OKUMA

TECHNICAL CENTRE

SINGAPORE

PROGRAMMING MANUAL

FOR CNC LATHE

PAGE

Programming Format

Model 1S (Single Turret Model)

N	G	X	Z	I,K,F	S,T,M	
O0100						
N0000	G50				S0000	
N0001	G00	Xoooooo	Zoooooo		Soooo Toooo Moo	
N0002		Xoooooo	Zoooooo		M03(M04) M08	
N0003						
: : :	(Cutting Program)					
N0050	G00	Xoooooo	Zoooooo		M05 M09	
N0051					M02	

Explanation:

O0100 : Program name

N0000 : G50 Maximum Spindle Speed Designation

N0001 : Dimension words X and Z specify the turret indexing position.

Four-digit S word specifies the spindle speed.

T word with four-digit number when tool nose radius compensation is not used, or with six-digit number when it is used.

Two-digit M code specifies the spindle speed range.

N0002 : M03 starts the spindle rotation in the forward direction. (M04 in reverse)

M08 starts coolant supply.

N0050 : Dimension words X and Z return the turret to the indexing position.

M05 stops spindle rotation and M08 coolant supply.

N0051 : Provide M02 at the end of a part program.

Program Process Sheet

						PROCESS SHEET		
N	G	X(U)	Z(W)	I	K	F	S,T,M	
O104								
N001	G00	X800	Z2000					
N002		X250	Z100				M41	
N003			Z53				S120 M03 M08	
N004			Z43					
N005	G01	X205				F0.3		
N006	G00	X250	Z50					
N007			Z40					
N008	G01	X205				F0.25		
N009	G00	X230	Z58					
N010		X192						
N011	G01		Z48			F0.4		
N012		X200						
N013			Z40					
N014	G00	X210	Z49					
N015		X196						
N016	G01							

Shown above is an example of a program process sheet. It is a listing which details part programming instructions comprising alphabetic and numeric characters arranged in a definite programming scheme. With these instructions, operations of the machine are expressed.

Each line of the process sheet starts with Sequence Number (N word)
A group of commands written in one line is referred to as a "block" and it describes one machine operation.
Usually, the machine executes the programmed commands block by block in the order as programmed to produce a part.

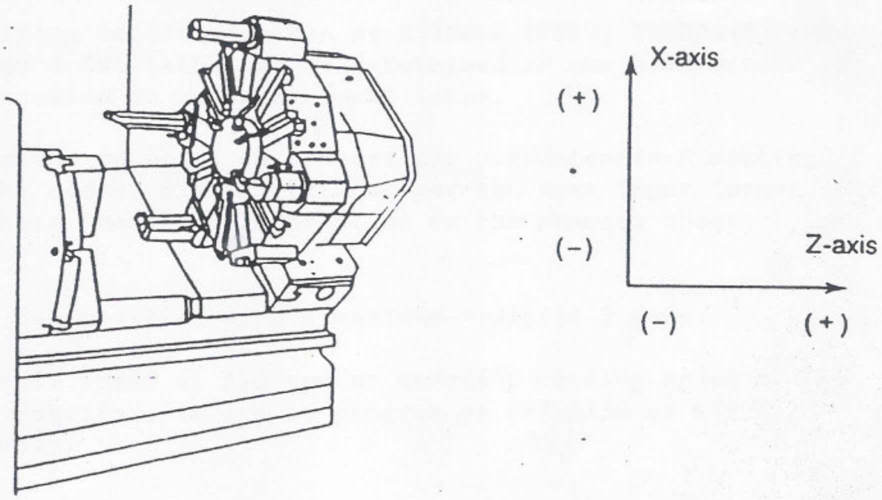
The steps preparing such program process sheet is called "programming".

AXIS AND MOTION NOMENCLATURE

Models LB9/LB12/LB15/LB25

The axis designation of LB9/LB12/LB15/LB15-II /LB25 CNC Lathes is as follows:

LB9/LB12/LB15/LB25



Transverse Turret Movement (infeeding direction) .... X-axis

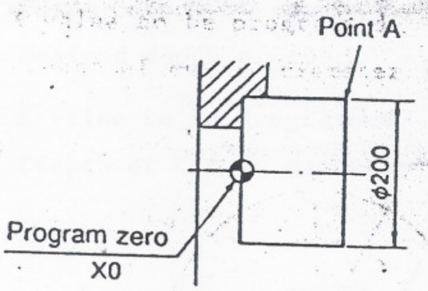
Longitudinal Turret Movement .... Z-axis

Direction of axis movement is defined by "+" and "−" sign characters.

X-Axis Command (1 mm unit command)

X-axis command is given in terms of diameter as indicated on a part drawing.

Example:

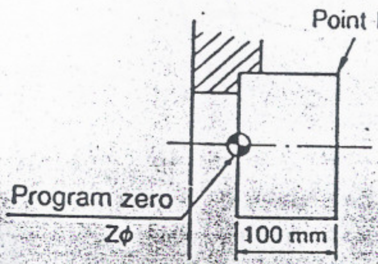


When "X200" command is specified, the turret moves along X-axis to position the cutting tool to Point A.

Z-Axis Command (1 mm unit command)

Z-axis command is given in terms of longitudinal dimensions referenced to the program zero.

Example:



When "Z100" command is specified, the turret moves along Z-axis to position the cutting tool to Point B.

HOW TO PROGRAM CUTTING CONDITION

Basically, the cutting conditions such as SPINDLE SPEED, FEEDRATE, and DEPTH OF CUT use on a CNC Lathe can be determined in the same manner as in the turning operation on a conventional lathe.

All the programmer has to do is to convert the pre-determined cutting conditions into the coded, digital form as per the data input format and to register these numerical information on the process sheet.

a) SPINDLE SPEED is specified with a maximum 4-digits S code.

   e.g If a spindle speed of 350 rpm or constant cutting speed of 120 m/min is required, it can be program as G97 S350 or G96 S120 respectively.

b) FEEDRATE is specified with a maximum of 5-digits in Metric system F code.

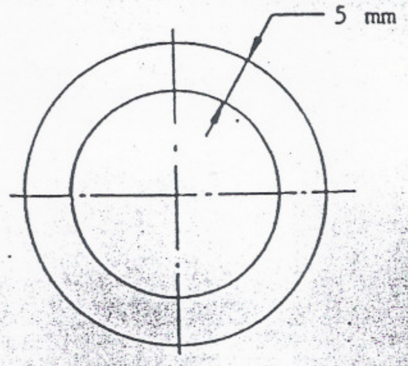
   e.g If a feedrate of 0.35 mm/rev is required, it can be programmed as G95 F0.35.

c) DEPTH OF CUT is half the difference tween the stock diameter and the programmed X value (in diameter).

Example :

Where, stock diameter = 100 mm
Desired depth of cut = 5 mm

X value to be programmed = 100 - (5x2)
(depth of cut in diameter) = 90mm



Determining Cutting Conditions

Cutting conditions such as spindle speed, feedrate and depth of cut are selected more or less in the same manner as with a conventional lathe. With CNC lathes, these conditions are entered in a program process sheet using alphanumerics in the predetermined format.

Formulas used to determine cutting conditions:

$$V = \frac{\pi \times D \times n}{1000}$$

where, V = cutting speed, m/min
π = circular constant (3.14)
D = workpiece diameter, mm
n = spindle speed, rpm

$$HP = \frac{V \times K \times t \times f}{75 \times 60}$$

$$kW = \frac{V \times K \times t \times f}{6000}$$

where, HP = required horse power, hp
kW = required power, kW
V = cutting speed, m/min
K = cutting resistance, kg/mm2
Example: 200 for S45C (JIS, carbon steel)
120 for FC20 (JIS, cast iron)
t = depth of cut, mm
f = feedrate, mm/rev.

Reference Point

On the CNC Lathes, there are two reference points as below:

(1) Machine Origin

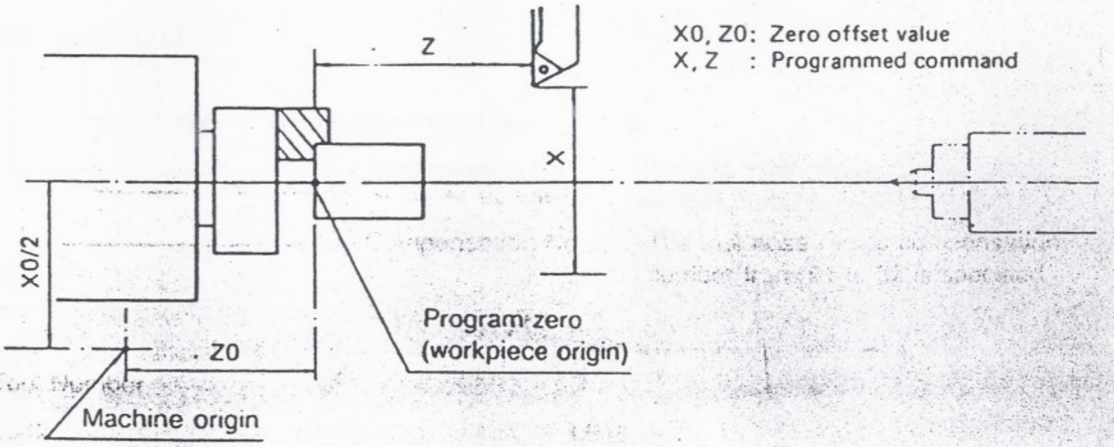
This is the fixed origin inherent in respective machines, where output values from the OSP position encoders of X- and Z-axis become zero. Axis movement of the machine is referenced to this fixed machine origin.

(2) Program Zero (Workpiece Origin)

The zero point that can be set at any point as needed by "Zero Offset" operation; once the workpiece zero is set, axis motion is controlled on the coordinate system that has the origin at the set workpiece zero.

Cutting is carried out taking this point as the reference point.

A programmer should inform a machine operator of the established programming zero position either on the part drawing or with a memorandum.

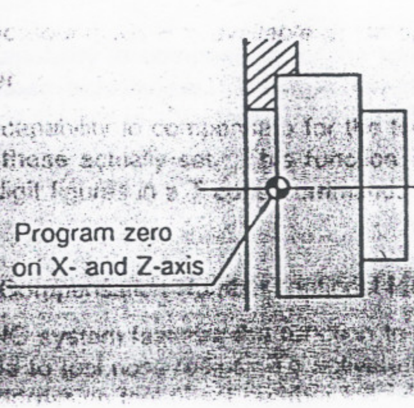


Program Zero

The Program Zero is the reference point for programming and may be established at any required point within a specified range.

Usually, it is taken at the center of the workpiece for X-axis and at the left-hand end of it for Z-axis, and programming is made in the coordinate system which has the origin at such point.

See the figure below showing the preferable zero position.



Tool Selection

LB Series

Selection of a cutting tool is made by four-digit figures following address character T.

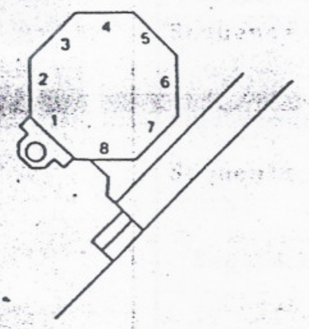
```
T  OO  OO
   ↑   ↑
   |   └──────────── Tool Offset No.  :  32 pairs (01 to 32)
   |                                     For two-spindle model, 32 pairs of tool
   |                                     offsets are used on the individual
   |                                     turrets.
   └──────────────── Tool No.         :  The number identifying the turret face
                                         position is used as a tool number.
```

When the control features the tool nose radius compensation function, a T word comprises six-digit figures. (optional for OSP500L-G)

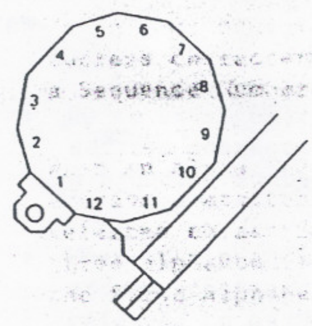
```
T  OO  OO  OO
   ↑   ↑   ↑
   |   └───┴──────── Same as above
   └──────────────── Compensation No. :  The tool nose radius compensation
                                         number from 01 to 32 is specified.
```

(1) Tool Number

LB12



LB15



Note 1: *As a tool number, specify the turret face position number indicated by a name plate.*

Note 2: *The octagonal turret is available as an option for LB12.*

(2) Tool Offset Number

The control has a capability to compensate for the tool dimensions between those assumed in programming and those actually set. This function is called the tool offset function and it is called out by two-digit figures in a T code command. Programmable tool offset number is from "1" to "32".

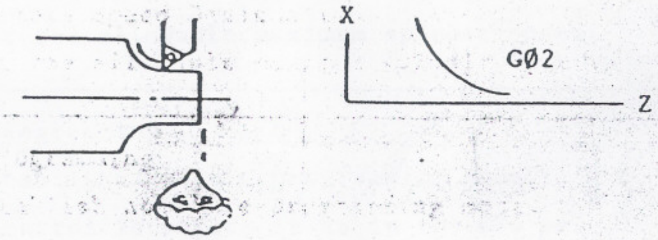
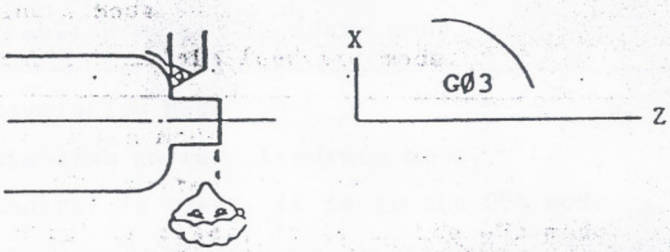
(3) Tool Nose Radius Compensation Number (optional for OSP500L-G)

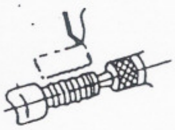
The OSP5020L CNC system features the function to automatically compensate for the error in finished contour due to tool nose radius. To activate this function, specify two-digit figures, 1 through 32, as addressing tool offset number following address character T.

WORD FORMAT

Word Format	Associated Information and Function
Ooooo	Program Number or Program Name Entered at the beginning of a part program to identify respective programs. Program Number : Up to four numeric characters following address character "O" are used to indicate a Prôgram Number. Program Name : When an alpha character appears following address character "O", such expression is referred to as "Program Name". Up to three alphanumerics can be used following the first aiphabetic character.
Noooo	Sequence Number or Sequence Name Entered at the beginning of each block to identify respective blocks in a part program. Numbers are usually used to indicate the execution order of blocks . Although program numbers are generally assigned with consecutive numbers, such number may not necessarily be consecutive. Sequence name is used to identify a specific block in a part program. Sequence Number : Up to four numeric characters following address character "N" are used to indicate a Sequence Number. Sequence Name : When an alpha character appears following address character "N", such expression is referred to as "Sequence Name". Up to three alphanumerics can be used following the first alphabetic character.
X±ooo.ooo	Dimension Word : Diameter Used to specify X-axis coordinate in diameter
Z±ooo.ooo	Dimension Word : Longitudinal Dimension Used to specify Z-axis xoordinate

G CODES

Code	Associated Information and function
Goo	G Code: Three numeric characters following address character G establishes the mode of axis movements.
G00	Rapid Feed Used to feed the axes at a rapid feedrate to the commanded coordinate position.
G01	Linear Interpolation Used to cut a straight line parallel to X- or Z-axis or a taper. Feedrate to be employed in this mode is commanded by an F word, Foooo.
G02	Circular Interpolation, CW Used to cut an arc in the clockwise direction. Feedrate to be employed is commanded by an F word, Fooooo, as in G01 mode. 
G03	Circular Interpolation, CCW Used to cut an arc in the counterclockwise direction. Feedrate to be employed is aommanded by an F word, Foooo, as in G01 mode. 
G04	Dwell Used to activate dwell function which stops axis motion for any required duration of time during a machining cycle. Duration of dwell movements is programmed in an F word : G04 Foooo. G04 F12.3 stops axis motion for 12.3 seconds, for instance.

Code	Associated Information and Function
G33	Fixed Thread Cutting Cycle : Longitudinal  Automatic thread cutting cycle as shown at the left is executed.
G40	Tool Nose Radius Compensation : Cancel Used to cancel the tool nose radius compensation function
G41	Tool Nose Radius Compensation: ID Ordinally Cutting Used to call out the tool nose radius compensation mode for ordinally ID cutting cycle.
G42	Tool Nose Radius Compensation: OD Ordinally Cutting Used to call out the tool nose radius compensation mode for ordinally OD cutting cycle.
G50	Maximum Spindle Speed Designation Used to set the allowable maximum spindle speed.
G90	Absolute Programming Used to establish absolute programming mode. When the control is reset, it is in the G90 mode.
G91	Incremental Programming Used to establish incremental programming mode.
G94	Feed per Minute Mode Used to establish mm/min feedrate mode.
G95	Feed per Revolution Mode Used to establish mm/rev. feedrate mode. When the control is reset, it is in the G95 mode.
G96	Constant Speed Cutting ON Used to establish the constant speed cutting mode.
G97	Constant speed cutting OFF Used to cancel the constant speed cutting mode.

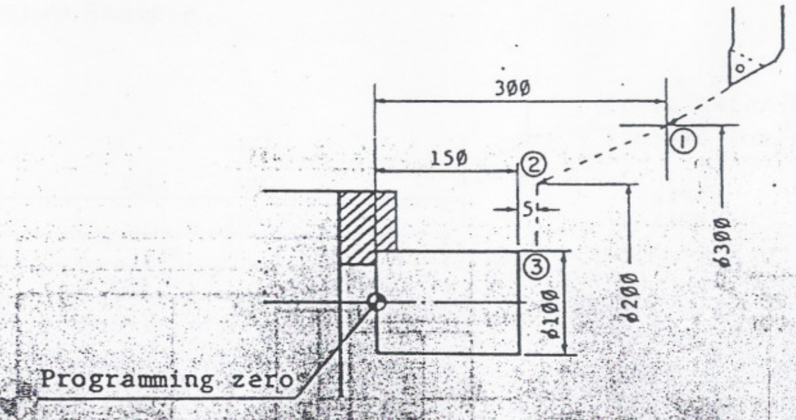
GOO POSITIONING

(1) Format

GØØ Xoooo.ooo Zoooo.ooo

With the commands indicated above, positioning to the programmed coordinate point is carried out at a rapid traverse rate.

(2) Example Program




```
NØØ1  GØØ   XЗØØ  Z3ØØ
NØØ2 (GØØ)  X2ØØ  Z155
NØØ3 (GØØ)  X1ØØ (Z155)
```

Commands in ( ) may be omitted.

NØØ1: Positioning is made to X3ØØ, Z3ØØ position at a rapid traverse rate.

NØØ2: Positioning is made to X2ØØ, Z155 position at a rapid traverse rate.

NØØ3: Positioning is made to X1ØØ along X-axis at a rapid traverse rate. No Z-axis movement occurs.

Note 1: For rapid traverse rates along individual axes,

Note 2: Tool path during positioning is not always straight.

Note 3: In the GØØ positioning mode, acceleration and deceleration are automatically activated.

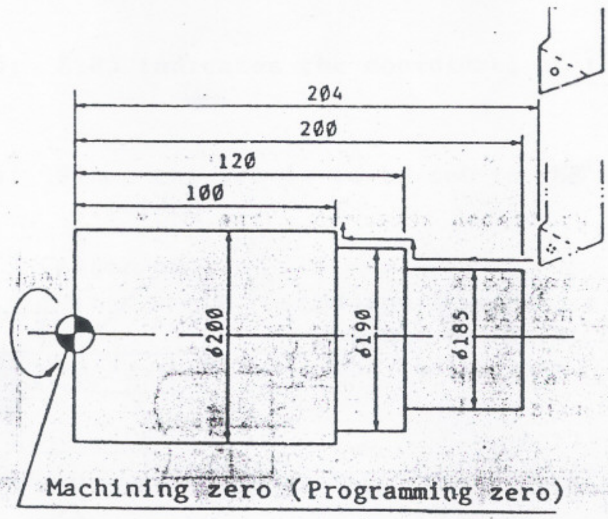
G01 STRAIGHT-LINE CUTTING

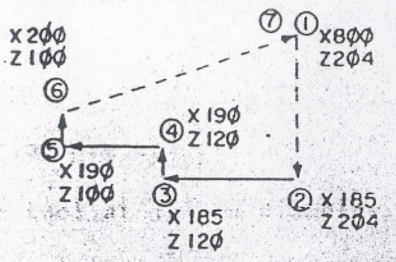
(1) Format

GØ1 Xoooo.ooo (Zoooo.ooo) Fo.ooo

With the commands above, axis movement from the current position to the commanded position is performed along the straight line parallel to either X- or Z-axis at a feedrate specified by an F word.

(2) Program Example





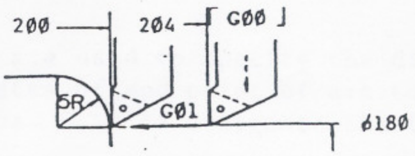
The reference point for cutting (programming zero) may be taken at any position convenient for programming. However, it is recommended to take such a point at the center of the workpiece left end face. This will ease programming since such a point permits the programmer to directly enter the dimensions indicated on a part drawing.

Feedrate:

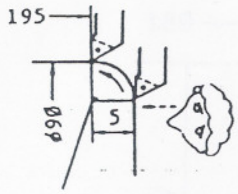
a) A desired feedrate is commanded by an F word. With FØ.3 command, axis is fed at Ø.3 mm/rev. For an F word, numerical data smaller than the selected unit system can be specified.

b) To feed the axis at a rate of Ø.1 mm/rev., specify FØ.1.

N0003: G01 Z200 feeds the cutting tool to the starting point of the arc to be cut at the specified feedrate.



N0004: Since the arc is to be cut in the counterclockwise direction, G03 is provided.



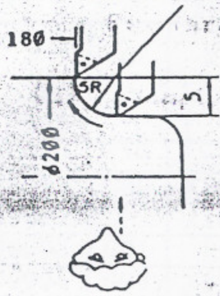
Center of arc

X and Z words are used to specify the coordinates of the end point of the arc. L word is to specify the radius size.

N0005: Z185 indicates the coordinate of the starting point of arc.

N0006: Since the arc is to be cut in the clockwise direction, G02 code is provided.

Center of arc



X and Z words are used to specify the coordinates of the end point of the arc. L word is to specify the radius size.

N0007: X800 Z204 returns the cutting tool to the starting point.

M05 stops the spindle rotation.

M09 stops the coolant supply.

N0008: M02 resets the control.

Note 1: In the block containing either G02 or G03 calling for circular interpolation mode, L word must be commanded.

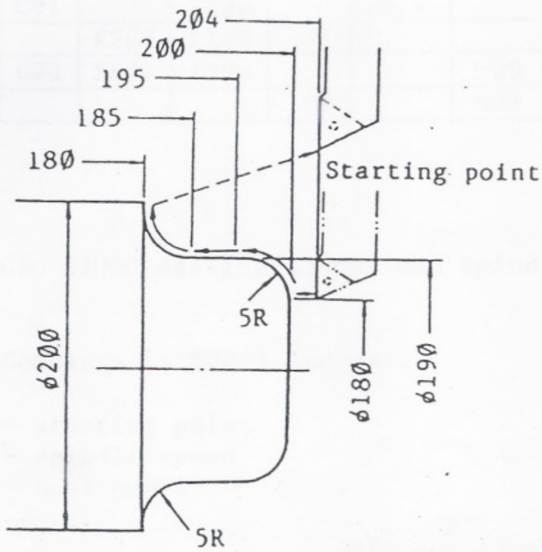
Note 2: L word must be specified in radius.

GO2/GO3 ARC CUTTING

Format

GO2/GO3 X1 Z1 L F

GO2 and GO3 are used to specify the direction of arc. X and Z words indicate the coordinates of end point of arc to be cut and L is to specify the size of the radius. F is to designate the feedrate.



N	G	X	Z	L	F	S,T,M
N0000	G50					S3000
NØØØ1	GØØ	X8ØØ	Z2Ø4			S2ØØ TØ1Ø1
NØØØ2		X18Ø				MØ8 MØ3
NØØØ3	GØ1		Z2ØØ		FØ.2	
NØØØ4	GØ3	X19Ø	Z195	L5		
NØØØ5	GØ1		Z185			
NØØØ6	GØ2	X2ØØ	Z18Ø	L5		
NØØØ7	GØØ	X8ØØ	Z2Ø4			MØ9 MØ5
NØØØ8						MØ2

N0000: G50 S3000 designates maximum spindle speed.

NØØØ1: Commands in NØØØ1 indicate:

- starting point
- spindle speed
- tool number

NØØØ2: X18Ø moves the cutting tool to X18Ø position, X coordinate of the starting point of the arc to be cut.

MØ8 starts coolant supply.

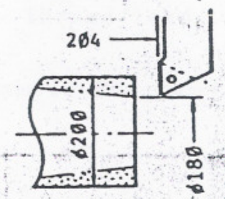
N	G	X	Z	I,K	F	S,T,M
N0000	G50					S3000
NØØØ1	GØØ	X8ØØ	Z2Ø4			S2ØØ TØ1Ø1
NØØØ2		X185				MØ8 MØ3
NØØØ3	GØ1		Z2ØØ		FØ.5	
NØØØ4		X2ØØ	Z1ØØ			
NØØØ5	GØØ	X8ØØ	Z2Ø4			MØ9 MØ5
NØØØ6						MØ2

N0000: G50 S3000 designates maximum spindle speed.

NØØØ1: Commands in NØØØ1 indicate:

- starting point
- spindle speed
- tool number

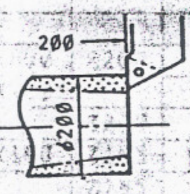
NØØØ2:



X185 positions the tool at 185 mm diameter position.

MØ3 starts spindle rotation in the forward direction.

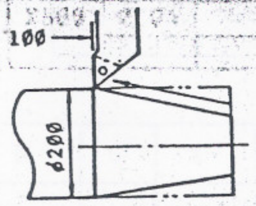
NØØØ3:



GØ1 Z2ØØ feeds Z-axis up to 2ØØ mm position at a rate commanded by an F word.

FØ.5 determines the feedrate in GØ1 mode as Ø.5Ø mm/rev.

NØØØ4:



With X2ØØ Z1ØØ command, cutting tool is fed to the end point for taper cutting.

NØØØ5: GØØ X8ØØ Z2Ø4 returns the cutting tool to the starting point.

MØ5 stops spindle rotation.

MØ9 stops coolant supply.

NØØØ6: MØ2 resets the control.

G04 DWELL TIME

With G04, the tool is stopped for a designated period. The numerals are after F word indicate the unit in seconds.

Format: G04 F....

A dwell for 2.5 seconds.

N1000 G04 F2.5

Note: G04 function is effective only for the block commanded.

G33 THREAD CUTTING (STRAIGHT, CONSTANT LEAD THREAD

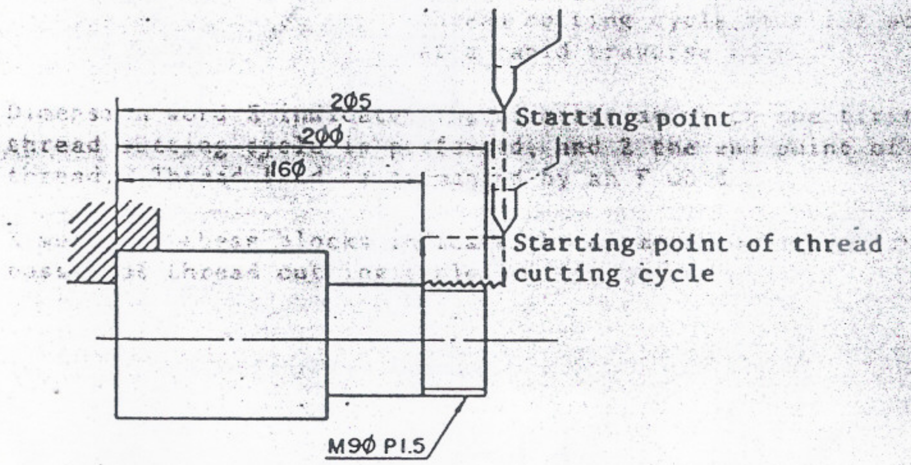
(1) Format

```
G00  Xoooo.ooo  Zoooo.ooo
G33  Xoooo.ooo  Zoooo.ooo  Foo.ooo
     Xoooo.ooo
     Xoooo.ooo
```

Thread cutting is performed in G33 mode.

X: Diameter of each thread cutting cycle
Z: End point of thread in longitudinal direction.
F: Thread lead

(2) Example Program



N	G	X	Z	I,K	F	S,T,M
N000	G50					S3000
NØØ1	GØØ	X8ØØ	Z2Ø5			S35Ø TØ1Ø1
NØØ2		X12Ø				MØ8 MØ3
NØØ3	G33	X89	Z16Ø		F1.5	
NØØ4		X88.5				
NØØ5		X88.3				
NØØ6		X88.1				
NØØ7		X88				
NØØ8	GØØ	X8ØØ	Z2Ø5			MØ9 MØ5
NØØ9						MØ2

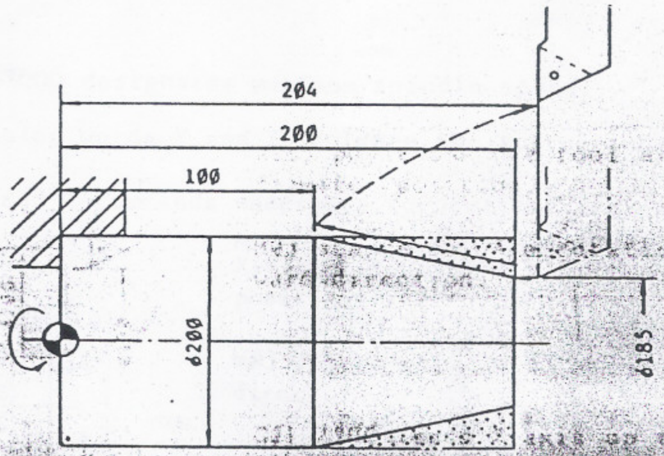
G01 TAPER CUTTING

(1) Format

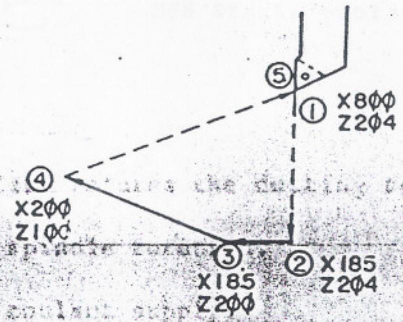
GØ1 Xoooo.ooo Zoooo.ooo Fo.ooo

The same commands as straight-line cutting are used for cutting a taper.

(2) Example Program



(Coordinates Commanded on Process Sheet)



N	G	X	Z	I,K	F	S,T,M
NØØØØ	G50					S3000
NØØØ1	GØØ	X8ØØ	Z2Ø4			S2ØØ TØ1Ø1
NØØØ2		X185				MØ3
NØØØ3	GØ1		Z12Ø		FØ.5	MØ8
NØØØ4	[ ]	X19Ø				
NØØØ5		[ ]	Z1ØØ			
NØØØ6		X2ØØ	[ ]			
NØØØ7	GØØ	X8ØØ	Z2Ø4			MØ5 MØ9
NØØØ8						MØ2

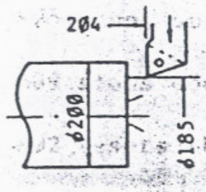
Commands in [ ], where the same command as provided in the previous block is to be entered, can be omitted.

N0000: G50 S3000 designates maximum spindle speed.

NØØØ1: Dimension words X and Z indicate the turret indexing position.

M, S and T commands necessary for cutting are entered.

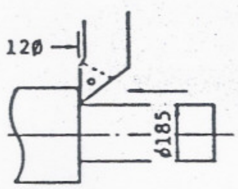
NØØØ2:



X185 positions the tool at 185 mm diameter position.

MØ3 starts spindle rotation in the forward direction.

NØØØ3:

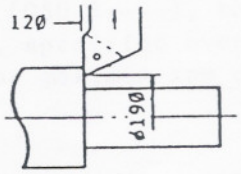


GØ1 Z12Ø feeds Z-axis up to 12Ø mm position at a rate commanded by an F word.

FØ.5 determines the feedrate in GØ1 mode as Ø.5Ø mm/rev.

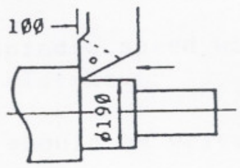
MØ8 starts coolant supply.

N0004:



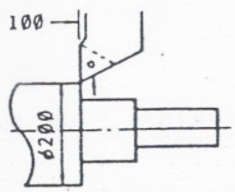
G01 X190 feeds X-axis up to 190 mm diameter position at 0.5 mm/rev. which is specified in the preceding block to finish the shoulder.

N0005:



G01 Z100 feeds Z-axis up to 100 mm position at 0.5 mm/rev.

N0006:



G01 X200 feeds X-axis up to 200 mm diameter position at 0.5 mm/rev. to finish the shoulder.

N0007: G00 X800 Z204 returns the cutting tool to the starting point.

M05 stops spindle rotation.

M09 stops coolant supply.

N0008: M02 resets the control.

G50 SETTING OF MAXIMUM SPINDLE SPEED

If the maximum spindle speed is specified with a 4 digit S value preceded by G50 code (G50 S----), the actual spindle speed will not exceed the maximum RPM, specified even when the programmed S value calls for the speed exceeding the maximum rpm of that range.

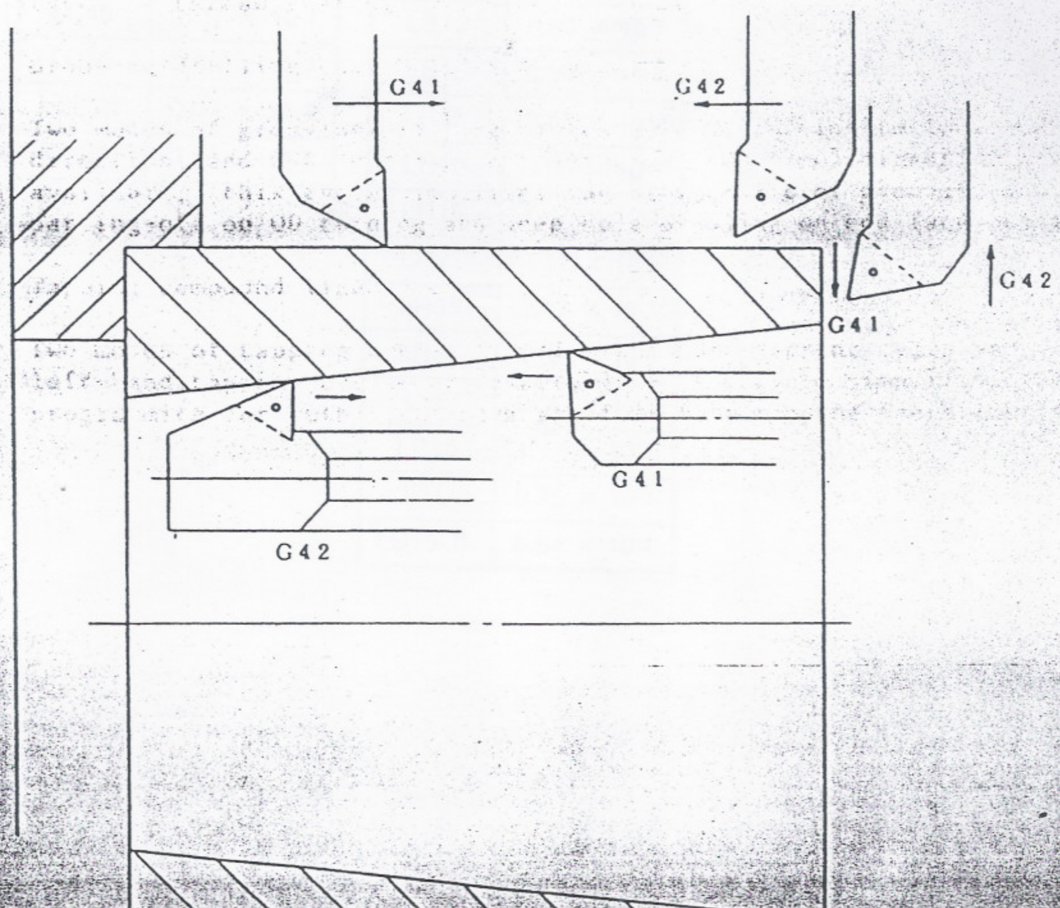
Format :

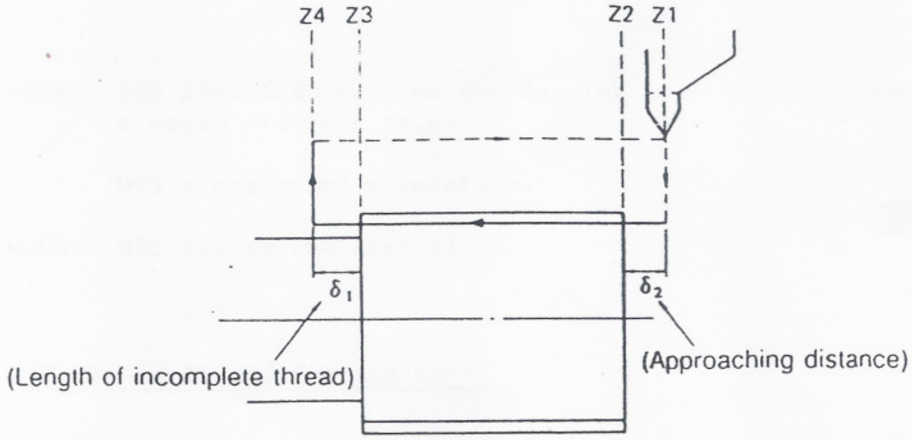
N0002: G50 S3000

The actual spindle speed will not exceed 3000 rpm if the speed is specified as S4000 by mistake.

G50 command should be written in an individual block by itself.

G41 & G42 TOOL NOSE RADIUS COMPENSATION





Values $\delta 1$ and $\delta 2$ vary depending on cutting conditions. Generally, values $\delta 1$ and $\delta 2$ must satisfy the following equations:

$\delta 1 > K \cdot N \cdot P$
$\delta 2 > K \cdot N \cdot P$

where,

N: spindle speed
P: lead
K: machine model dependent constant

Values for constant K for individual models are indicated below:

Model	K
LC30	1.07×10^{-3}
LC40	1.07×10^{-3}
LC50	0.96×10^{-3}
LS30N	0.87×10^{-3}
LH35	0.96×10^{-3}
LH55	0.96×10^{-3}

Model	K
LB9	0.96×10^{-3}
LB12	0.96×10^{-3}
LB15	0.96×10^{-3}
LB25	0.85×10^{-3}
LR10	0.85×10^{-3}
LR15	1.07×10^{-3}
LR25	1.17×10^{-3}
LR35	1.07×10^{-3}
LR45	1.07×10^{-3}
LP6	0.96×10^{-3}
LP15	1.17×10^{-3}
FTL	0.75×10^{-3}
LB15-II	0.64×10^{-3}

Example:

For the LB15-II, with a peripheral speed of 100 m/min, a 10 mm diameter and a thread lead of 1.5 mm, spindle speed and feedrate are calculated as below.

Spindle speed : $N = \dfrac{100 \times 10^3}{10} = 3{,}183$ (rev/min)

Feedrate : $N \times P = 3{,}183 \times 1.5 = 4{,}775$ (mm/min)

Since $K = 0.64 \times 10^{-3}$, $\delta 1$ and $\delta 2$ must be greater than 3.05 mm which is calculated as below:

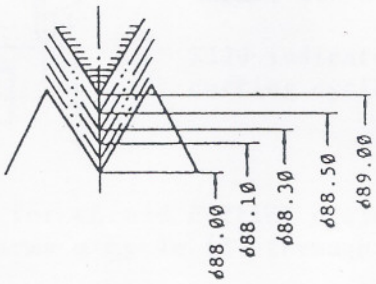
$0.64 \times 10^{-3} \times 4{,}775 = 3.05$ (mm)

NØØ8: GØØ X8ØØ Z2Ø5 returns the cutting tool to the starting point at a rapid traverse rate.

MØ5 stops spindle rotation.

NØØ9: MØ2 resets the control.

X Words in Thread Cutting Cycle



Note 1: Number of infeeds in thread cutting cycle should be selected according to material to be cut, thread lead, etc.

Note 2: NEVER CHANGE SPINDLE SPEED WHILE THREAD CUTTING CYCLE.

PRECAUTIONS FOR PROGRAMMING THREAD CUTTING CYCLES

Observe the following points when programming thread cutting cycles:

1) Spindle Speed Change During Thread Cutting Cycle

If the spindle speed change is intended while thread cutting cycle, it will shift the starting point of the thread cutting cycle, thus damaging the thread being cut.

Therefore, NEVER CHANGE SPINDLE SPEED WHILE THREAD CUTTING CYCLE.

2) Feedrate Override

The feedrate override dial is inoperative while thread cutting cycle.

3) Extra Length in Thread Cutting Program

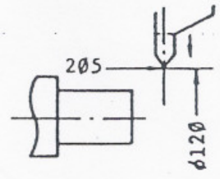
Since certain length of incomplete thread is usually near start and end point of the cut, it is necessary to add proper amount $\delta 1$ and $\delta 2$ to the start and from the end of the thread to be cut for cutting proper shape of thread.

NOOO: G50 S3000 designates maximum spindle speed.

NØØ1: Commands in NØØ1 indicate:

- starting point
- spindle speed
- tool number

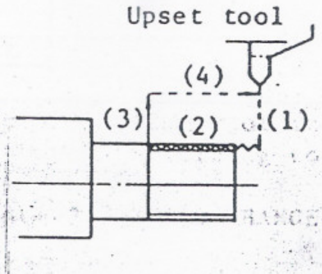
NØØ2:



MØ3 starts spindle rotation in the normal direction.

X12Ø indicates the X coordinate of thread cutting cycle starting point.

NØØ3: G33 calls for thread cutting cycle in which the cutting tool performs a cycle (1) through (4).



(1) The cutting tool moves to the first thread cutting diameter position at a rapid traverse rate.

(2) The cutting tool is then fed along Z-axis at a feedrate specified by the F word (1.5 for thread lead of 1.5 mm).

(3) The cutting tool retracts from the workpiece at the designated feedrate.

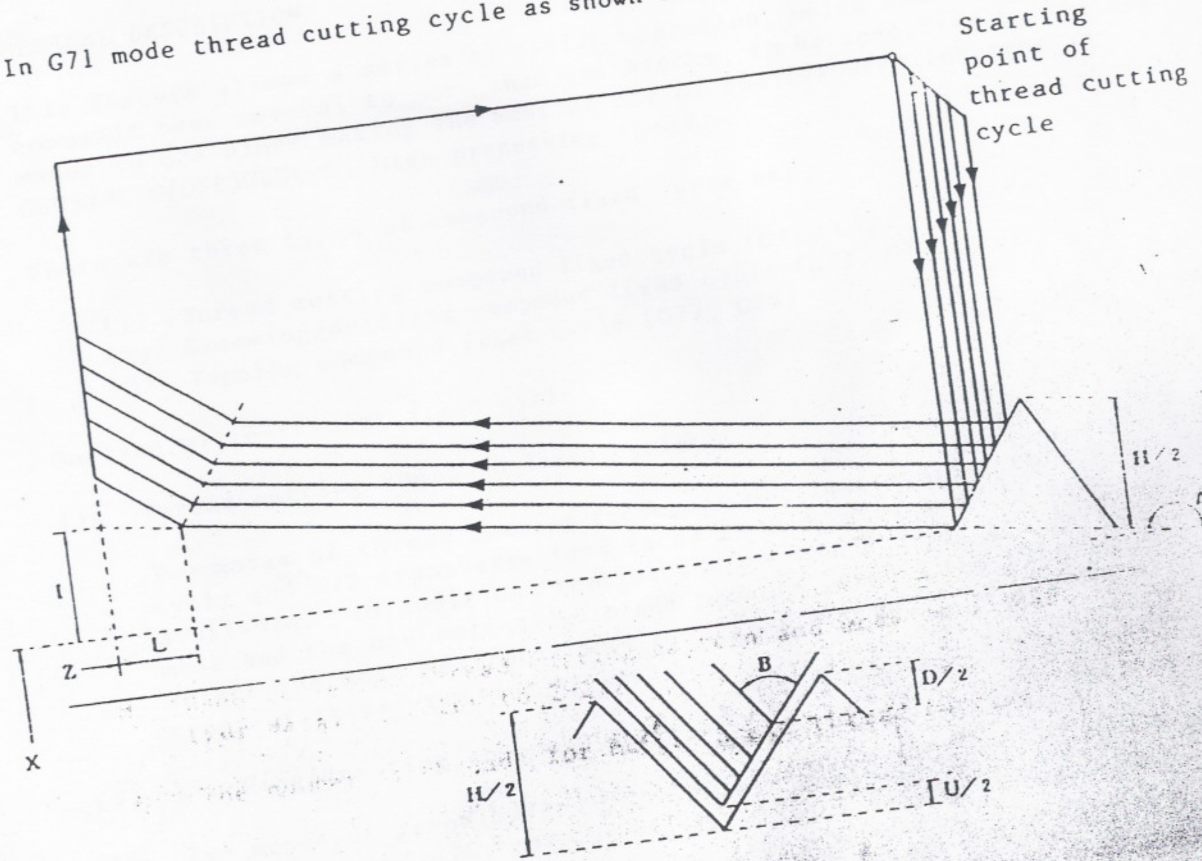
(4) The cutting tool returns to the thread cutting cycle starting point at a rapid traverse rate.

Dimension word X indicates the diameter at which the first thread cutting cycle is performed, and Z the end point of the thread. Thread lead is commanded by an F word.

NØØ4: X words in these blocks indicate the diameter of respective
:
NØØ7 passes of thread cutting cycle.

G71 LONGITUDINAL THREAD CUTTING CYCLE

In G71 mode thread cutting cycle as shown below is performed:



Format:

N∅∅∅1 G71 X Z A(I) B D U H L F J M Q

Description of each word:

X: Final diameter of thread

Z: Z coordinate of end point of thread

A: Taper angle

I: Difference in radius between starting point and end point for taper thread (expressed in radius)

For taper thread, use either A or I word.

COMPOUND FIXED CYCLE (SPECIAL FIXED CYCLE)

GENERAL DESCRIPTION

This feature allows a series of cyclic operation, which usually requires commands over several to more than ten blocks, to be specified by the commands in one block making the most of one of the features inherent in OSP500L-G/OSP5000L-G, high processing speed.

There are three types of compound fixed cycle as:

(1) Thread cutting compound fixed cycle (G71, G72)
(2) Grooving/drilling compound fixed cycle (G73, G74)
(3) Tapping compound fixed cycle (G77, G78)

Outline of individual fixed cycle

(1) Thread cutting compound fixed cycle

Two modes of thread cutting cycles as G71 longitudinal thread cutting cycle and G72 transverse (end face) thread cutting cycle are available. In addition, combination of M code designating cutting mode and the one selecting infeed pattern permits the programmer to select the most desirable mode of thread cutting from available six types of thread cutting cycles.

(2) Grooving/drilling compound fixed cycle

Two modes of grooving/drilling cycles as G73, cutting in longitudinal direction, and G74, cutting in transverse (end face) direction are available; this cycle simplifies the programming of grooving and parting-off on OD turning and deep hole drilling on end face cutting.

(3) Tapping compound fixed cycle

Two modes of tapping cycles as G77 right-hand tapping cycle and G78 left-hand tapping cycles are available. This cycle simplifies the programming for both right-hand and left-hand tapping operations.

Format:

NØØØ1 G72 X Z A (K) B D W H L E F J M Q

Description of each word:

X: X coordinate of end point of thread

Z: Z dimension of final thread cutting cycle

A: Taper angle

K: Distance between starting point and end point for taper thread

For taper thread, use either A or K word.

B: Cutter tip point angle
($0° \leq B < 180°$; $0°$ if no B command is provided.)

D: Depth of cut in the first thread cutting cycle

W: Finishing allowance
(No finishing cycle is performed if a W word is not provided.)

H: Thread height

L: Chamfering distance in final thread cutting cycle
(Effective in M23 mode; if no L word is provided in M23 mode, L is assumed to the distance equivalent to one lead.)

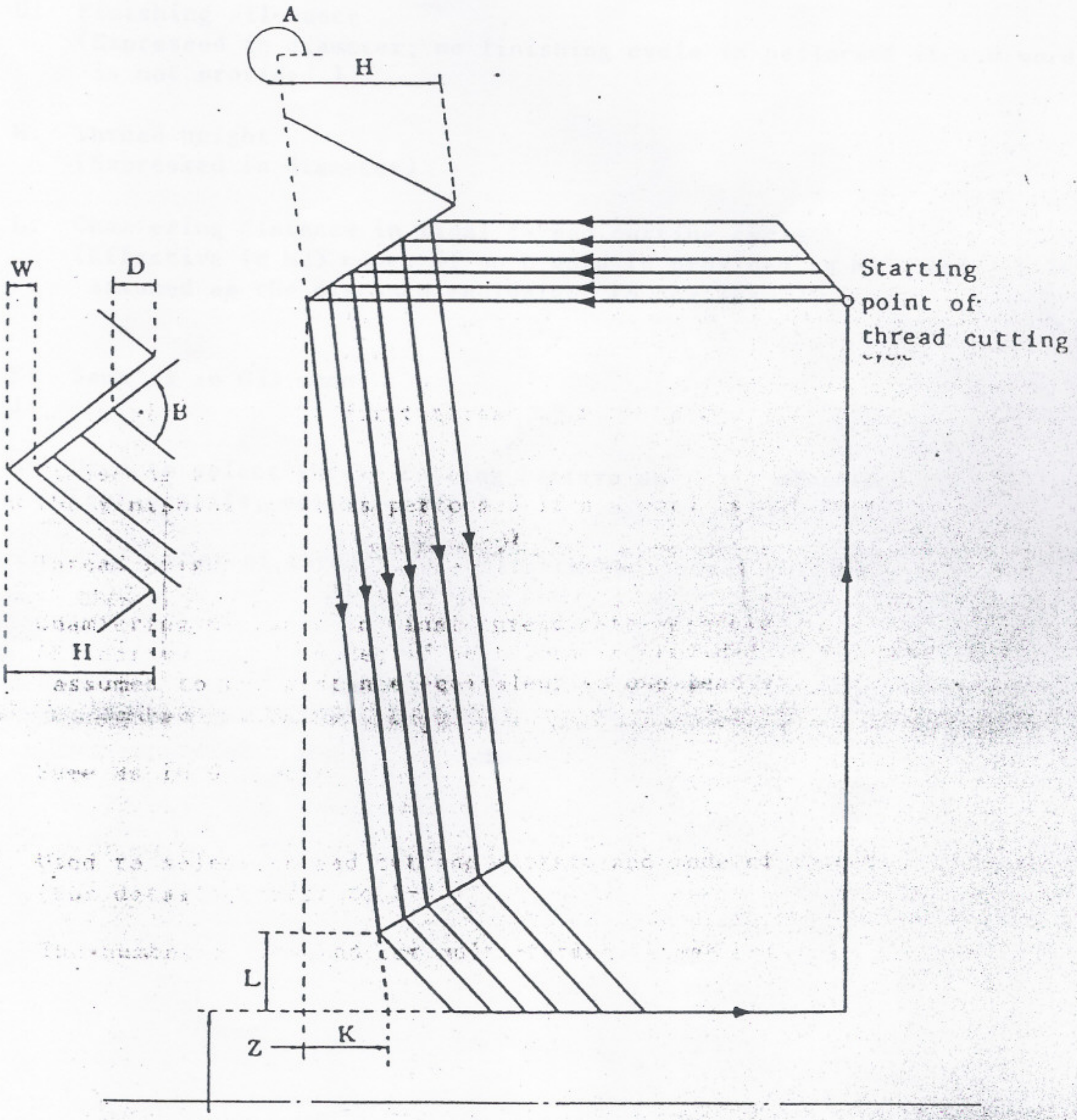
E:
F: Same as in G32 mode
J:

M: Used to select thread cutting pattern and mode of infeed.
(For details, refer to 2-3.)

Q: The number of threads for multi-thread thread cutting

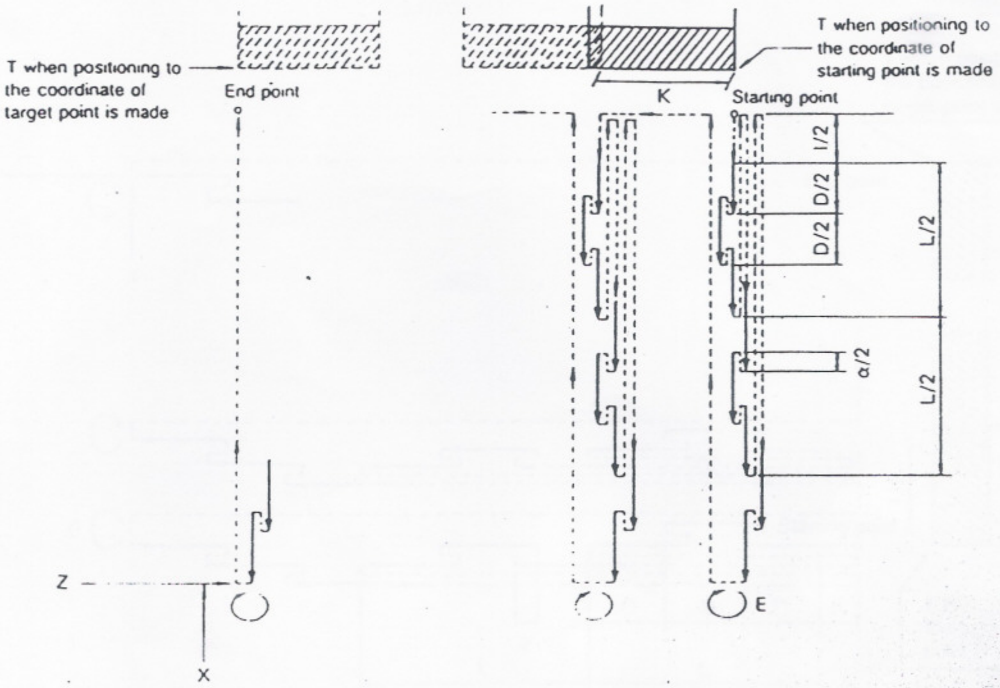
G72 TRANSVERSE THREAD CUTTING COMPOUND FIXED CYCLE

In this fixed cycle, thread cutting cycle as shown below is performed.



Longitudinal Grooving Fixed Cycle (G73)

In G73 mode, grooving cycle as shown below is performed.



Format:

N0001 G73 X Z I K D L F E T DA

Description of each word:

X : X coordinate of target point

Z : Z coordinate of target point

I : Shift amount in X-axis direction
(in diameter; 0 if no I word is provided)

K : Shift amount in Z-axis direction
(0 if no K word is provided)

D : Depth of cut (infeed amount)

L : Total infeed amount for tool withdrawal motion (in diameter; tool sequence is not performed when L word is not specified.)

DA: Retraction amount "α" is specified. When no DA word is provided, the amount set with the optional parameter (long word) No. 7 is used as the retraction amount. This applies both in the G94 and G95 modes.

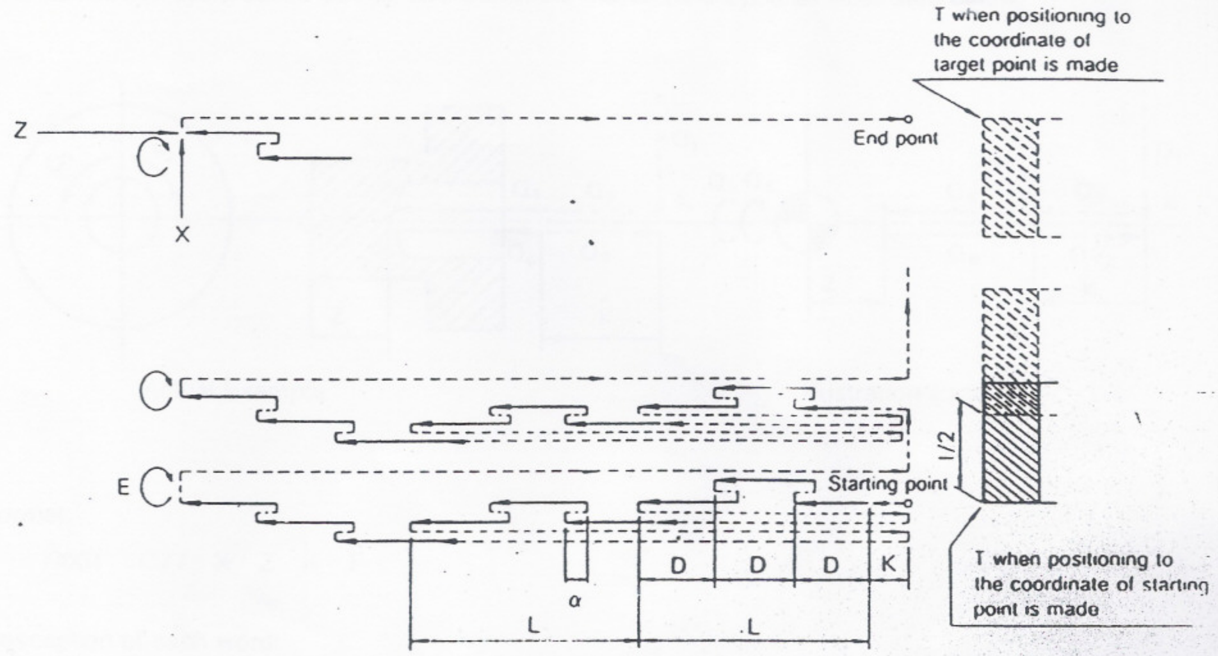
Note that DA command is not effective for A specification.

E : Duration of dwell motion when target point on X-axis is reached (Command unit is the same as an F word in G04 mode.)
(If no E word is provided, this sequence is not performed.)

T : Tool offset number determining the tool offset amount when target point on Z-axis is reached.
(If no T word is provided, tool offset number selected at positioning to the starting point of grooving cycle is selected.) T command after this block is the one designated when positioning to the starting point is performed.)

Transverse Grooving Fixed Cycle (G74)

In G74 mode, grooving cycle as shown below is performed.



Format:

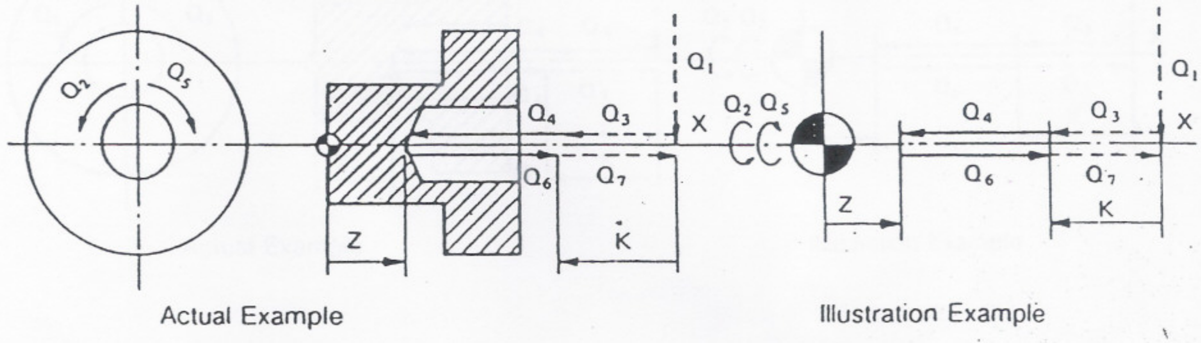
N0001 G74 X Z I K D L F E T DA

Description of each word:

- X : X coordinate of target point
- Z : Z coordinate of target point
- I : Shift amount in X-axis direction
  (in diameter; 0 if no I word is provided)
- K : Shift amount in Z-axis direction
  (0 if no K word is provided)
- D : Depth of cut (infeed amount)
- L : Total infeed amount for tool withdrawal motion
  (The sequence is not performed when L word is not specified.)
- DA: Retraction amount "α" is specified. When no DA word is provided, the amount set with the optional parameter (long word) No. 7 is used as the retraction amount. This applies both in the G94 and G95 modes.
- E : Duration of dwell motion when target point on Z-axis is reached (command unit is the same as an F word in G04 mode.)
  (If no E word is provided, this sequence is not performed.)
- T : Tool offset number determining the tool offset amount when target point on X-axis is reached.
  (If no T word is provided, tool offset number selected at positioning to the starting point of grooving cycle is selected. T command after this block is the one designated when positioning to the starting point is performed.)

Right-hand Tapping Cycle (G77)

The compound cycle called out by G77 conducts the tappind cycle as illustrated below.



Format:

N001 G77 X Z K F

Description of each word:

G77: G code to call out tapping compound fixed cycle.

Specify this G code in the next place following a sequence number (name).

X : X coordinate of tapping cycle start point (target point)

Z : Z coordinate of tapping cycle end point (target point)

K : Rapid axis feedrate for axis feed from the cycle start point to the cutting start point

F : Feedrate

Axis movements:

Q1 : Positioning of X-axis at the specified positioning target point (cycle start point) at a rapid feedrate. In this positioning cycle, no Z-axis movement occurs and thus the turret must have been positioned at a position where not interfere with the workpiece during this positioning before calling out the G77 cycle.

Q2 : The spindle rotates clockwise at a speed active before the G77 cycle is called. Therefore, it is necessary to specify required spindle speed before calling the G77 cycle.

If this compound fixed cycle is called without designating a spindle speed, axis infeed does not occur since the spindle does not rotate and thus the cycle is halted.

Q3 : Positioning of Z-axis at a position designated by a K word at a rapid feedrate.

Q4 : Tapping from the point reached in Q3 to the depth specified by a Z word at a specified feedrate (F).

Q5 : The spindle stops once and then starts in the reverse direction at the same speed as used in infeeding.

Q6 : Z-axis retraction to a point reached in Q4 cycle at a cutting feedrate.

Q7 : Z-axis retraction to a point reached in Q3 cycle at a rapid feedrate.

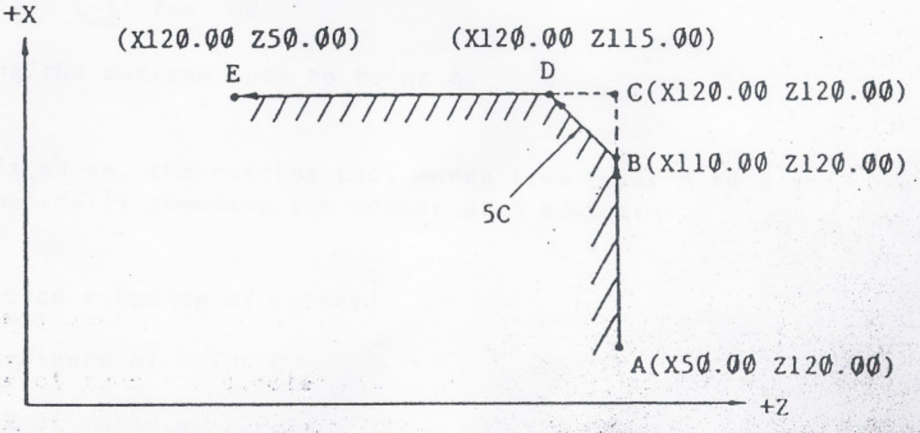
G75 AUTOMATIC CHAMFERING

When cutting a workpiece, it is often necessary to chamfer the sharp edge (either 45 deg. chamfering or rounding). Such chamfering can be accomplished using conventional taper and circular interpolation G codes such as GØ1, GØ2 and GØ3. However, this Automatic Chamfering Function permits chamfering to be done in a simple programming.

Commands used in this feature are:

G75: Chamfering at 45 deg.
G76: Rounding
L : Size of chamfering

45 DEG. CHAMFERING (G75)



To cut the above shown contour along the Points A, B, D and E, program as:

<u>G75</u> GØ1 <u>X12Ø</u> <u>L-5</u> Foo CR

after positioning the cutting tool to Point A.

With the commands above, the cutting tool moves from Point A to B and then to D, thus automatically chamfering the corner at 45 deg. with the size of 5 mm.

G75 : Specifies chamfering at 45 deg.

X12Ø: X coordinate of Point C

L-5 : Size of chamfered face

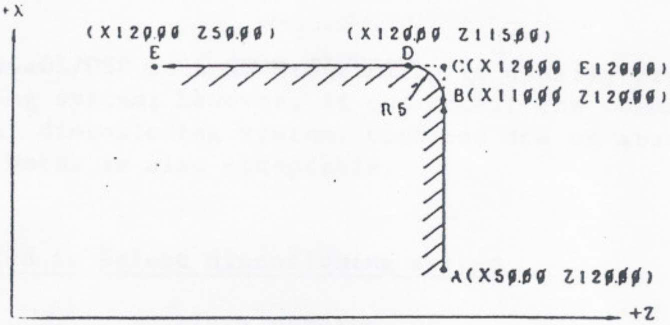
Its sign is determined by the direction of axis movement;

"+" ..... when Z-axis (X-axis) moves in the positive direction after X-axis (Z-axis) moved.

"-" ..... when Z-axis (X-axis) moves in the negative direction after X-axis (Z-axis) moved.

By commanding the coordinates of Point E, the cutting tool moves from Point D to Point E.

G76 ROUNDING



To cut the above shown contour along the Points A, B, D and E, program as:

G76 GØ1 X12Ø L-5 Foo CR

after positioning the cutting tool to Point A.

With the commands above, the cutting tool moves from Point A to B and then to D, thus automatically rounding the corner at 5 mm radius.

G76 : Specifies rounding of corner

X12Ø: X coordinate of Point C

L-5 : Radius of rounding circle

Its sign is determined by the direction of axis movement;

"+" ..... when Z-axis (X-axis) moves in the positive direction after X-axis (Z-axis) moved.

"-" ..... when Z-axis (X-axis) moves in the negative direction after X-axis (Z-axis) moved.

By commanding the coordinates of Point E, the cutting tool moves from Point D to Point E.

NOTES

1. G75/G76 is effective only in G01 mode. If it is specified in other than G01 mode, it causes an alarm.

2. G75/G76 is non-modal and active only in the commanded block.

3. If the axis movement dimension specified in the block calling for automatic chamfering is smaller than the absolute value of the L word, and alarm results.

4. If the axis movement dimensions specified in the block calling for automatic chamfering are zero both for X and Z, or if neither X nor Z value is zero in such block, it results in an alarm.

   The block calling for automatic chamfering mode can contain only one dimension word, either X or Z.

5. The automatic chamfering program is effective in:
   LAP and Tool Nose radius compensation mode

G90/G91 INCREMENTAL PROGRAMMING

With OSP 5020L/OSP 500L-G, programming is usually prepared in absolute dimensioning system; however, it can accept the commands expressed in incremental dimensioning system. Combined use of absolute and incremental dimension words is also acceptable.

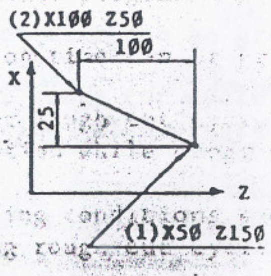
G codes used to select dimensioning system

G90 .... Absolute programming
(cancel of incremental programming)

when the control is reset, it is in the G90 mode.

G91 .... Incremental programming

Example (positioning from point (1) to point (2)):



Absolute	Incremental
GØØ X5Ø Z15Ø .... (1)	GØØ X5Ø Z15Ø .... (1)
X1ØØ Z5Ø .... (2)	*G91 X5Ø Z-1ØØ .... (2)

* Designate dimensional differences between points (2) and (1).

Note: In incremental programming, X word should be expressed in diameter.

LAP LATHE AUTO-PROGRAMMING FUNCTION

GENERAL DESCRIPTION

LAP (Lathe Auto-Programming) is the function to make use of high speed processing capability which characterizes the OSP500L-G/OSP5000L-G series. With this function, the control automatically generates tool path to produce the required part contour.

In this function the program comprising, dimension data of the final contour to be finished including rough cut conditions is prepared as the Contour Definition Program; when it is called out with the cutting conditions specified, the control automatically generates tool path for respective rough cut cycles, and then finish the workpiece to the programmed dimensions.

This feature permits the programmer to complete the part program simply by picking up the dimensions specified in an engineering drawing and, therefore, it simplifies programming as well as it reduces programming time; this furthermore facilitates tape check procedure and also tape punch procedure.

Various cutting modes available with the LAP can cope with any type of cutting intended.

Features of LAP are:

1) No special programming language is needed. The same programming manner as in conventional programming technique can handle the LAP function.

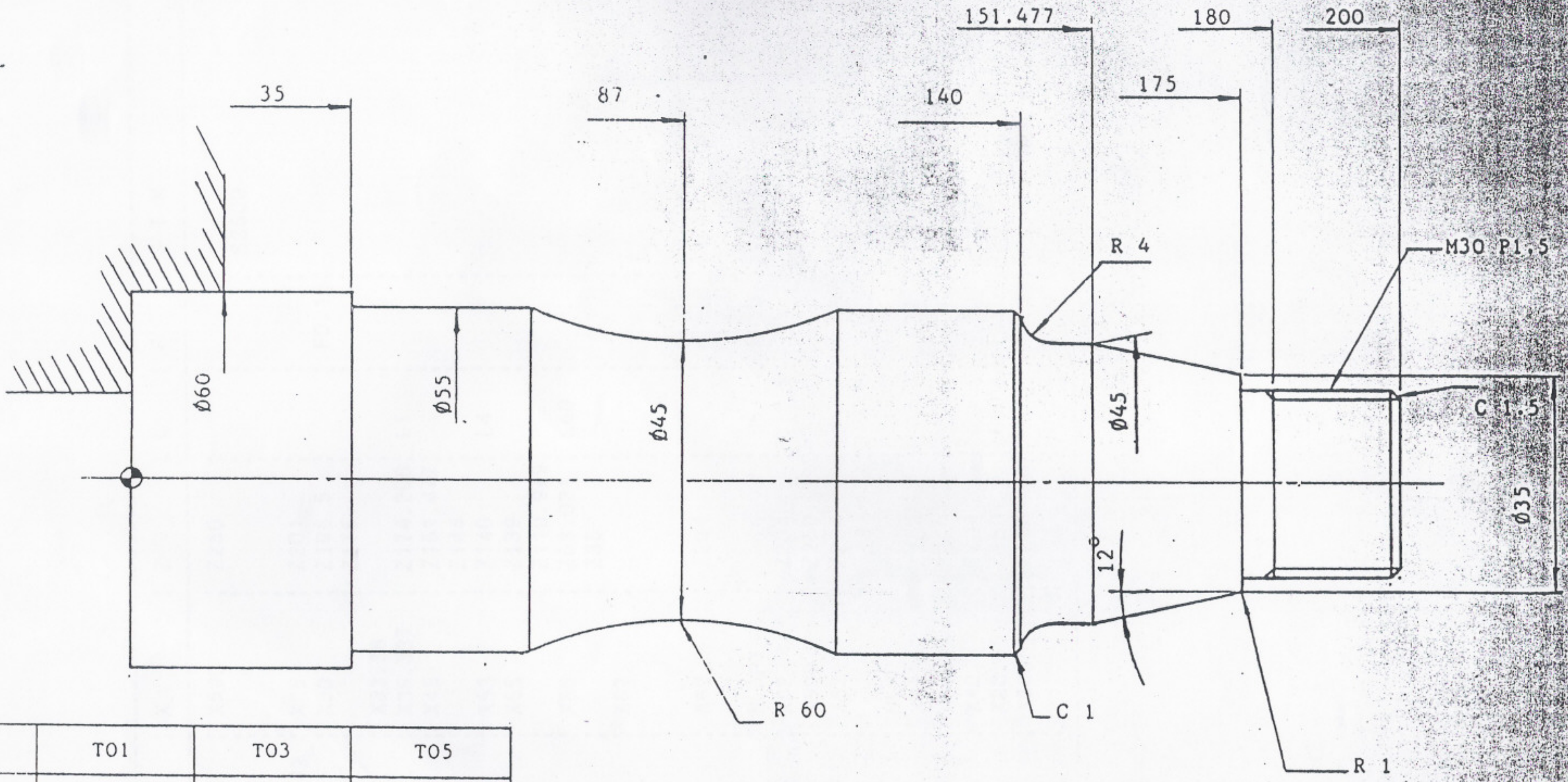
2) Tape preparation time can be greatly reduced.

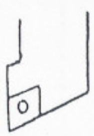
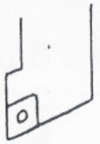
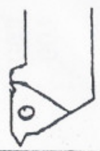
3) Programming for rough cut cycle can be eliminated, and this simplifies manual calculation while programming.

4) Change of cutting conditions such as depth of cut and feedrate is possible during rough cut cycle.

FORMAT (G CODES)

G80	End of contour definition
G81	Start of contour definition, longitudinal
G82	Start of contour definition, Transverse
G85	Bar turning rough cut cycle
G87	Finish cut cycle



TOOL NO.	T01	T03	T05
TOOL BIT			
CUTTING SPEED	150 M/MIN	200 M/MIN	120 M/MIN
CUTTING SELECTION	OD ROUGH	OD FINISH	THREADING

MATERIAL : CARBON STEEL (S45C)
ALL DIMENSION IN MM

N	G	X	Z	L	F	S T M
	G00	X500	Z250			
	G50					S3500
NLAP1	G81					
	G00 G42	X25	Z201			
	G01	X30	Z198.5		F0.15	
			Z175			
		X33.38				
	G03	X35.337	Z174.208	L1		
	G01	X45	Z151.477			
			Z144			
	G02	X53	Z140	L4		
	G01	X55	Z139			
			Z110.979			
	G02	X55	Z63.021	L60		
	G01		Z35			
		X62				
	G40					
	G80					
NAT01	G00	X62	Z204			S670 T010101 M42 M03 M08
	G96					S130
	G85	NLAP1		D6	F0.3	U0.4 W0.1
	G00	X62	Z204			
	G97	X500	Z250			S930
NAT02		X62	Z204			T030303
	G96					S180
	G87	NLAP1				
	G00	X62	Z204			
	G97	X500	Z250			S1135
NAT03		X40	Z204			T0505
	G71	X28.05	Z180	B60	F1.5	D0.6 H1.95 M23 M73 M32
	G00	X500	Z250			M05 M09
						M02

G87 FINISH CUT CYCLE

Format

N∅2∅3 G87 NLAP1 ␣ U W

N∅2∅3: Sequence number

G87 : G code calling out finish cut cycle
To be provided right after sequence number (name)

NLAP1: Sequence name in the first block of contour defining blocks

Blank: Enter either tab or space code

U : Stock removal in finish cut cycle, X component

W : Stock removal in finish cut cycle, Z component

With the commands above, the control starts searching of contour definition program beginning with the sequence name NLAP1. After assigning parameter data of U and W of NLAP1, the control starts the finish cut cycle.

Note 1: No S, T or M code may be provided in the G87 block.

Note 2: As a feedrate, the one provided in the contour definition program is effective.

If no F word is provided in the contour definition program, the feedrate effective before this block becomes effective.

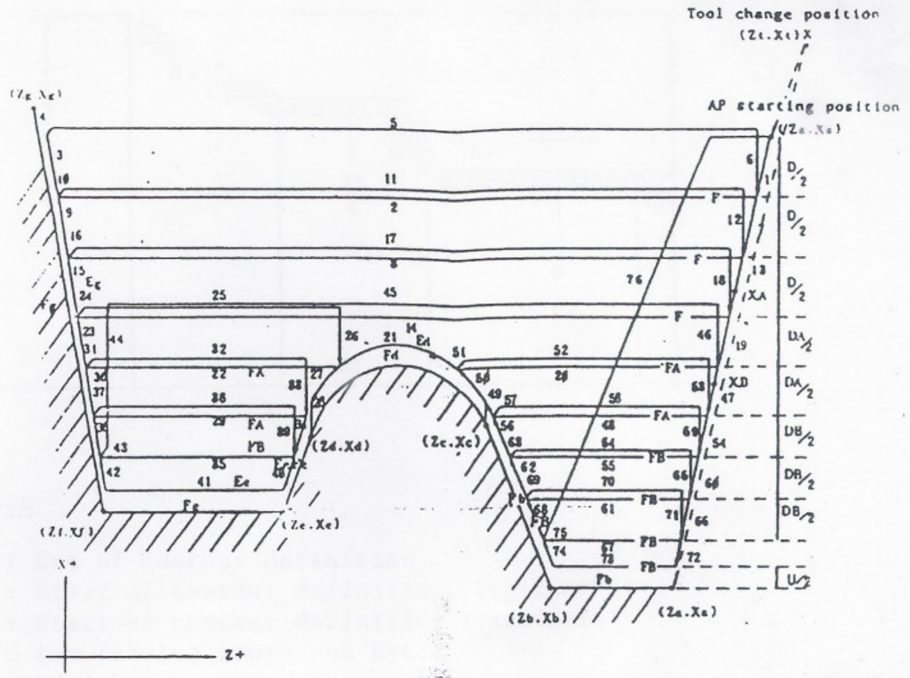
NOTE THAT THE G87 BLOCK CANNOT CONTAIN AN F WORD.

Note 3: When U and/or W word is not provided, U and/or W is assumed "∅".

U and W words must be positive or zero. If not, an alarm results.

EXPLANATION OF LAP FUNCTIONS AND PROGRAM

Tool Path and Program - Longitudinal Turning



Contour Definition

```
NLAP1  G81                                    ---- Start of contour definition
NØØØ1  GØØ  Xa  Za
NØØØ2  GØ1  Xb  Zb                Fb
NØØØ3       Xc  Zc
NØØØ4  GØ3  Xd  Zd   Ia   Kd      Fd   Sd   Ed
NØØØ5  GØ1  Xe  Ze                Fe   Se   Ee
NØØØ6       Xf  Zf
NØØØ7       Xg  Zg                Fg   Sg   Eg
NØØØ8  G8Ø                                    ---- End of contour definition
```

- F, S: Feedrate and spindle speed for finish cut cycle
- E: Feedrate for rough cut cycle, rough cut along contour

Rough Cut Cycle

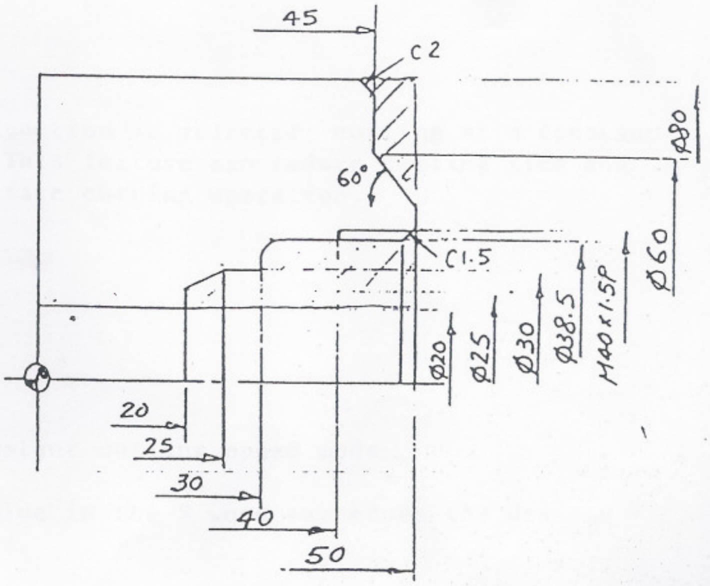
```
NØ1Ø1  GØØ  Xt   Zt                           Tool change point
NØ1Ø2       Xs   Zs                 STM       Starting point of AP
                                              S, T and M for rough cut cycle
NØ1Ø3  G85  NLAP1        D   F   U   W        Calls for rough cut cycle
$      G84  XA=          DA=  FA=             Continued line; Point XA
                                                cutting condition change point
$           XB=          DB=  FB=             Continued line; Point XB
                                                cutting condition change point
```

- D, F: Depth of cut and feedrate up to Point A (XA)
- U, W: Stock removal amount in finish cut cycle
- DA=, FA= / DB=, FB=: Depth of cut and infeed amount from Point A/B (XA/XB)

Finish Cut Cycle

```
NØ2Ø1  GØØ  Xt  Zt                            Tool change point
NØ2Ø2                  STM                    S, T and M for finish cut cycle
NØ2Ø3  G87  NLAP1                             Calls for finish cut cycle
```

Bar turning cycle(G85) - Longitudinal,ID(G81) and End face(G82)




```
G00 X800 Z800
G50 S3500
NAP1 G82
G00 G41 X82 Z42
G01 Z45 A135 F0.18
X60
Z50 A120
X37
Z52
G40
G80
NAP2 G81
G00 G41 X43.5 Z51
G01 X38.5 A45 F0.15
Z33
G03 X32.5 Z30 L3
G01 X30
Z25
X25 Z20
X18
G40
G80
G00 X800 Z800 M41 M03 S600 M63
NTA1 X90 Z51 T010101 M08
G96 S150
G85 NAP1 D2 U0.4 W0.1 F0.3
G00 X90 Z51
G97 X800 Z800 S600 M05 M63
NTA2 X90 Z52 S800 M42 M03 T020202 M63
G96 S200
G87 NAP1
G00 G97 X800 Z800 S800 M63
NTA3 X18 Z55 T030303 S2300 M63
G96 S150
G85 NAP2 D4 U0.3 W0.1 F0.25
G00 G97 X18 Z55 S2300 M63
X800 Z800
NTA4 X18 Z55 T040404 S2800 M63
G96 S180
G87 NAP2
G00 G97 X18 Z55 S2800 M63
X800 Z800 S1500 M63
NTA5 X34 Z56 T0505
G71 X40 Z40 B60 D0.4 U0.05 H1.5 F1.5 L1 M33 M73 M23
X800 Z800 M05 M22 M63
M02
```

(G96/G97) PROGRAMMING CONSTANT SPEED CUTTING OPERATION

When the constant speed cutting function is selected, cutting at a constant cutting speed can be performed. This feature can reduce cutting time and also assure stable finish in end face cutting operation.

Format:

G96 Soooo

G96 .... Entry of constant cutting speed mode

Soooo .... Numerical value in the S word expresses the desired cutting speed.

S1ØØ means 1ØØ m/min.

G97 Soooo

G97 .... Cancel of G96

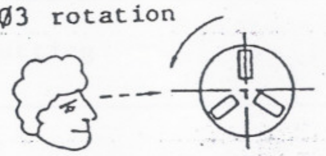
Soooo .... Numerical value of the S word expresses the desired spindle speed.

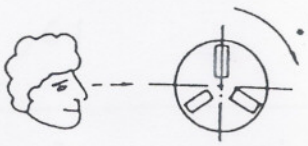
When the control is reset, it is in the G97 mode.

Example

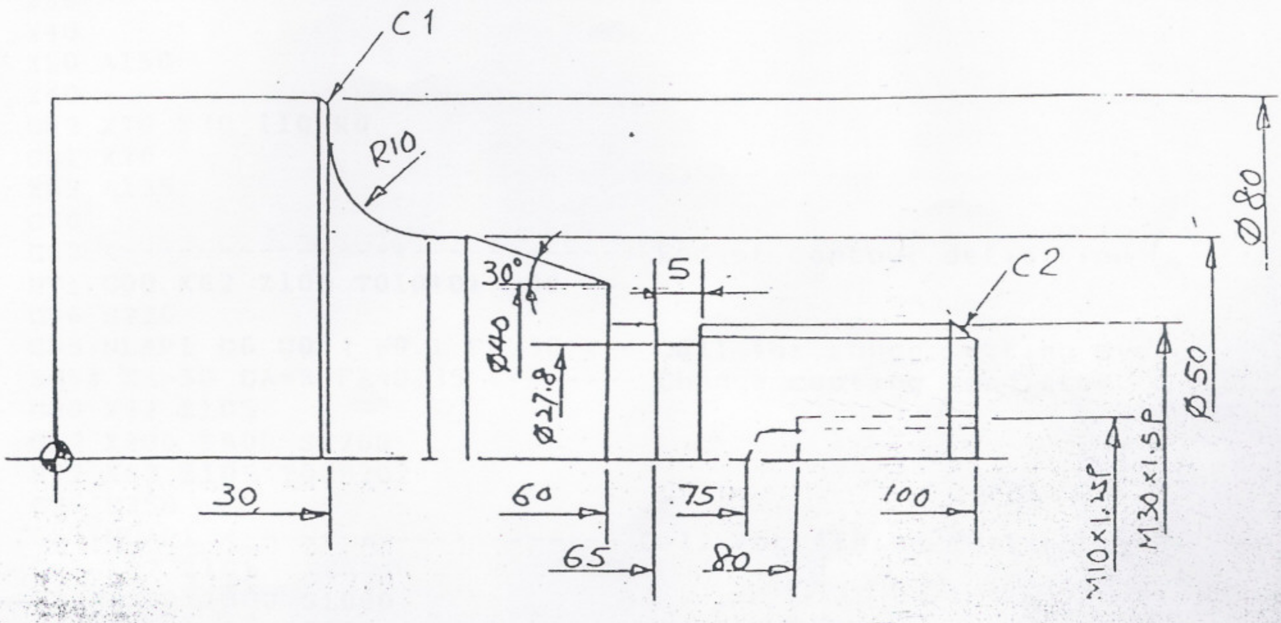
```
Nooo  G96 S1ØØ     G96 calls for constant speed cutting mode and the
  :                commands following this block are all executed in
  :                this mode.
  :
  :                S1ØØ .... 1ØØ m/min.
  :
Nooo  G97 S5ØØ     G97 cancels G96 mode, and cutting after this block
  :                is carried out at a spindle speed of 5ØØ rpm.
  :
```

5. M CODES

Code	Associated Information and function
M00	M Code : Up to three numeric characters following address character M are used for specifying various miscellaneous machine functions such as spindle CW/CCW, collant ON/OFF, etc.
M00	Program Stop When M00 is executed, machine operation goes into cycle stop state; spindle rotation and collant supply are also brought to a stop. To continue execution of the part program, press the CYCLE START button. This program stop function is effectively used for measuring finished dimensions and also for removing chips during cycle.
M01	Optional Stop M01 performs the same function as M00 Program Stop, except that the control ignores programmed M01 codes unless the OPTIONAL STOP switch is turned ON.
M02	End of Program M02 provided at the end of a part program resets the control.
M03	CW Rotation M03 starts the spindle rotation to advance a right handed screw into the workpiece. 

Code	Associated Information and function
M04	CCW Rotation M04 starts spindle rotation to retract a right-handed screw from the workpiece.  Viewing spindle
M05	Spindle Stop M05 stops spindle rotation
M08	Coolant ON
M09	Coolant OFF
M22	Cancel of M23
M23	**Chamfering ON** **M23 executes chamfering using a fixed cycle, in G31 through G33 thread cutting cycle.**
M73	**Pattern of thread cutting**
M74	**Pattern of thread cutting**
M75	Pattern of thread cutting

PROGRAMMING EXAMPLE



TOOL NO. 1 - O.D. ROUGH CUT
2 - O.D. FINISH CUT
3 - CENTRING (CENTRE DRILL)
4 - GROOVING (WIDTH 3mm)
5 - DRILLING (DIA. 6.8)
6 - THREADING (M30x1.5P)
7 - TAPPING (M8x1.25P)

```
EXAMPLE :
G00 X800 Z800 <------------------ TURRET RETURN TO LIMIT
G50 S3500  <-------------------- MAXIMUM SPINDLE SPEED CONTROL
G00 X800 Z800 M42 M03 S1000
NLAP1 G81 <-------------------- Start of contour definition
G00 G42 X24 Z101
G01 X30 A135 F0.15
Z60
X40
X50 A150
Z40
G02 X70 Z30 I10 K0
G01 X78
X82 A135
G40
G80 <------------------------- End of contour definition
NT1 G00 X82 Z105 T010101 M08
G96 S220
G85 NLAP1 D6 U0.4 W0.1 F0.30 <-- Call for rough cutting cycle
$G84 XA=50 DA=8 FA=0.35 <------- Change cuuting condition
G00 X82 Z105
G97 X800 Z800 S1200
NT2 X82 Z105 T020202
G96 S250
G87 NLAP1 <------------------- Call for finish cutting cycle
G00 X82 Z105
G97 X800 Z800 S1000
NT3 X0 Z110 T0303 <----------- Centre drill
G01 Z102 F0.6
Z95 F0.05
G04 F0.5
G00 Z110
X800 Z800 S1200
NT4 X40 Z65 T0404
G73 X27.8 Z70 I9 K2.5 F0.1 D2 E0.5 T14 <----- Grooving cycle
G00 X800 Z800 S1400
NT5 X0 Z110 T0505
G74 X0 Z75 K9 F0.08 D3 L6 E1 <--------------- Drilling cycle
G00 X0 Z110
X800 Z800 S1500
NT6 X40 Z106 T0606            <----------- Thread cutting cycle
G71 X28.05 Z68 B60 D0.3 U0.05 H1.95 L0.5 F1.5 M32 M73 M23
G00 X800 Z800 M22 M05
M41 M03 S350
NT7 X0 Z115 T0707
G77 X0 Z80 K8 F1.25 <------------------------- Tapping cycle
G00 X800 Z800 M05 M09 M63
M02
```

TOOL NOSE RADIUS COMPENSATION FUNCTION (Standard for OSP5020L only)

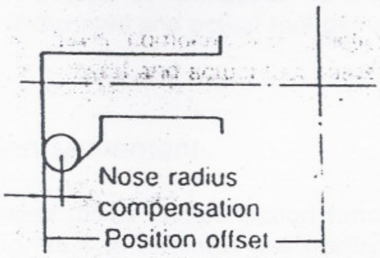
The data processing performance of the OSP5020L/OSP500L-G series is all the more enhanced by one distinctive feature: The Tool Radius Compensation Function.

The tool tip point radius of most cutting tools used in turning operation is the cause of inconsistencies between the designated tool paths and the actually finished workpiece contour. With the tool radius compensation function, such geometric error is automatically compensated by simple programming.

In die sinking or milling operation, cutting is performed by a large diameter milling cutter. There, control systems are equipped with the Cutter Offset function where the actual tool path generated is offset from the designated path by the amount equivalent to the radius of the cutter used. This is a standard function with OSP5000 series controllers. However, in comparison with the cutter offset function, the tool nose compensation function offers special features that are requisites for lathes.

Tool Offset and Nose Radius Compensation

In turning operation, various types and different shapes of tools are used to finish one workpiece. ID cutting tools, OD cutting tools, rough cut tools, finish cut tools, drill, etc. Accordingly, the tool nose radius compensation function has to be activated simultaneously with the tool offset function.



The tool nose radius function is activated by a six-digit T command:

T 00 00 00

- Tool offset number (01 to 32)
- Tool number (01 to 12)
- Tool nose radius compensation number (01 to 32)

Geometrical Cutting Error Due to Tool Nose Radius

If cutting along paths A-B-C-D-E in Fig. 16-3 is intended without activating the tool nose radius compensation function, the portions indicated by hatching lines will remain uncut and cause a geometrical errors. This is because the tool setting is made to locate the imaginary cutting point P in Fig. 16-2 at the datum point and trace programmed path as controlled by NC commands. However, the actual cutting tip point is not precisely located on that datum point because of the tool nose radius and this produces geometrical errors.

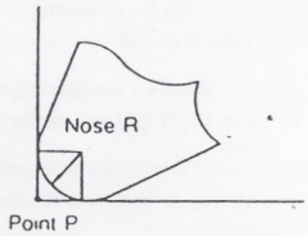


Fig. 16-2 Tool Setting Point

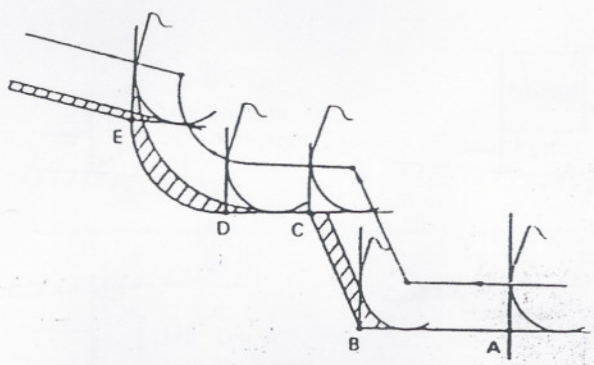


Fig. 16-3 Tool Path and Resulting Error
- Without Tool Nose Radius Compensation -

The tool nose radius compensation function automatically compensates the inconsistency between the designated and actual tool paths caused by the tool nose radius.

Compensation Movement

With the tool nose radius compensation function activated, the tool path is compensated as illustrated in Fig. 16-4 to eliminate the portions left uncut, shown in Fig. 16-2. This assures accurate finish as programmed.

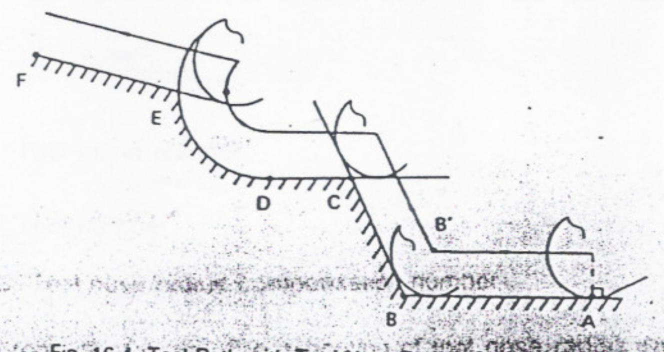


Fig. 16-4 Tool Path with Tool Nose Radius Compensation

Programming

Programming commands, G, M and T codes, used to activate the tool nose radius compensation function, are detailed in this section.

G Codes

G40: Used to cancel the tool nose radius compensation mode.

G41: Tool nose radius compensation - Left
Used when the tool moves on the left side of the workpiece.

G42: Tool nose radius compensation - Right
Used when the tool moves on the right side of the workpiece.

Note 1: The term indicating the side of the workpiece, right or left, is determined according to the direction in which the tool is advancing.

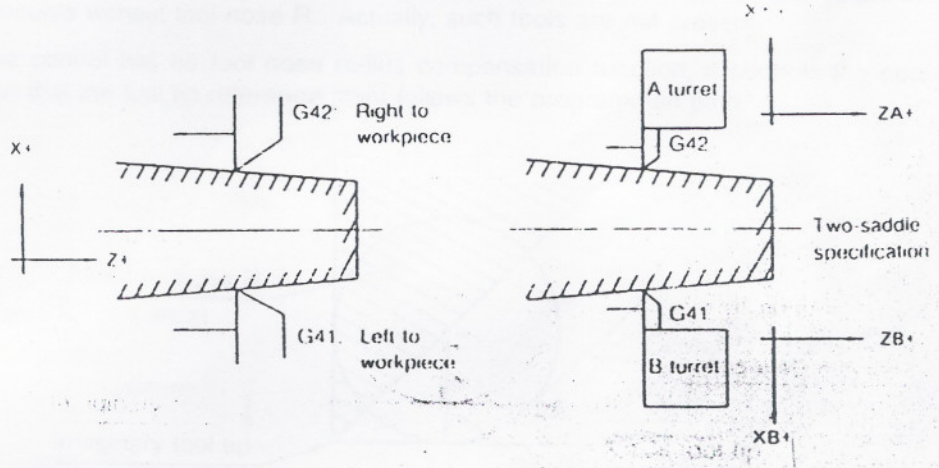
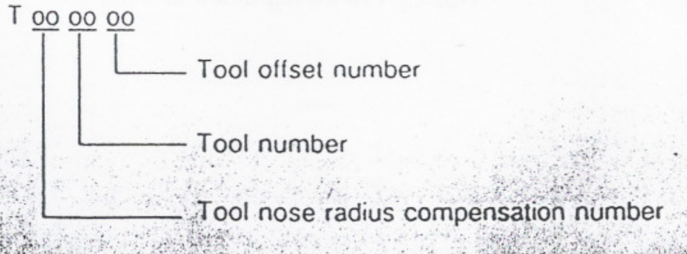


Fig. 16-5 Designation of G41 & G42

T Codes

Six numerical characters following address character "T" specifies the nose radius compensation number, tool number, and tool offset number.

To set offset data through the keyboard, 32 pairs of compensation data for the tool nose radius and tool offset function can be entered (01 through 32). For the tool number, 01 through 12 are available.



Note: To change tool offset during the execution of tool nose radius compensation, designate the tool nose radius compensation number and the tool number.

Tool Nose Radius Compensation Data

As seen in the previous séction, programming procedure to activate the tool nose radius compensation function has been simplified. However, to finish workpieces accurately using this function, the tool nose radius of the tool to be used must be measured precisely and the measured value entered correctly in the NC memory.

Measuring Nose Radius

Measure the center of the tool nose circle with respect to the tool tip reference point which is taken as the imaginary tool tip point for tool presetting. See Fig. 16-6 below.

The imaginary tool tip point indicates the ideal tool tip point which can be expressed only by the tool offset amounts without tool nose R. Actually, such tools are not present.

When the control has no tool nose radius compensation function, it controls the coordinated axis motion so that the tool tip reference point follows the programmed path.

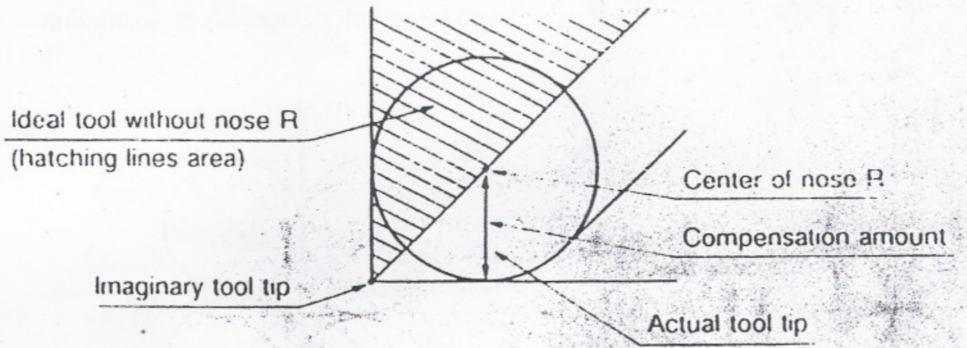


Fig. 16-6

In the measurement of nose R compensation data, both the nose R compensation amount and the direction of nose R center in reference to the imaginary tool tip.

Direction of nose R center in reference to the imaginary tool tip is expressed in the following two ways:

a) By positive and negative signs of X, Z compensation amounts

b) By a P number

If method a) is used, positive and negative signs are determined from the position of the nose R center in reference to the origin as in Fig. 16-7.

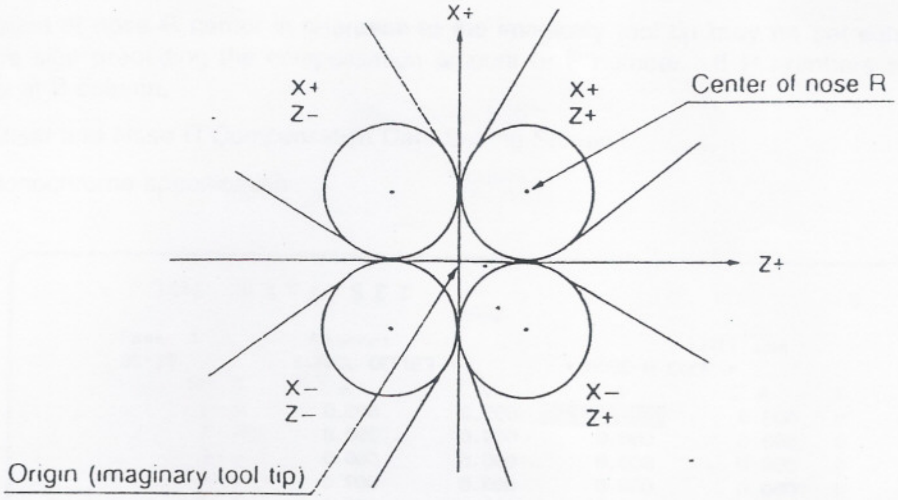
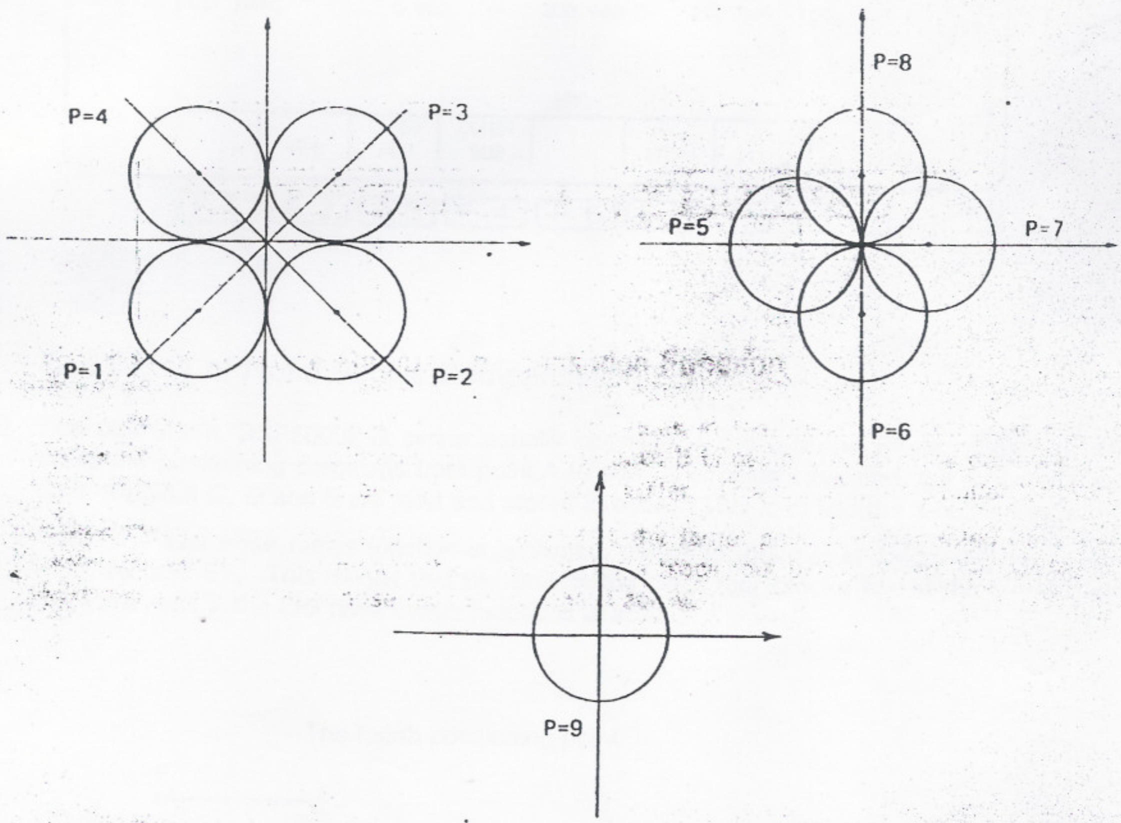


Fig. 16-7 Signs of Compensation Amount by Nose R Center Position

With method b), coded numbers are assigned in advance for individual nose R center position orientation to distinguish the directions



When P = 9, nose R center is set at the origin (imaginary tool tip).

Fig. 16-8 P Numbers for Nose R Center Positions

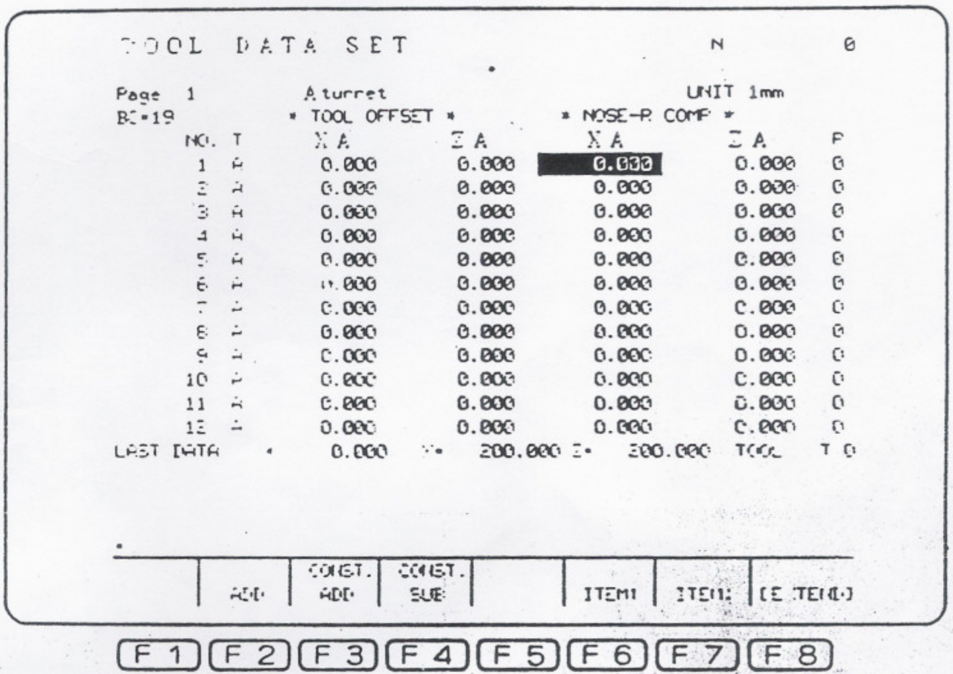
Setting Compensation Amounts

Set the tool compensation amount at the NOSE R COMP columns at the TOOL DATA SET screen. The compensation amounts can be set in the same manner as setting tool offset amounts.

Orientation of nose R center in reference to the imaginary tool tip may be set either by positive or negative sign preceding the compensation amount or P number. If P numbers are used, set the number at P column.

Tool Offset and Nose R Compensation Data Setting Screen

a) Monochrome specification



TOOL DATA SET N 0

Page 1 A turret UNIT 1mm

BI•19 * TOOL OFFSET * * NOSE-R COMP *

NO.	T	X A	Z A	X A	Z A	P
1	A	0.000	0.000	0.000	0.000	0
2	A	0.000	0.000	0.000	0.000	0
3	A	0.000	0.000	0.000	0.000	0
4	A	0.000	0.000	0.000	0.000	0
5	A	0.000	0.000	0.000	0.000	0
6	A	0.000	0.000	0.000	0.000	0
7	A	0.000	0.000	0.000	0.000	0
8	A	0.000	0.000	0.000	0.000	0
9	A	0.000	0.000	0.000	0.000	0
10	A	0.000	0.000	0.000	0.000	0
11	A	0.000	0.000	0.000	0.000	0
12	A	0.000	0.000	0.000	0.000	0

LAST DATA • 0.000 X• 200.000 Z• 300.000 TOOL T 0

F1	F2	F3	F4	F5	F6	F7	F8
	ADD	CONST. ADD	CONST. SUB		ITEM1	ITEM2	[EXTEND]

Conditions of Nose Radius Compensation Function

(1) The OSP5020L/OSP5000L-G series usually operates in the 3-buffer mode. See Fig. 16-9. While the positioning command from point A to point B is being executed, the positioning point data of points C, D and E are read and stored in buffer. This is called the 3-buffer function.

When the tool nose radius function is activated, the target point E is calculated from straight lines DE and EF. This means that the data in the block four blocks ahead the current target point are read if the tool nose radius functions is active.