

Computer Aided Manufacturing



**Dr.S.RAMABALAN,
PRINCIPAL,
E.G.S. PILLAY ENGINEERING COLLEGE,
NAGAPATTINAM.**



Unit III

PROGRAMMING OF CNC LATHE

Coordinate system - structure of a part program -G & M Codes - Programming for FANUC and SIEMENS controller -Single pass and canned cycle -Turning, facing and threading -Multi-pass canned cycle -Rough and Finish turning, facing, pattern repeating, grooving, threading, drilling, boring, peck drilling, high speed drilling cycle - Subprogram and Macro programming -Tool length and nose radius compensation - offset -Tool, work and coordinate -Insert -Materials, Classification, Nomenclature and Selection -Tool and Work holding devices -Automatic tool changer -Turret and drum type -Tool holder nomenclature and selection -CNC part programming using CAD/CAM software and interfacing with CNC machines

Cleansing through Breathing

Nadi Suddhi
(Alternate
Nostril breathing)



1 min

Recap and review of previous class

Let's
Recap



5 mins

Prerequisite Knowledge

- Knowledge on conventional machining
- Fundamentals of metal removal processes

3.EVOCATION



4. GENERAL OBJECTIVE (GO)

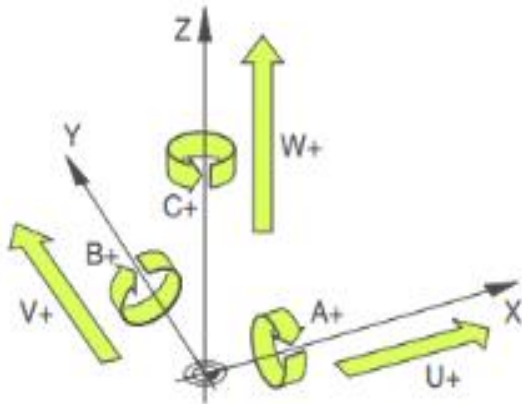
- Students will be able to apply the knowledge to create the CNC part program for circular components in FANUC & SIEMENS controller.

5. SPECIFIC OBJECTIVE (SO) MAPPED WITH STEM

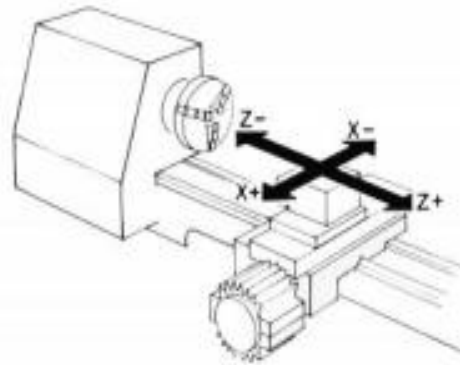
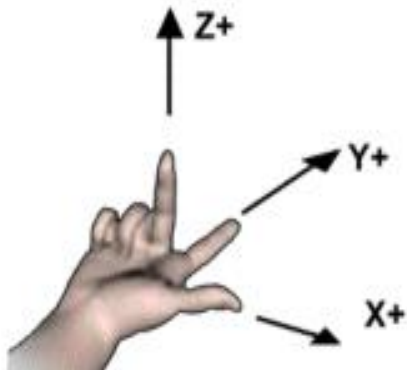
The students will be able to

1. Recall the coordinate systems used in CNC machines. (R-F) (E)
2. Explain the structure of part programming in CNC machines. (U-C) (E)
3. Exemplify the canned cycles used in CNC Lathe programming in FANUC and SIEMENS controller. (U-C) (E)
4. Construct the part program for given cylindrical component. (Ap-C) (E)

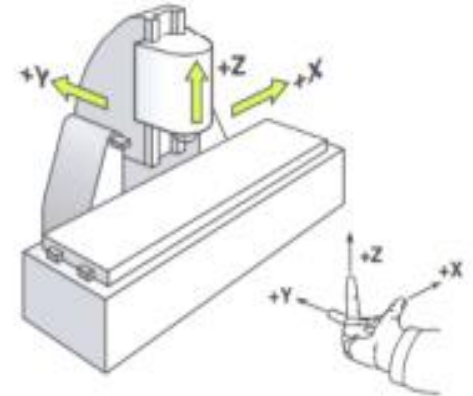
Axis nomenclature



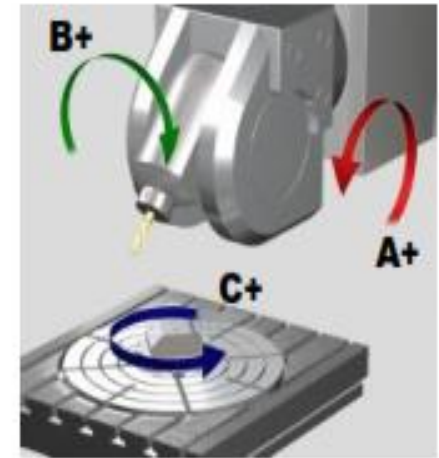
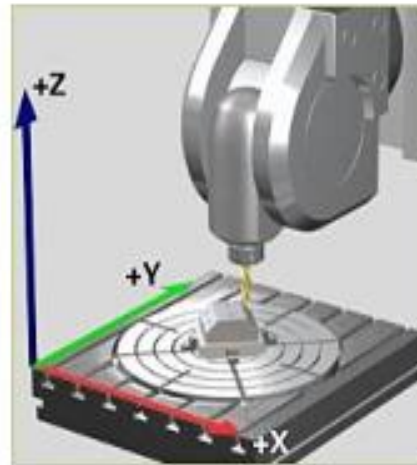
The axes are named according to DIN 66217.



Turning machine

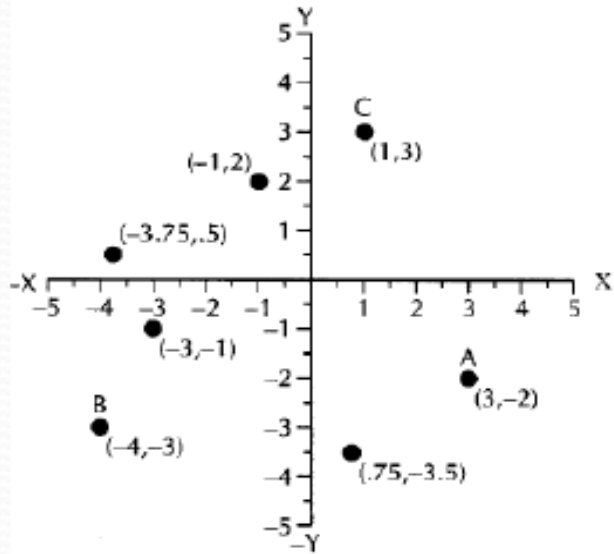


Three-axes milling machine

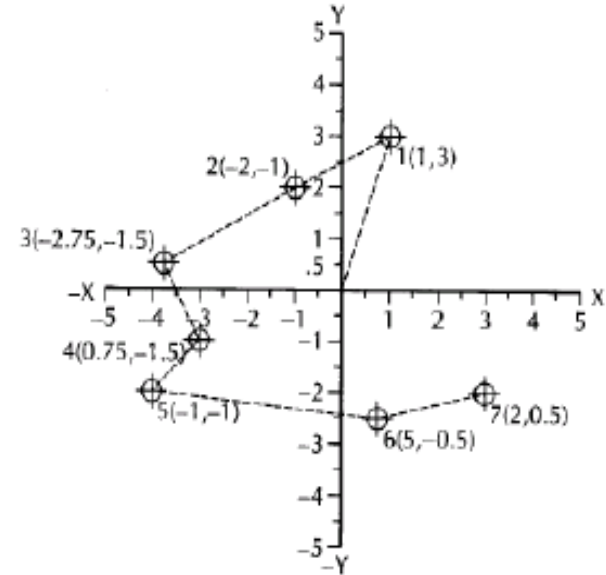


Six-axis milling machine

Based on Coordinate system



Absolute Coordinate System

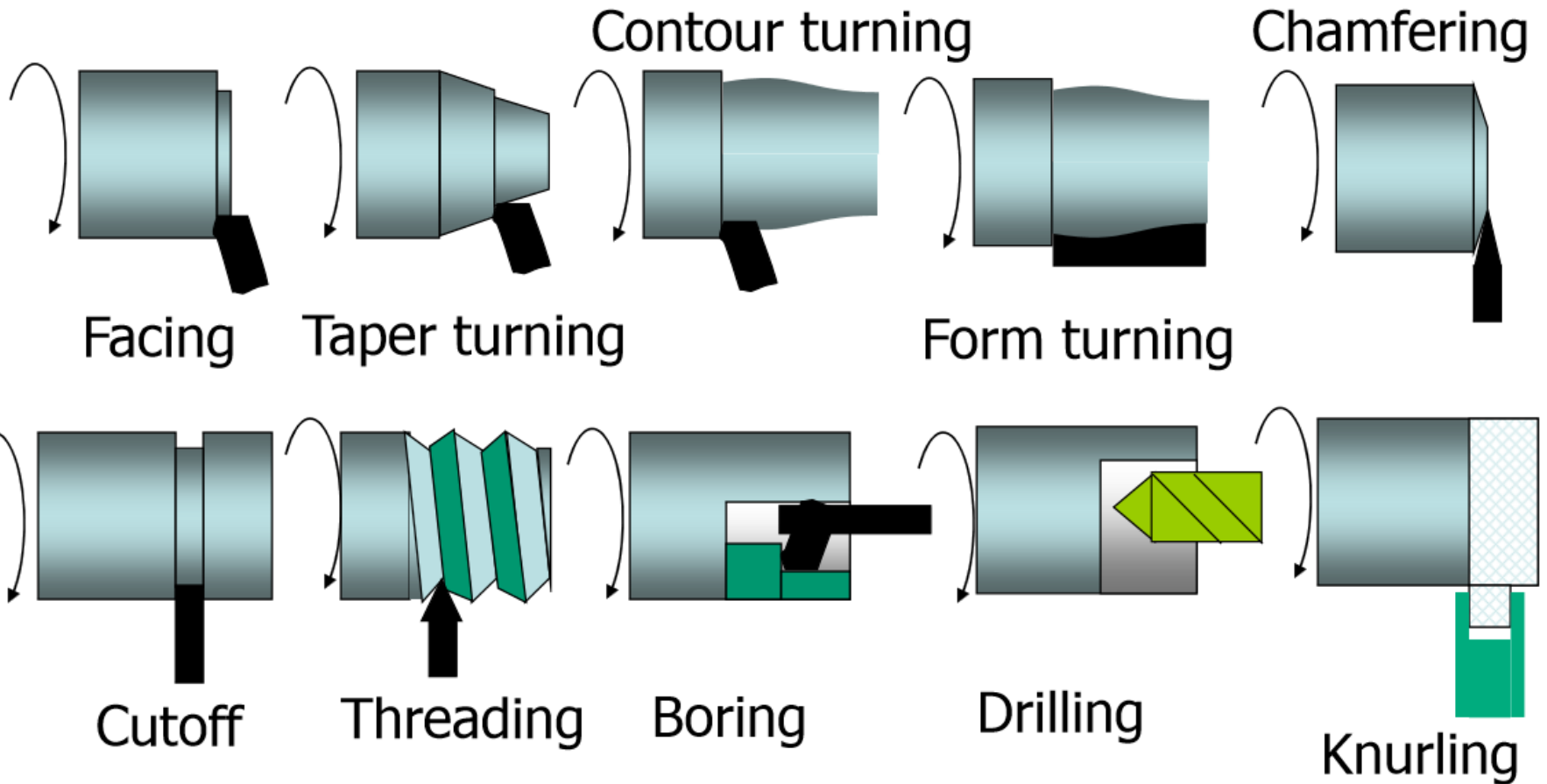


Incremental Coordinate System

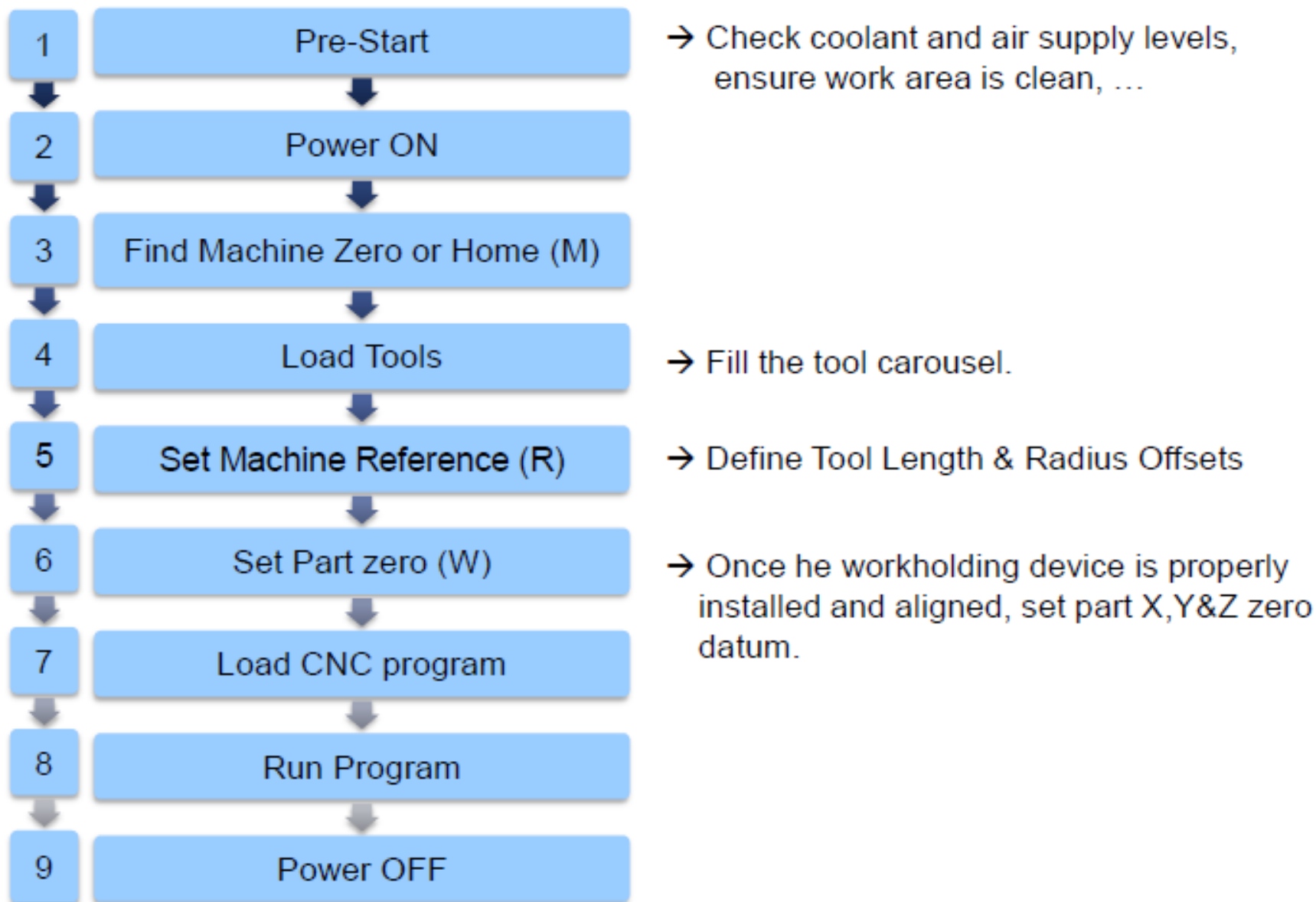
CNC Programming – Turning



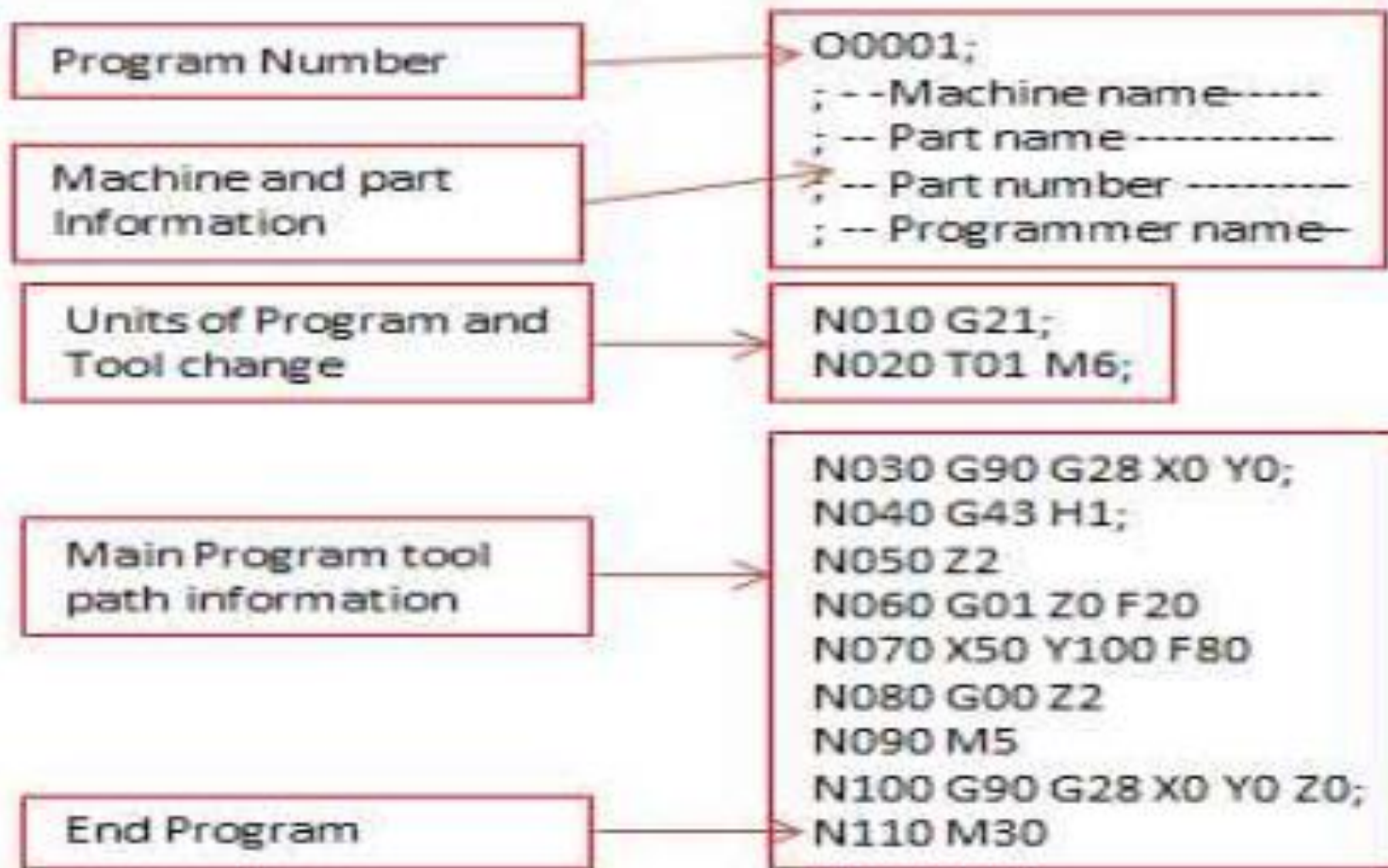
Sequence of operation



CNC machine setup and operation



Structure of a part program



Programming Key Letters

O - Program number (Used for program identification)

N - Sequence number (Used for line identification)

G - Preparatory function or Geometry code

X - X axis designation

Y - Y axis designation

Z - Z axis designation

R - Radius designation

F – Feed rate designation

S - Spindle speed designation

H - Tool length offset designation

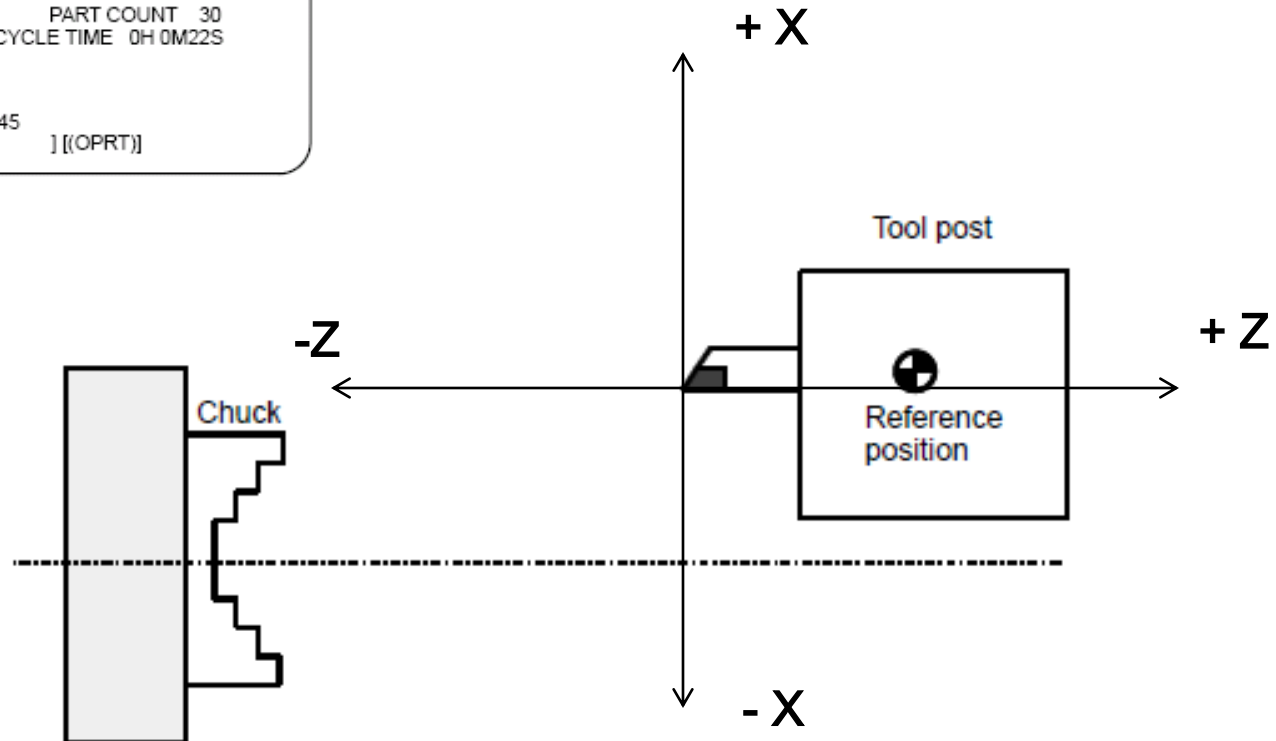
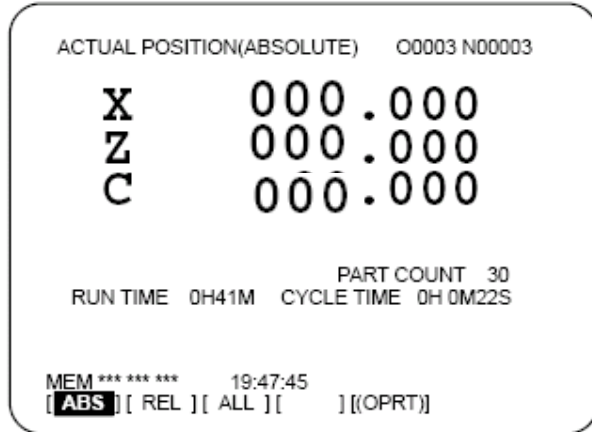
D - Tool radius offset designation

T - Tool Designation

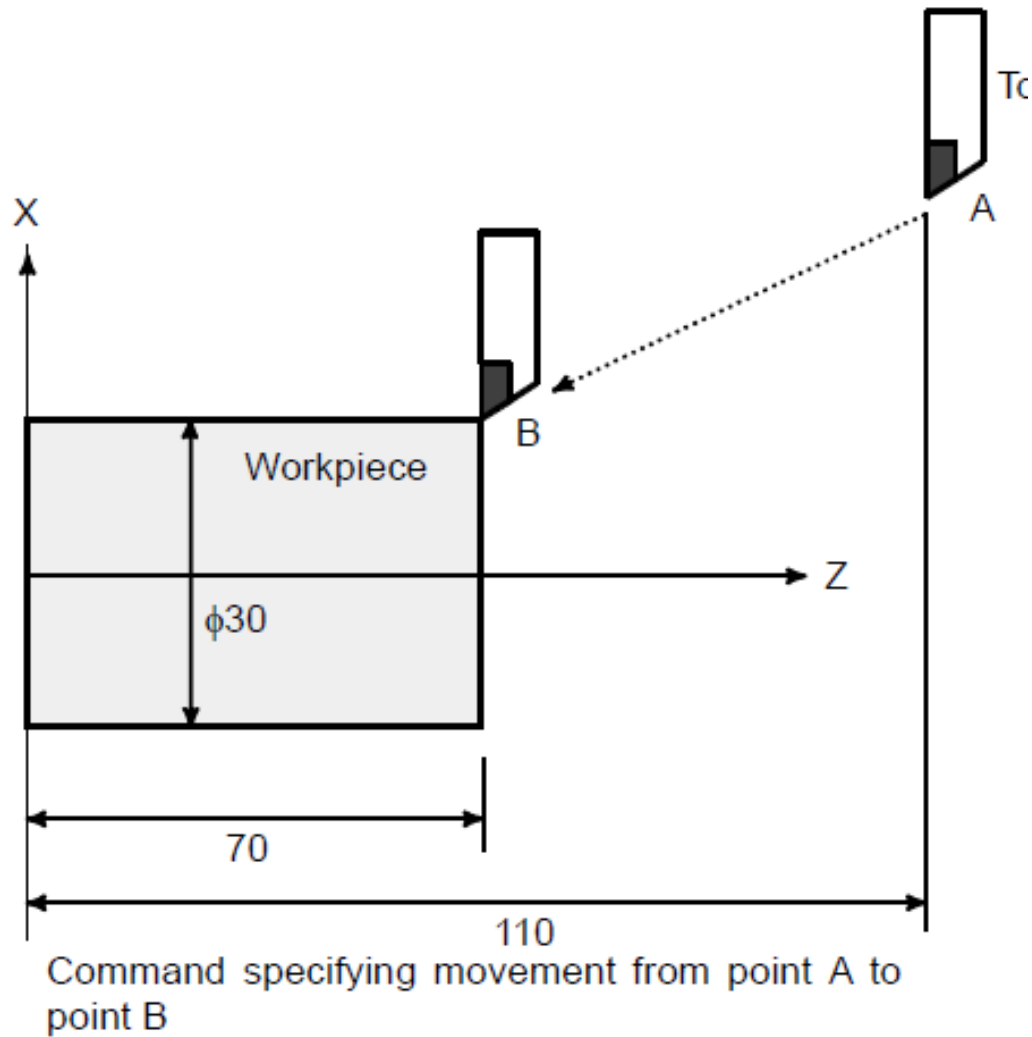
M - Miscellaneous function or Machine function

MACHINE COORDINATE SYSTEM

This coordinate system is established in CNC controller after referencing

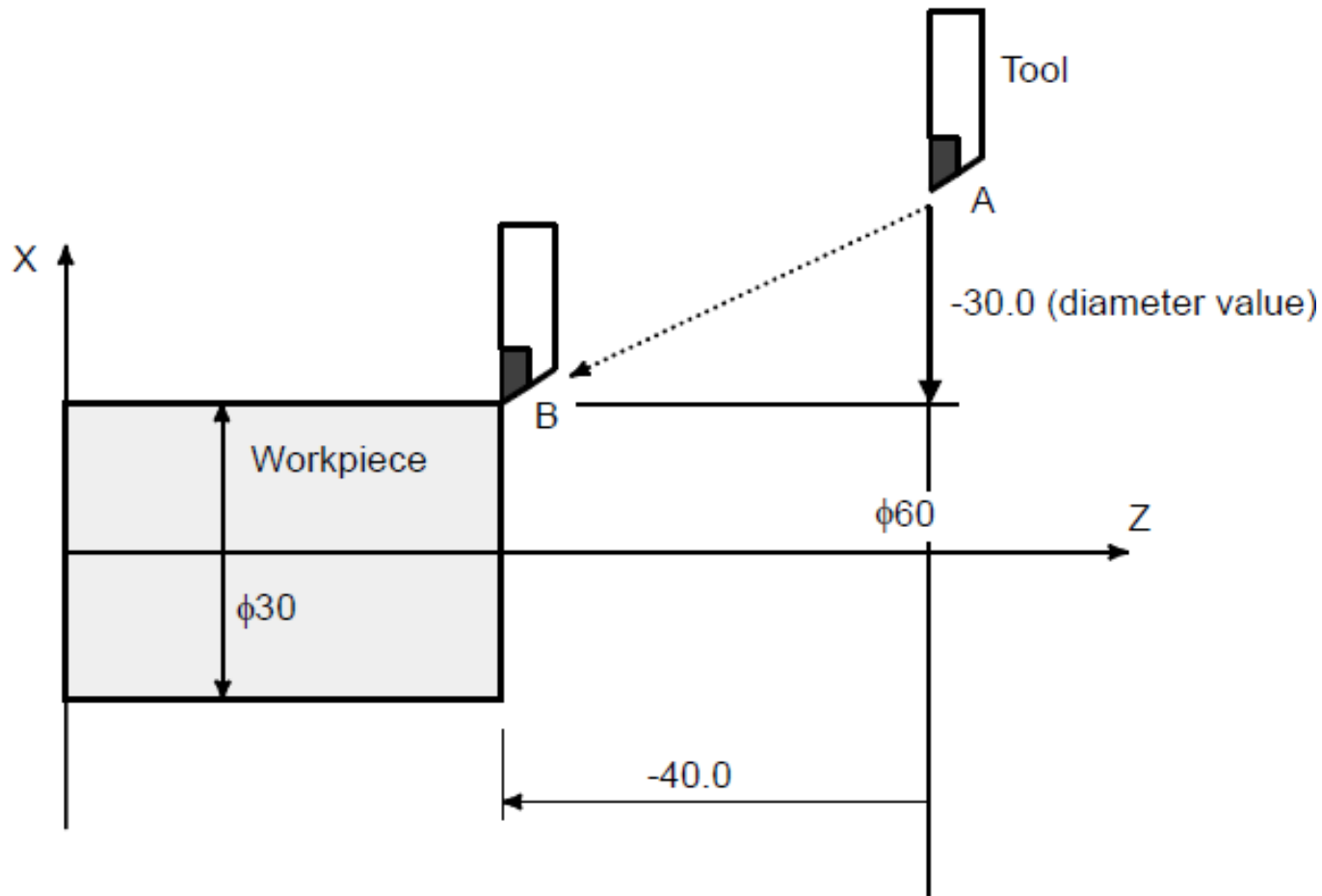


Absolute Command



X30.0Z70.0;
Coordinates of point B

Incremental command

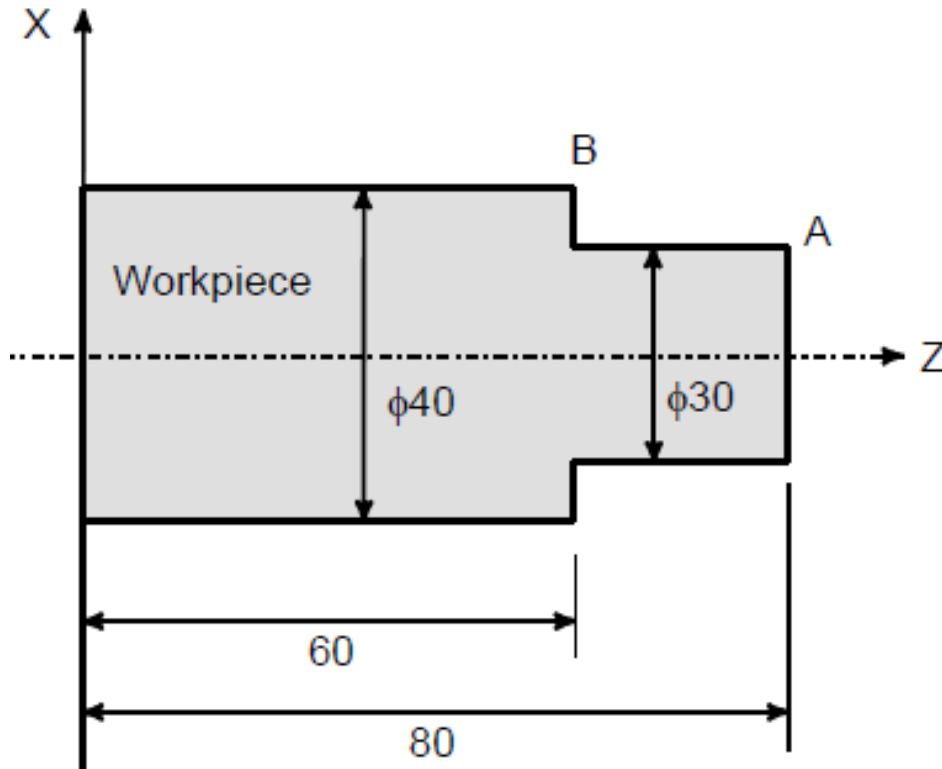


Command specifying movement from point A to point B

U-30.0 W-40.0

Distance and direction for movement along each axis

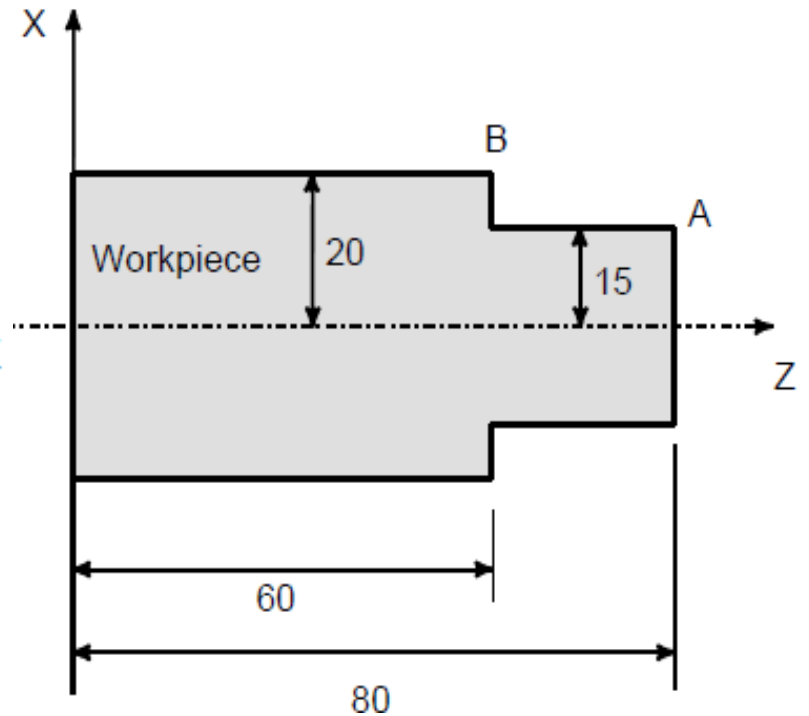
Diameter programming



Coordinate values of points A and B

A(30.0, 80.0), B(40.0, 60.0)

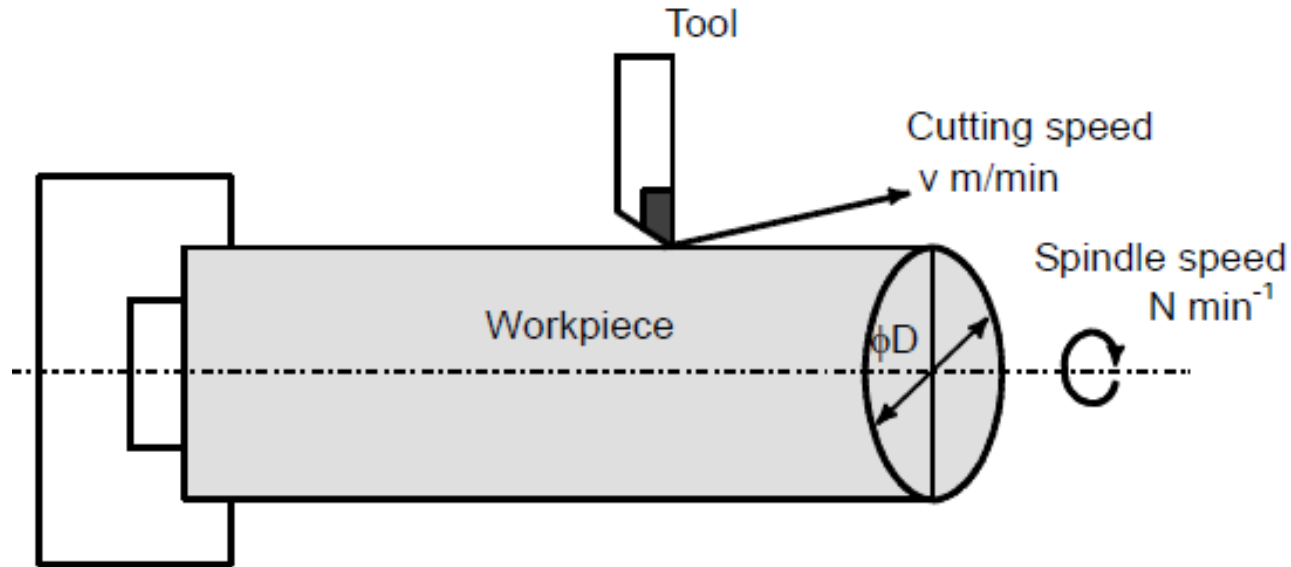
Radius programming



Coordinate values of points A and B

A(15.0, 80.0), B(20.0, 60.0)

Cutting Speed and Spindle Speed



Cutting Speed is nothing but **linear speed of the tool over the job**

Cutting speed depends on **tool material and job material**

Spindle speed is the number of rotation per minute made by Spindle

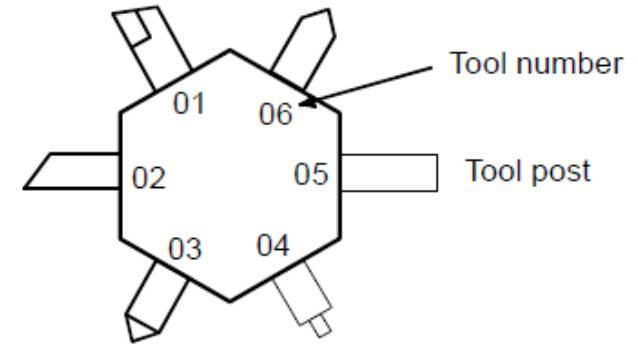
$$N = \frac{1000 * V}{\pi D} \quad N = \text{spindle speed (RPM)}, \quad V = \text{Cutting Speed (m/min)}$$

$D = \text{Diameter in mm}$

Ex. Cutting Speed = 30 Met/Min, Job Dia. = 15 mm, Spindle Speed = _____

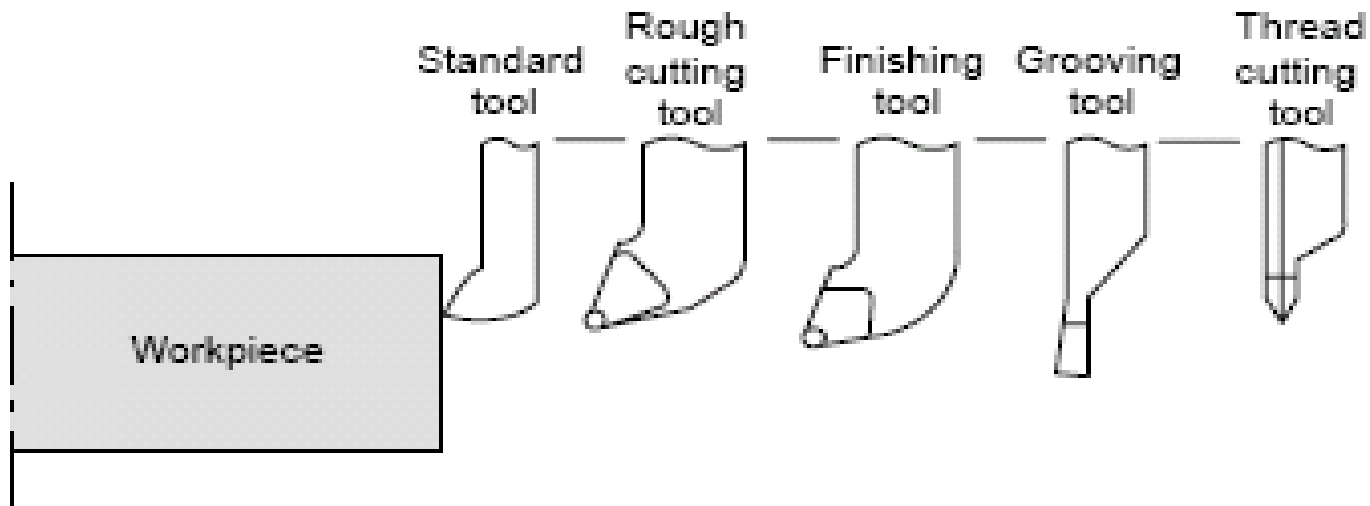
SELECTION OF TOOL

Tool 2 and offset 2 is referred as **T0202**
in the program

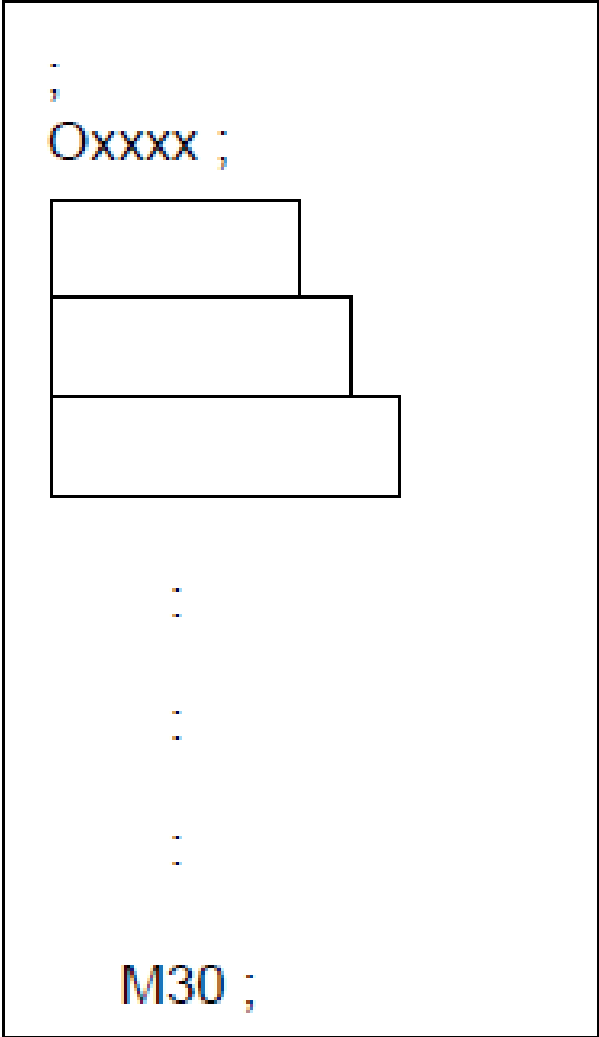


Below are commonly used tools

- Tool offsets are used to compensate for the length variation of different tools



PROGRAM CONFIGURATION



Program number

Block

Block

Block

:

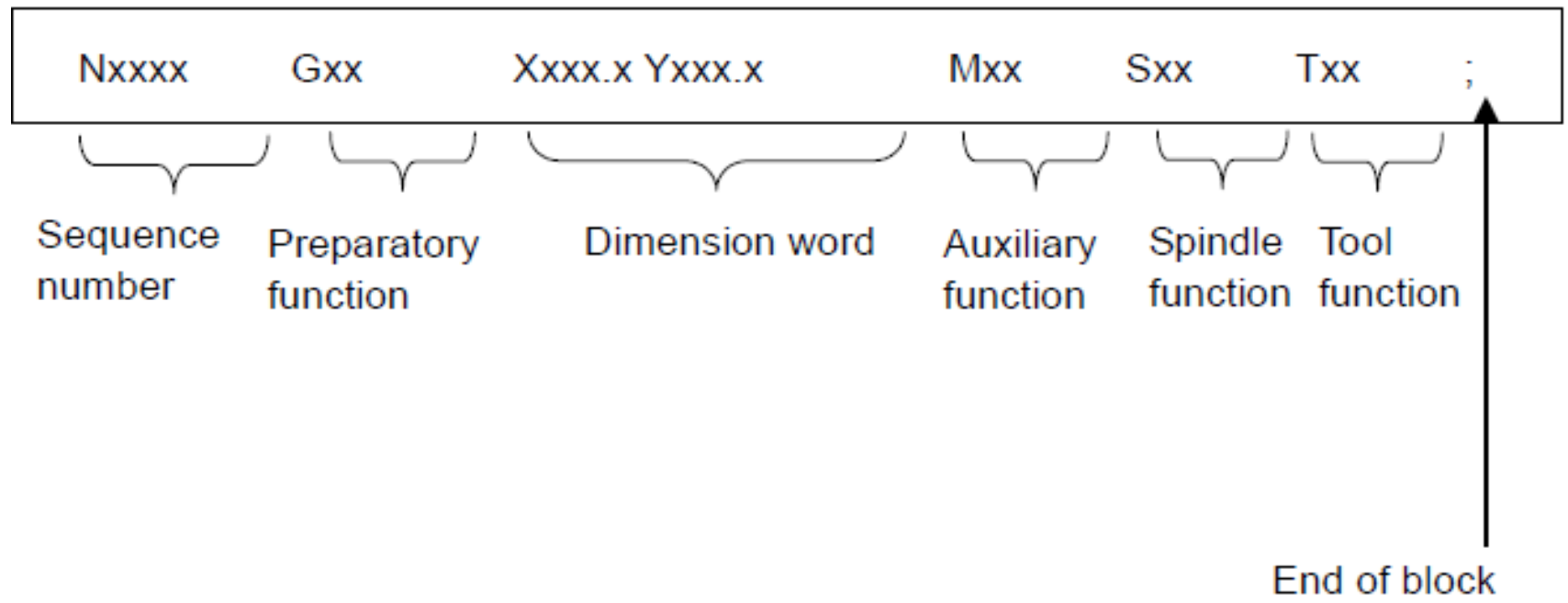
:

:

End of program

Block configuration

1 block



Program structure continued :

Program data

It is possible to set Decimal type entry or Calculator type using CNC parameter(P3401#0).

(Usual setting is decimal type data entry)

Decimal Entry (P3401#0=0)

X40.0 is treated as 40 mm

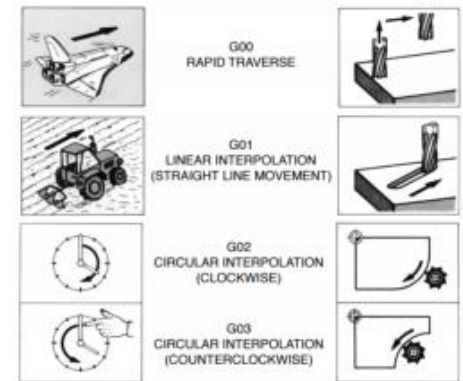
X40 is treated as 40 microns

Calculator type entry (P3401#0=1)

X40.0 is treated as 40 mm

X40 is also treated as 40 mm

G – Codes



G code system			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G31	G31	G31		Skip function
G40	G40	G40	07	Tool nose radius compensation : cancel
G41	G41	G41		Tool nose radius compensation : left
G42	G42	G42		Tool nose radius compensation : right

G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating cycle
G74	G74	G76		End face peck drilling cycle
G75	G75	G77		Outer diameter/internal diameter drilling cycle
G76	G76	G78	Multiple-thread cutting cycle	
G98	G94	G94	05	Feed per minute
G99	G95	G95		Feed per revolution
-	G90	G90	03	Absolute programming
-	G91	G91		Incremental programming

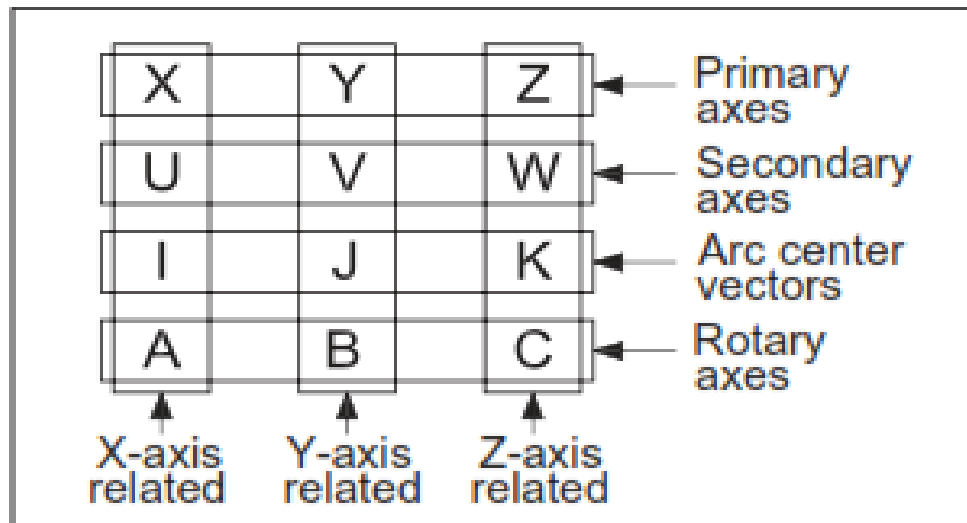
Table of Important M codes

M Codes are instructions describing miscellaneous functions like calling the tool, spindle rotation, coolant on/off etc.,

M00	Program Stop
M01	Optional Stop
M02	Program End
M03	Spindle Forward
M04	Spindle Reverse
M05	Spindle Stop
M06	Automatic Tool change
M08	Coolant On
M09	Coolant Off
M10	Vice / Chuck Open
M11	Vice / Chuck Close
M30	Program Stop & Rewind
M38	Door Open
M39	Door Close
M98	Sub program Call
M99	Subprogram Exit

Additional Axes

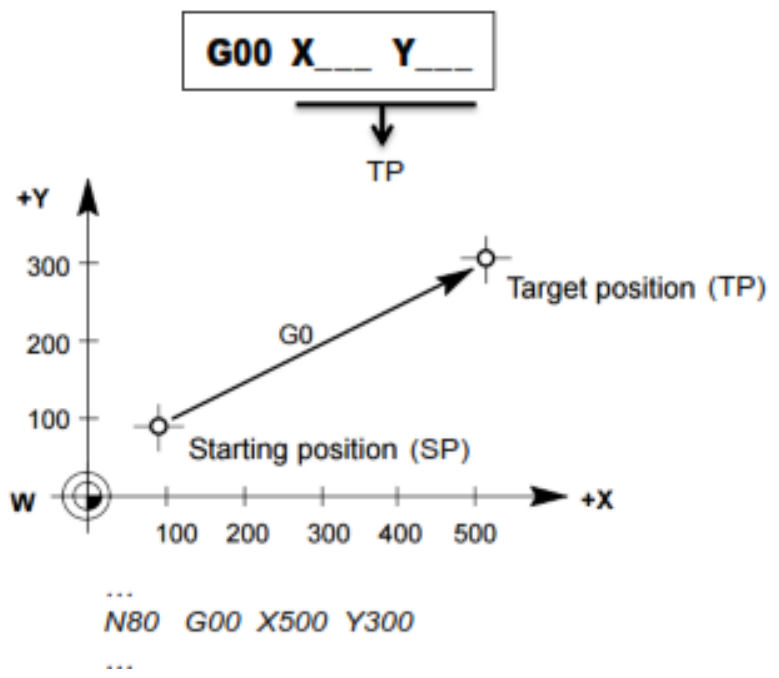
- A CNC machine of any type can be designed with one or more additional axes, normally designated as the secondary or parallel axes using the U, V and W letters.
- For a rotary or an indexing applications, additional axes are defined as A, B and C axes, as being rotated about the X, Y and Z axes, again in their respective order.
- Positive direction of a rotary (or an indexing) axis is the direction required to advance a right handed screw in the positive X, Y or Z axis.



Preparatory functions or G-codes

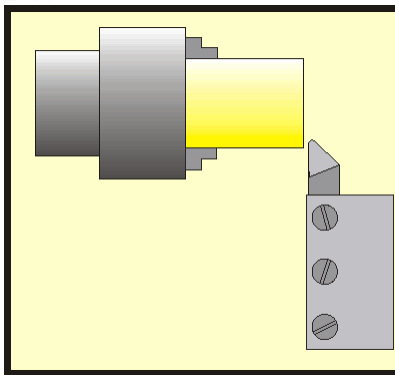
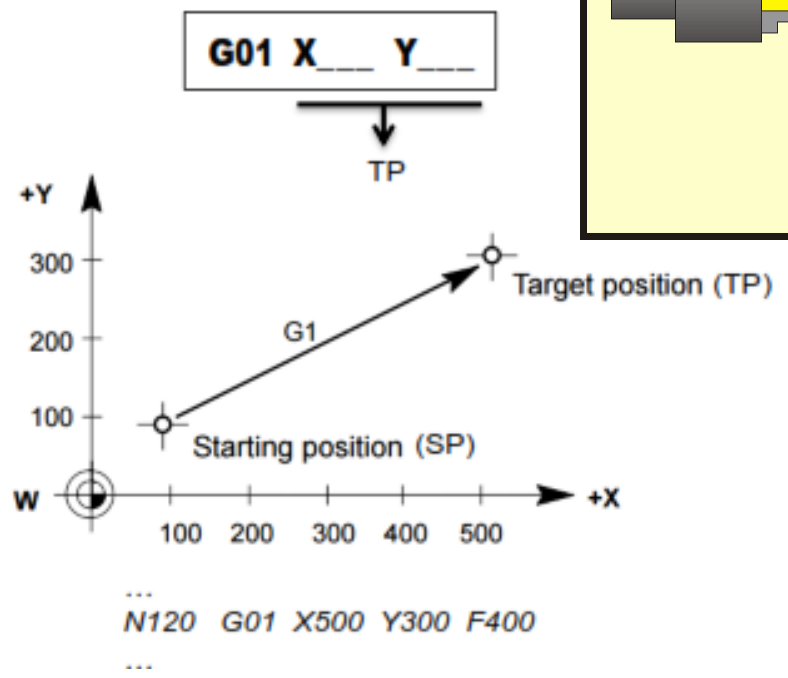
Rapid traverse (G00)

- It is a positioning linear movement at maximum F value defined in the machine parameters.
- Not valid for cutting.
- It can be programmed as G00, G0 or G.



Linear interpolation (G01)

- It is a working linear movement at the programmed F value.
- It can be programmed as G01 or G1.

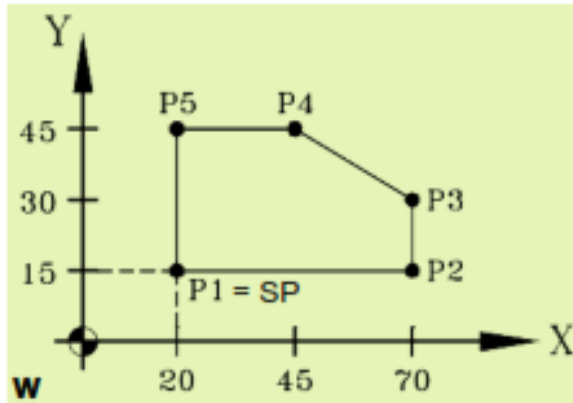


Preparatory functions or G-codes

Rapid traverse (G00)

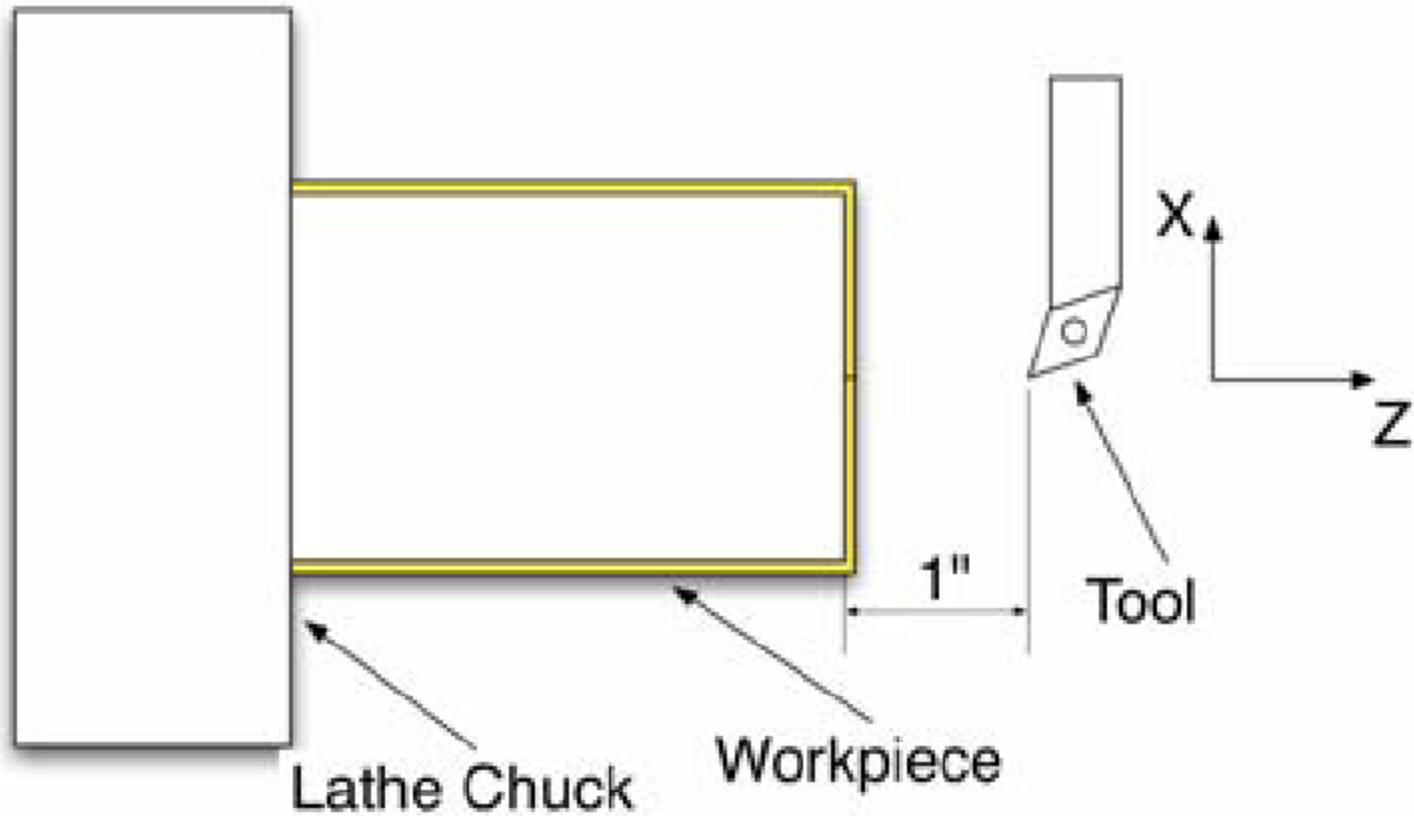
Linear interpolation (G01)

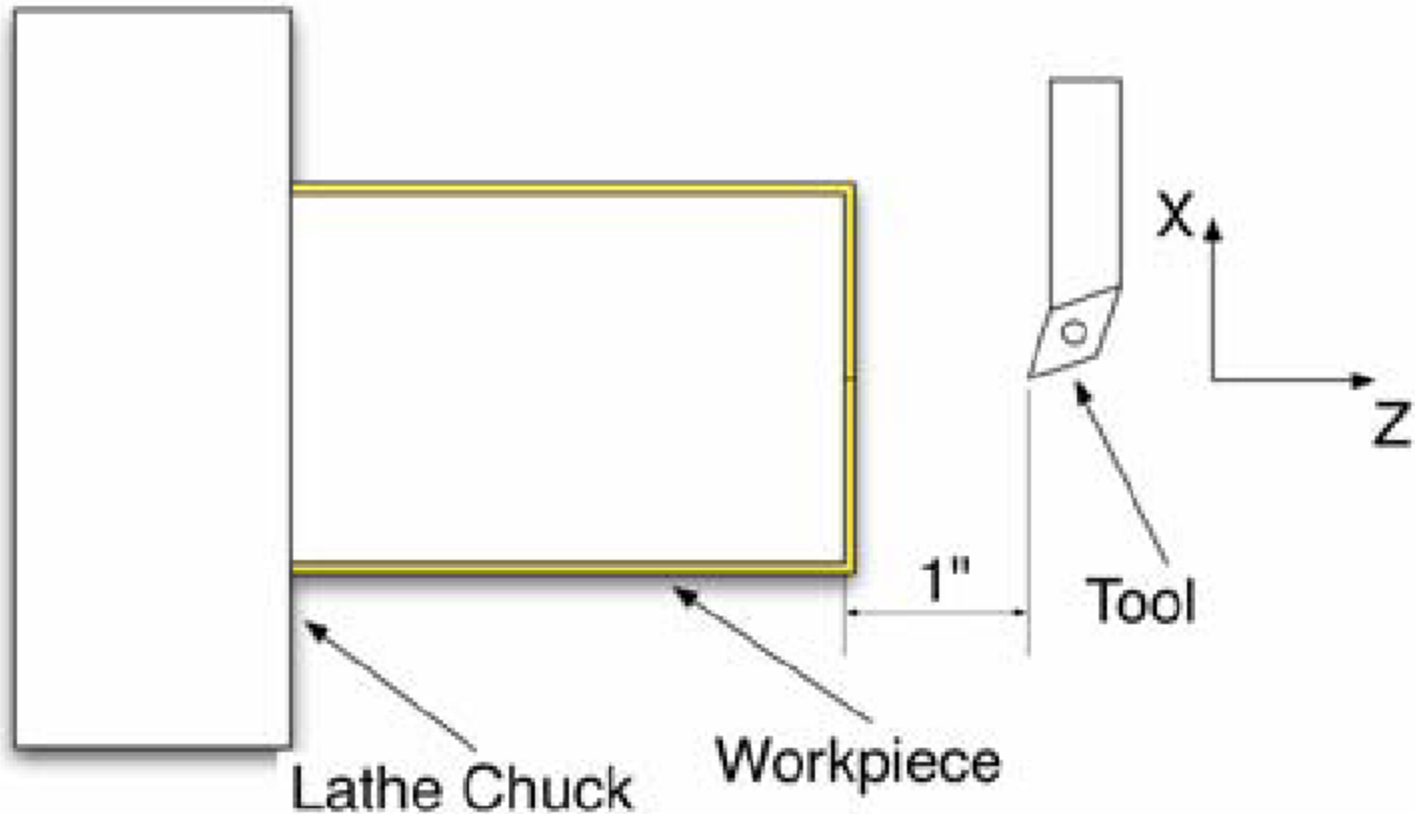
EXERCISE 1



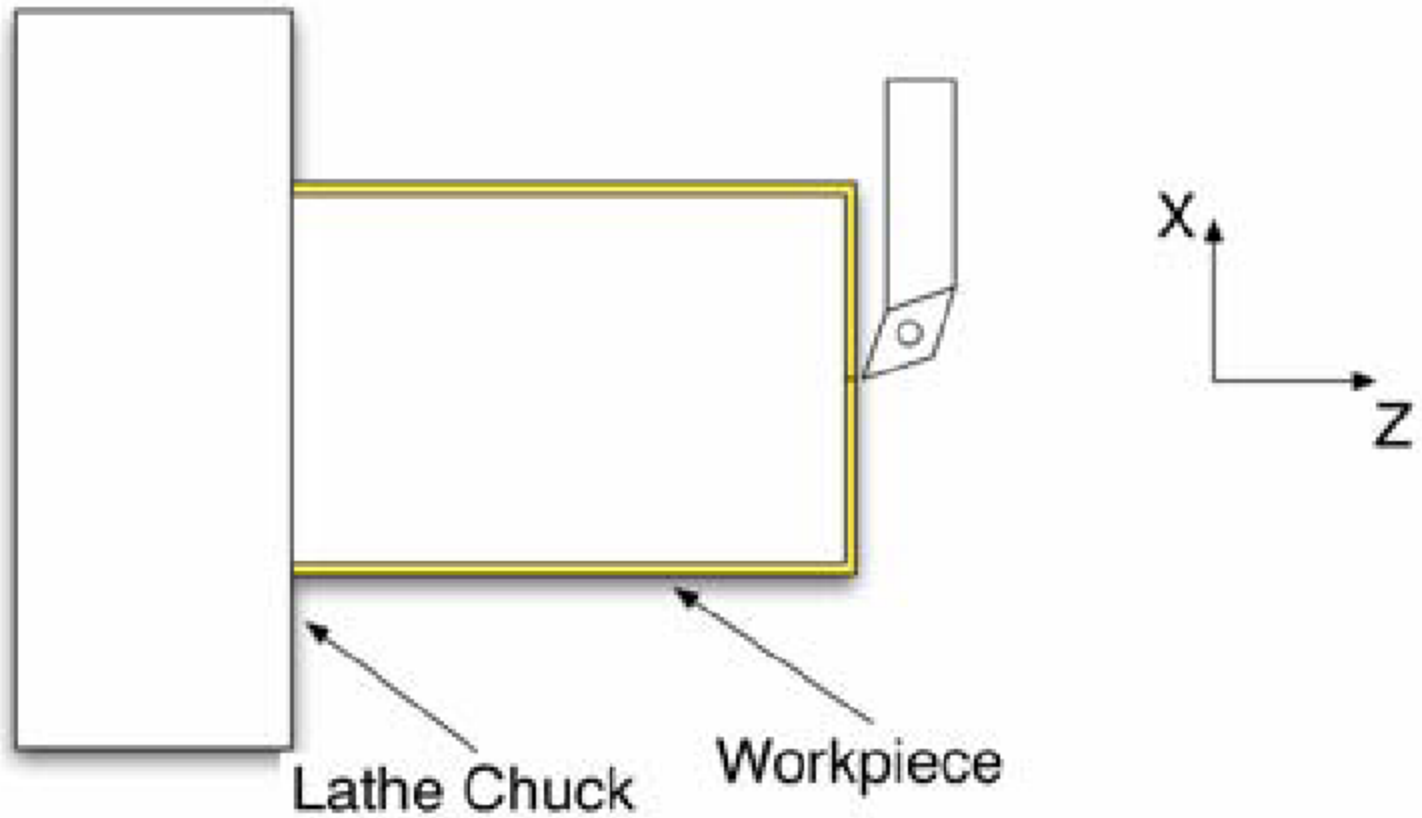
	X	Y
P1	20	15
P2	70	15
P3	70	30
P4	45	45
P5	20	45

Lathe Turning

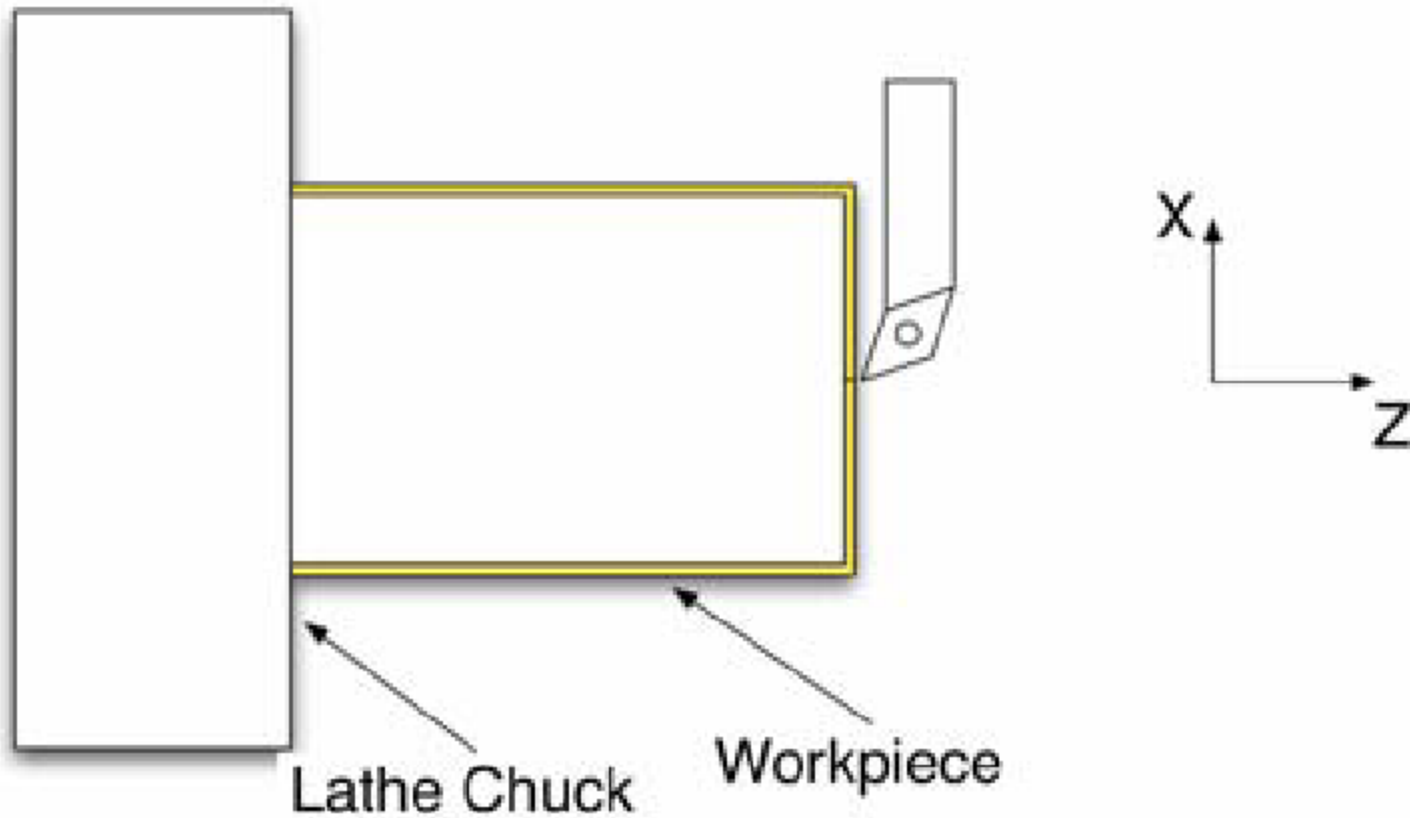




N001 G90 ; Absolute coordinate system
N002 M06 T01 ; Select tool #1 (turning tool)
N003 M03 S2000 ; Turn spindle on (CW), set 2000 rpm
N004 G00 X0.0 Z-0.9 ; Rapid traverse

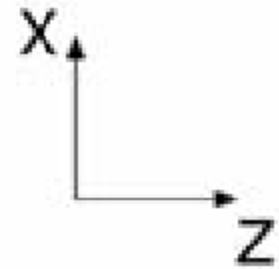
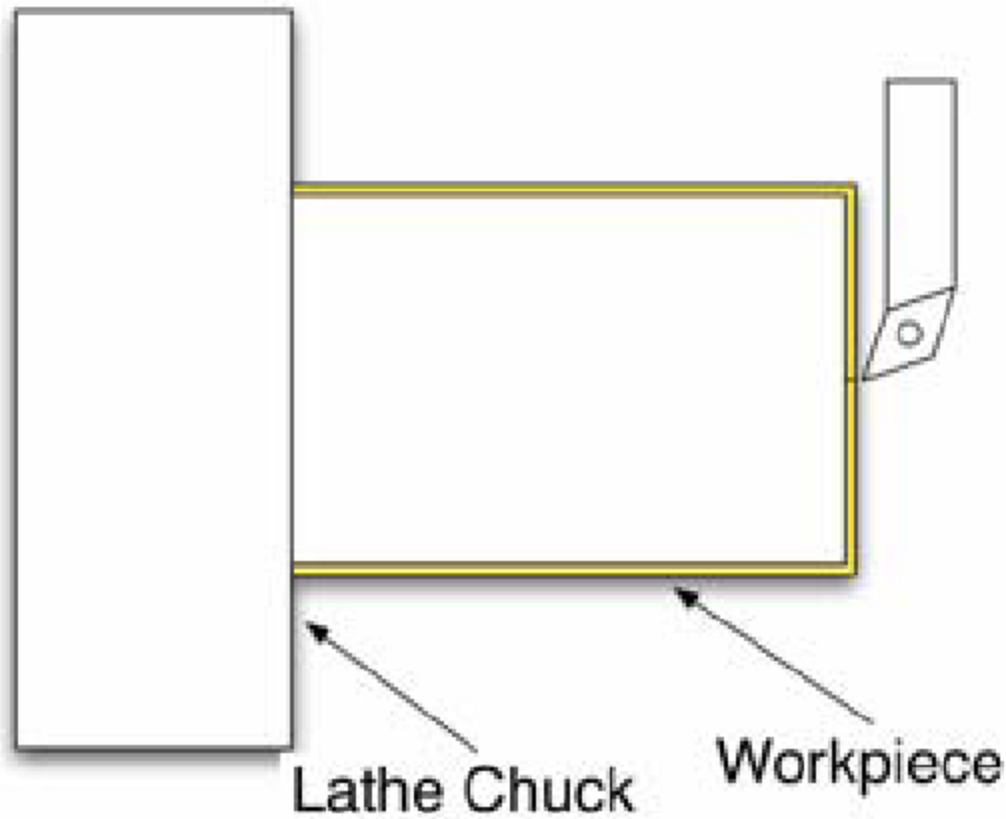


N004 G00 X0.0 Z-0.9 ; Rapid Traverse

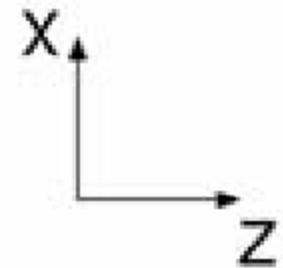
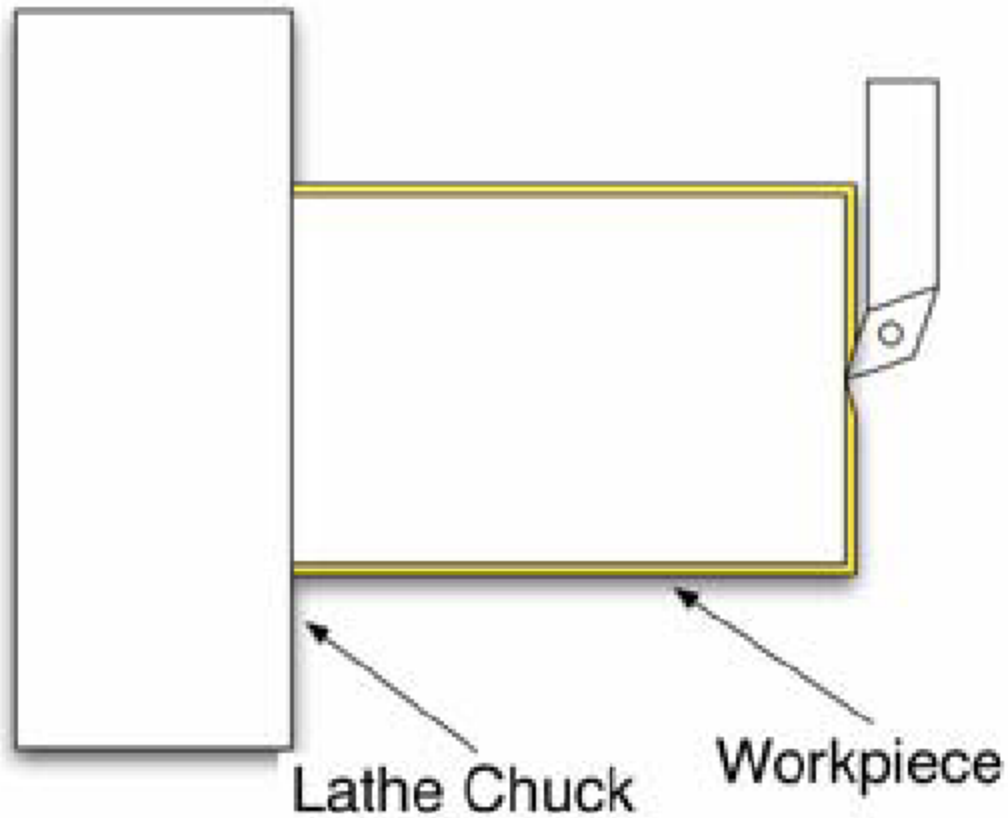


N004 G00 X0.0 Z-0.9 ; Rapid Traverse

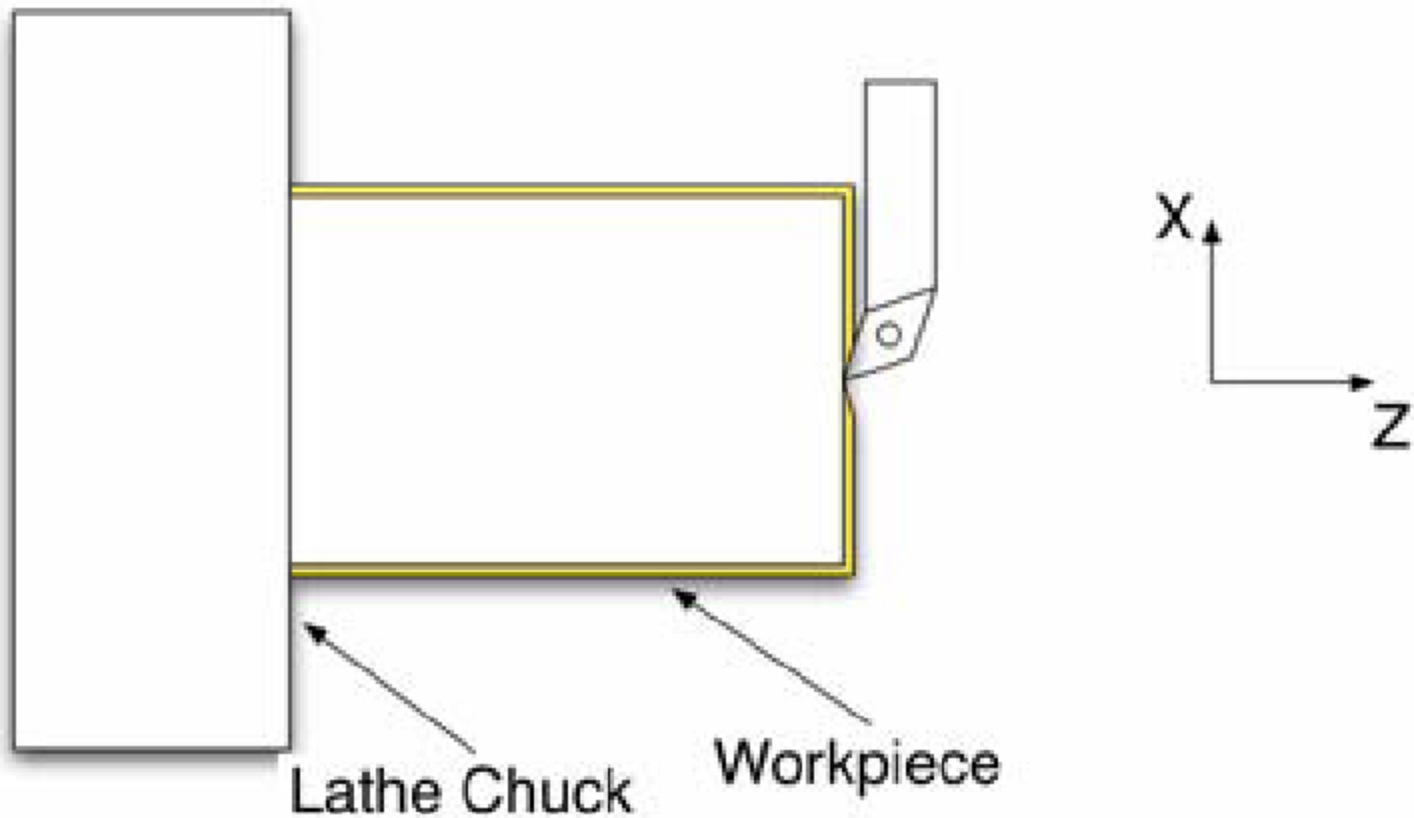
Do not run the tool into the workpiece during Rapid Traverse!



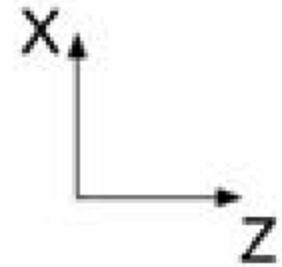
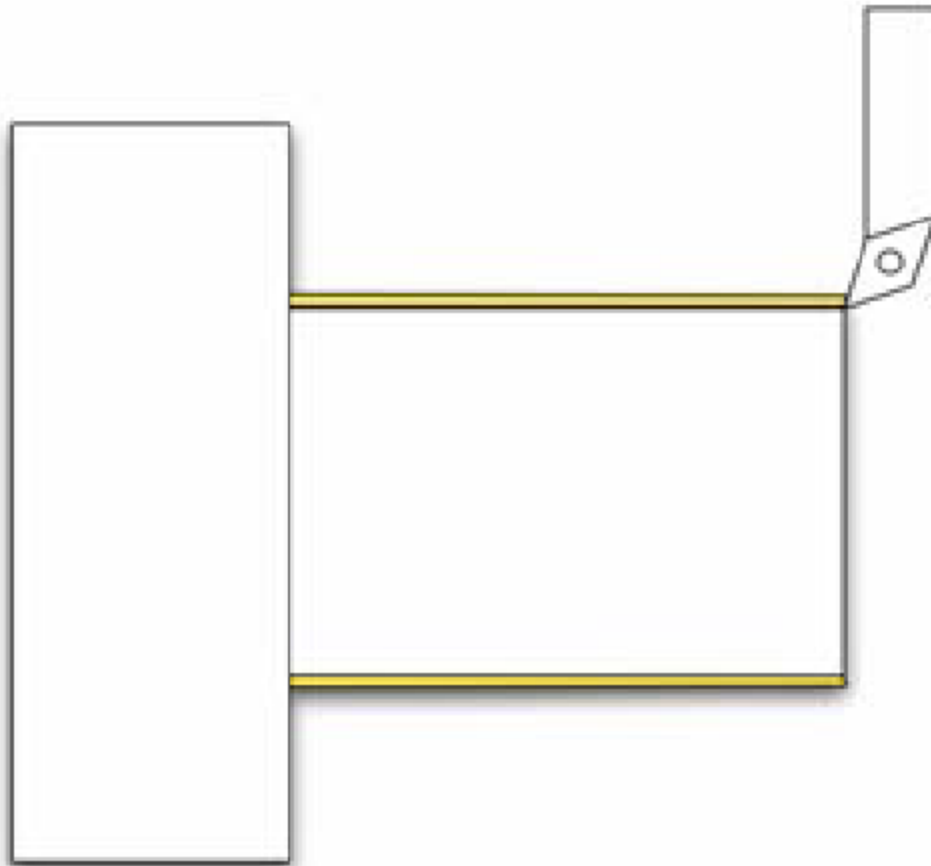
N005 G01 X0.0 Z-1.0 F10 ; enter part



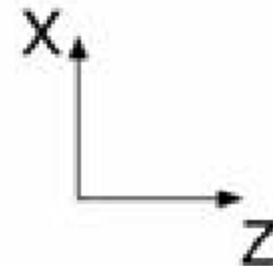
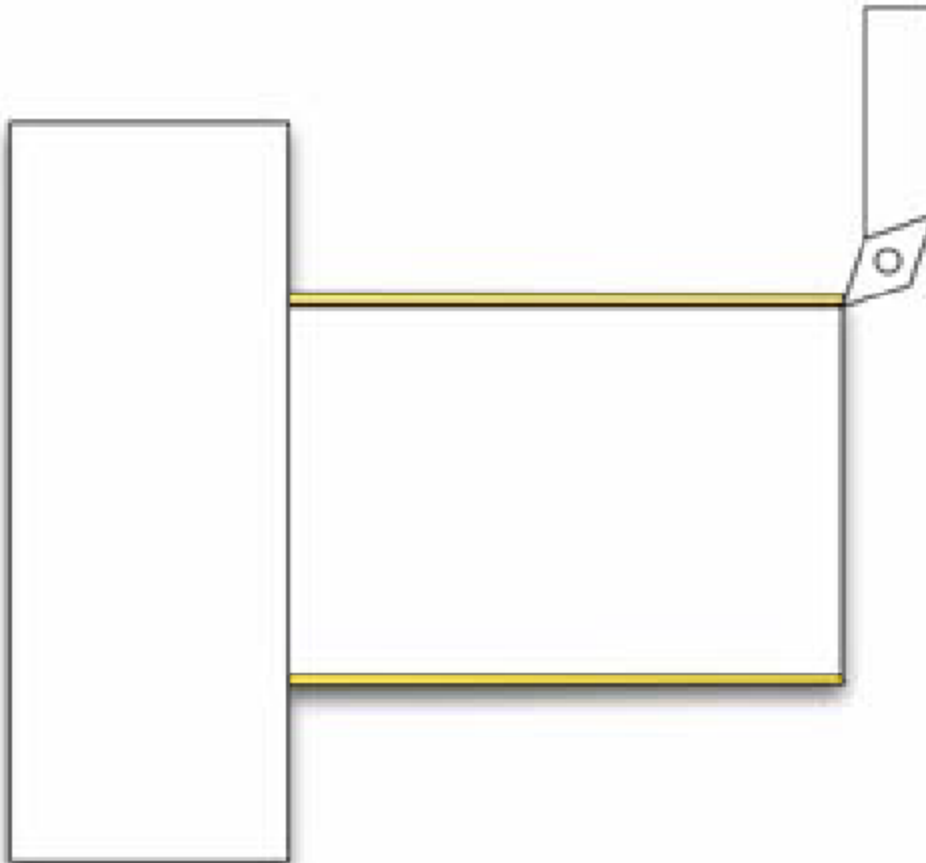
N005 G01 X0.0 Z-1.0 F10 ; enter part



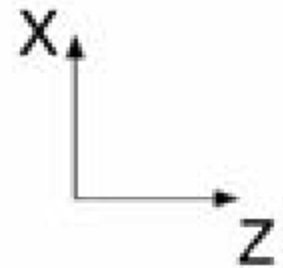
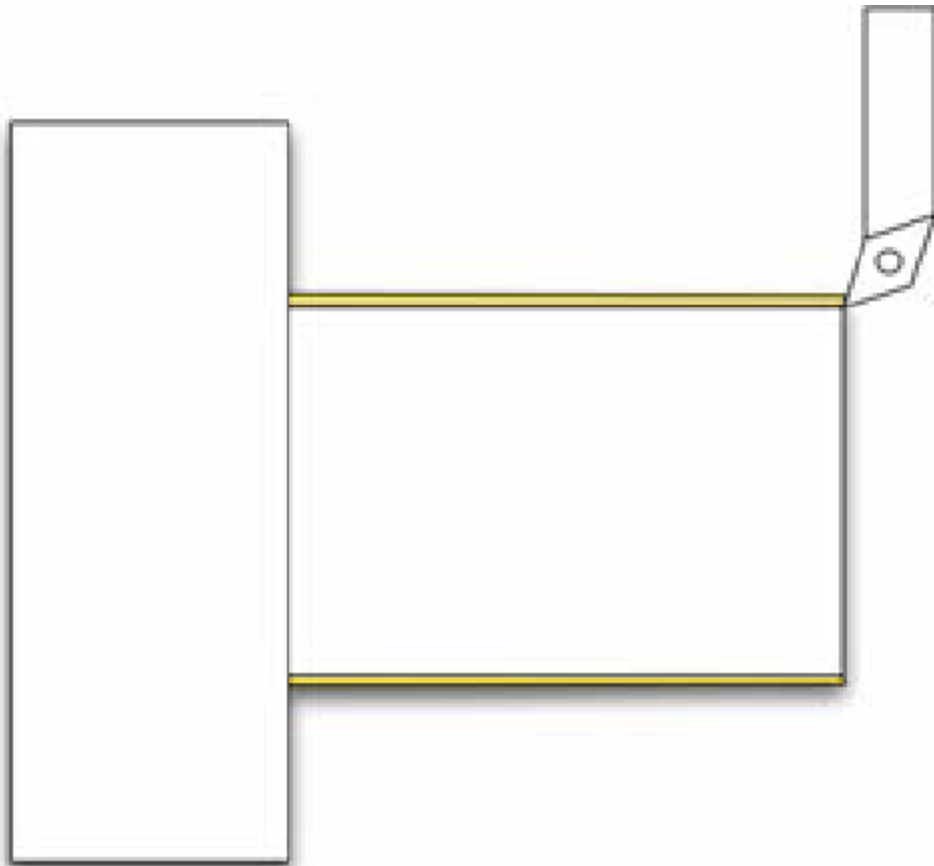
N005 G01 X0.0 Z-1.0 F10 ; enter part
N006 G01 X1.0 Z-1.0 F10 ; make facing cut



N005 G01 X0.0 Z-1.0 F10 ; enter part
N006 G01 X1.0 Z-1.0 F10 ; make facing cut

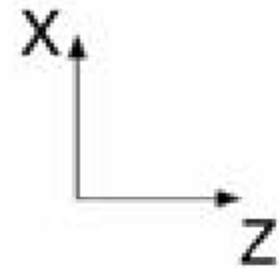
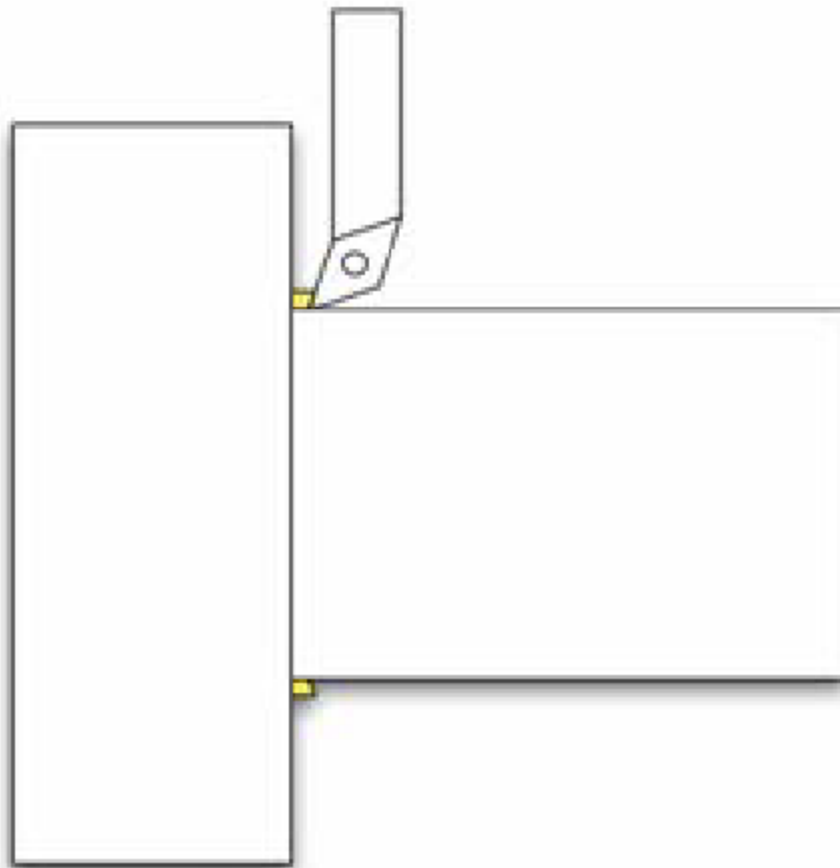


N005 G01 X0.0 Z-1.0 F10 ; enter part
N006 G01 X1.0 Z-1.0 F10 ; make facing cut
N006 G01 X1.0 ; can omit unchanged values



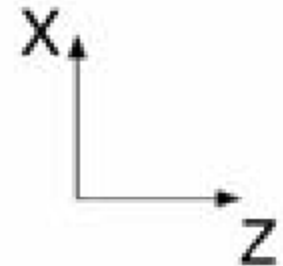
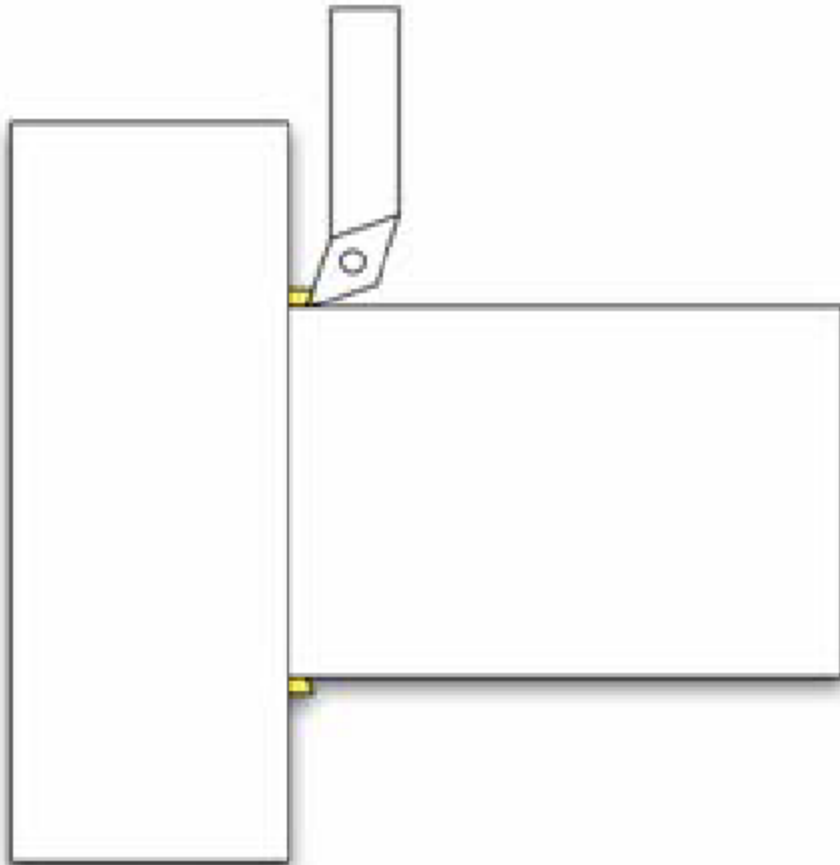
N006 G01 X1.0
N007 G01 Z-3.9

; make facing cut
; turn side



N006 G01 X1.0
N007 G01 Z-3.9

; make facing cut
; turn side



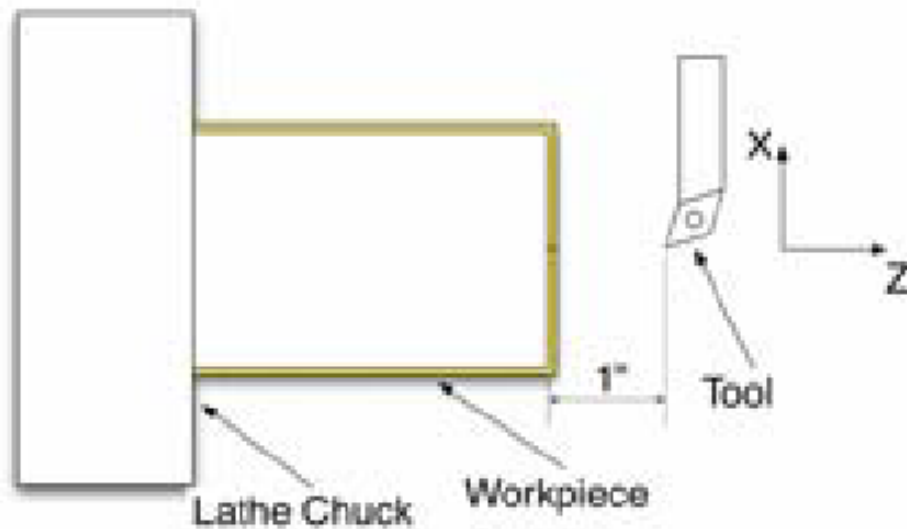
N006 G01 X1.0

; make facing cut

N007 G01 Z-3.9

; turn side

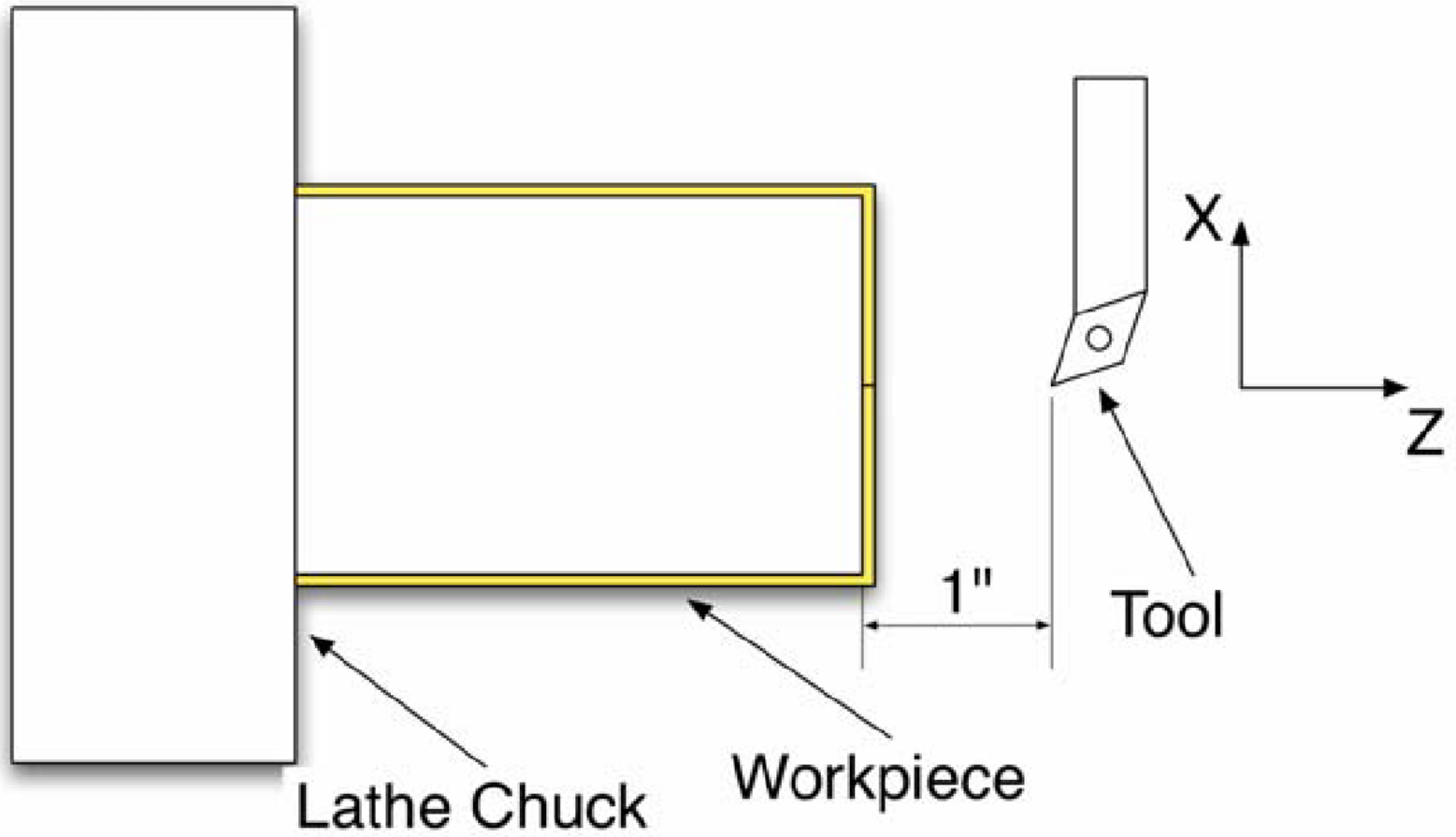
Do not run the tool into the chuck during machining!



```

N001 G90           ; Absolute coordinate system
N002 M06 T01      ; Select tool #1 (turning tool)
N003 M03 S2000    ; Turn spindle on (CW), set 2000 rpm
N004 G00 X0.0 Z-0.9 ; Rapid traverse
N005 G01 X0.0 Z-1.0 F10 ; enter part
N006 G01 X1.0     ; make facing cut
N007 G01 Z-3.9    ; turn side

```



N001 G90 ; Absolute coordinate system
N002 M06 T01 ; Select tool #1 (turning tool)
N003 M03 S2000 ; Turn spindle on (CW), set 2000 rpm
N004 G00 X0.0 Z-0.9 ; Rapid traverse
N005 G01 Z-1.0 F10 ; Enter part
N006 X1.0 ; Make facing cut
N007 Z-3.9 ; Make side cut
N008 G00 X1.1 Z-3.8 ; Leave part surface
N008 G00 Z0 ; Rapid move for retracting in Z-axis
N009 M05 X0 ; Spindle off, rapid move to (0,0)
N010 M30 ; End program

Preparatory functions or G-codes

Clockwise circular interpolation (G02) ↻

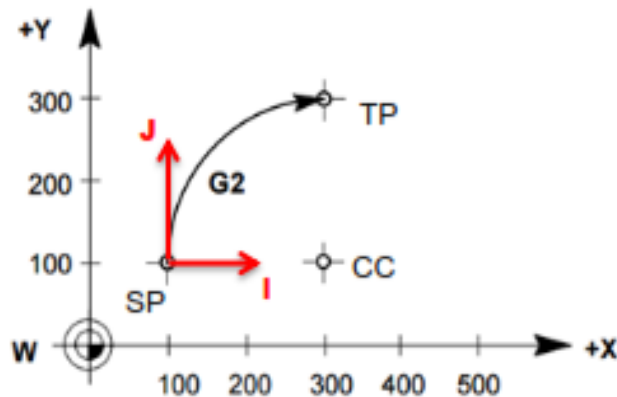
Counterclockwise circular interpolation (G03) ↺

- It is a working circular movement at the programmed F value.
- It can be programmed as G02 or G2 / G03 or G3.

CARTESIAN COORDINATES
WITH ARC CENTER

G02 X ___ Y ___ I ___ J ___

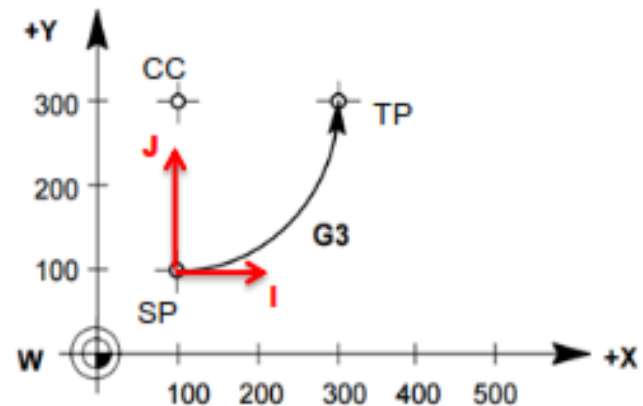
TP Distance from the SP to
the Circle Center (CC).



...
N60 G02 X300 Y300 I200 J0
...

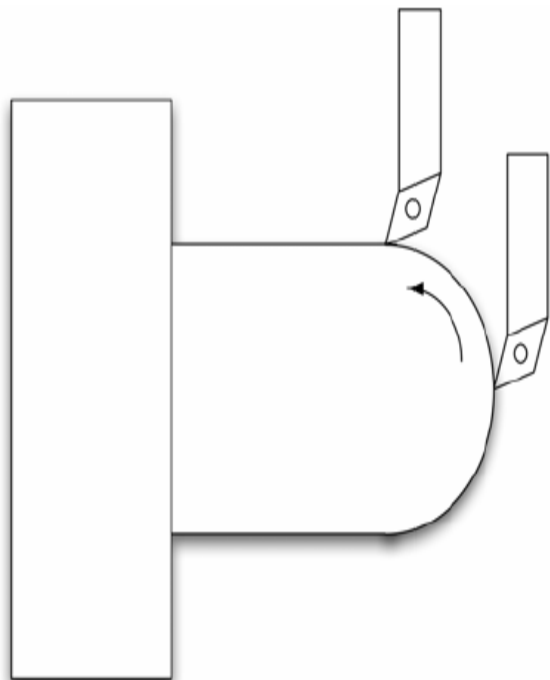
G03 X ___ Y ___ I ___ J ___

TP Distance from the SP to
the Circle Center (CC).

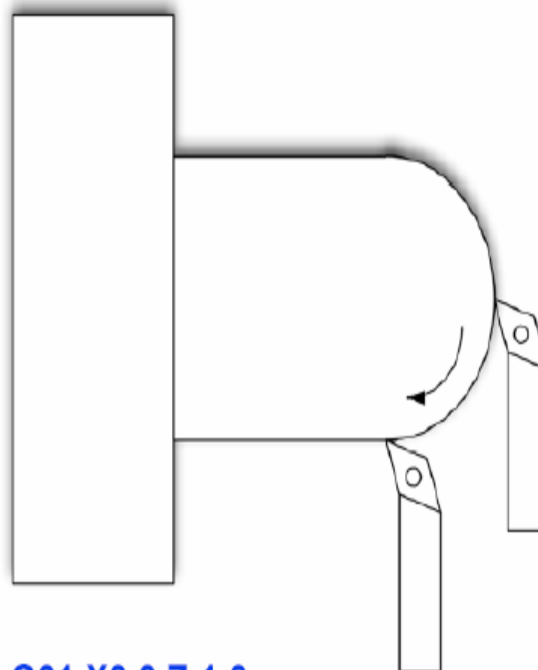


...
N60 G03 X300 Y300 I0 J200
...

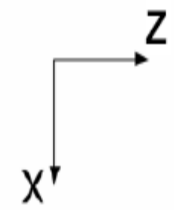
Circular Interpolation



N001 G01 X0.0 Z-1.0
N002 G02 X1.0 Z-2.0 I0.0 K-2.0 F5.0 ; cut CW arc
X,Z are end coordinates; I,K are arc center coordinates
CW/CCW Defined by -Y axis **NOT STANDARD!**



N001 G01 X0.0 Z-1.0
N002 **G02** X1.0 Z-2.0 I0.0 K-2.0 F5.0 ; cut CW arc
X,Z are end coordinates; I,K are arc center coordinates
CW/CCW Defined by **-Y axis**



CNC Turning - Clockwise or Counter Clockwise ?

Back cutting lathe (default)



Front cutting lathe



* CW - Clockwise direction as viewed from top of machine
CCW - Counter clockwise as viewed from top of machine

Preparatory functions or G-codes

Clockwise circular interpolation (G02) ↻

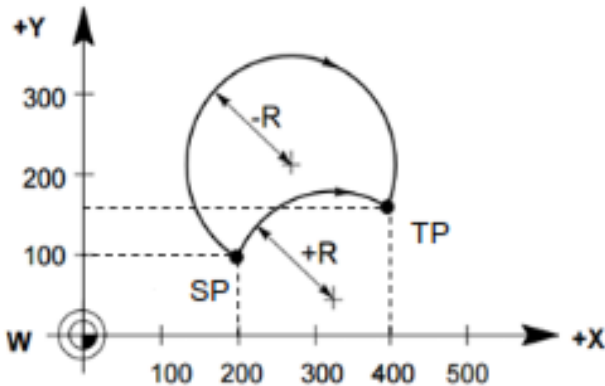
Counterclockwise circular interpolation (G03) ↺

CARTESIAN COORDINATES WITH ARC RADIUS

- A complete circle cannot be programmed.

G02 X___ Y___ R___

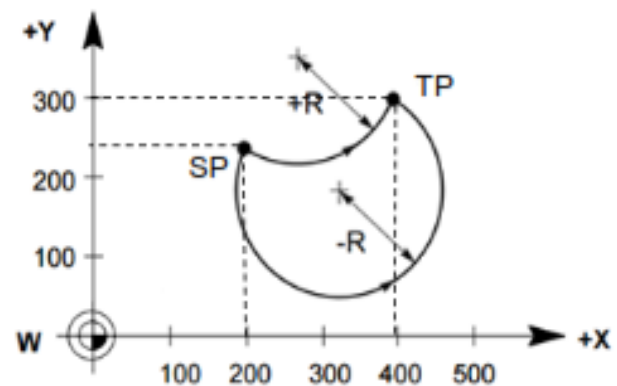
TP R + : Arc < 180°
R - : Arc > 180°



R+ {
...
N40 G02 X400 Y150 R150
...
R- {
...
N40 G02 X400 Y150 R-150
...

G03 X___ Y___ R___

TP R + : Arc < 180°
R - : Arc > 180°



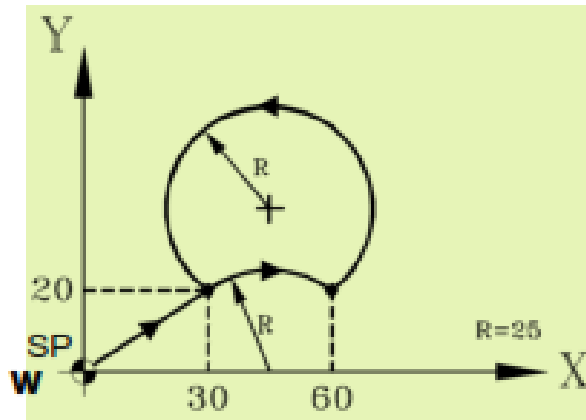
R+ {
...
N40 G03 X400 Y300 R150
...
R- {
...
N40 G03 X400 Y300 R-150
...

Preparatory functions or G-codes

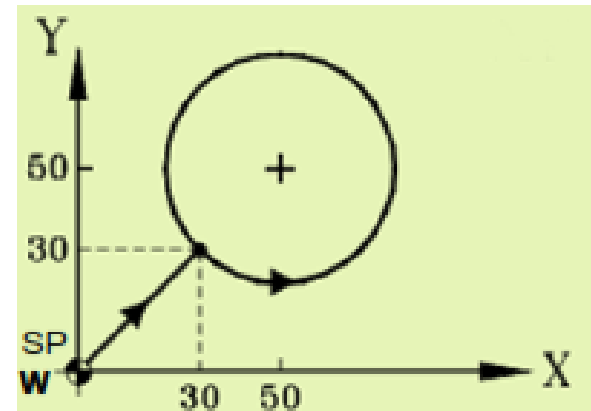
Clockwise circular interpolation (G02) ↻

Counterclockwise circular interpolation (G03) ↺

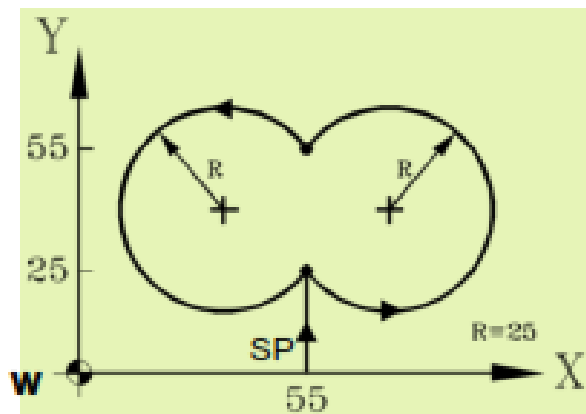
EXERCISE 2



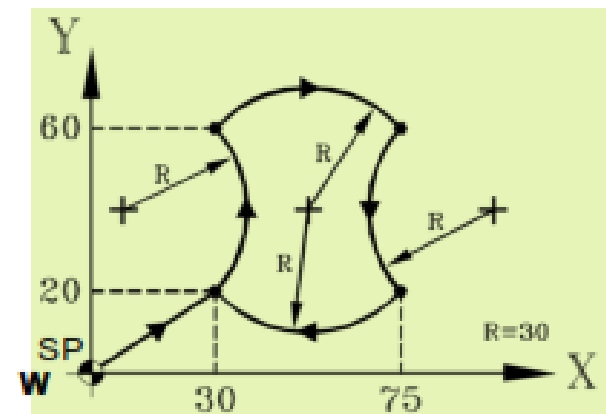
EXERCISE 3



EXERCISE 4



EXERCISE 5

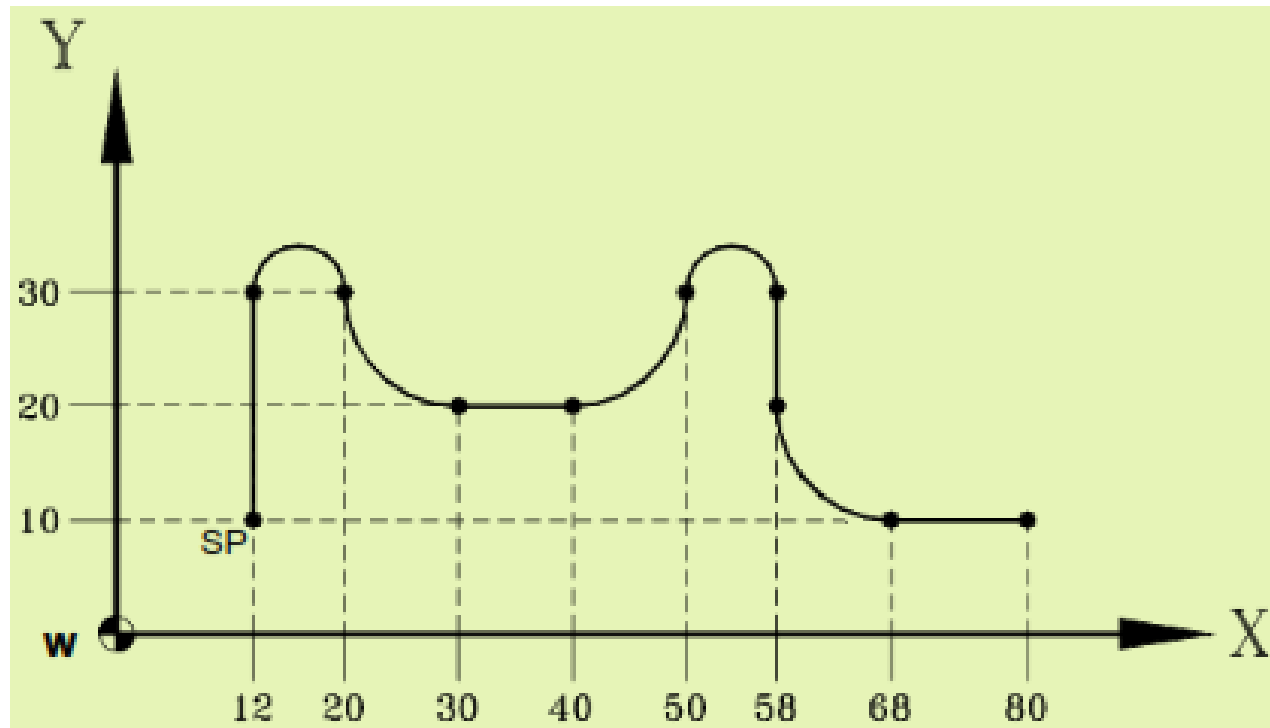


Preparatory functions or G-codes

Clockwise circular interpolation (G02) ↻

Counterclockwise circular interpolation (G03) ↺

EXERCISE 6



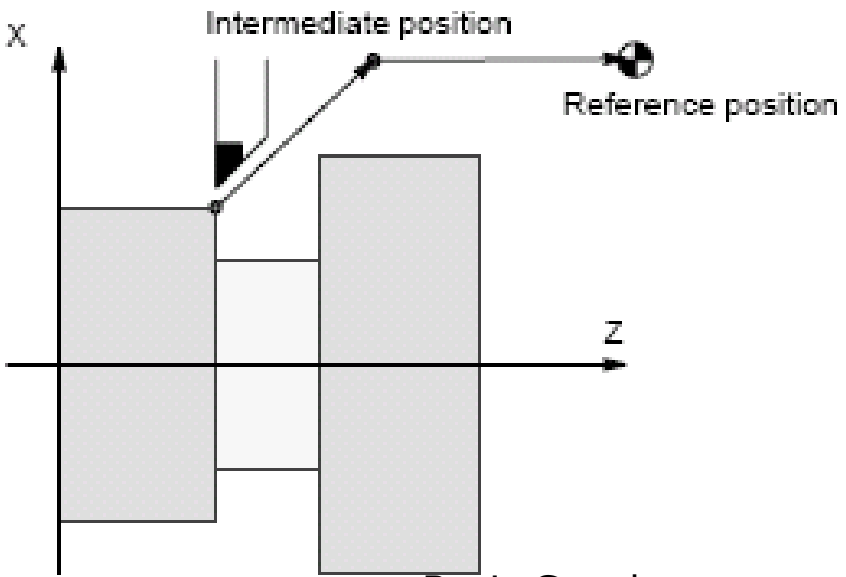
G28 Reference Position Return

- This code is used to move the turret towards home position for tool indexing with rapid traverse (No need to cancel offsets, while doing so)

Format is as follows

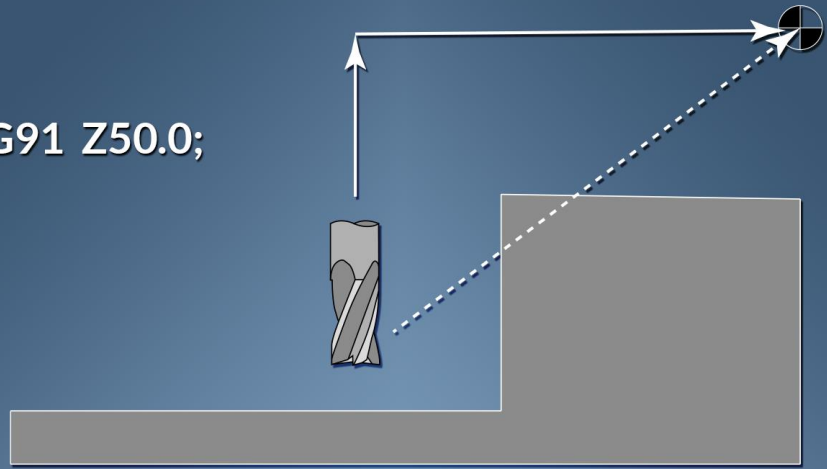
```
G28 IP _____
```

IP is intermediate position



G28 Reference Return

```
G28 G91 Z50.0;
```

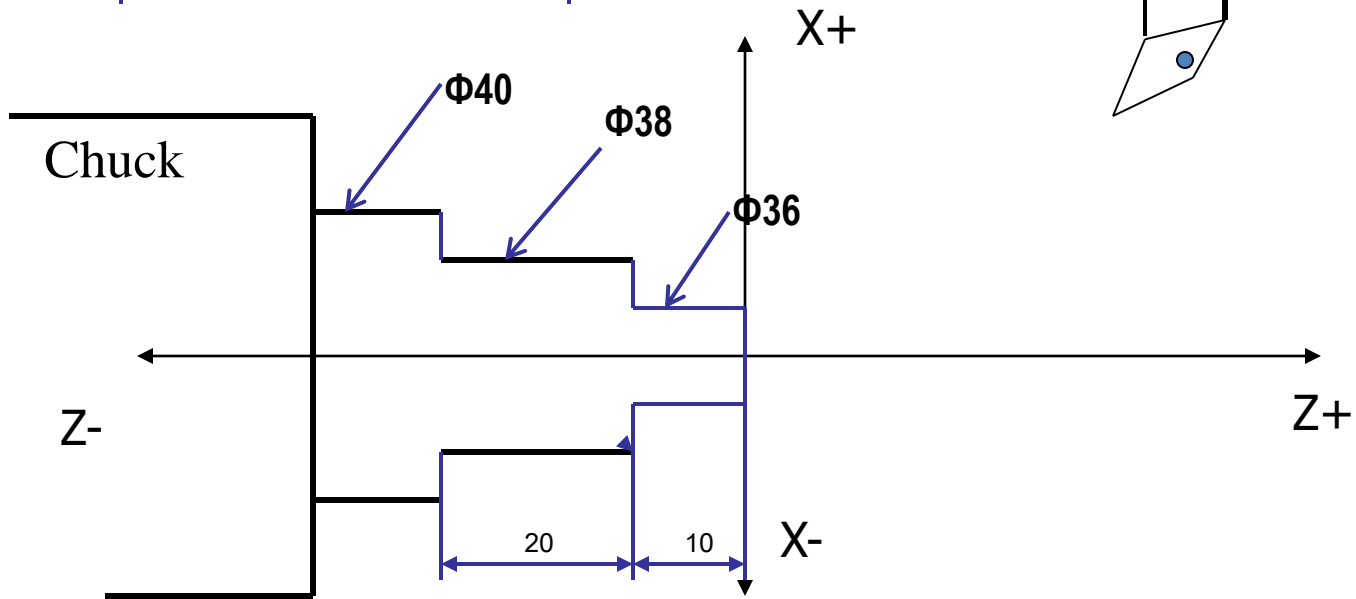
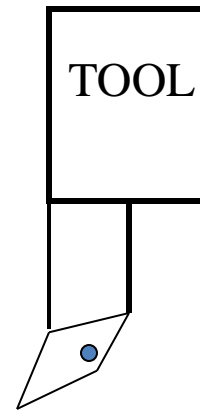
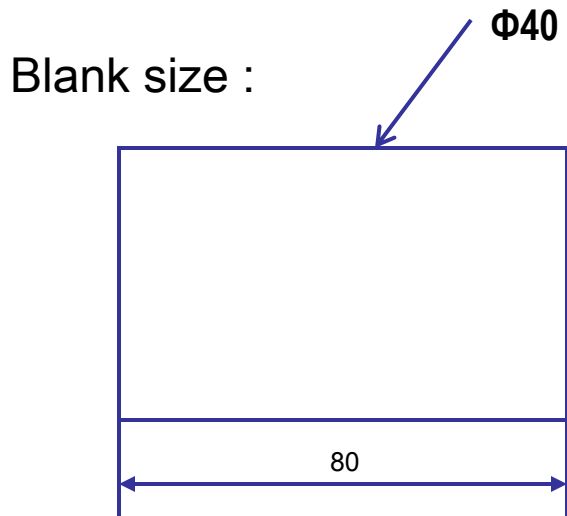


```
G28 G91 Z X Y;
```

- G28 - Reference Return
- G91 - Incremental Mode
- Z - Distance in Z before return
- X - Distance in X before return
- Y - Distance in Y before return

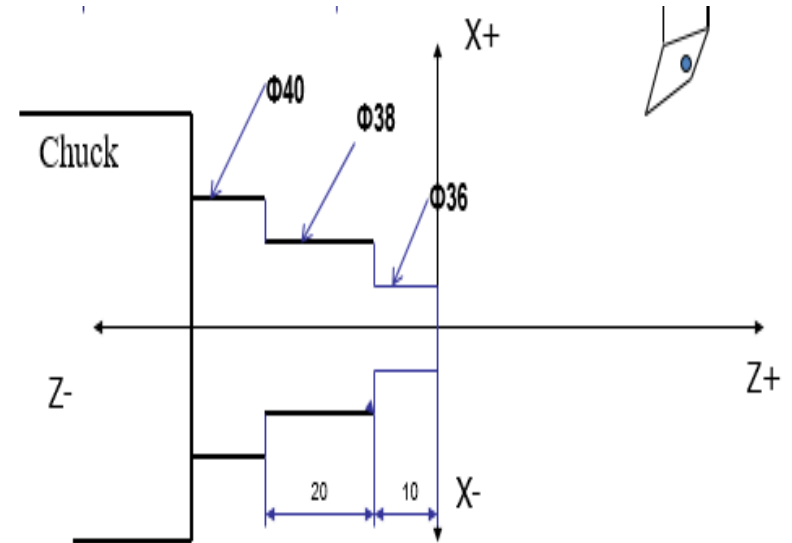
By stating a distance in an axis before returning to the home position, we can avoid obstacles in the machine

Basic part program



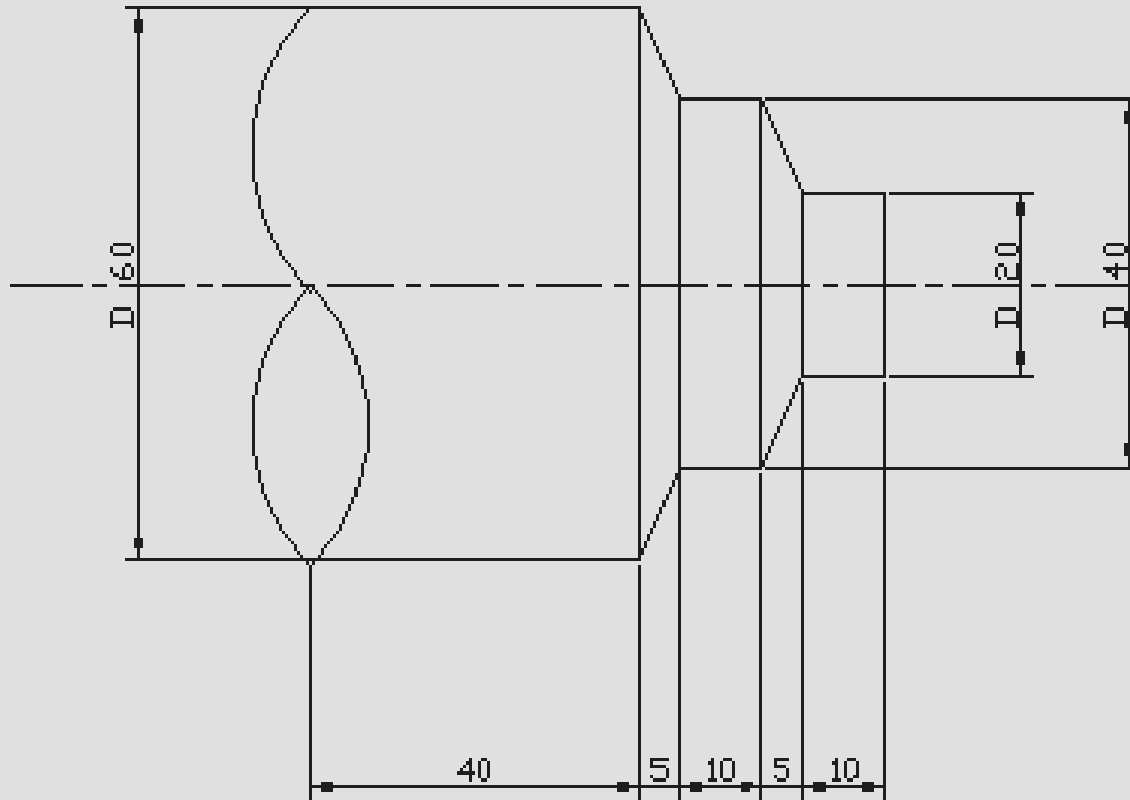
Program - G01 and G02

- O0001;
- N10 G21 G99;
- N20 G28 U0 W0;
- N30 T0101;
- N40 S1250 M03;
- N50 G00 X41.0;
- N60 G00 Z0.0;
- N70 G01 X-1.0 F0.3;
- N80 G00 Z2.0;
- N90 G00 X41.0;
- N100 G00 Z0.0;
- N110 G01 X38.0 F0.3;
- N120 G01 Z-30.0 F0.3;
- N130 G01 X41.0;
- N140 G00 Z0.0;
- N150 G01 X36.0 F0.3;
- N160 G01 Z-10.0 F0.3;
- N170 G01 X41.0;
- N180 G00 Z0.0;
- N190 G28 U0 W0;
- N200 M05;
- N210 M30;



CNCEZ TURNING - I

SPECIMEN DRAWING



ALL DIMENSIONS ARE IN MM

Program:

:%

: 1001

N05 G90 M06 T01 G21

N10 M03 S1500

N15 G00 X0 Z0

N20 G01 X20 Z0 F35

N25 G01 X20 Z-10

N30 G01 X40 Z-15

N35 G01 X40 Z-25

N40 G01 X60 Z-30

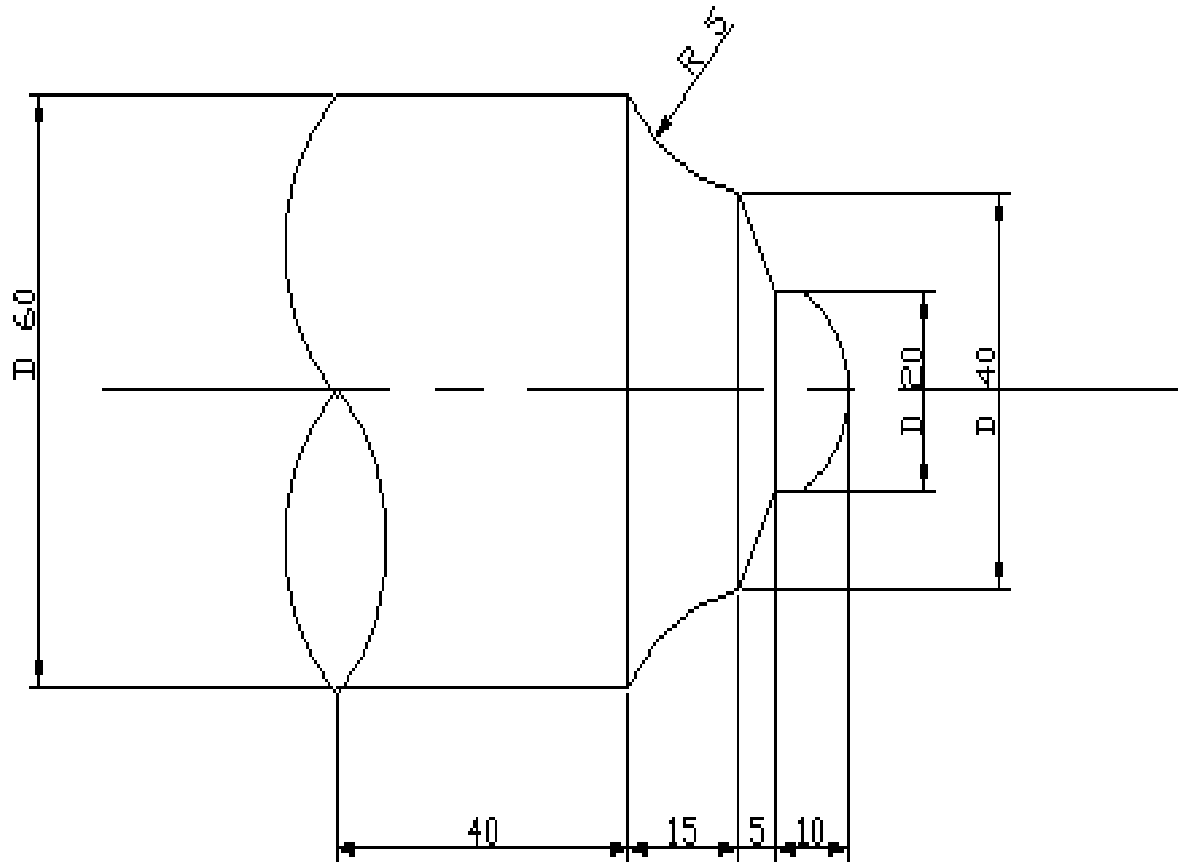
N45 G01 X60 Z-70

N50 G00 X85 Z10

N55 M05 M30

CNCEZ TURNING - II

SPECIMEN DRAWING



ALL DIMENSIONS ARE IN mm

Program:

:%

: 1002

N05 G90 M06 T01 G21

N10 M03 S1500

N15 G00 X0 Z0

N20 G03 X20 Z-5 R10 F40

N25 G01 X20 Z-10

N30 G01 X40 Z-15

N35 G02 X60 Z-25 R 15

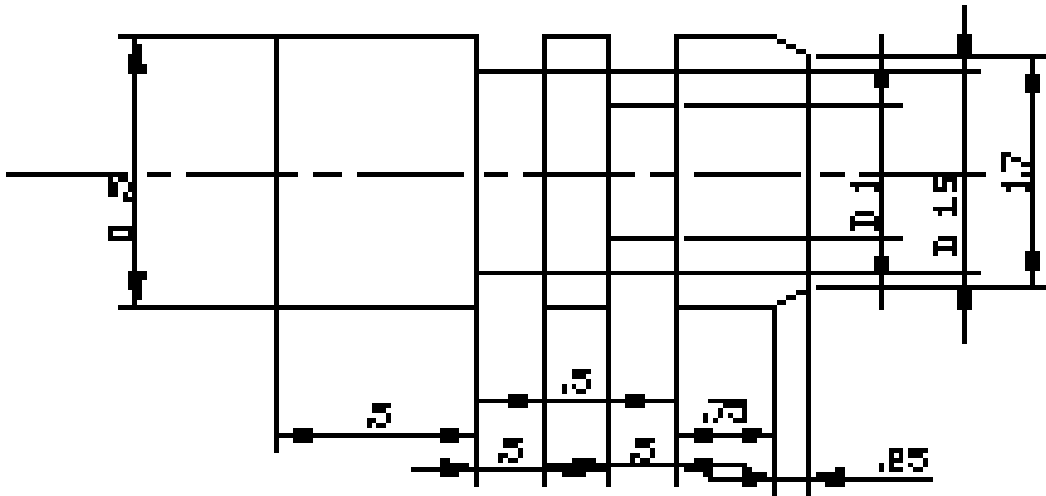
N40 G01 X60 Z-70

N45 G00 X80 Z10

N50 M05 M30

CNCEZ TURNING - III

specimen drawing



All Dimensions
Are In Inches

Program:

```
:%  
: 1003  
N05 G90 G20  
N10 M03 S1500  
N15 M06 T01  
N20 G00 X0 Z0  
N25 G01 X1.7 Z0 F40  
N30 X2 Z-0.25  
N35 X2 Z-2.5  
N40 X2.5 Z10  
N45 M06 T02  
N50 G00 X2 Z-1  
N55 G01 X1 Z-1 F40  
N60 G01 X1 Z-1.5  
N65 G01 X2 Z-1.5  
N70 G01 X2 Z-2  
N75 G01 X1.7 Z-2  
N80 G01 X1.7 Z-2.5  
N85 G01X2 Z-2.5  
N90 G00 X3 Z1  
N95 M05  
N100 M30
```

Programming Example

Cylindrical Part

O0013

N0005 G53

N0010 T0303

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X22.50 Z2.0 S500

N0080 G01 Z-30.0 F100

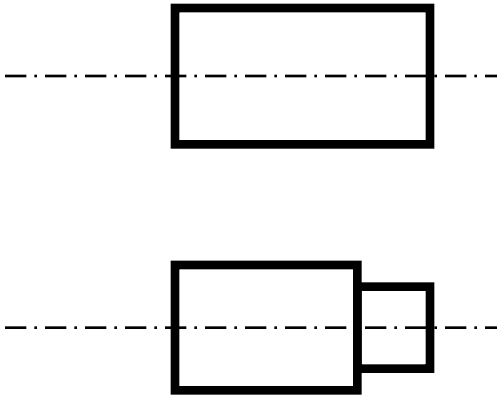
N0090 G00 X23.0 Z2.0 S500

N0100 G84 X17.5 Z-20.0 D₀=200 D₂=200 D₃=650

N0110 G00 Z2.0

N0120 X50.0 Z50.0

N0130 M30



Program Interpretation

00013

Program identification number

Program Interpretation

O0013

N0005 G53

To cancel any previous working zero point

Program Interpretation

O0013

N0005 G53

N0010 T0303

N0010 Sequence number

T0303 Select tool number 303

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.0 Z0.0 S500 M04

G57 To set the working zero point as saved

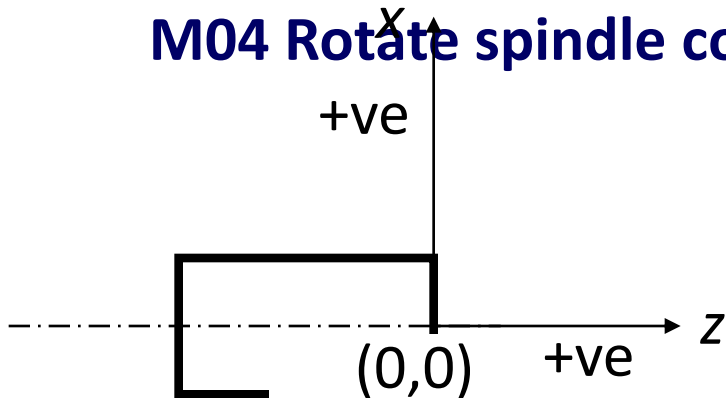
G00 Rapid movement (no cutting)

X26.0 X location (as a diameter; 13 form zero)

Z0.0 Z location

S500 Spindle speed is 500 rpm

M04 Rotate spindle counterclockwise



Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

G01 Linear interpolation (cutting)

**X-0.20 Move only in x direction until you pass
the center by 0.1 mm (facing)**

F100 Set feed rate to 100 mm/min.



Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

G00 Move rapidly away from workpiece (no cutting)

Z2.0 the movement is 2 mm away from the face.

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

Go to a safe location away from the workpiece [x = 50 (25 from zero), z = 50] to change the tool.

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

T0404 Select tool number 404

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X22.50 Z2.0 S500

G57 PS0

G00 Rapid movement (no cutting)

X22.50 X location (as a diameter; 11.25 from zero)

Z2.0 Z location

S500 Spindle speed is 500 rpm

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

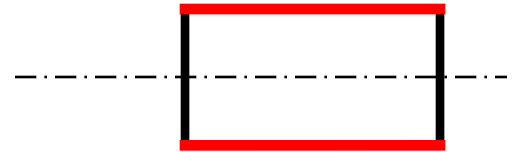
N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X25.00 Z2.0 S500 M04

N0080 G01 Z-30.0 F100



G01 Linear interpolation (cutting)

Z-30 Move only in z direction (external turning)

F100 Set feed rate to 100 mm/min.

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X25.00 Z2.0 S500 M04

N0080 G01 X22.5 Z-70.0 F100

N0090 G00 X23.0 Z2.0 S500

G00 Move rapidly away from workpiece (no cutting) to location $x = 23.0$ (11.50 from zero) and $z = 2.0$.

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X25.00 Z2.0 S500 M04

N0080 G01 X22.5 Z-70.0 F100

N0090 G00 X26.0 Z2.0 S500

N0100 G84 X17.5 Z-20.0 D0=200 D2=200 D3=650

G84 Turning cycle for machining the step

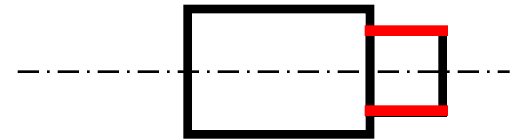
X17.5 final diameter

Z-20 length of step is 20 mm

D0=200 Finish allowance in X direction (0.2 mm) D2=200

Finish allowance in Z direction (0.2 mm)

D3=650 Depth of cut in each pass (0.65 mm)



Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X25.00 Z2.0 S500 M04

N0080 G01 X22.5 Z-70.0 F100

N0090 G00 X26.0 Z2.0 S500

N0100 G84 X17.5 Z-20.0 D₀=200 D₂=200 D₃=650

N0110 G00 Z2.0

G00 Move rapidly away from workpiece (no cutting)

Z2.0 the movement is 2 mm away from the face.

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X25.00 Z2.0 S500 M04

N0080 G01 X22.5 Z-70.0 F100

N0090 G00 X26.0 Z2.0 S500

N0100 G84 X17.5 Z-20.0 D₀=200 D₂=200 D₃=650

N0110 G00 Z2.0

N0120 X50.0 Z50.0

X50.0 Z50.0 Move to the tool changing location

Program Interpretation

O0013

N0005 G53

N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

N0040 G00 Z2.0

N0050 X50.0 Z50.0

N0060 T0404

N0070 G57 G00 X25.00 Z2.0 S500 M04

N0080 G01 X22.5 Z-70.0 F100

N0090 G00 X26.0 Z2.0 S500

N0100 G84 X17.5 Z-20.0 D₀=200 D₂=200 D₃=650

N0110 G00 Z2.0

N0120 X50.0 Z50.0 T00

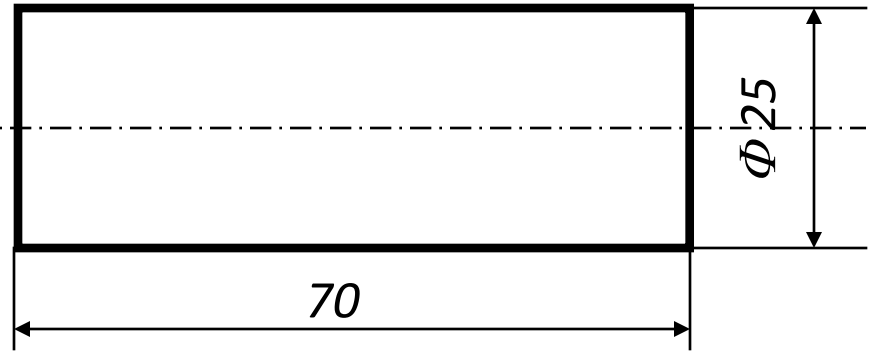
N0130 M05 M30

M30 Program End

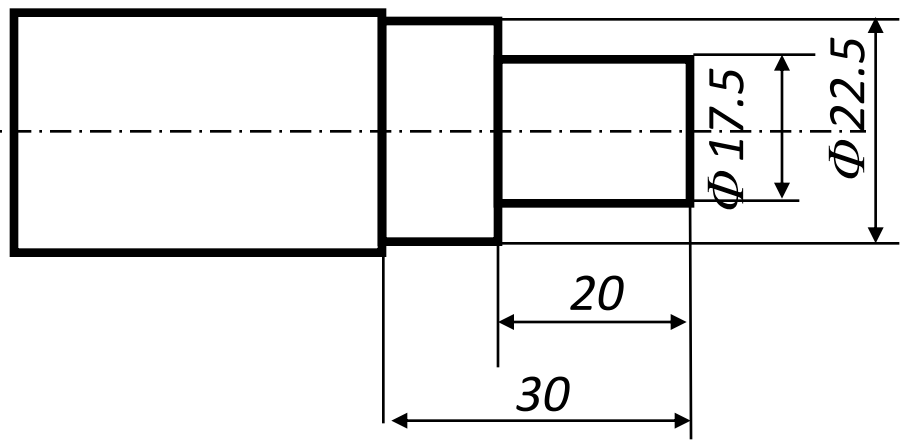
Programming Example

Cylindrical Part

Raw Material



Finished Part



Discussion



[Please refer http://www.helmancnc.com/](http://www.helmancnc.com/)

10 mins