Lathe Coordinate System

MORI SEIKI
THE MACHINE TOOL COMPANY

2. X-Axis and Z-Axis

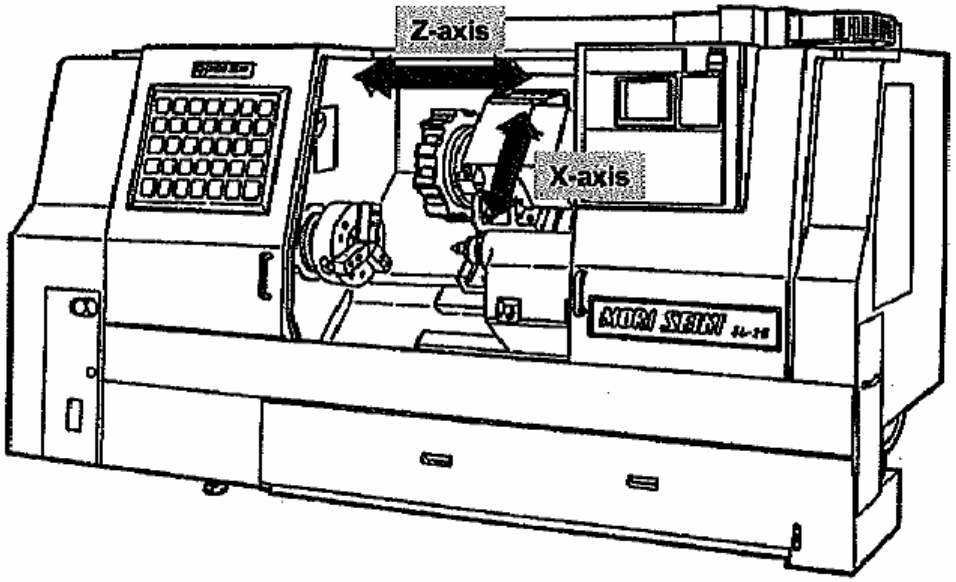
COORDINATES

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.

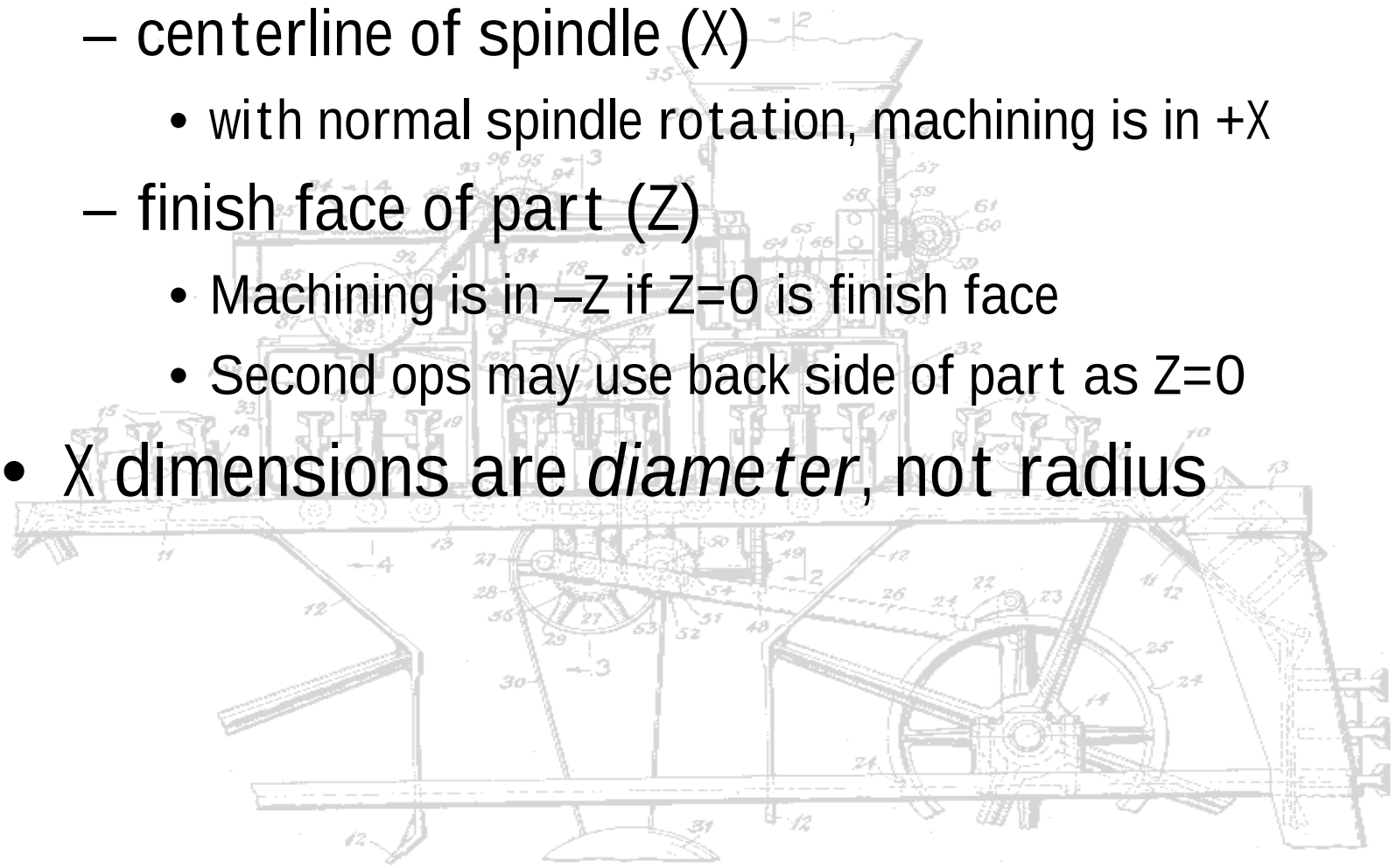
Z-axis ..... The axis along which the carriage moves.

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



Workpiece Zero Point

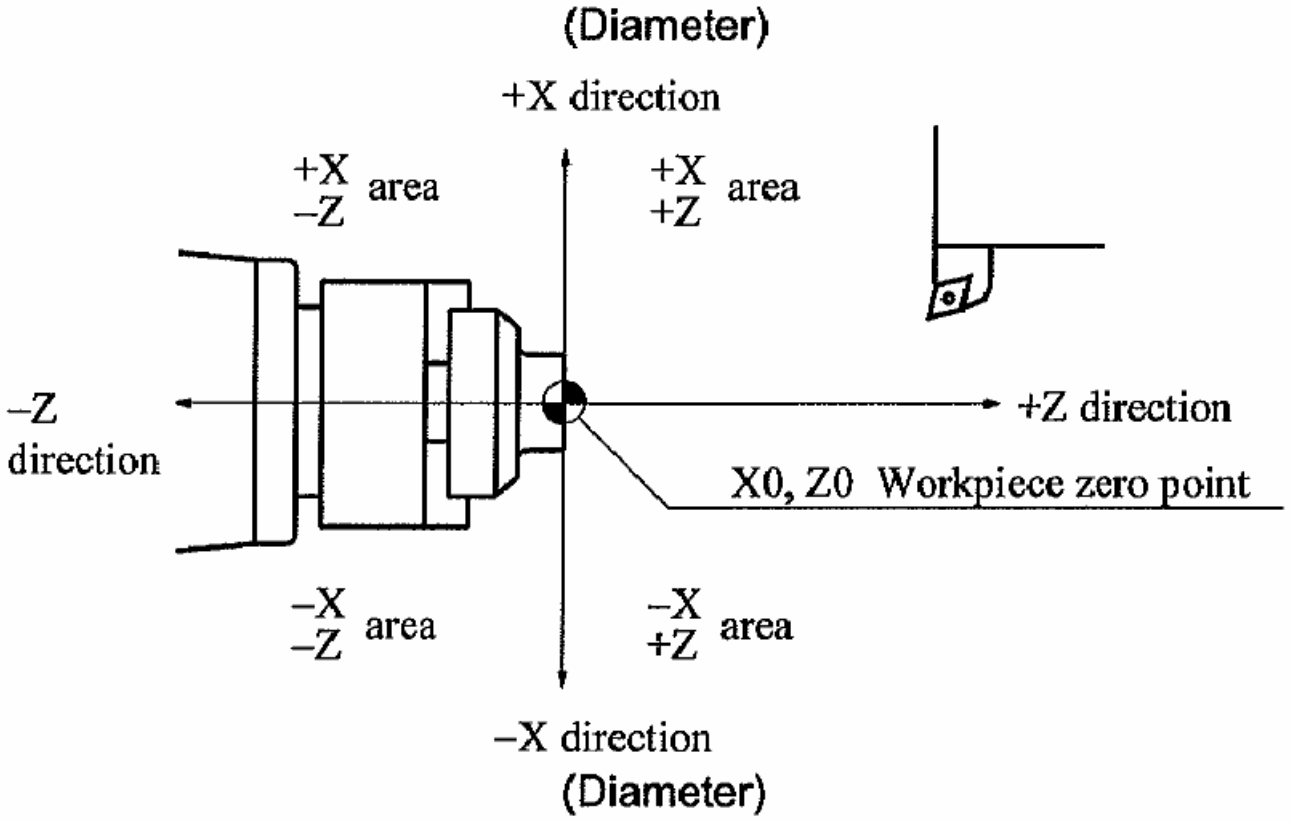
- Coordinate system zero point is
  - centerline of spindle (X)
    - with normal spindle rotation, machining is in +X
  - finish face of part (Z)
    - Machining is in –Z if Z=0 is finish face
    - Second ops may use back side of part as Z=0
- X dimensions are *diameter*, not radius



WORKPIECE ZERO POINT

a) Chuck-work

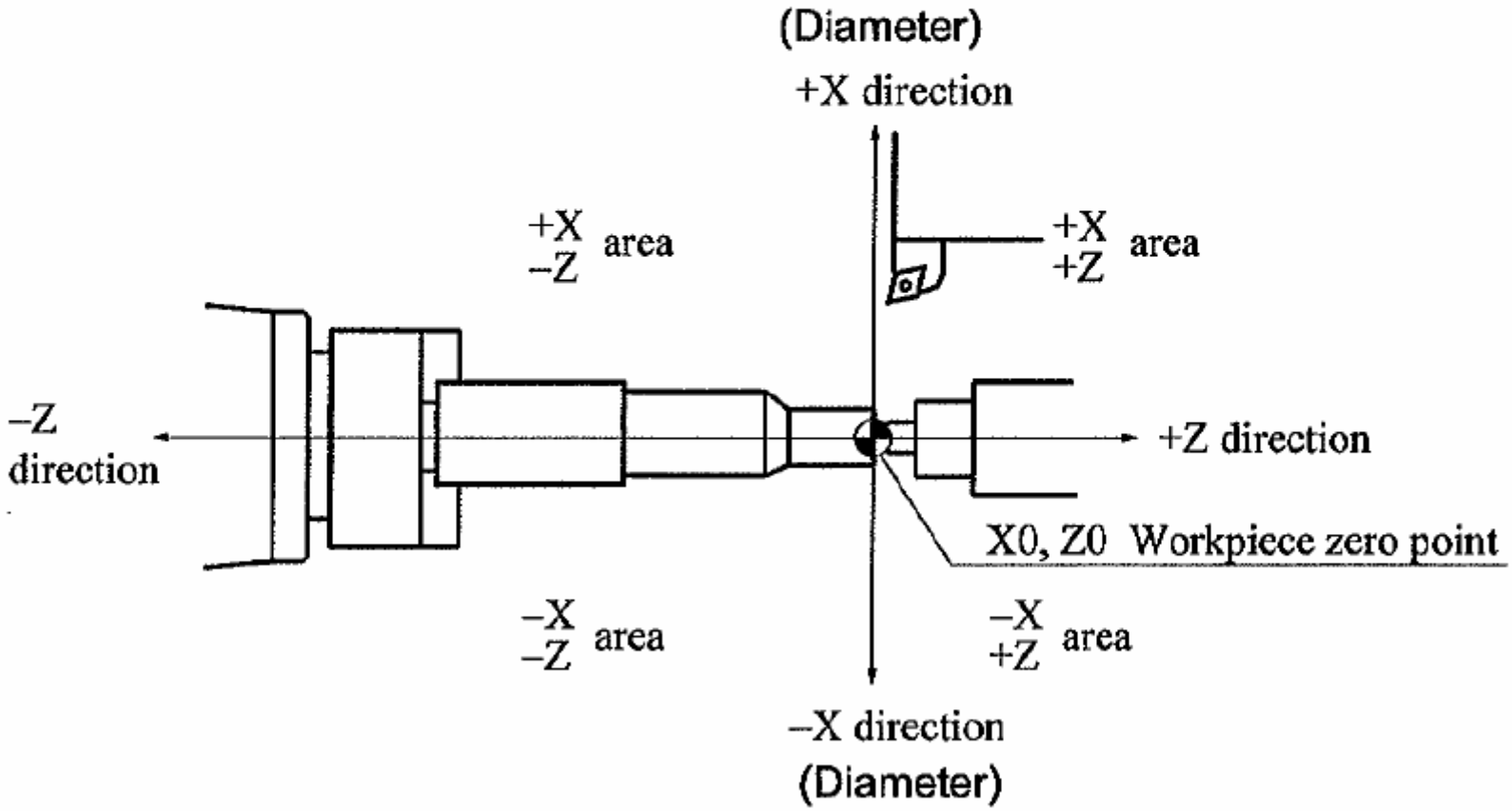
X-axis: Diametric commands



WORKPIECE ZERO POINT

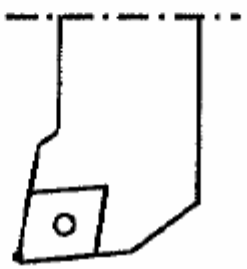
b) Center-work

X-axis: Diametric commands

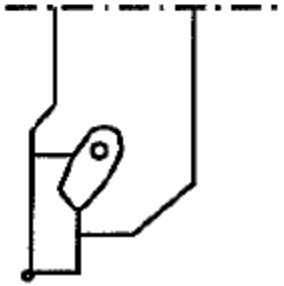


4. Tool Command Point

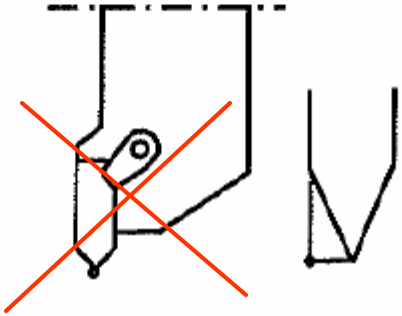
a) O.D./Face Turning



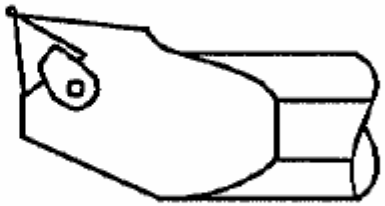
b) O.D. Grooving



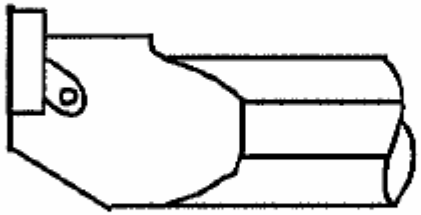
c) O.D. Threading



d) I.D. Turning

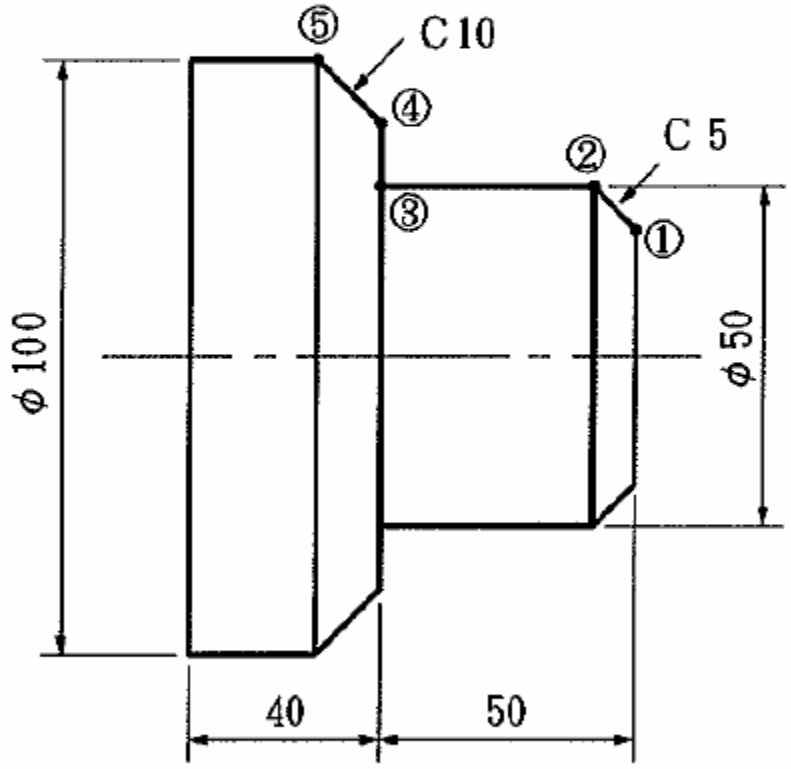


e) I.D. Grooving



Example

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

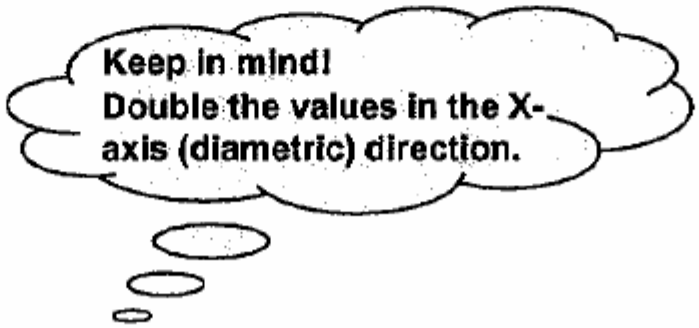


(Fig. 1-6)

```
O
_____

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

_____

_____
M30
```

Dimensions (coordinate values) of point (1) to point (5) are indicated below.

Point	Dimension		Program Command	
	Diametric Direction	Longitudinal Direction	Diametric Direction	Longitudinal Direction
(1)	40 mm	0 mm	X40.0	Z 0
(2)	50 mm	–5 mm	X50.0	Z–5.0
(3)	50 mm	–50 mm	X50.0	Z–50.0
(4)	80 mm	–50 mm	X80.0	Z–50.0
(5)	100 mm	–60 mm	X100.0	Z–60.0



6. Dimensions in Programming

6.1 Absolute Programming

Points (1) to (4) in Fig. 1-6 may be expressed as shown below.

(1) X40000	Z0	X40.0	Z0
(2) X50000	Z–5000	X50.0	Z–5.0
(3) X50000	Z–50000	X50.0	Z–50.0
(4) X80000	Z–50000	X80.0	Z–50.0
(5) X100000	Z–60000	X100.0	Z–60.0

That is, values as indicated below are used to express X and Z values.

X: Diameter of the workpiece (in reference to X0)
Z: Distance from the workpiece zero point (Z0)

The sign that immediately follows X or Z shows the area where the point lies.
If the sign is "+", it is usually omitted.
If the same coordinate value continues, the second and later appearance may be omitted.

6.2 Incremental Programming

In incremental programming, addresses U (in diameter) and W are used to express dimensions in the X-axis and Z-axis directions, respectively, to clearly identifying it from absolute programming. In this programming method, the plus sign (omitted) and the minus sign indicate the direction in which the tool (command point) moves. The numerical value that follows the sign indicate the distance.

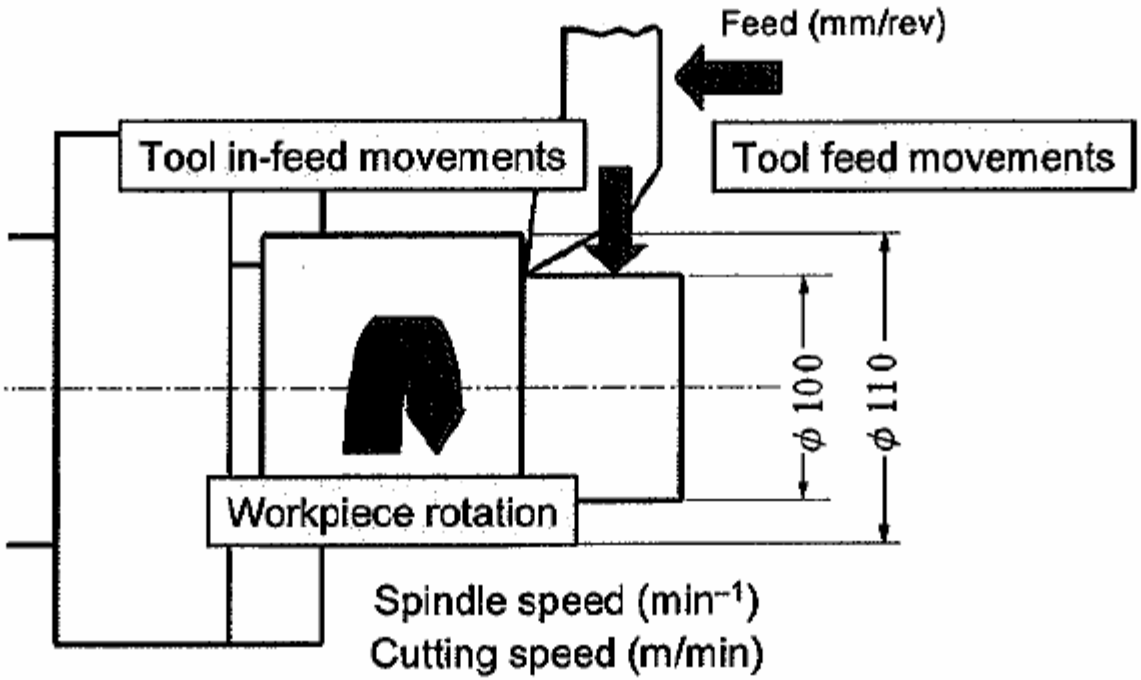
If you use the incremental commands to express point (1) to point (5) of the sample workpiece in Fig. 1-6, the commands are as shown below.

Point (1)	X40.0	Z0
Point (2)	U10.0	W–5.0
Point (3)		W–45.0
Point (4)	U30.0	
Point (5)	U20.0	W–10.0

<Use of Incremental Programming>

This programming method is usually used in retracting (moving away) the tool from the cut surface (O.D., I.D., end face).

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;
```

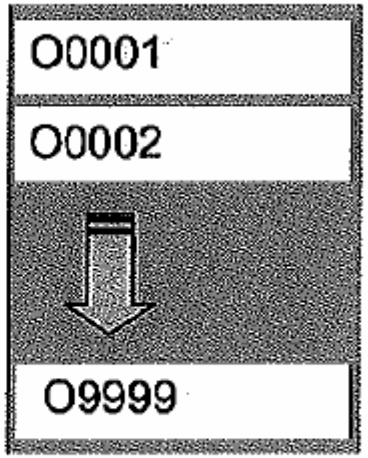
Program number (O1~O9999)

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

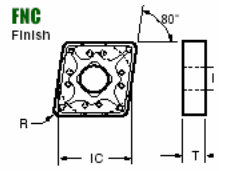
THANK YOU for your careful attention to my lecture.

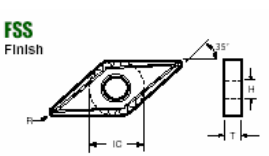
This ends the explanation of programming.

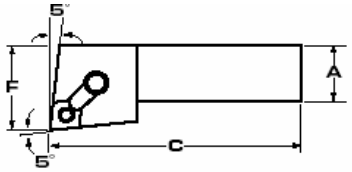
I hope all of you enjoyed to understand programming basics.

Lathe Tooling

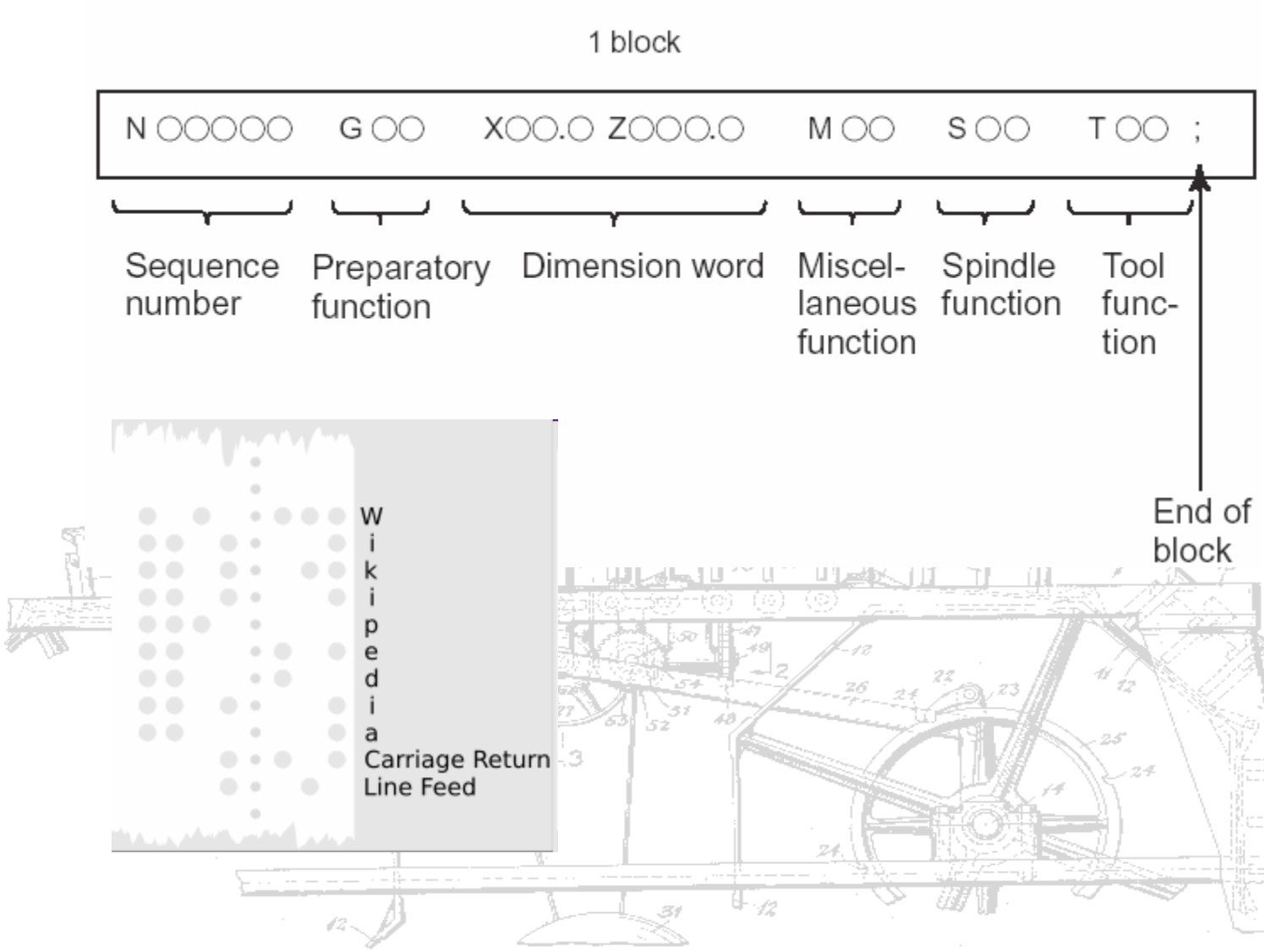
- Turning
- Boring
- OD Grooving
- ID Grooving
- Face Grooving
- OD Threading
- ID Threading







CNC Lathe Programming



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