

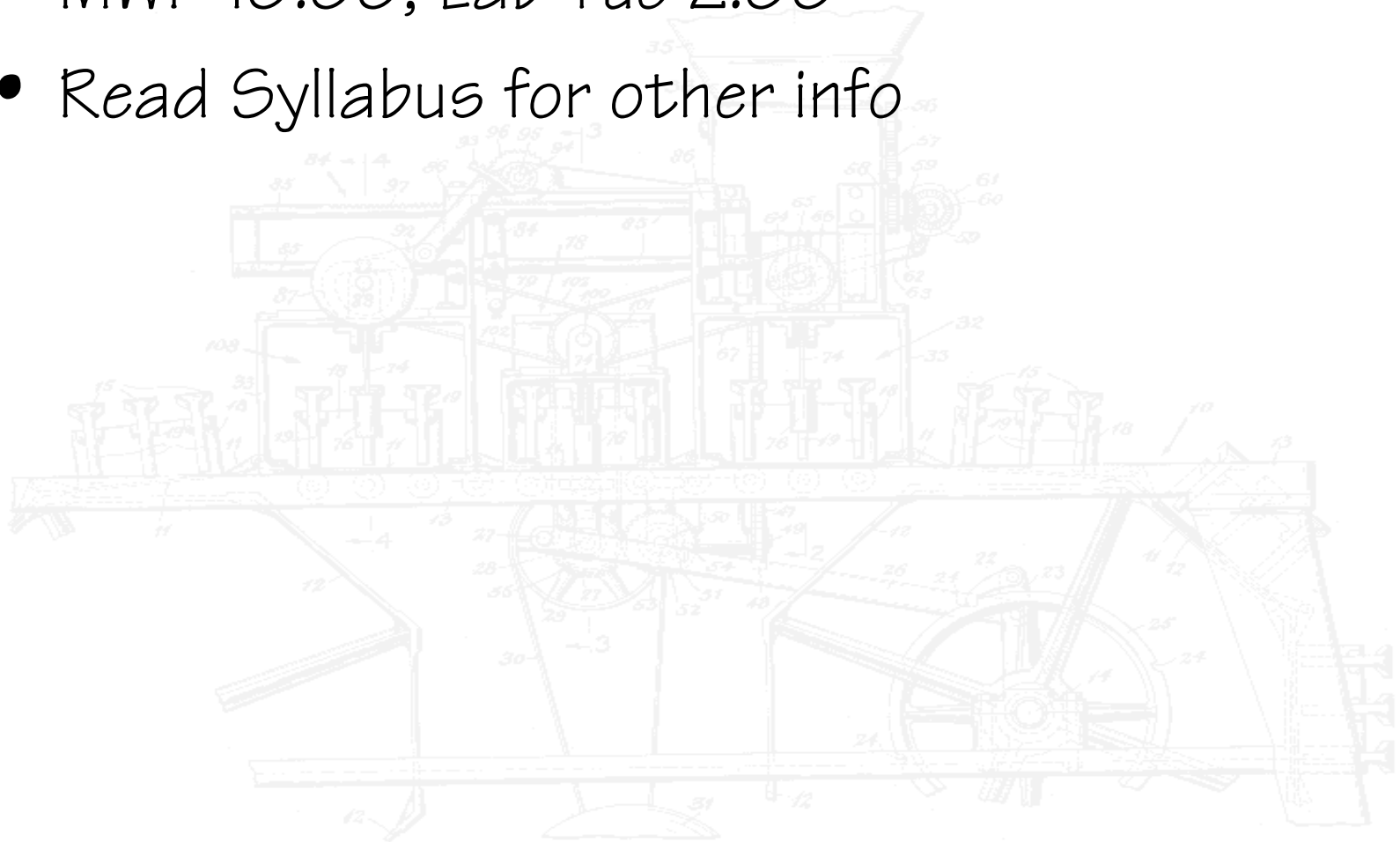
# ENGR480 MANUFACTURING SYSTEMS

Spring 2011



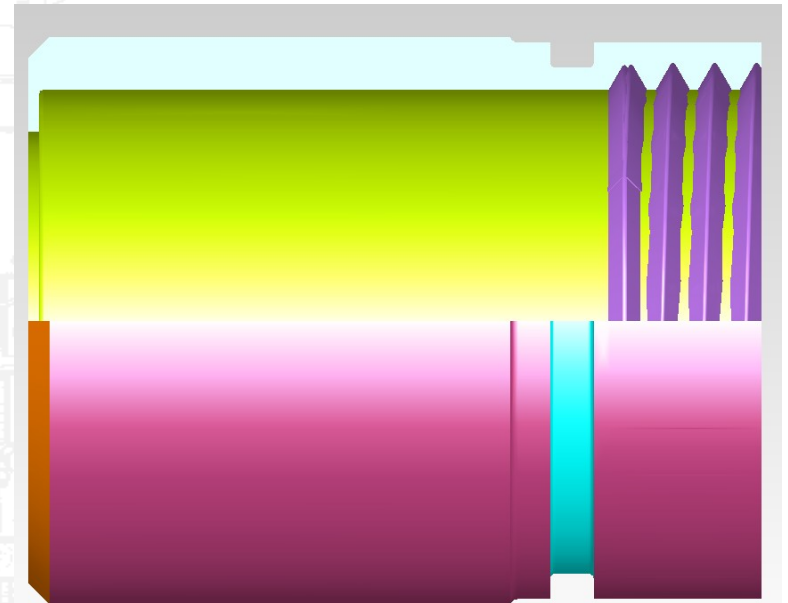
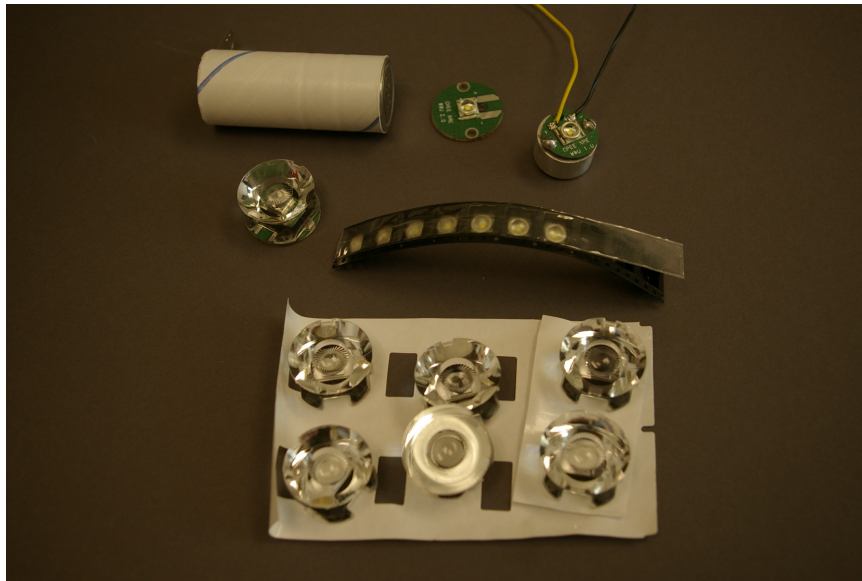
# ENGR480 MANUFACTURING SYSTEMS

- MWF 10:00, Lab Tue 2:00
- Read Syllabus for other info



# OUR MANUFACTURING COMPANY

## *FLASHOFGENIUS INC*



# LATHE COORDINATE SYSTEM

**MORI SEIKI**  
THE MACHINE TOOL COMPANY

## COORDINATES

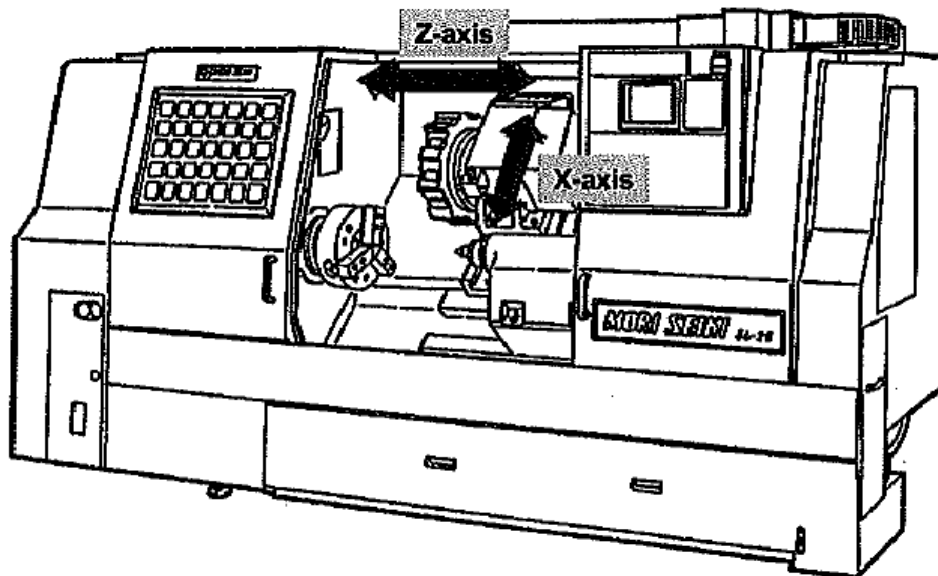
### 2. X-Axis and Z-Axis

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.

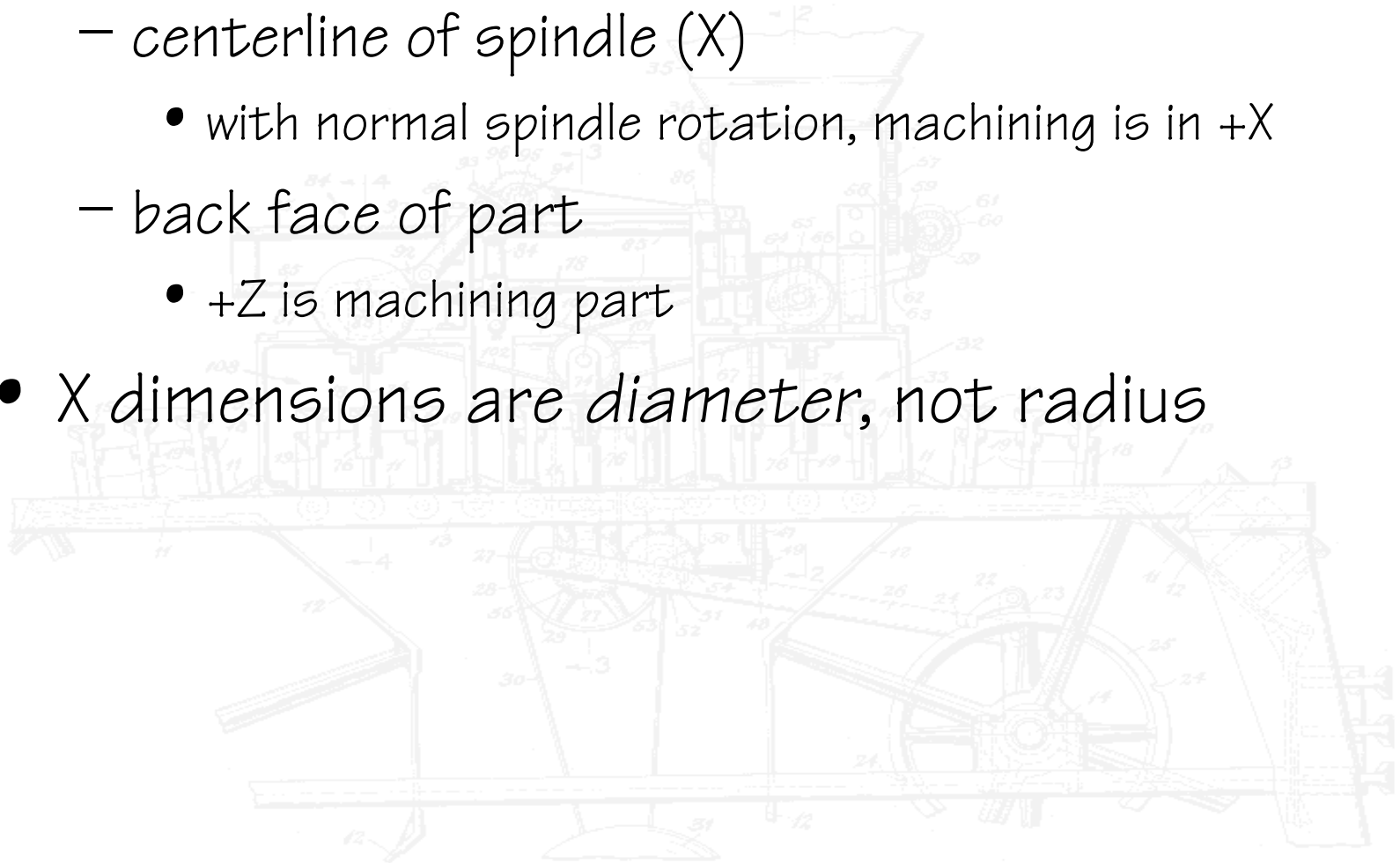
Z-axis ..... The axis along which the carriage moves.

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

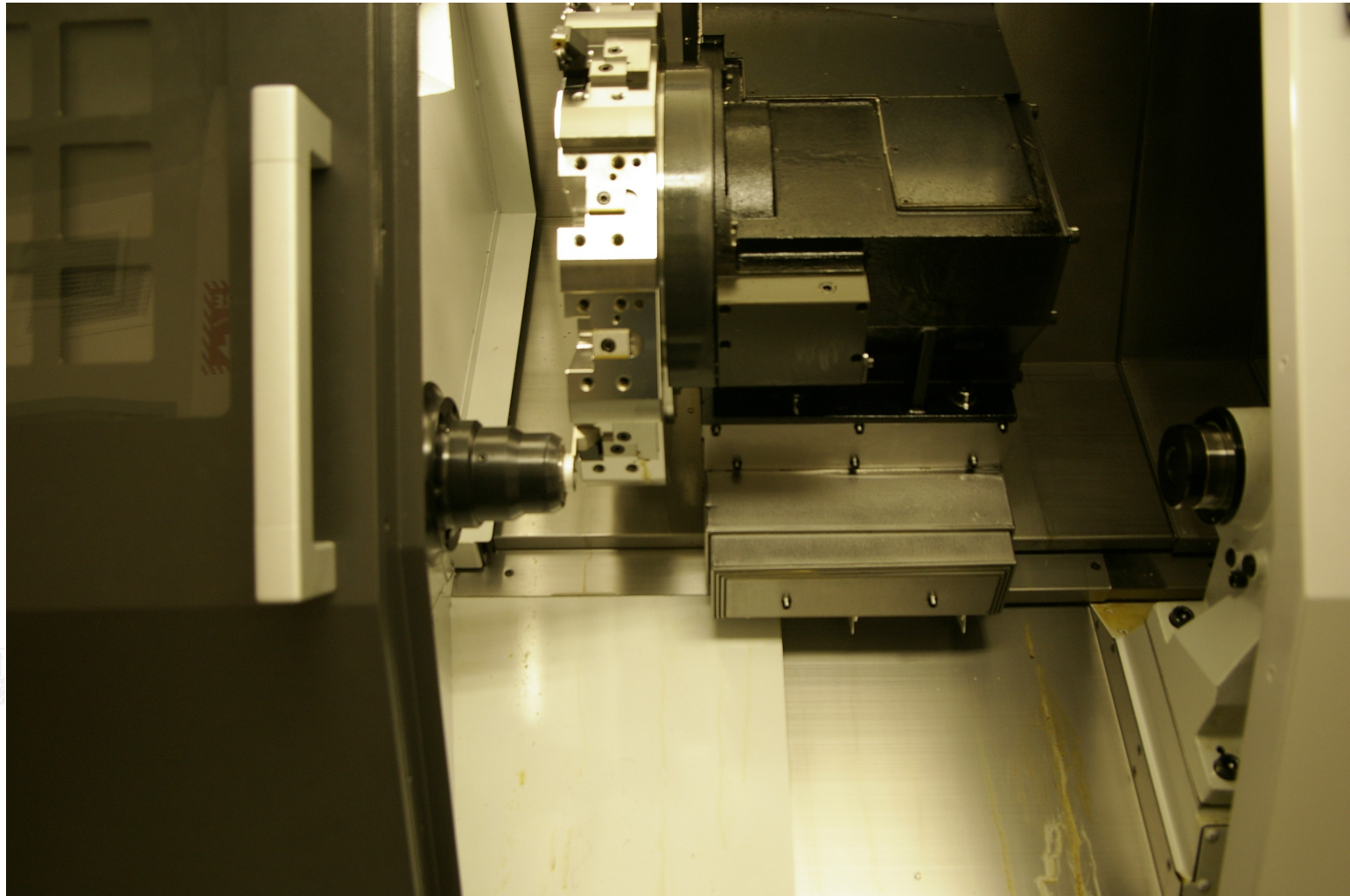


# WORKPIECE ZERO POINT

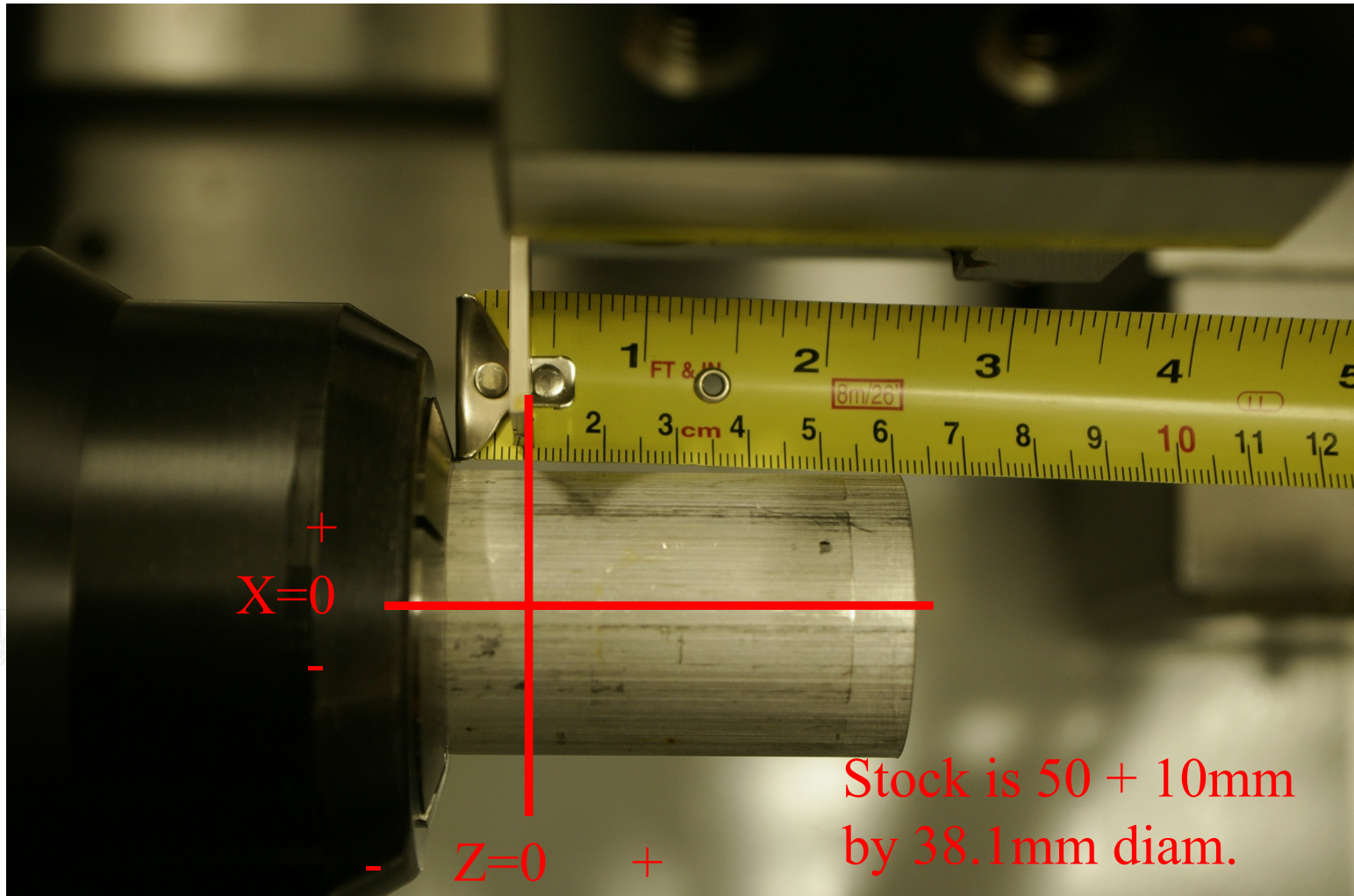
- Coordinate system zero point is
  - centerline of spindle (X)
    - with normal spindle rotation, machining is in +X
  - back face of part
    - +Z is machining part
- X dimensions are *diameter*, not radius



# WORKPIECE ZERO POINT

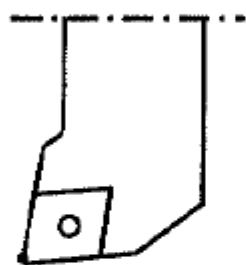


# WORKPIECE ZERO POINT

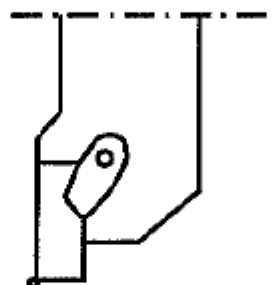


## 4. Tool Command Point

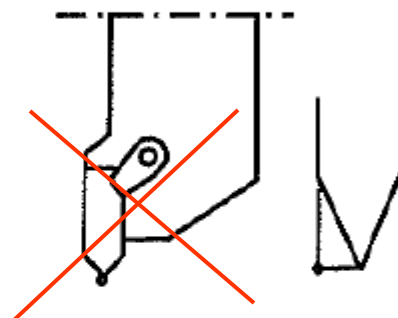
a) O.D./Face Turning



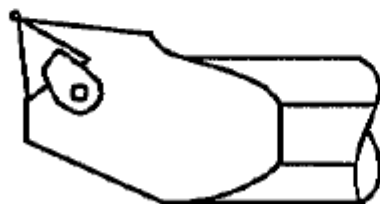
b) O.D. Grooving



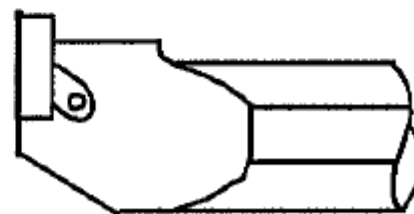
c) O.D. Threading



d) I.D. Turning



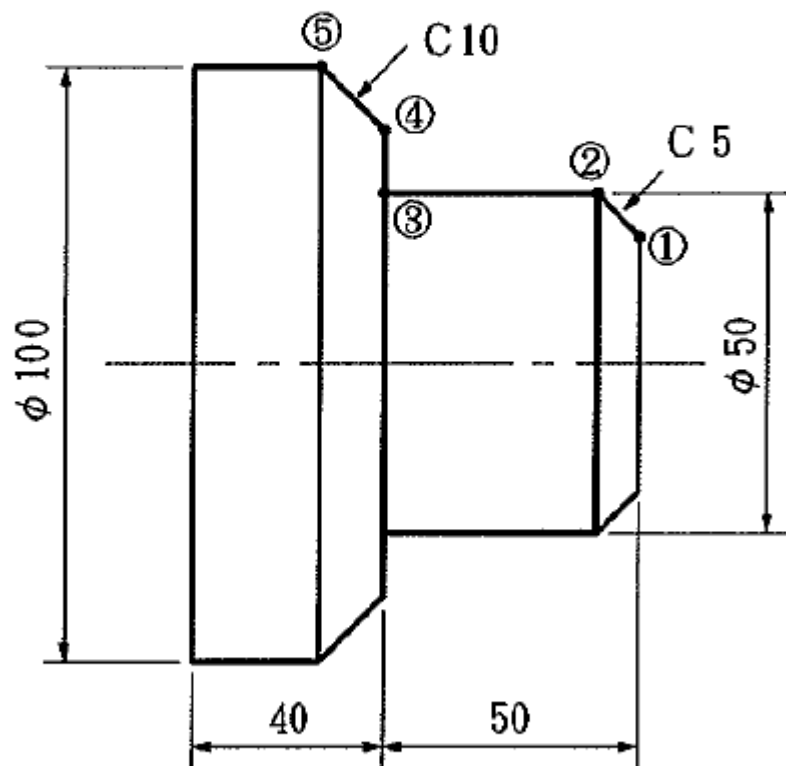
e) I.D. Grooving





**Example**

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



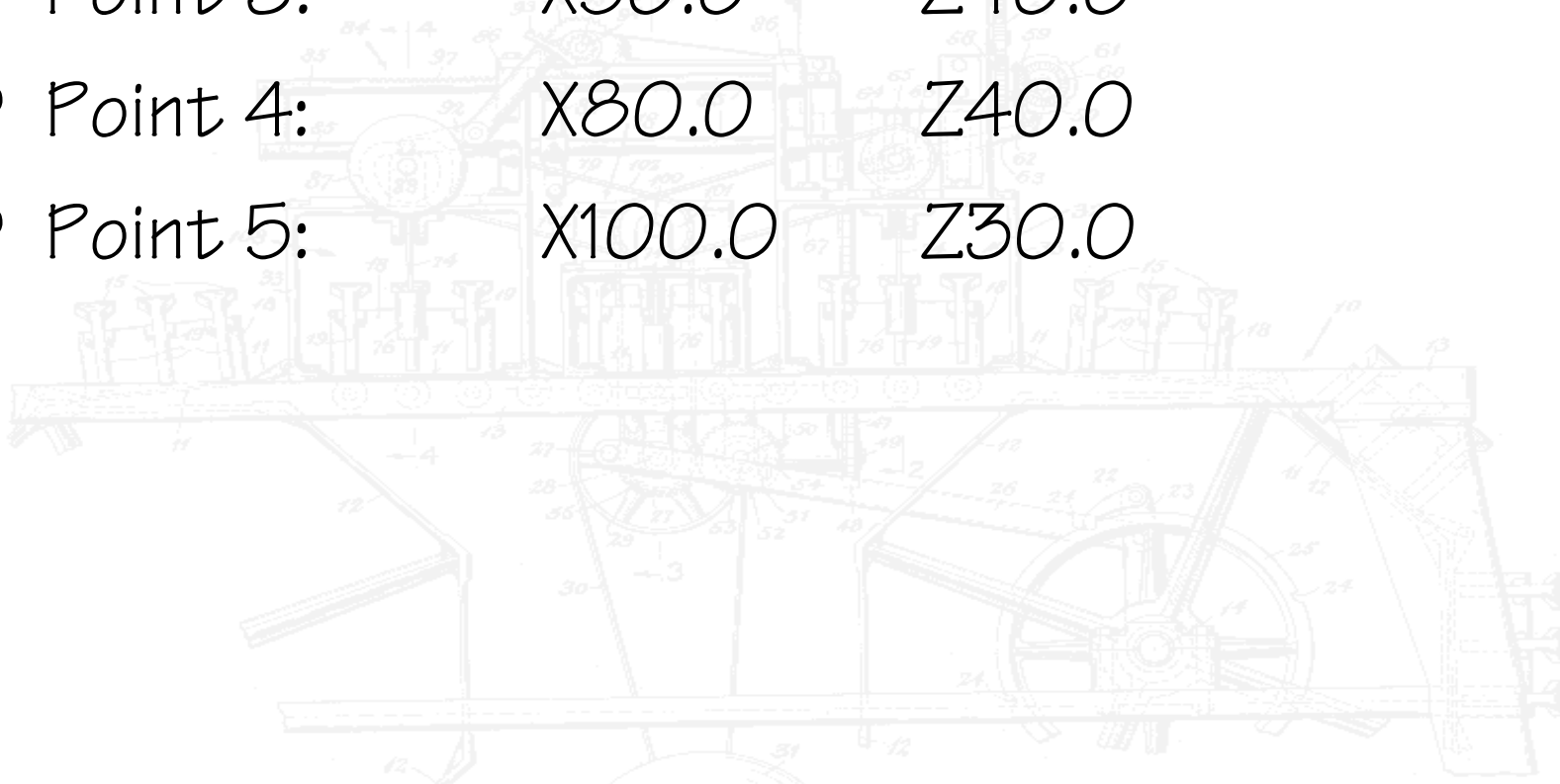
Z=0 (Fig. 1-6)

- 0 \_\_\_\_\_
- ① X\_\_\_\_\_ Z\_\_\_\_\_
- ② X\_\_\_\_\_ Z\_\_\_\_\_
- ③ X\_\_\_\_\_ Z\_\_\_\_\_
- ④ X\_\_\_\_\_ Z\_\_\_\_\_
- ⑤ X\_\_\_\_\_ Z\_\_\_\_\_

M30

# ABSOLUTE PROGRAMMING

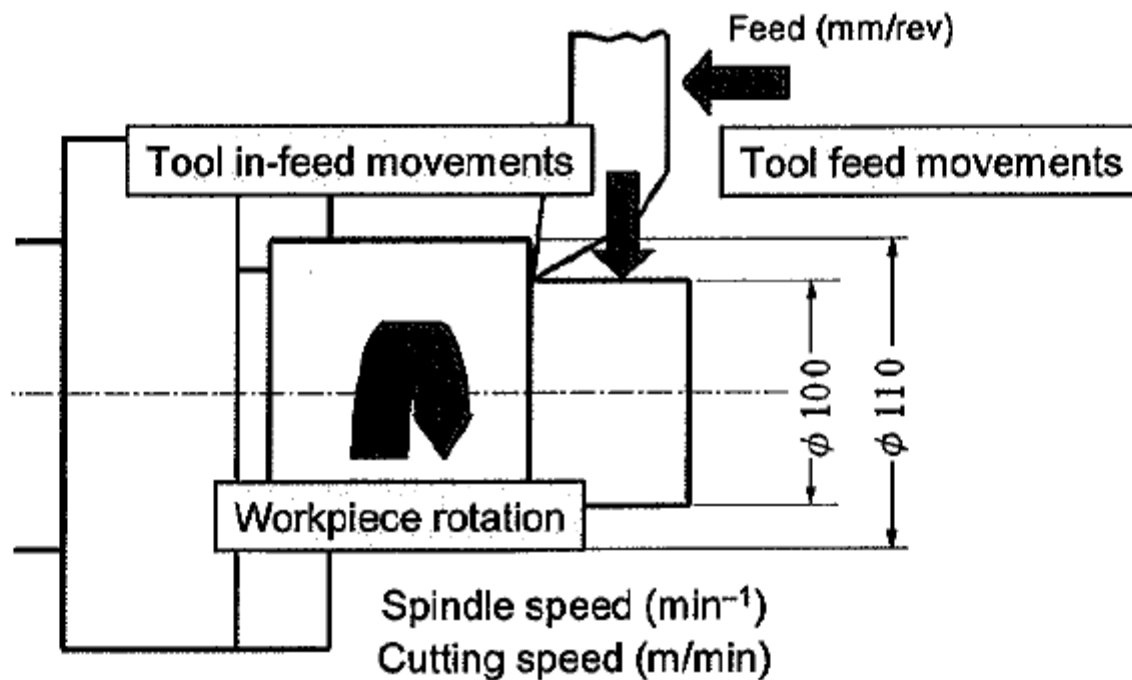
- Point 1: X40.0 Z90.0
- Point 2: X50.0 Z85.0
- Point 3: X50.0 Z40.0
- Point 4: X80.0 Z40.0
- Point 5: X100.0 Z30.0



# 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 ( $F_{max.} \leq Nr/2$ )  
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 ( $D_{max} \leq$  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 ( $\text{min}^{-1}$ ) and cutting speed (m/min)**

Spindle speed command/S500 (= 500  $\text{min}^{-1}$ )

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



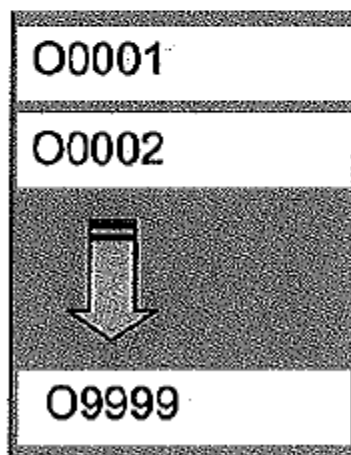
G00U1.0Z10.0  
X300.0Z150.0  
M01  
M30;

Program number (O1~O9999)

Program name

Must be specified at the beginning of all programs.

In the storage area



NC reset & rewind

# 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

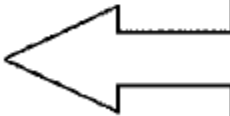


G00U1.0Z10.0

X300.0Z150.0

M01

;



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



# THE LAST BLOCK

$\bar{O}1$

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.

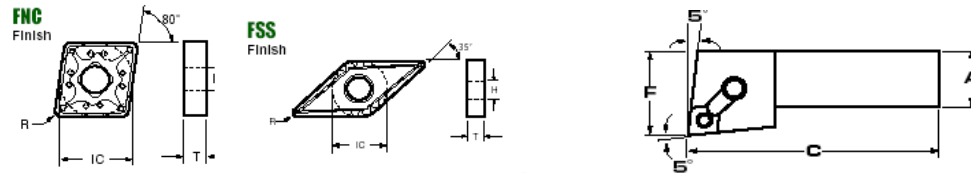
This ends the explanation of  
programming.

*I hope all of  
you enjoyed  
to understand  
programming  
basics.*



# LATHE TOOLING

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



| Cutting process<br>Cutting procedure                 | 1                | 2                      | 3        |
|--|------------------|------------------------|----------|
|  | End face cutting | Outer diameter cutting | Grooving |
| 1. Cutting method<br>: Rough<br>Semi<br>Finish       |                  |                        |          |
| 2. Cutting tools                                     |                  |                        |          |
| 3. Cutting conditions<br>: Feedrate<br>Cutting depth |                  |                        |          |
| 4. Tool path   |                  |                        |          |



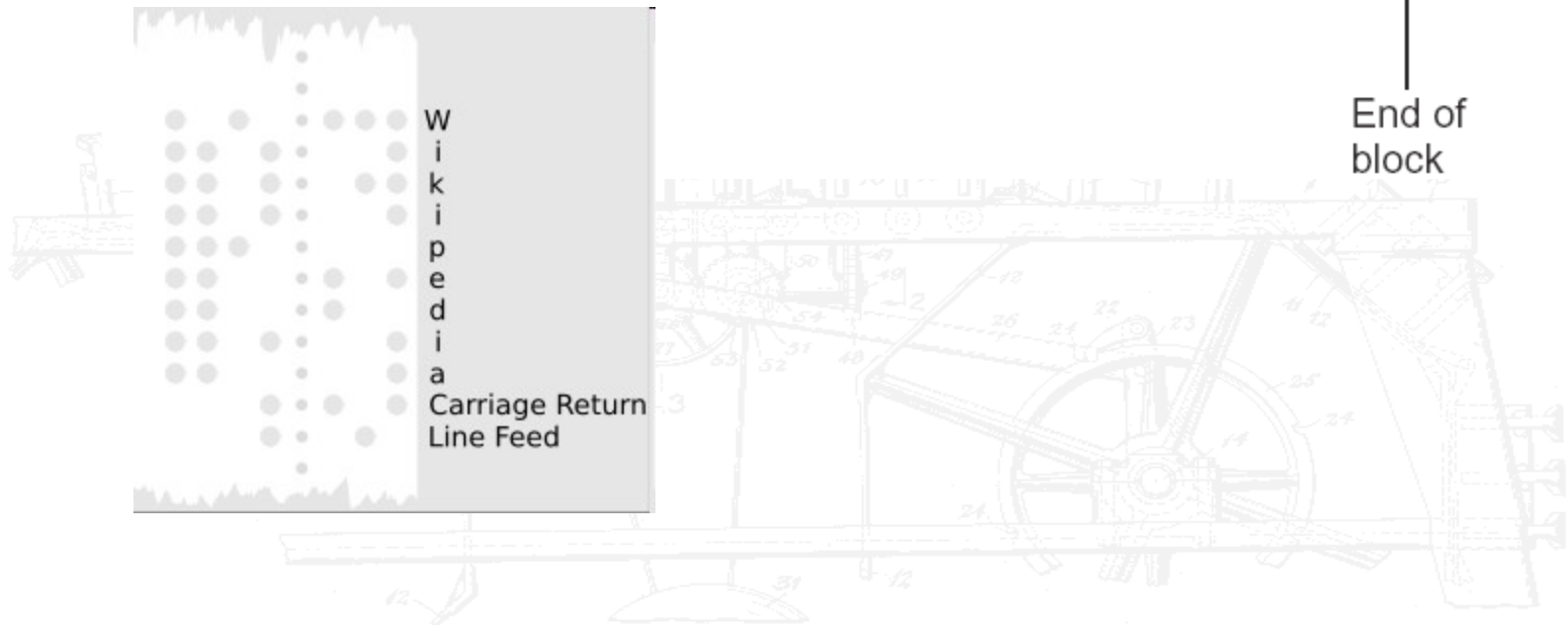
# CNC LATHE PROGRAMMING

1 block

```
N 000000 G 00 X00.0 Z000.0 M 00 S 00 T 00 ;
```

Sequence number    Preparatory function    Dimension word    Miscellaneous function    Spindle function    Tool function

End of block



# 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

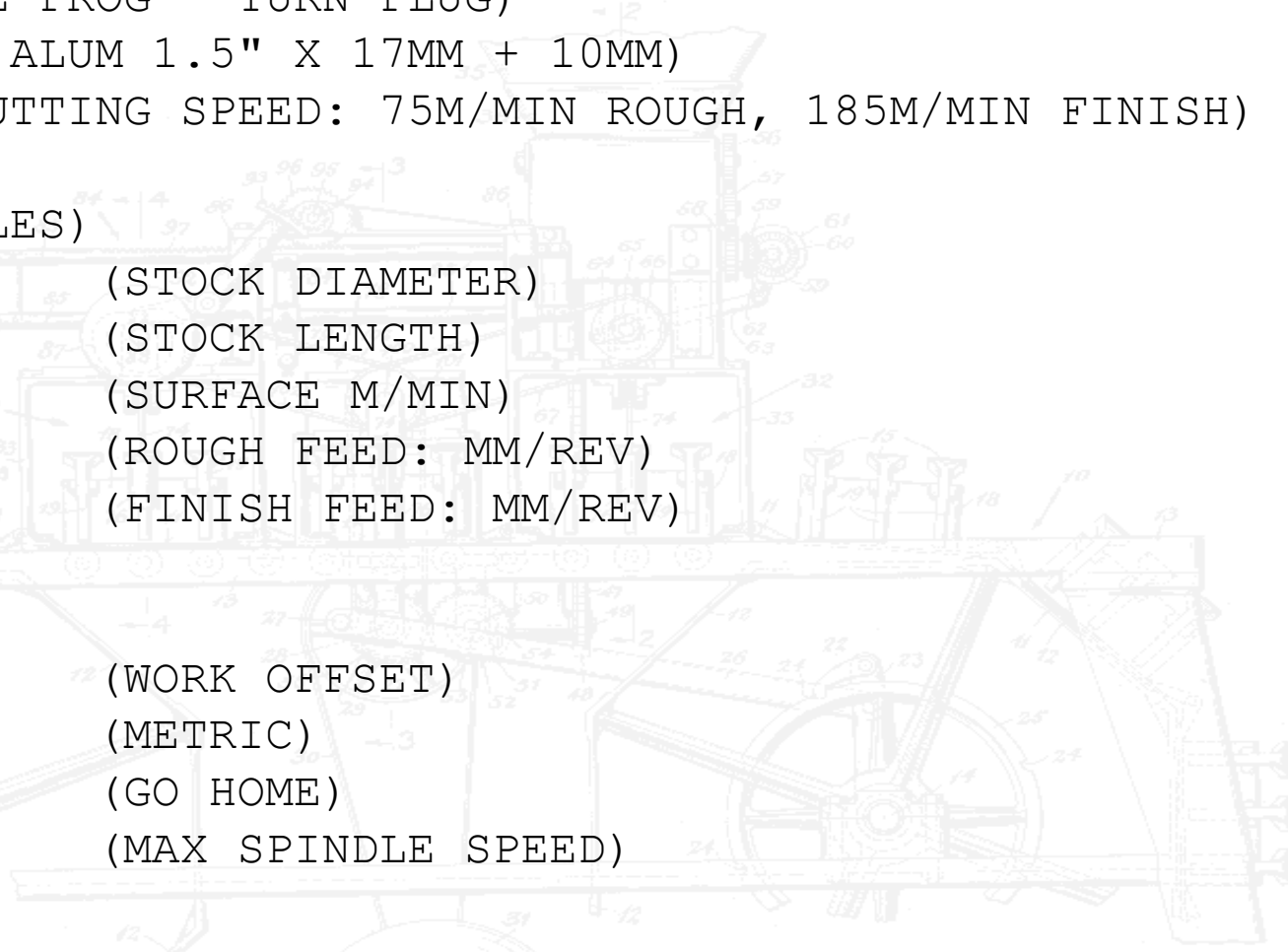
|     |                             |
|-----|-----------------------------|
| G54 | Select work coord system #1 |
| G70 | Finishing cycle             |
| G71 | Roughing cycle              |
| G72 | Facing cycle                |
| G76 | Threading cycle             |
| G96 | Constant surface speed mode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

# M-CODES FOR TURNING

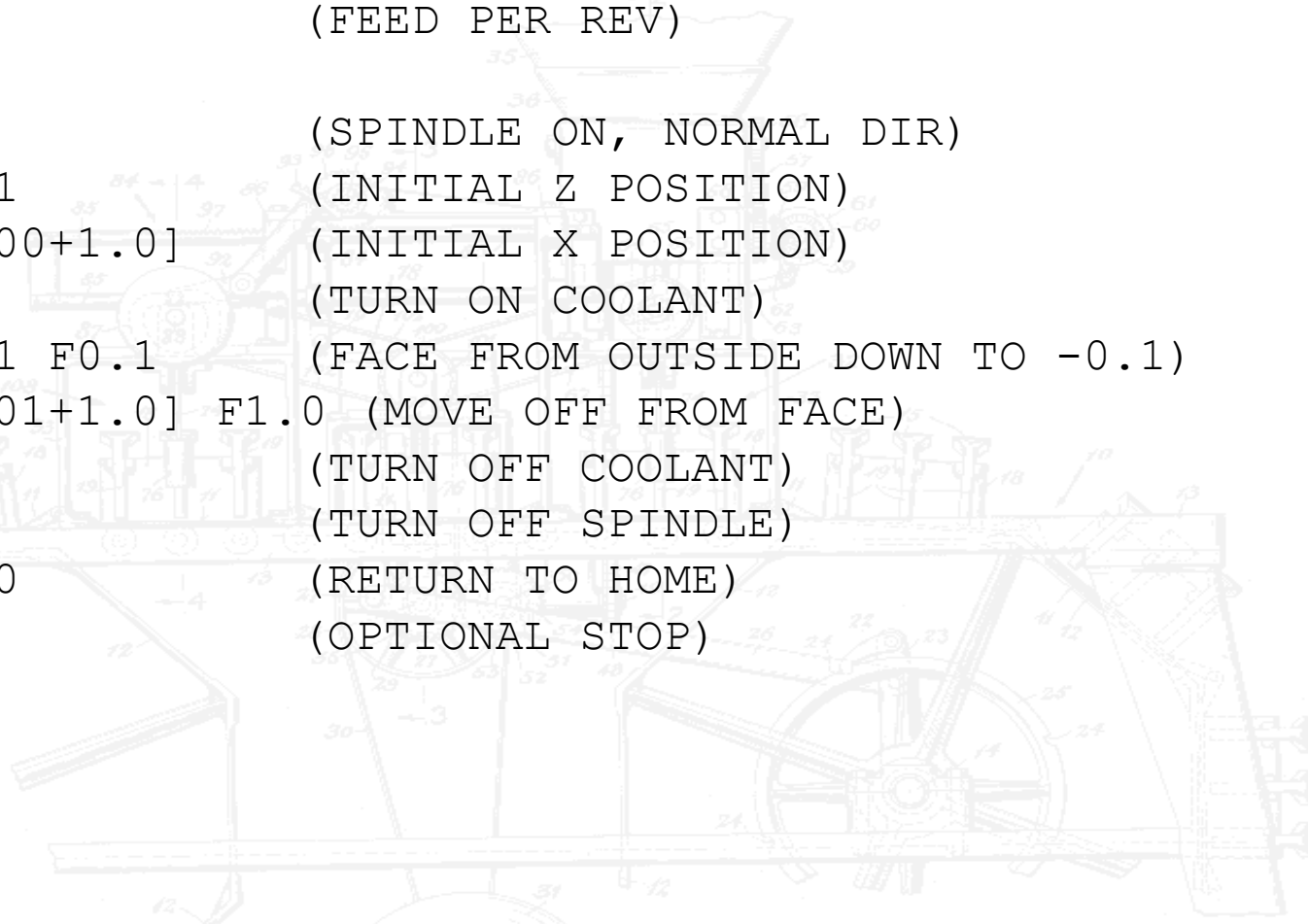
|     |                                      |
|-----|--------------------------------------|
| M00 | Program Stop                         |
| M01 | Opt. Program Stop (panel controlled) |
| M03 | Start spindle (normal rotation)      |
| M04 | Start spindle (reverse rotation)     |
| M05 | Stop spindle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

# EXAMPLE PROGRAM

```
%  
O1  
; (EXAMPLE PROG - TURN PLUG)  
; (STOCK: ALUM 1.5" X 17MM + 10MM)  
; (ALUM CUTTING SPEED: 75M/MIN ROUGH, 185M/MIN FINISH)  
;  
; (VARIABLES)  
#500=38.1 (STOCK DIAMETER)  
#501=17.0 (STOCK LENGTH)  
#502=75.0 (SURFACE M/MIN)  
#503=0.4 (ROUGH FEED: MM/REV)  
#504=0.1 (FINISH FEED: MM/REV)  
;  
N1  
G54 (WORK OFFSET)  
G21 (METRIC)  
G28 U0 W0 (GO HOME)  
G50 S2000 (MAX SPINDLE SPEED)
```

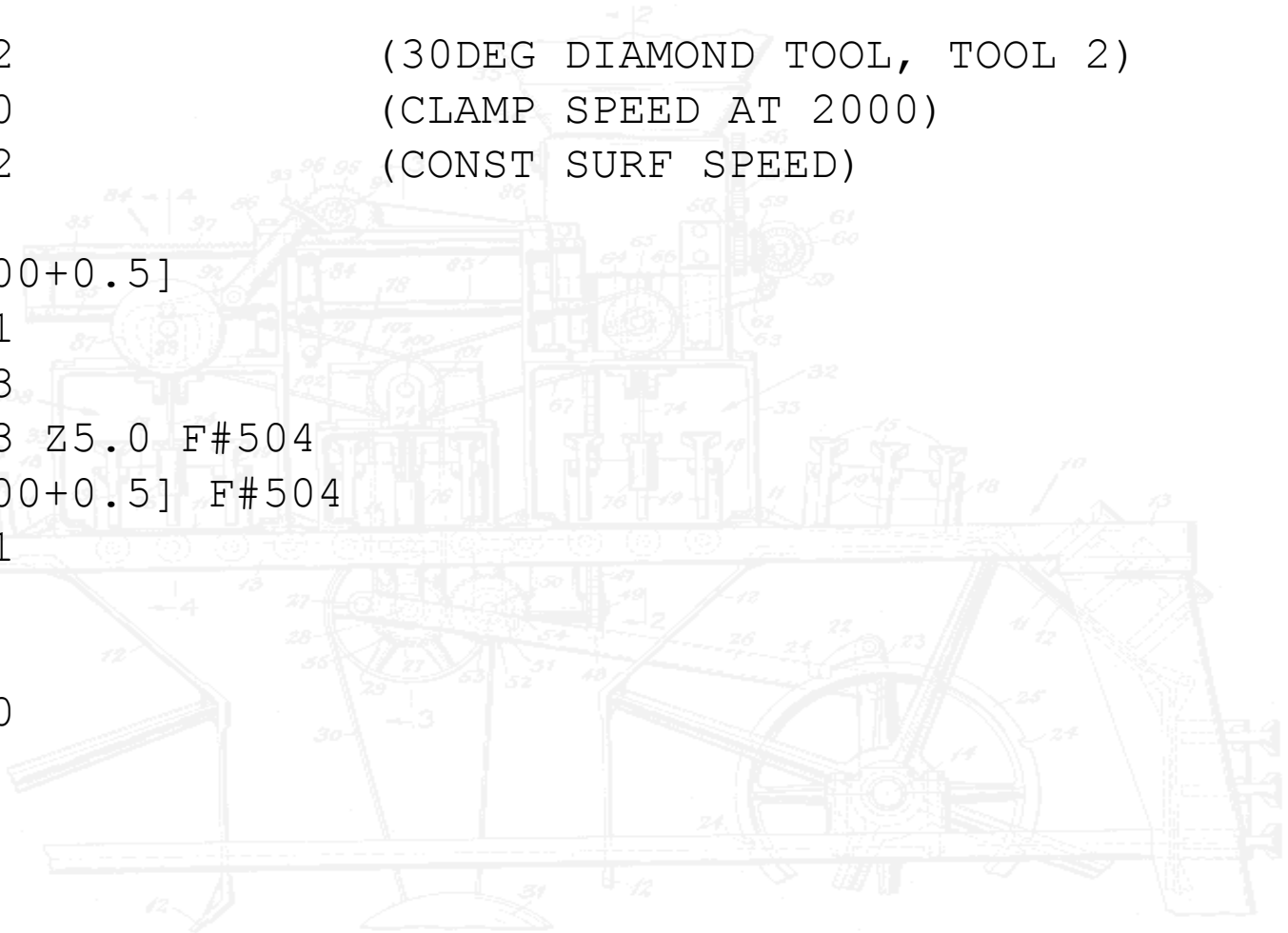


```
;
; (FACE WITH 80DEG DIAMOND TOOL AND SPEED LIMITING)
;
G00 T0101          (80DEG DIAMOND TOOL, TOOL 1)
G50 S1000          (CLAMP SPEED AT 1000RPM)
G96 S#502          (CONST SURF SPEED)
G99                (FEED PER REV)
;
M03                (SPINDLE ON, NORMAL DIR)
G00 Z#501          (INITIAL Z POSITION)
G00 X[#500+1.0]    (INITIAL X POSITION)
M08                (TURN ON COOLANT)
G01 X-0.1 F0.1     (FACE FROM OUTSIDE DOWN TO -0.1)
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 DOC -> 8 PASSES)
;
N2
G00 T0101          (55DEG DIAMOND TOOL, TOOL 1)
G50 S2000          (CLAMP SPEED AT 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 Z5.0 F#503 (THIRD PASS)
```

```
;
; (FINISH OD 20.3MM)
;
N3
G00 T0202          (30DEG DIAMOND TOOL, TOOL 2)
G50 S2000          (CLAMP SPEED AT 2000)
G96 S#502          (CONST SURF SPEED)
G99
G00 X[#500+0.5]
G00 Z#501
G00 X20.3
G01 X20.3 Z5.0 F#504
G01 X[#500+0.5] F#504
G00 Z#501
M09
M05
G28 U0 W0
M01
```





```
;
; (CUTOFF)
G00 T1010
G00 X[#500+2.0]
G00 Z-3.0 (CUTOFF BLADE IS 3.0MM WIDE)
M03
M08
G50 S1000 (CLAMP SPEED AT 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)
%
```

