25	1.5	1.75	2.0	2.5	3.0	3.5	4.0	4.5	5.0
CUTTING DEPT	H2	0.6	0.74	0.89	1.05	1.19	1.49	1.79	2.08	2.38	2.68	2.98
CORNER ROUND	R	0.07	0.09	0.11	0.13	0.14	0.18	0.22	0.25	0.29	0.32	0.36
SCREW CUTTING NUMBER OF TIMES	1	0.25	0.30	0.30	0.30	0.30	0.30	0.35	0.35	0.35	0.40	0.45
	2	0.20	0.20	0.20	0.25	0.25	0.28	0.30	0.35	0.35	0.35	0.35
	3	0.10	0.11	0.14	0.16	0.20	0.24	0.26	0.30	0.30	0.30	0.32
	4	0.05	0.08	0.12	0.12	0.14	0.20	0.22	0.25	0.26	0.28	0.30
	5		0.05	0.08	0.10	0.11	0.15	0.18	0.20	0.23	0.25	0.25
	6			0.05	0.07	0.08	0.11	0.13	0.15	0.20	0.22	0.25
	7				0.05	0.06	0.09	0.10	0.12	0.17	0.20	0.20
	8					0.05	0.07	0.08	0.10	0.14	0.15	0.17
	9						0.05	0.07	0.08	0.10	0.12	0.15
	10							0.05	0.05	0.10	0.10	0.15
	11							0.05	0.05	0.08	0.08	0.10
	12								0.05	0.05	0.08	0.10
	13									0.05	0.05	0.08
	14										0.05	0.06
	15										0.05	0.06


+X
-Z
M
W
+Z
WORK SHIFT VALUE
-X


OFFSET / GEOMETRY
NO. X Z T
G 01 10.000 1.000 0
G 02 1.486 -49.561 0
G 03 1.486 -49.561 0
G 04 0.000 0.000 0
G 05 1.486 -49.561 0
G 06 1.486 -49.561 0
G 07 1.486 -49.561 0
G 08 1.486 -49.561
ACT. POSITION(RELATIVE)
U 0.000 W 0.000
NUM. MZ 120. MDI 0T
WEAR GEOM W.SHIFT MRCRO
RESET
CURSOR
PAGE
ALTER
INSRT
DELET
EOB
CAN
INPUT
POS
PRGRM
MENU OFSET
DGNOS PARAM
OPR ALARM
AUX GRAPH
OUTPT START



Work shift method using the tool measure

1.Return to the reference manually.

2. Install the work piece to the JAW and move the TURRET to appropriate position, and then prepare the basic tools to work.

3. On the section of material, TOUCH of process in facing the basic tool

∴At this, it is absolutely not allowed to move the Z spindle.

4. Select WORK/SHIFT screen.

Method) MENU OFSET → PAGE ↑ ↓ Push the bottun to select the WORK/SHIFT

5. Inpit the DATA.

Method) $\overline{M}$ → W 5 Z → DATA push bottuns one by one, and push MEASURE on the console, and push INPUT, then identify the input.

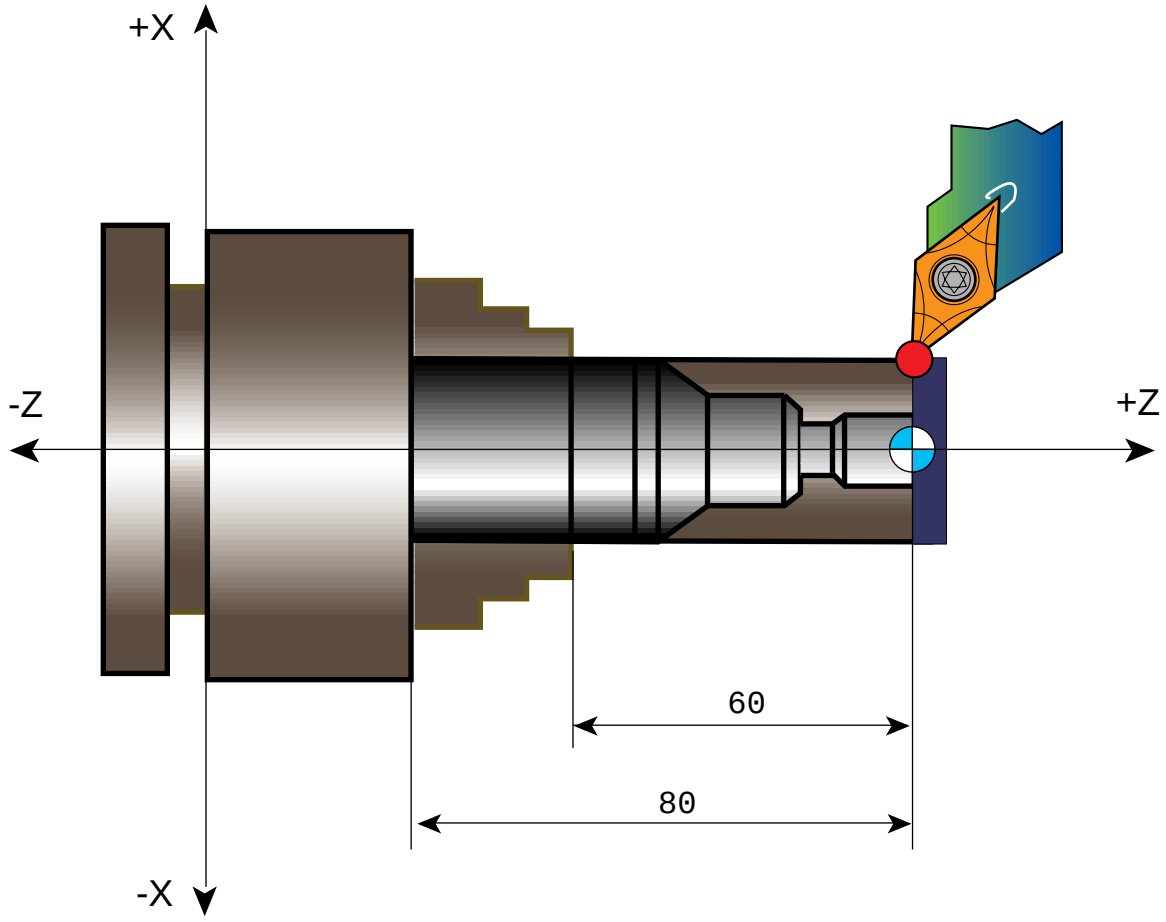
* DATA → Z coordinate value in the program (Touched position)

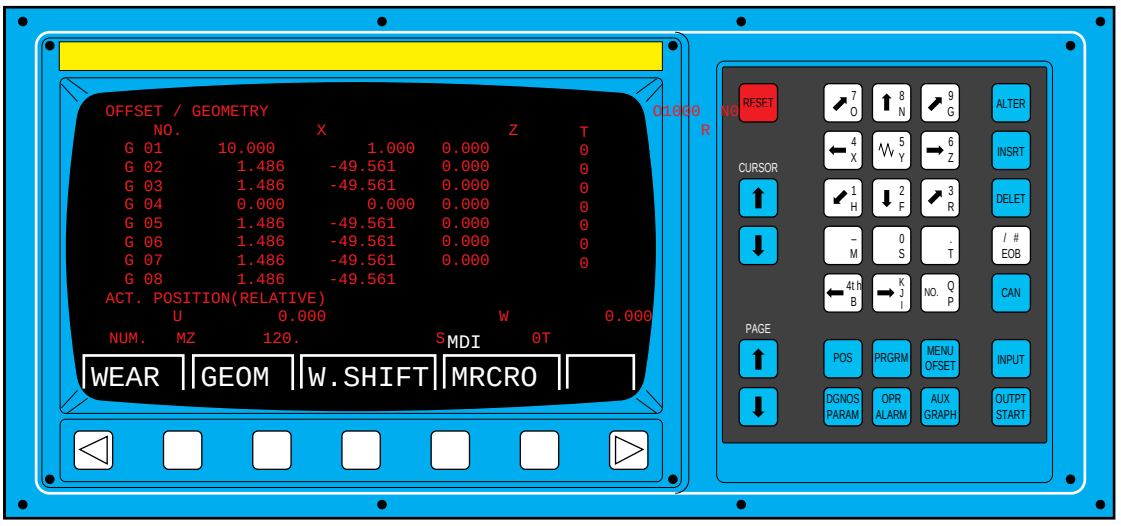
* After input, Z value on the screen of WORK/SHIFT is automatically calculated and input.

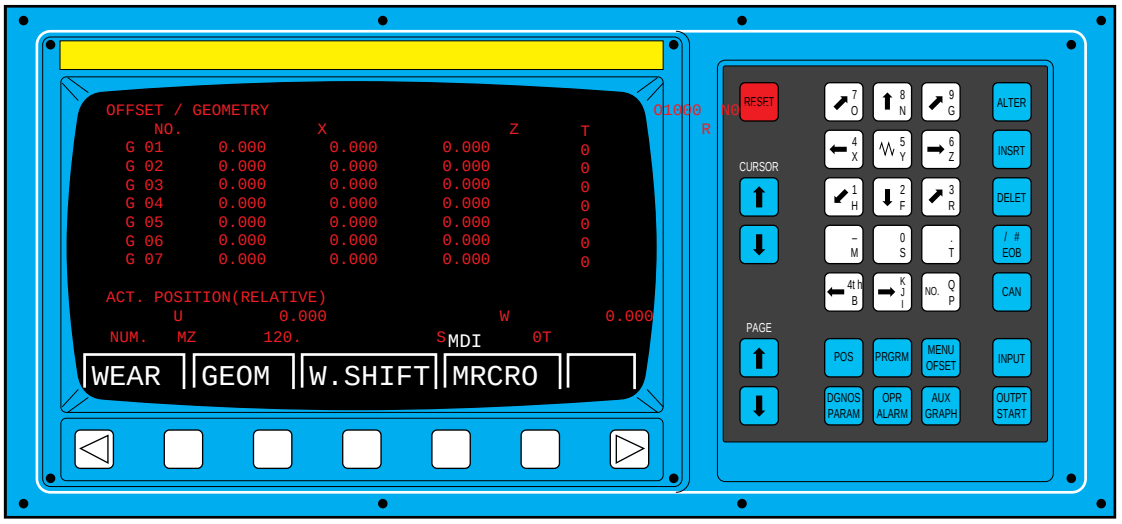
6. As the input is completed,

Push PAGE ↑ ↓ to select the OFFSET screen.

Offs.


+X
-Z
+Z
60
80
-X

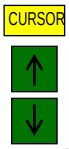

OFFSET / GEOMETRY
NO. X Z T
G 01 10.000 1.000 0.000 0
G 02 1.486 -49.561 0.000 0
G 03 1.486 -49.561 0.000 0
G 04 0.000 0.000 0.000 0
G 05 1.486 -49.561 0.000 0
G 06 1.486 -49.561 0.000 0
G 07 1.486 -49.561 0.000 0
G 08 1.486 -49.561
ACT. POSITION(RELATIVE)
U 0.000 W 0.000
NUM. MZ 120. S MDI 0T
WEAR GEOM W.SHIFT MRCRO
O1000 N0
R
RESET
CURSOR
PAGE
ALTER
INSRT
DELET
EOB
CAN
INPUT
POS
PRGRM
MENU OFSET
DGNOS PARAM
OPR ALARM
AUX GRAPH
OUTPT START



OFFSET method using Tool measure

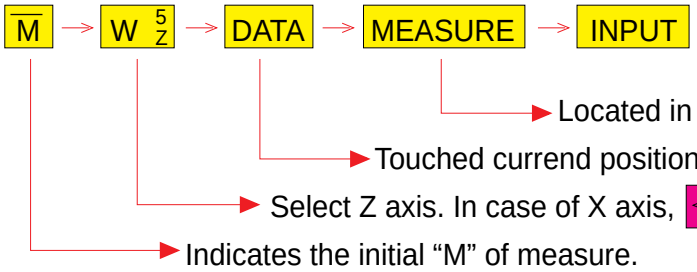
→ Z axis OFFSET

1. After selecting OFFSET screen

   push  to move the OFFSET No. of the basic tool .

   ∗ In gereral, tool no. and OFFSET No. shall be the same

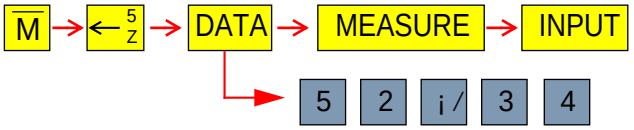
2. After selecting numbers, input the coordinate value of Z in the current position which is touched. The method shall be the same as work shift. For summary,



   After input as above, Z value of OFFSET selected by the cursor is automatically input, but the basic tool becomes “ 0 ”(zero). If another value is given, start from the begining again.(Work shift end)

→ X axis OFFSET

3. Continuously, process the outside diameter with the basic tool, and retreat the Z spindle to + direction(right hand), stop rotating, then measure the processed outside diameter(Xvalue). If the measured value is ø52.34, the position of tool is X52.34 therefor, input the X value.



∗ As you input with above method, X value on OFFSET screen is automatically input.

4. prepare another tool which you want to OFFSET to the work position.
5. Touch slightly on the section of the material.
6. If you input with the same method as finding the OFFSET value of Z spindle written previously, Z OFFSET value of this tool is autonatically input. (Difference of length compared with the basic tool)
7. Find the X OFFSET value with the same method as 3.
8. For all other tools, OFFSET with repeating above method(1~3).

(Attention)

1. On WORK/SHIFT screen, input only Z value, not X value.
   (∗ Except the GANG TYPE)
2. For the drill and a kind of center drill, input only the OFFSET of Z spindle, leave the X value as " 0 ".
3. Above explanation to find the value of OFFSET is the method when input only the Z value on WORK/SHIFT screen.
   If you input the X axis with the Z axis on WORK/SHIFT screen, you should input the OFFSET value of X spindle for all tools which are processed in the center of main spindle like the drill and the center drill.
5. If you OFFSET with above method with using the function of tool measure, you don,t have to designate the coordinate as G50 during the programming.
   Example)
   (When using TOOL MEASURE)
   O 3333 :
   N1 G50 T0100 S1800 M42 :
   G96 S100 M03 :
   (When not using TOOL MEASURE)
   O 3334 :
   N1 G50 T100. Z100. T0100 S1800 M42 :
   G96 S170 M03 :

M-FUNCTION

M00 : PROGRAM STOP

When M00 is commanded in automatic operation mode(MDI or MEM mode), the automatic operation will stop after completion of the command in the block containing M00.
When the machine is stopped by M00 code. Manual operation can be done if the mode selector switch is turned to JOG position.
To restart cycle, select the mode selector switch to previous automatic operation mode and then depress the CYCLE START button.

NOTE1)

Spindle stops after completion of M00, then chuck open-close can be done by manual without changing the MODE.

M01 : OPTIONAL STOP

This command is used to stop the machine temporarily by slash(/) and check workpiece at the end of each tool operations. OPTIONAL STOP switch(toggle switch) is used to selection this code.

M02 : END OF PROGRAM

This code is used in the last block of chucking work part program to end the program.
When this code occurs during the automatic operation of the machine, the program returns to the head after performing the other command in the block, the control is reset, this automatic mode ends and the machine stop.

M03 : MAIN-SPINDLE FORWARD DIRECTION

Specifies to start the main spindle rotation in counterclockwise direction. S code should be specified in the same block or previous.
If M03 code is specified when the chuck is open, the sequence error will occur.

M04 : MAIN-SPINDLE REVERSE DIRECTION

Specifies to start the main spindle rotation in clockwise direction. S code should be specified in the same block or previous.
If M04 code is specified when the chuck is open, the sequence error will occur.

M05 : MAIN-SPINDLE STOP

Specifies to stop the main spindle rotation. Even M05 is specified, the command spindle speed remains effective. Therefore, if M03 or M04 is specified again, the spindle will rotate by the same speed as the previous speed.

M07 : HIGH PRESSURE COOLANT ON (optional)

Specifies to start the high pressure coolant pump.

M08 : COOLANT ON

Specifies to start the coolant pump. The coolant pump will start when the COOLANT switch on the operating panel is set to ON position.

M09 : COOLANT OFF

Specifies to stop the high pressure coolant pump and coolant pump.

M10: PART CATCHER1 ADVANCE (optional)

This command moves the part catcher1 advance.

M11 : PART CATCHER1 RETRACT (optional)
This command moves the part catcher1 retract.
M13 : AIR BLOW FOR TURRET (optional)
Air blow for turret when M13 is commanded.
M14 : AIR BLOW FOR MAIN SPINDLE (optional)
Air blow for main spindle when M14 is commanded.
M15 : AIR BLOW OFF (optional)
Air blowing stops. This command is available on M13, M14.
M17 : MACHINE LOCK ON
Specifies to machine lock on. This command is specified only MDI mode.
M18 : MACHINE LOCK OFF
Specifies to machine lock off. This command is specified only MDI mode.
M19 : MAIN- SPINDLE ORIENTATION (optional)
This code stops main-spindle at fixed angle. M19 Sxxx : Main-spindle multi orientation (ORIENTATION “B”) When M19 code and S code should be specified in the same block, the spindle stops position is determined by S code.
M24 : CHIP CONVEYOR RUN (optional)
Specifies to run the chip conveyor.
M25 : CHIP CONVEYOR STOP (optional)
Specifies to stop the chip conveyor.
M30 : PROGRAM END & REWIND (continuous running)
Return to head of the memory by M30 command, reset and stop. The program is restarted by cycle start and specifies at last block.
M31: INTERLOCK BY-PASS (MAIN-SPINDLE & TAILSTOCK)
This code is used when cycle start is available the spindle unclamp and the tail stock quill operation during spindle rotating
M32 : STEADY REST CLAMP/UNCLAMP DURING SPINDLE ROTATION
This code is interlock by-pass of spindle rotating when STEADY REST is used. STEASY REST clamp(M38 or M58) and unclamp(M39 & M59) is valid during spindle rotating with M66.
M33 : REVOLVING TOOL-SPINDLE FORWARD DIRECTION
Revolving tool-spindle starts forward rotation.
M34 : REVOLVING TOOL-SPINDLE REVERSE DIRECTION
Revolving tool-spindle starts reverse rotation.
M35 : REVOLVING TOOL STOP
Revolving tool-spindle stops.

M38 : STEADY REST CLAMP(optional-right side), M58 : STEADY REST CLAMP(optional-left side)
Specifies to clamp the steady rest.
M39 : STEADY REST CLAMP(optional-right side), M59 : STEADY REST CLAMP(optional-left side)
Specifies to unclamp the steady rest.
M40 : GEAR CHANGE NEUTRAL
M41 : GEAR CHANGE LOW
M42 : GEAR CHANGE MIDDLE
M43 : GEAR CHANGE HIGH
Specifies to change the each gear range.
M46 : Prog. TAIL STOCK BODY UNCLAMP & TRACTION BAR ADVANCE (optional)
Simultaneous start of prog. Tail stock body unclamp and traction bar retract with this command.
M47 : Prog. TAILSTOCK BODY CLAMP & TRACTION BAR RETRACT (optional)
Simultaneous start of prog. Tail stock body clamp and traction bar advance with this command.
M50 : BAR FEEDING (optional)
When automatic bar feeder is attached, feed of material is performed.
M52 : SPLASH GUARD DOOR OPEN (optional)
The splash guard is opened with this command.
M53: SPLASH GUARD DOOR CLOSE (optional)
The splash guard is closed with this command.
M54 : PARTS COUNT (optional)
When M54 is commanded, pieces counter.
M61 : SWITCHING LOW SPEED (only aP60)
When the aP60 spindle motor is use, output torque and speed range of spindle is difference by power line switching. M61 is used to low speed rpm(Y-CONNECTION). 400 ~ 500 rpm(18.5kw)
M62 : SWITCHING HIGH SPEED (only aP60)
M62 is used to high speed rpm(⊿-CONNECTION). 750 ~ 4500 rpm(22kw)
M63 : MAIN-SPINDLE CW & COOLANT ON
Simultaneous start of main-spindle forward rotation and coolant. Spindle forward and coolant are preformed by one(M63) command. Coolant comes out only when operation panel switch is “on”.
M64 : MAIN-SPINDLE CCW & COOLANT ON
Simultaneous start of main-spindle reverse rotation and coolant. Spindle reverse and coolant are preformed by one(M64) command. Coolant comes out only when operation panel switch is “on”.

M65 : MAIN-SPINDLE & COOLANT STOP
Stop of main-spindle rotation, coolant is stopped by one command.
M66 : DUAL CHUCKING LOW CLAMP (optional)
Main-chuck is closed by low pressure.
M67 : DUAL CHUCKING HIGH CLAMP (optional)
Main-chuck is closed by high pressure.
M68 : MAIN-SPINDLE CLAMP
Specified to open the main-chuck automatically such as bar work.
M69 : MAIN-SPINDLE UNCLAMP
Specified to close the main-chuck automatically such as bar work.
M70 : DUAL TAILSTOCK LOW ADVANCE (optional)
Tailstock bar is advanced by low pressure.
M74 : ERROR DETECT ON
When M74 is in effect, the control proceed to the next block regardless of the pulse lag of servo between block for liner and circular interpolation except positioning (G00). The permits the machine to move smoothly between blocks. However, the corner of the workpiece may not be quite sharp. M74 command is modal, and it will remain effective until M75 is command.
M75 : ERROR DETECT OFF
Specifies to release the state of error detection ON. When the power is turned on, M75 will be in effect, and it will remain effective until M74 is command.
M76 : CHAMFERING ON
When M76 is specified before the command of thread cutting cycle G76 or G92, the threading tool will pull out at the terminating thread portion.
M77 : CHAMFERING OFF
Cancel the command of pull out threading function which as specified by M77 code. M77 code is the modal code.
M78 : TAIL STOCK QUILL ADVANCE
The tail stock quill is advanced with this command.
M79 : TAIL STOCK QUILL RETRACT
The tail stock quill is retracted with this command.
M80 : QUICK-SETTER SWING ARM DOWN (optional)
Specifies to up the quick-setter swing arm.
M81 : QUICK-SETTER SWING ARM UP (optional)
Specifies to up the quick-setter swing arm.

M82 : MIRROR IMAGE ON
Specifies to mirror image on.
M83 : MIRROR IMAGE OFF
Specifies to mirror image off.
M84 : TURRET CW ROTATION
This code is used to switch the direction of turret indexing to CW when it is set in the automatic selection mode. As this code is as non-modal code, it should be used in the same block the T-code.
M85 : TURRET CCW ROTATION
The turret indexes in clockwise by specifying M85 in the same block of T-code. This M85 is a non-modal code.
M86 : TORQUE SKIP ACT
This code is used to skip the torque of moving axis. As this code is a modal code until M87 command, only valid the sub-spindle with B-axis. EX) G00 B-500.0 ; M86 ; G98 G31 P99 V-20.0 F100.0 ; G01 B-500.0 ; M87 ;
M87 : TORQUE SKIP CANCEL
This code is used to cancel torque skip function of M86.
M88 : C-AXIS LOW CLAMP
Specified to clamp the C-axis by low pressure. Only valid the C-axis control.
M89 : C-AXIS HIGH CLAMP
Specified to clamp the C-axis by high pressure. Only valid the C-axis control.
M90 : C-AXIS UNCLAMP
Specified to unclamp the C-axis. Only valid the C-axis control.
M91,M92,M93,M94 : EXTERNAL M-CODE COMMAND (optional)
There code spare M code.
M98 : SUB-Prog. CALL
This code is used to enter a sub-program.
M99 : END OF SUB-PROGRAM
This code shows the end of a sub-program. Executing M99 take the control back to the main program.

M103 : SUB-SPINDLE FORWARD DIRECTION
Specifies to start the sub spindle rotation in counterclockwise direction. S code should be specified in the same block or previous. If M103 code is specified when the sub-chuck is open, the sequence error will occur.
M104 : SUB-SPINDLE REVERSE DIRECTION
Specifies to start the sub spindle rotation in clockwise direction. S code should be specified in the same block or previous. If M04 code is specified when the sub-chuck is open, the sequence error will occur.
M105 : SUB-SPINDLE STOP
Specifies to stop the sub spindle rotation. Even M05 is specified, the command spindle speed remains effective. Therefore, if M103 or M104 is specified again, the spindle will rotate by the same speed as the previous speed.
M110 : PART CATCHER2 ADVANCE (optional)
This command moves the part catcher2 advance.
M111 : PART CATCHER2 RETRACT (optional)
This command moves the part catcher2 retract.
M114 : AIR BLOW FOR SUB SPINDLE (optional)
Air blow for sub spindle when M114 is commanded.
M119 : SUB-SPINDLE ORIENTATION (optional)
This code stops sub-spindle at fixed angle. M119 Sxxx : sub-spindle multi orientation (ORIENTATION “B”) When M19 code and S code should be specified in the same block, the spindle tops position is determined by S code.
M131 : INTERLOCK BY-PASS (SUB-SPINDLE)
This code is used when cycle start valid on sub spindle unclamp.
M163 : SUB-SPINDLE CW & COOLANT ON
Simultaneous start of sub spindle forward rotation and coolant. Spindle forward and coolant are preformed by one(M163) command. Coolant comes out only when operation panel switch is “on”.
M164 : SUB-SPINDLE CW & COOLANT ON
Simultaneous start of sub spindle forward rotation and coolant. Spindle forward and coolant are preformed by one(M164) command. Coolant comes out only when operation panel switch is “on”.
M165 : SUB-SPINDLE & COOLANT STOP
The sub spindle rotation & coolant is stopped by one command.
M168 : SUB-SPINDLE CLAMP
Specifies to open the sub-chuck automatically such as bar work.

M169 : SUB-SPINDLE UNCLAMP
Specified to close the sub-chuck automatically such as bar work.
M203 : FORWARD SYNCHRONOUS COMMAND
Main and sub spindle start simultaneously for forward rotation. It is synchronized with forward rotation of main and sub spindle.
M204 : REVERSE SYNCHRONOUS COMMAND
Main and sub spindle start simultaneously for reverse rotation. It is synchronized with reverse rotation of main and sub spindle.
M205 : SYNCHRONOUS STOP
The synchronous rotation of main and sub spindle is stop.
M206 : SPINDLE ROTATION RELEASE
Specified to release the speed control of main and sub spindle. If you want to the main and sub spindle is rotate by difference rpm, M206 is commanded before S-code. Spindle override on operating panel valid last selected spindle. EX) M03 S1000 ; M206 ; M103 S500 ;