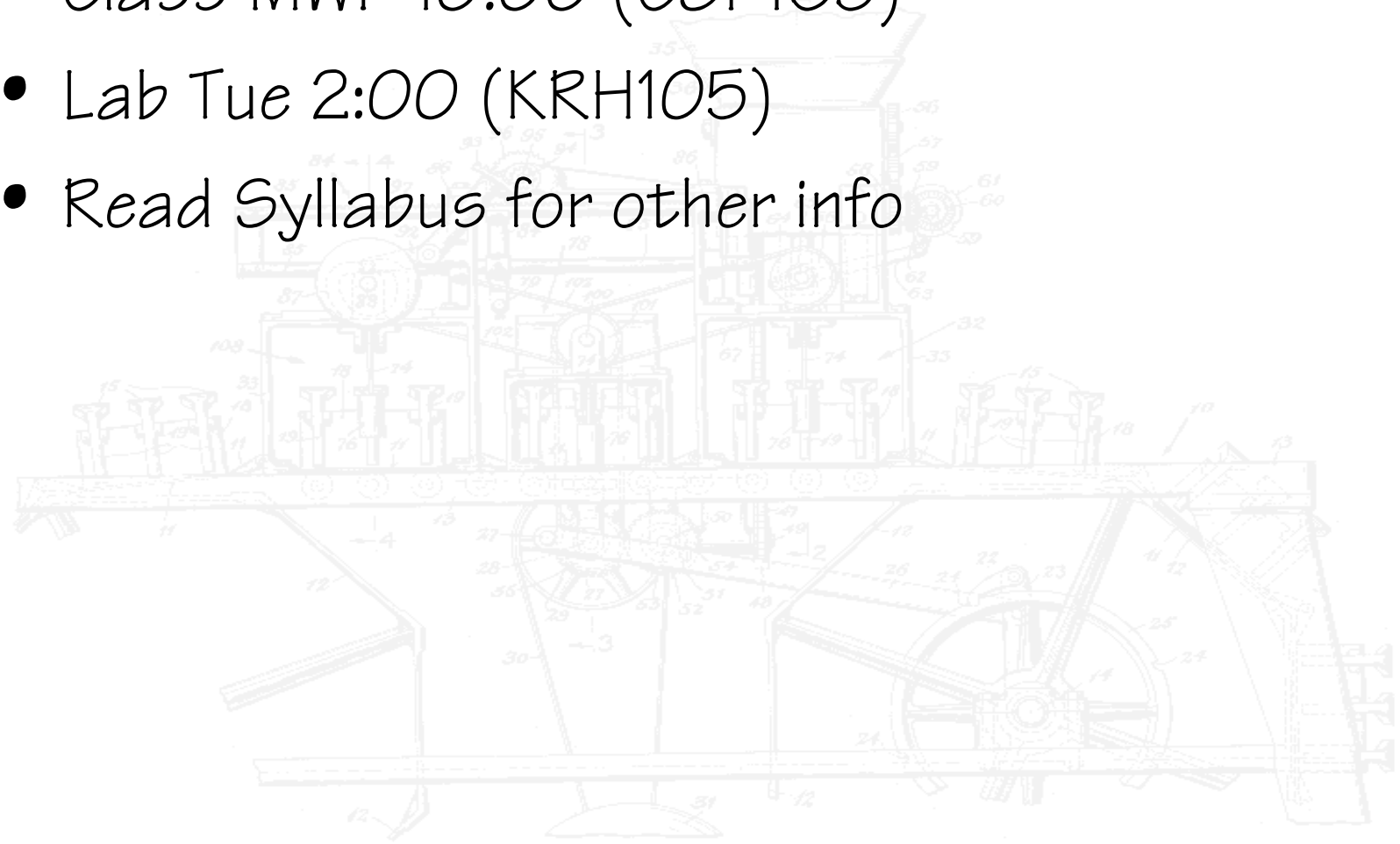
ENGR480 Manufacturing Systems

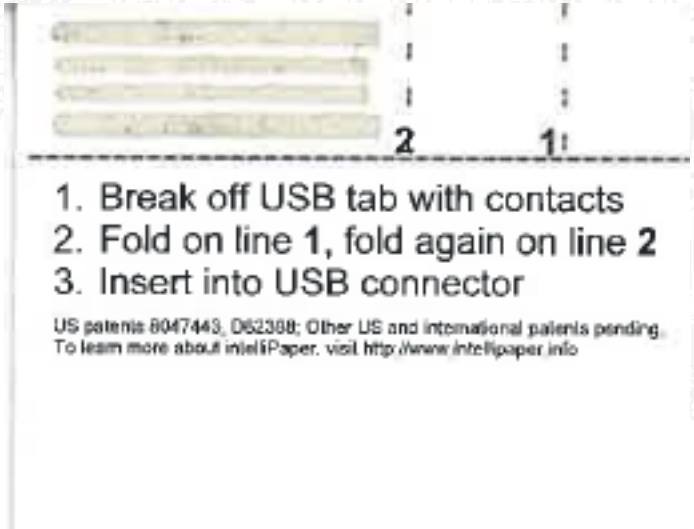
Spring 2013

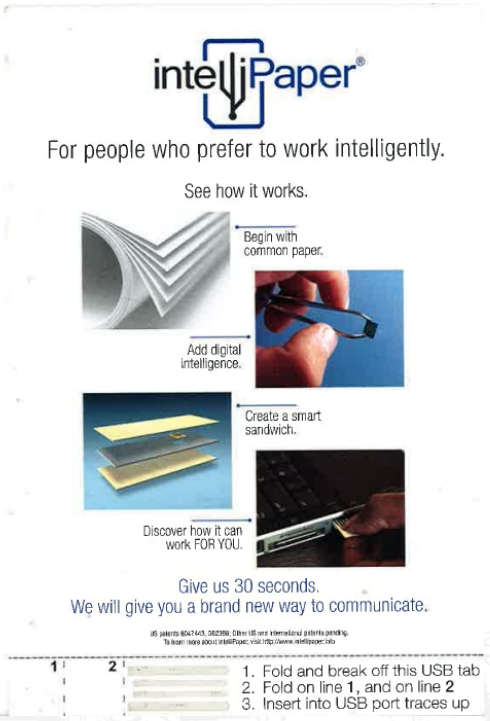
ENGR480 Manufacturing Systems

- Class MWF 10:00 (CSP165)
- Lab Tue 2:00 (KRH105)
- Read Syllabus for other info



Our Customer
intelliPaper LLC





Lathe Coordinate System

MORI SEIKI
THE MACHINE TOOL COMPANY

2. X-Axis and Z-Axis

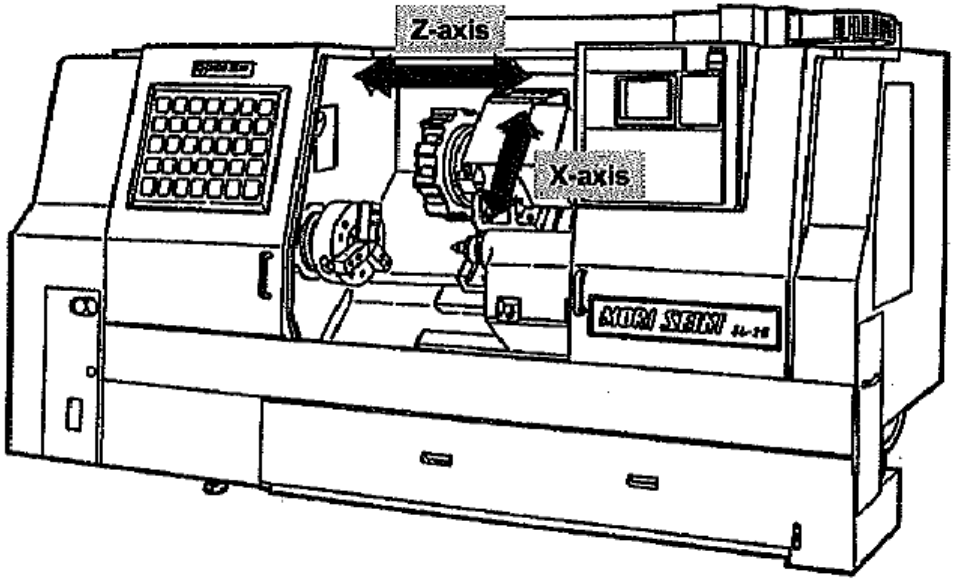
COORDINATES

Basic model of NC lathe has two numerically controlled axes, called X-axis and Z-axis.

X-axis ..... The axis along which the cross slide moves.

Z-axis ..... The axis along which the carriage moves.

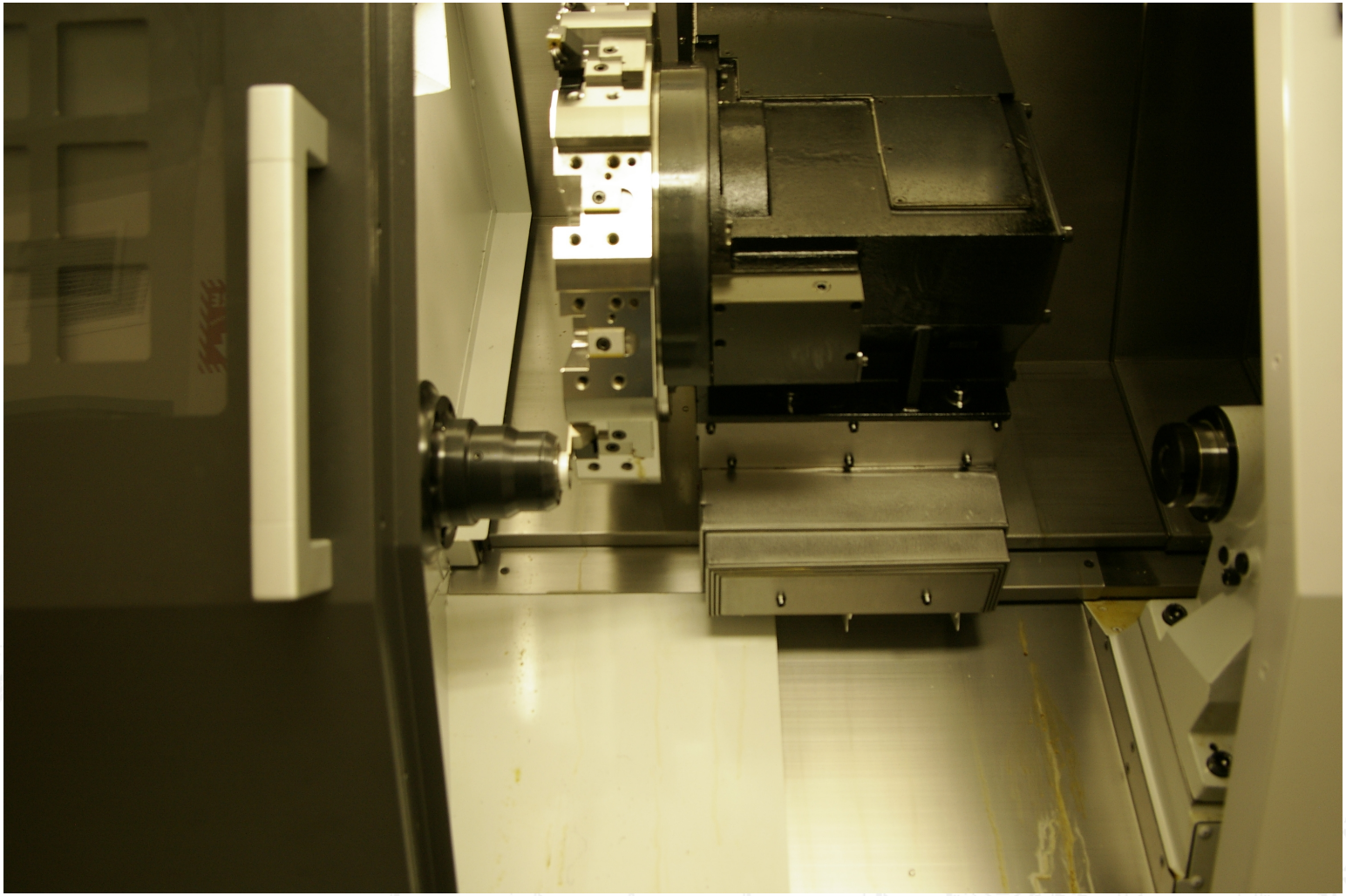
The direction of an axis is determined by the positive (plus) and negative (minus) signs.



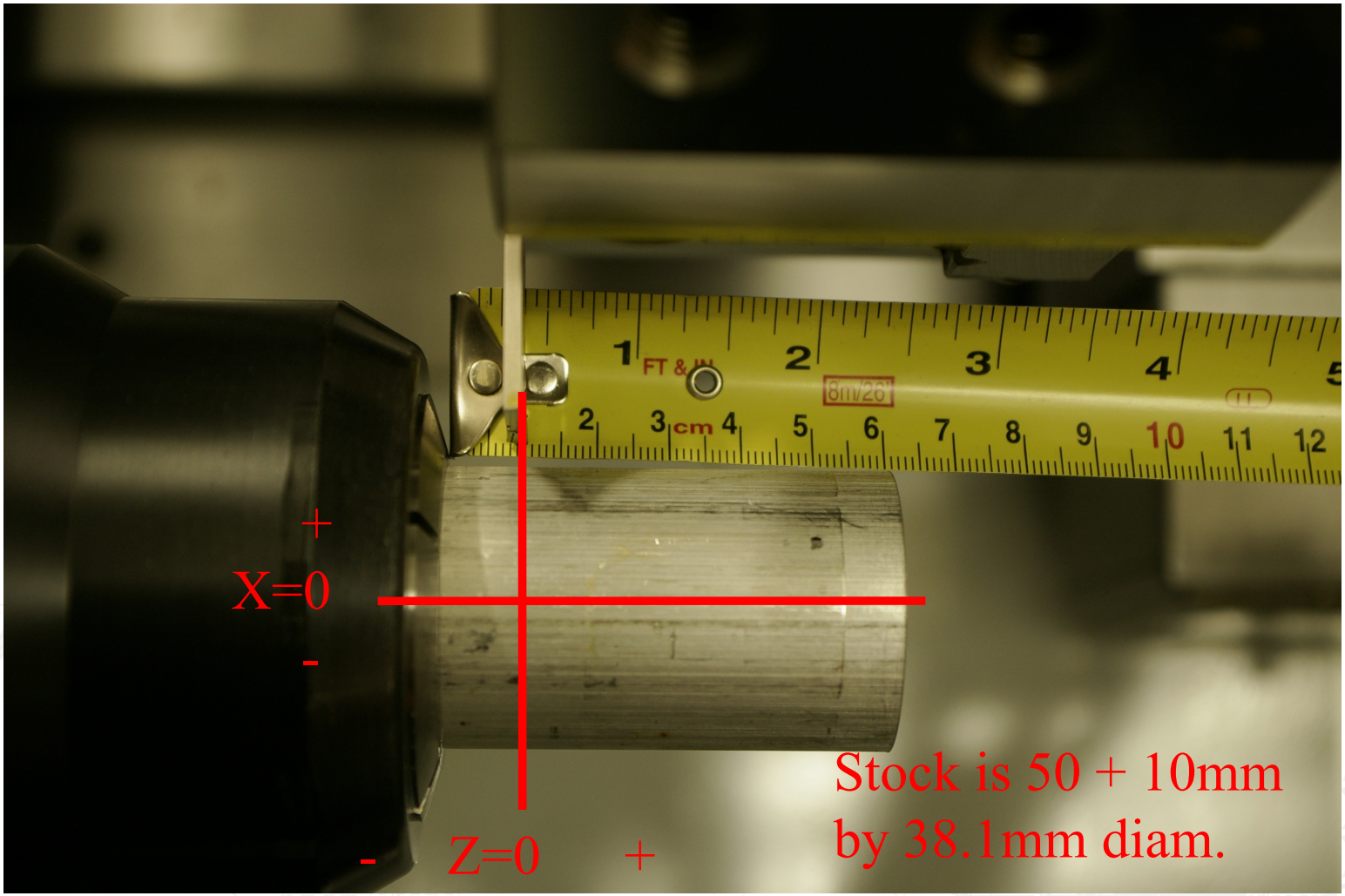
Workpiece Zero Point

- Coordinate system zero point is
  - centerline of spindle (X zero)
    - with normal spindle rotation, machining is in +X
  - back face of part (Z zero)
    - +Z is machining part
- X dimensions are *diameter*, not radius

Workpiece Zero Point

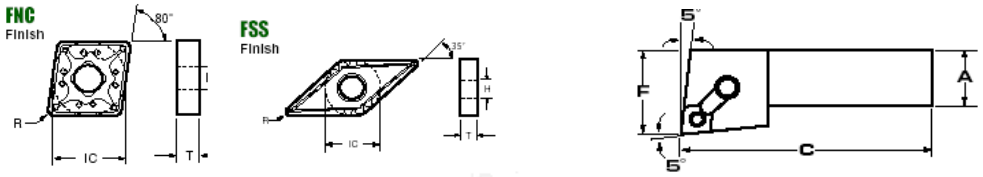


Workpiece Zero Point



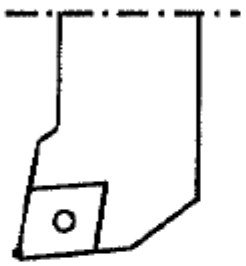
Lathe Tooling

- Turning
- Boring
- OD Grooving
- ID Grooving
- Face Grooving
- OD Threading
- ID Threading

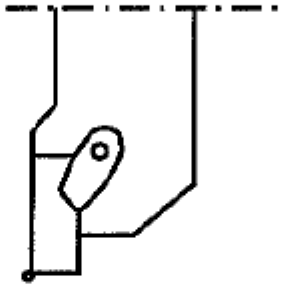


4. Tool Command Point

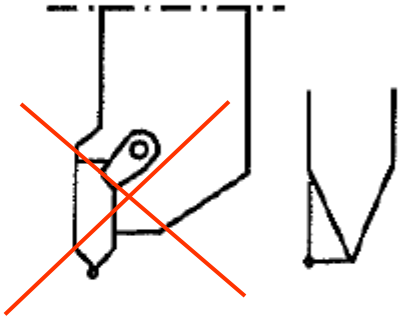
a) O.D./Face Turning



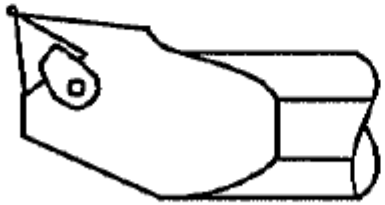
b) O.D. Grooving



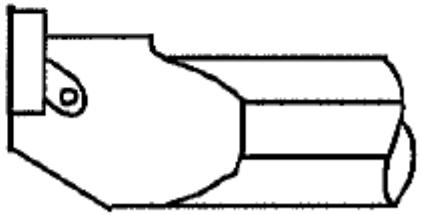
c) O.D. Threading



d) I.D. Turning

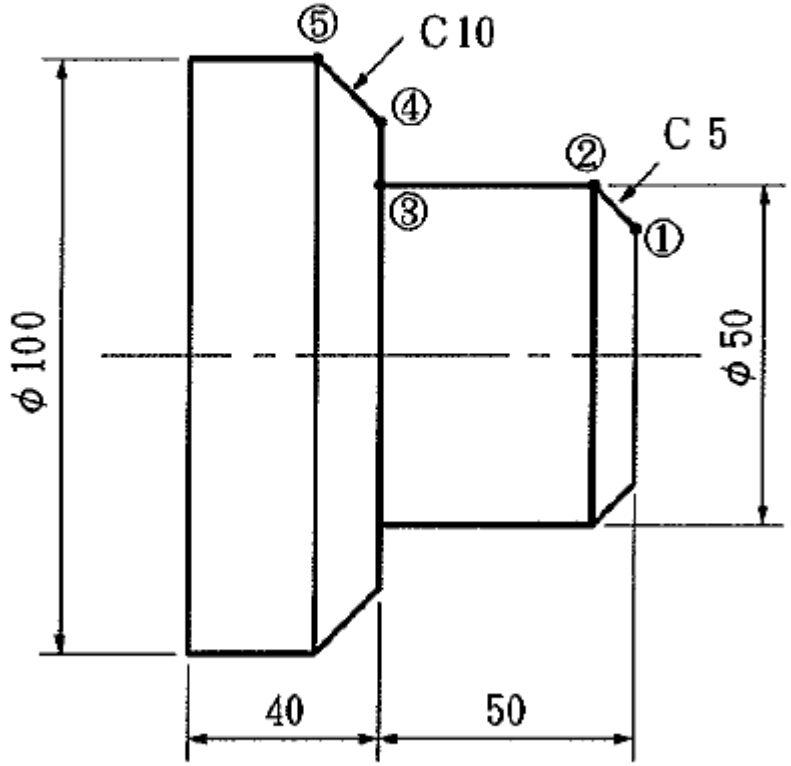


e) I.D. Grooving



Example

Let's find out the dimensional data (actually, coordinate values) of five points (① to ⑤) in the drawing below.

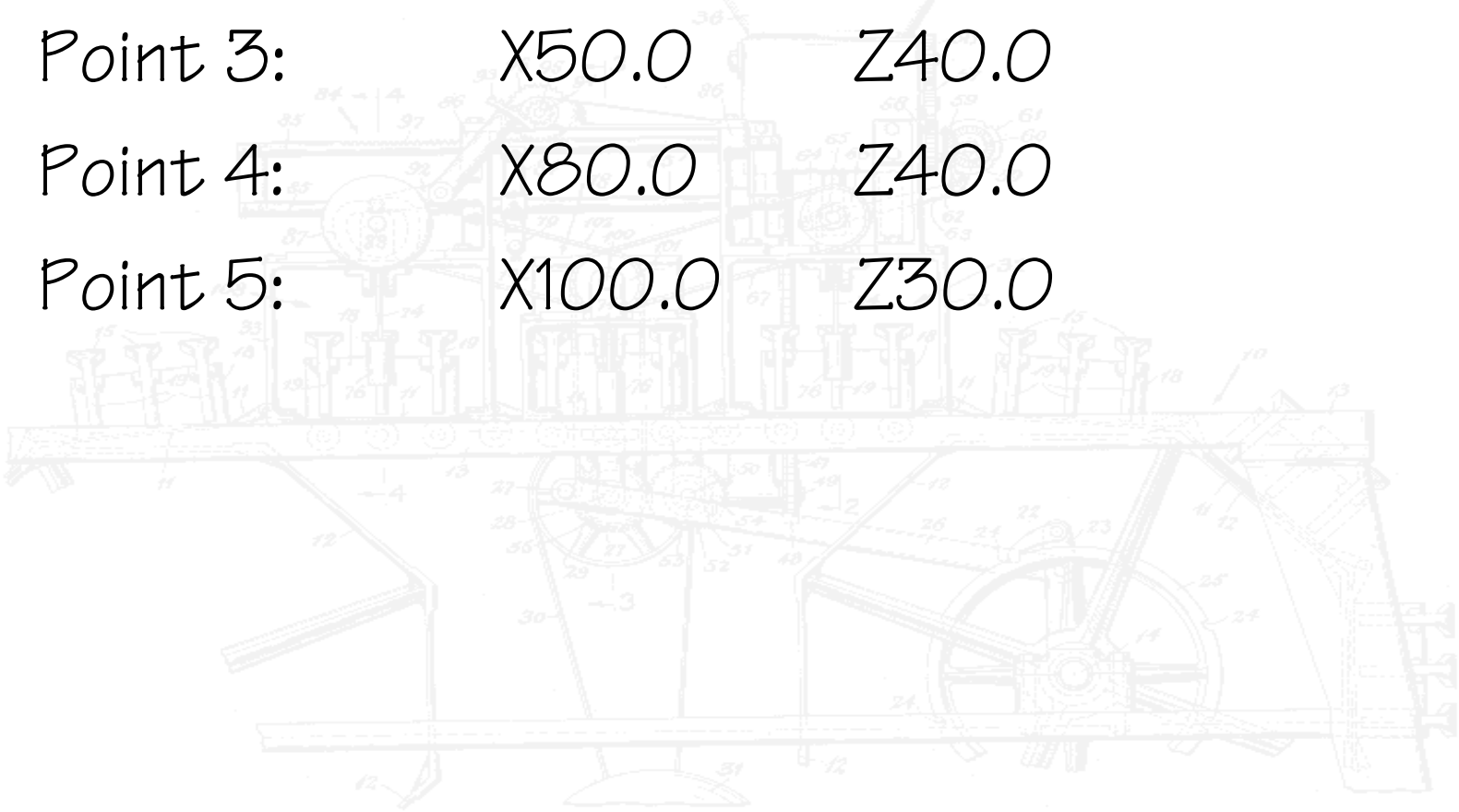


(Fig. 1-6)

```
O
_____

① X____ Z____
② X____ Z____
③ X____ Z____
④ X____ Z____
⑤ X____ Z____

_____

_____

M30
```

Absolute Programming

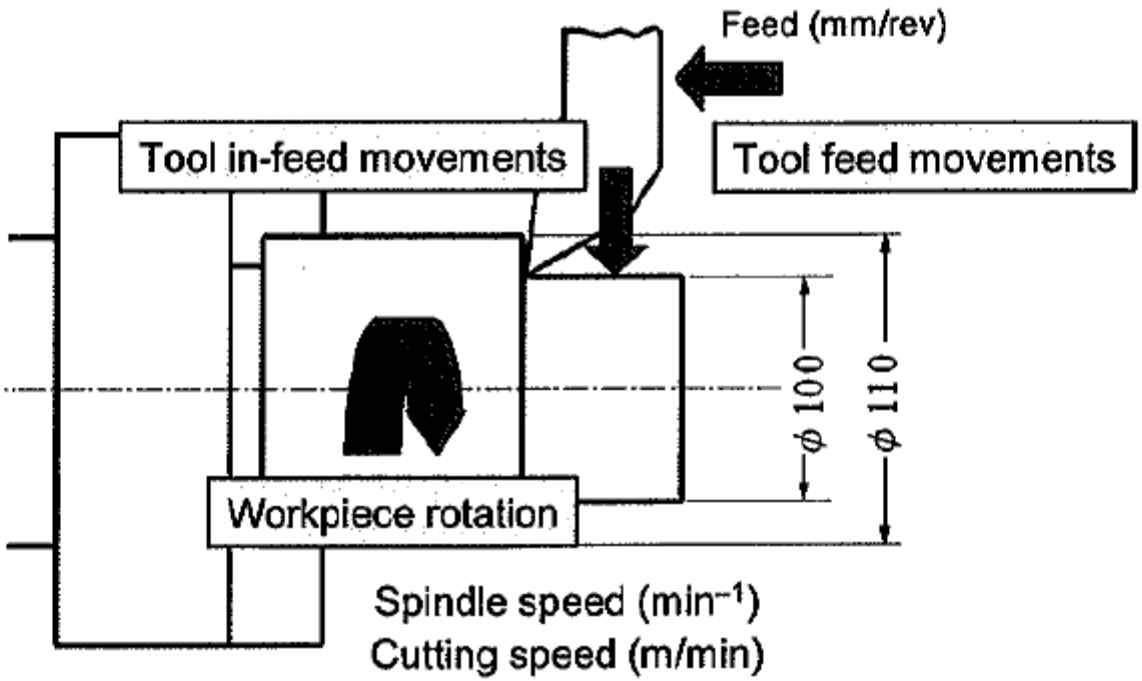
- Point 1 : X40.0 Z90.0
- Point 2: X50.0 Z85.0
- Point 3: X50.0 Z40.0
- Point 4: X80.0 Z40.0
- Point 5: X100.0 Z30.0



Incremental Programming

- In incremental programming, only the change in X and Z are given. Change in X is U, change in Z is W.
- ONLY USE FOR MOVING AWAY FROM PART, NOT CUTTING!
- Point 1: X40.0 Z90.0
- Point 2: U10.0 W-5.0
- Point 3: W-45.0
- Point 4: U30.0
- Point 5: U20.0 W-10.0

7. Cutting Conditions



7.1 How to Determine Cutting Conditions

(1) Cutting speed (spindle speed)	:	Material and shape of workpiece
		Insert material (hardness in high temperature and wear resistivity)
(2) Cutting feedrate	:	Rigidity of the machine and tools
		Nose radius of insert (Fmax. ≤ Nr/2)
		Chip breaker shape
		Surface finish specified in the drawing → Nose radius
(3) Depth of cut	:	Rigidity of the machine and tools
		Main motor output
		Insert thickness (Dmax ≤ Insert thickness)
		Cutting edge length of insert
		Chip breaker shape
(4) Workpiece holding conditions	:	Workpiece holding status and method
		Workpiece shape
		Size and balance of jaws
		Chucking pressure
		Tailstock spindle thrust force

Major Five Functions That Control NC Lathe

1. **G Function: Preparatory function**
   Commands related to X-/Z-axis feed control/G00, G01, G02

2. **M Function: Miscellaneous function**
   ON/OFF commands output from the CNC to the NC lathe/M00, M03, M08

3. **T Function: Tool selection function**
   Calling the tool to be used/T0101 (4-digit T code)

4. **S Function: Spindle speed (min^{-1}) and cutting speed (m/min)**
   Spindle speed command/S500 (= 500 min^{-1})
   Cutting speed command/S200 (= 200 m/min)

5. **F Function: Feedrate command (mm/rev)**
   Tool feedrate command/F0.3 (= 0.3 mm/rev)

BASIC PATTERNS OF PROGRAM (1)

```
O1
N1
G50S2000
G00T0101
G962S00M03
        :
G00U1.0Z10.0
X300.0Z150.0
M01
M30;
```

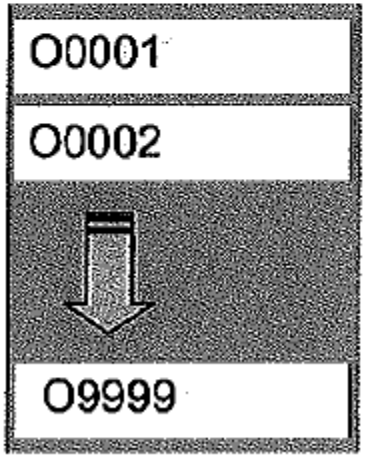
Program number (O1~O9999)

Program name

Must be specified at the beginning of all programs.

NC reset & rewind

In the storage area



BASIC PATTERNS OF PROGRAM (2)

```
O1
N1
G50S2000
G00T0101
G96S200M03

      ⋮

G00U1.0Z10.0
X300.0Z150.0
M01
;
```

Sequence number (to be specified at the beginning of a part program)

Usually, a sequence number is specified at the beginning of individual machining processes to identify the processes.

- Rough O.D. turning N1
- Rough I.D. turning N2
- Finish O.D. turning N3
- N4
- N5
- Thread cutting N6

BASIC PATTERNS OF PROGRAM (3)

```
O1
N1
G50 S2000
G00 T0101
G96S200M03
        :
G00U1.0Z10.0
X300.0Z150.0
M01
;
```

The command specifying the allowable maximum spindle speed

BASIC PATTERNS OF PROGRAM (4)

```
O1
N1
G50S2000
G00 T0101
G96S200M03
      :
      :
G00U1.0Z10.0
X300.0Z150.0
M01
```

The command specifying the tool number and the tool position offset number

BASIC PATTERNS OF PROGRAM (5)

```
O1
N1
G50S2000
G00T0101
G96      S200M03
```

The command specifying the cutting speed, spindle rotation in the normal direction and spindle revolving speed

```
G00U1.0Z10.0
X300.0Z150.0
M01
```

BASIC PATTERNS OF PROGRAM (6)

```
O1
N1
G50S2000
G00T0101
G96S200M03
      :
G00U1.0Z10.0
X300.0Z150.0
M01
;
```

Moving the tool away from the workpiece after finishing the machining

BASIC PATTERNS OF PROGRAM (7)

```
O1
N1
G50S2000
G00T0101
G96S200M03
        :
G00U1.0Z10.0
X300.0Z150.0
M01
;
```

Moving the tool to the position where turret rotation is possible
Optional stop

THE LAST BLOCK

```
O1
N1
G50S2000
G00T0101
G96S200M03
      ⋮
G00U1.0Z10.0
X300.0Z150.0
M01
      ⋮
M30
```

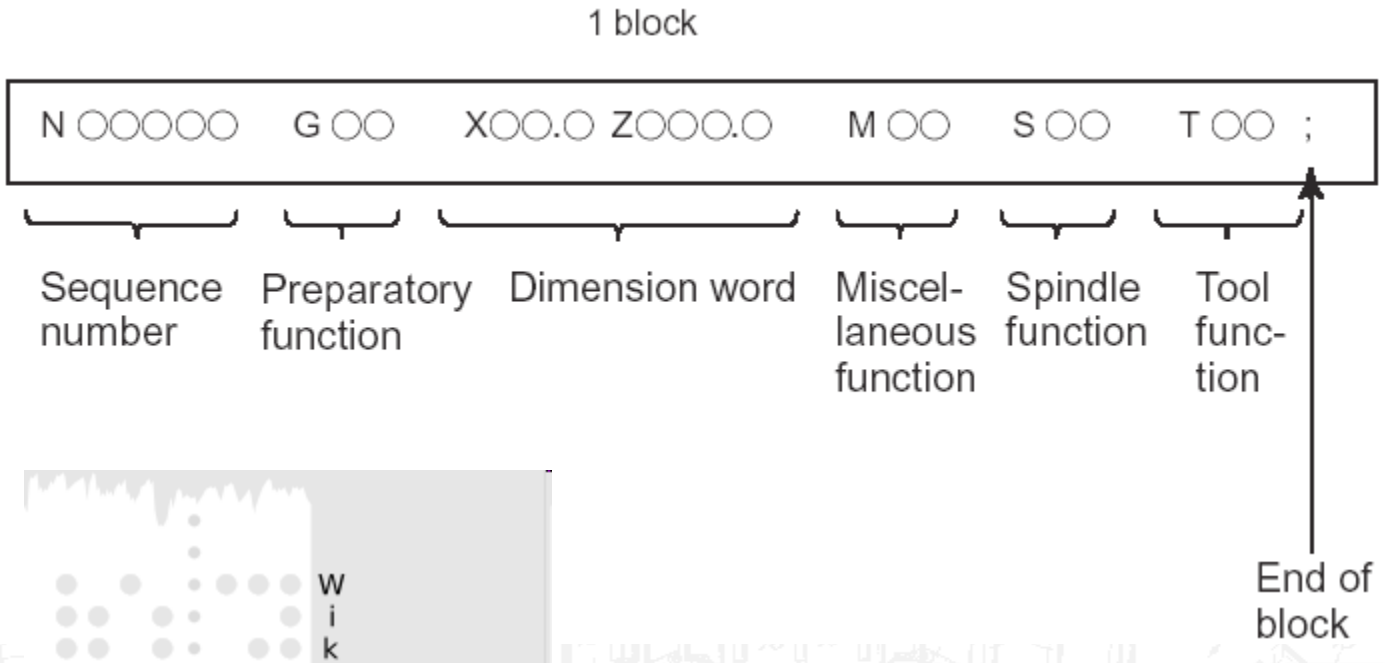
⇦ NC reset and rewind

THANK YOU for your careful attention to my lecture.

This ends the explanation of programming.

I hope all of you enjoyed to understand programming basics.

CNC Lathe Programming



Wikipedia
Carriage Return
Line Feed

G-Codes for Turning

G00	Rapid positioning
G01	Linear interpolation (feeding)
G02	CW Circular interpolation
G03	CCW Circular interpolation
G04	Dwell
G20	Inch system
G21	Metric system
G28	Return to reference point
G50	Limit spindle speed

G-Codes for Turning

G54	Select work coord system #1
G70	Finishing cycle
G71	Roughing cycle
G72	Facing cycle
G76	Threading cycle
G96	Constant *surface* speed mode
G97	Constant *spindle* speed mode
G98	Feed per minute mode
G99	Feed per revolution mode

M-Codes for Turning

M00	Program Stop
M01	Opt. Program Stop (panel controlled)
M03	Start spindle (normal rotation)
M04	Start spindle (reverse rotation)
M05	Stop spindle
M08	Start coolant
M09	Stop coolant
M10	Close chuck
M11	Open chuck
M30	Program end

Example Program

```
%
O1 (THAT'S AN OH NOT A ZERO)
; (EXAMPLE PROG - TURN PLUG 20.0MM OD X 17MM LONG)
; (STOCK: ALUM 1.5" X 17MM + 10MM)
;
; (T1- CNMG 55DEG DIAMOND)
; (T2- VNMG 15DEG DIAMOND)
; (T10- 3.175MM CUTOFF)
;
; (VARIABLES)
#500=38.1      (STOCK DIAMETER)
#501=17.0      (STOCK LENGTH)
#502=150.0     (SURFACE M/MIN FOR CUTTING ALUM)
#503=0.2       (ROUGHING FEED: MM/REV)
;
N1
G54            (WORK OFFSET)
G21            (METRIC)
G28 U0 W0      (GO HOME)
G50 S2000      (MAX SPINDLE SPEED)
```

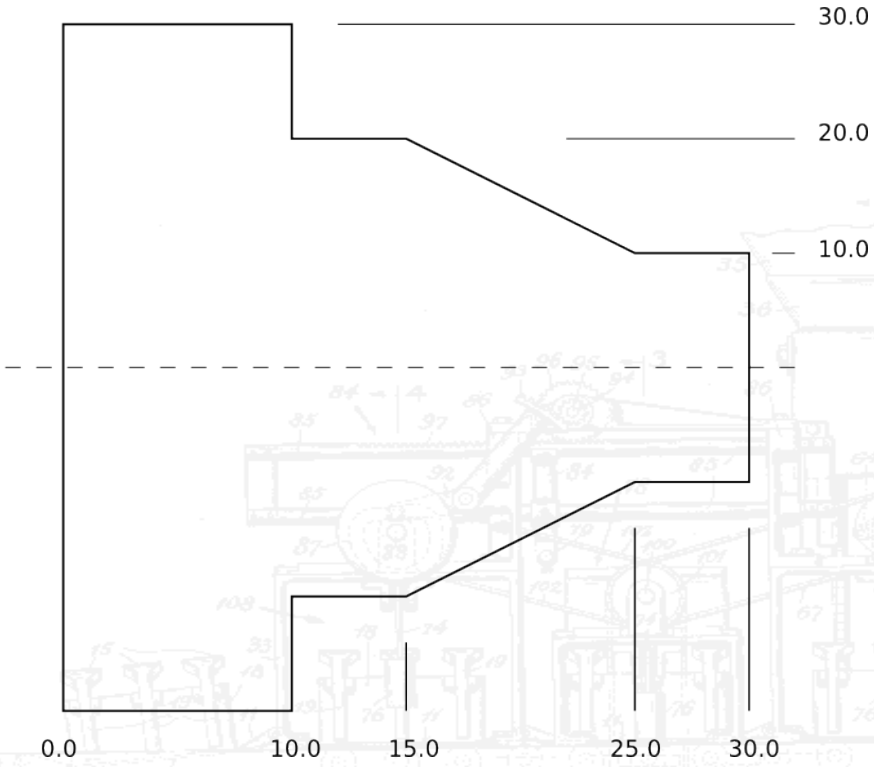
```
;
; (FACE WITH 55DEG DIAMOND TOOL AND SPEED LIMITING)
;
G00 T0101              (CUTOFF TOOL)
G50 S1000              (CLAMP SPEED AT MAX 1000RPM)
G96 S#502              (CONST SURF SPEED)
G99                    (FEED PER REV)
;
M03                    (SPINDLE ON, NORMAL DIR)
G00 Z#501              (INITIAL Z POSITION)
G00 X[#500+1.0]        (INITIAL X POSITION)
M08                    (TURN ON COOLANT)
G01 X-1.0 F#503        (FACE FROM OUTSIDE DOWN TO -1.0MM)
G01 Z[#501+1.0] F1.0 (MOVE OFF FROM FACE)
M09                    (TURN OFF COOLANT)
M05                    (TURN OFF SPINDLE)
G28 U0 W0              (RETURN TO HOME)
M01                    (OPTIONAL STOP)
```

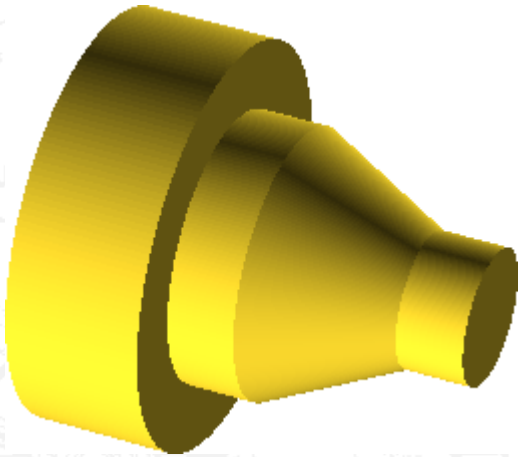
```
;
; (ROUGH OD 38.0 TO 20.5MM)
; (USE 2.0MM DEPTH OF CUT -> 8 PASSES)
;
N2
G00 T0101           (55DEG DIAMOND TOOL, TOOL 1)
G50 S2000           (CLAMP SPEED AT MAX 2000)
G96 S#502           (CONST SURF SPEED)
G99                 (FEED PER REV)
G00 X[#500 + 0.5] Z#501   (INITIAL POINT FOR ROUGHING)
M03                 (SPINDLE ON)
M08                 (TURN ON COOLANT)
G00 X36.0 Z#501
G01 X36.0 Z5.5 F#503  (FIRST PASS)
G01 X[#500+0.5] F#503 (RETRACT X)
G00 Z#501           (RETRACT Z)
G00 X34.0
G01 X34.0 Z5.5 F#503  (SECOND PASS)
G01 X[#500+0.5] F#503
G00 Z#501
G00 X32.0
G01 X32.0 Z5.5 F#503  (THIRD PASS)
G01 X[#500+0.5] F#503
...
```

```
;
; (FINISH OD 20.0MM)
;
N3
G00 T0202              (VNMG 15DEG DIAMOND)
G00 X19.0                    (POSITION FOR START OF
CHAMFER)
G00 Z[#501+0.5]
G01 X20.0 Z[#501-0.5] F0.1 (CHAMFER 0.5MM)
G01 Z0.0 F0.1                (FINISH TURN)
G01 X[#500+0.5] F0.1         (BACK OFF)
M09
M05
G28 U0 W0                    (GO HOME)
M01
```

```
;
; (CUTOFF)
G00 T1010
G00 Z-3.175           (CUTOFF BLADE IS 3.175MM WIDE)
G00 X[#500+2.0]
M03
M08
G50 S1000             (CLAMP SPEED AT MAX 1000RPM)
G96 S#502             (CSS)
G01 X-0.1 F0.05       (CUTOFF)
G01 X[#500+2.0] F4.0 (RETRACT)
M09
M05
G28 U0 W0
M30                   (END PROGRAM)
%
```

Assignment for Lab





Special Tips

- NEVER DO A TOOL CHANGE AWAY FROM HOME! (G28 U0 W0)
- ALWAYS PUT A DECIMAL POINT AFTER DIMENSIONAL NUMBERS (no decimal --> microns)
- USE ALL CAPS IN YOUR PROGRAM (lowercase gets dropped)
- DON'T GET "OHS" AND "ZEROS" MIXED UP. PROGRAM NAME STARTS WITH "OH", NOT ZERO