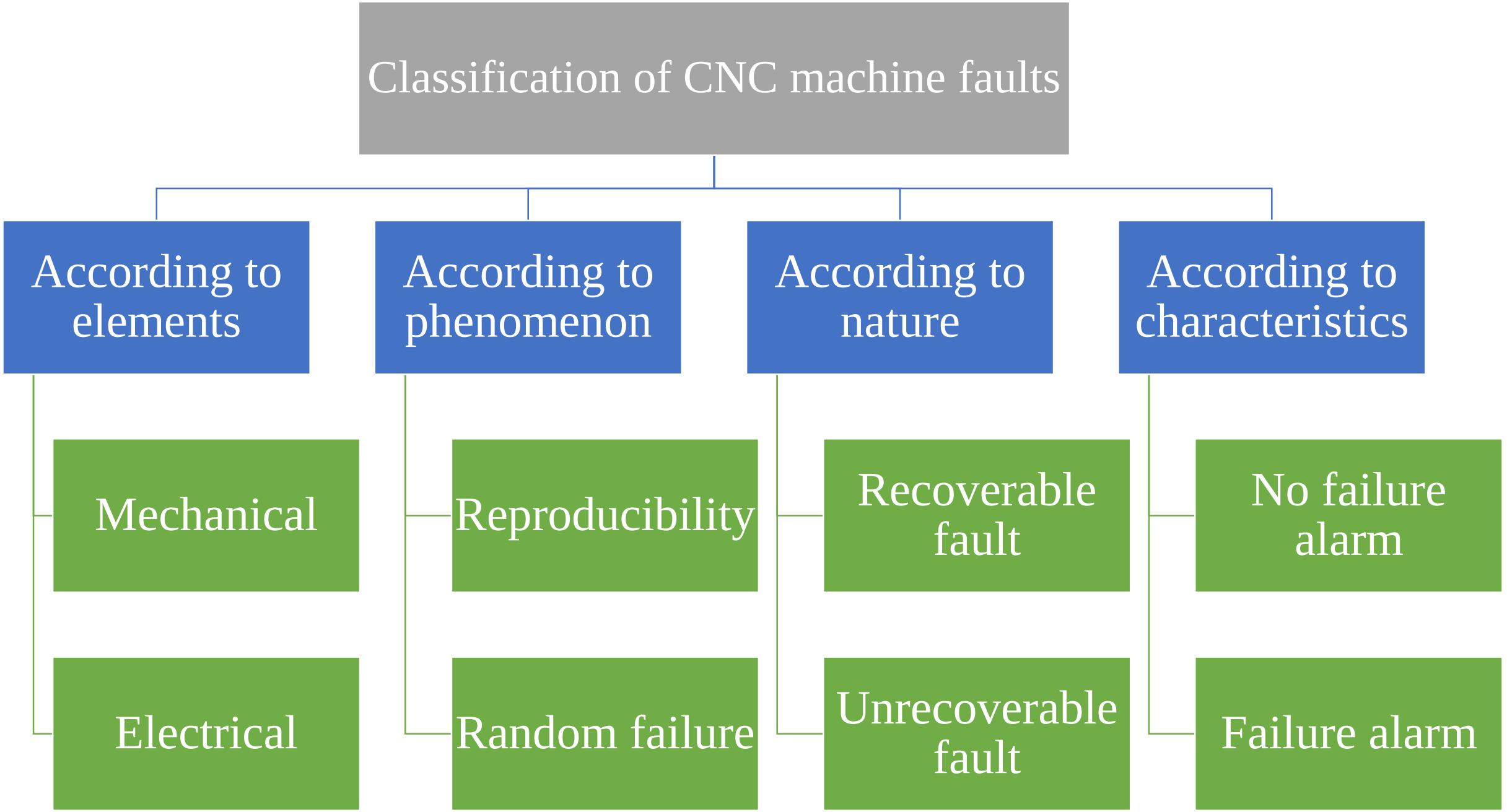
CNC MACHINES AND AUTOMATION

AMIT JANGRA
Lecturer
Mechanical Engineering Department
GP HISAR

Chapter 5
(Problems in CNC machines)


Classification of CNC machine faults
According to elements
According to phenomenon
According to nature
According to characteristics
Mechanical
Electrical
Reproducibility
Random failure
Recoverable fault
Unrecoverable fault
No failure alarm
Failure alarm

Common Problems in Mechanical Components

i) Bed

ii) Tail stock and Head stock

iii) Slideways Tool Pallettes

iv) Mechanical Switches

v) Host part

Common Problems in Electrical Components

i) Switching Devices

ii) Solenoids

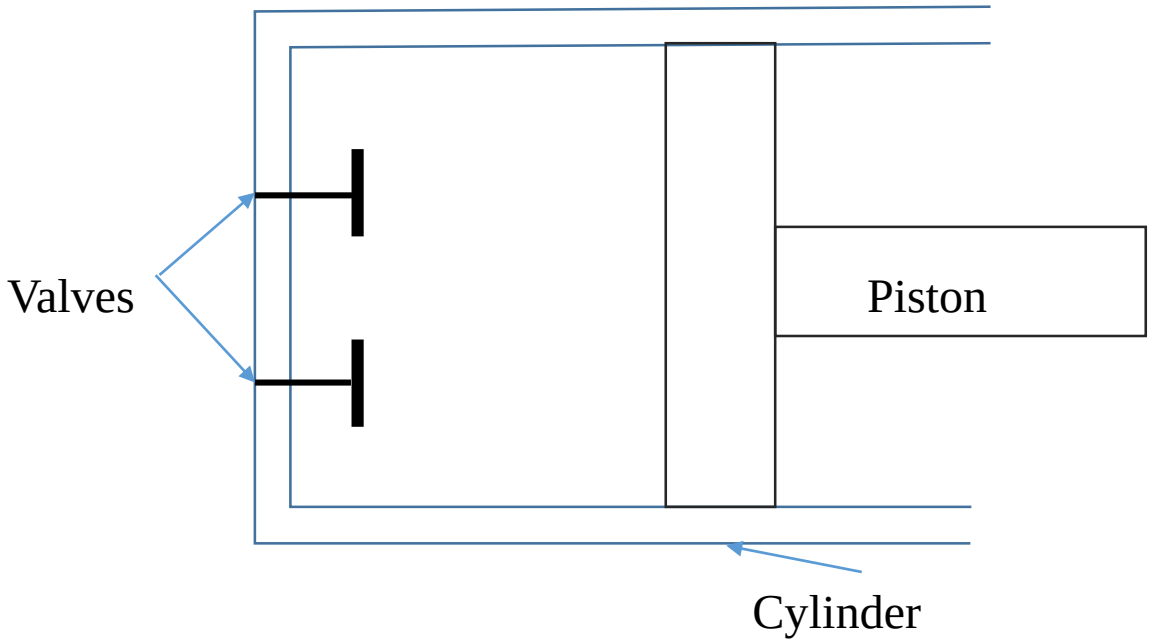
iii) Drive System

Common Problems in Pneumatic Components

i) Cylinder

ii) Piston and Piston rings

iii) Valves



Common Problems in Electronic Components

i) Display Screen/Monitor

ii) Tele Communication Lines

iii) Sensors

iv) Feedback Unit

v) CPU

vi) Converters

vii) Control Panel

Common Problems in Electronic Components

- Visual Inspection
- Oscilloscope
- Logic Probe
- Current Tracer
- Logic Pulser
- Logic Clip
- Logic Comparator
- Arbitrary Waveform Generator
- Signature Analyzer
- Digital Multimeter

Fault	Cause	Remedies
1. Panel is not Working	Main Supply is not given to panel. Inbuilt panel short circuit protected M.C. BTO OFF Emergency Stop is pressed. Key switch off.	Main supply is given to panel. M.C.B is always on. Emergency stop is released. Key switch is on.
2. Machine is not working	Supply is not given to Panel CNC switch is off.	Proper supply is given to panel. CNC switch is on.
3. We are in CNC software but progress in not exciting.	Power failure Emergency Stop is pressed. Limit switched operated. Communication cable is not connected or it is broken.	Power is on. Emergency stop is released limit switched released communication cable is connected properly.
4. Lubrication motor not working	Proper 240 volt supply not given. Signal relay not working.	Proper 240 volt supply given. Signal relay is changed.
5. X-motor and Z-motor drive not working.	Main supply is not given to drive. Wire is not connected from interfacing card to drive. Motor wires are broken. Communication fail.	Main supply is given to drive. Interface wire is connected properly. Check Motor wires are proper connect.
6. The material barns on outside edge.	Tool is blunt Feed speed too low	Replace with sharp tool Increase feed speed
7. Burnt edge of holes	Tool is blunt Tool put in wrong spindle	Replace with sharp tool Insert tool in correct spindle
8. Rough edge , cutter marks visible	Machine is travelling too fast	Reduce the feed speed
9. Component not the correct size	Tool data has been entered incorrectly Component size is entered incorrectly	Edit tool data setting Edit component data

Online Time Fault finding/Diagnosis Tools in CNC machines

i) Direct Observation Method

ii) Make use of the CNC system hardware, software and alarm function

iii) Hammering method

iv) Theory analysis method

v) Measurement comparison method

vi) Interface signal method

vii) Self diagnosis technique

viii) Parameter test method

ix) Malfunction and failure analysis

x) Control start-up diagnosis

xi) Preventive maintenance notices

xii) Tool life monitoring Programming diagnostics

NC words

i) **n-words**: - They denote the sequence number to identify the block. The complete word usually consist of three digits with ‘n’ as a prefix.

ii) **g-words**: - These are called preparatory words i.e., the words used to prepare the controlling unit for the operating instructions, which are to follow.

iii) x, y, z, a and b words: - They are knowns as coordinate words or dimension data words. The first three words x, y, z followed by actual dimensions, represent the coordinate position of tool along the three principal axis while the words ‘a’ and ‘b’ indicate the angular positions.

iv) **f-words**: - These words carry the alphabet ‘f’ as prefix and may contain upto 8 digit maximum. They are used to specify feed rate in mm/min.

v) **s-words**: - These words carry the alphabet ‘s’ as prefix and specify cutting speed in rev./min of the spindle.

vi) **t-words**: - These words carry the alphabet ‘t’ as prefix and may contain upto 5 digit maximum. They are known as tool selection words and used only for those NC machines which carry a tool turret or an ATC.

vii) **m-words**: - These are known as Miscellaneous Function words. They consists of three digits as a maximum, including the alphabet ‘m’ as a prefix. Such function is always the last word in the block to indicate an operation.

viii) **EOB**: - It means the End of Block and it indicates the end of instructions contained in the block.

Machine tool Zero Point Setting

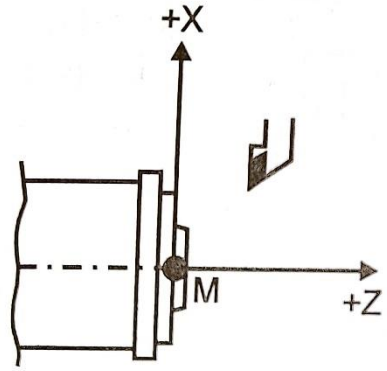
i) **Manual Setting**: - The operator can used MCU controls to locate the spindle over the desired part zero and then set X and Y coordinate registers on the console to zero.

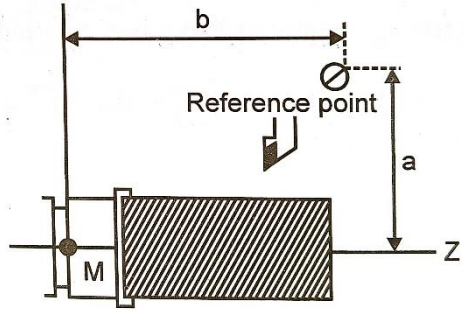
ii) **Absolute zero shift**: - This method can change the position of the coordinate system by a command line in the CNC program.

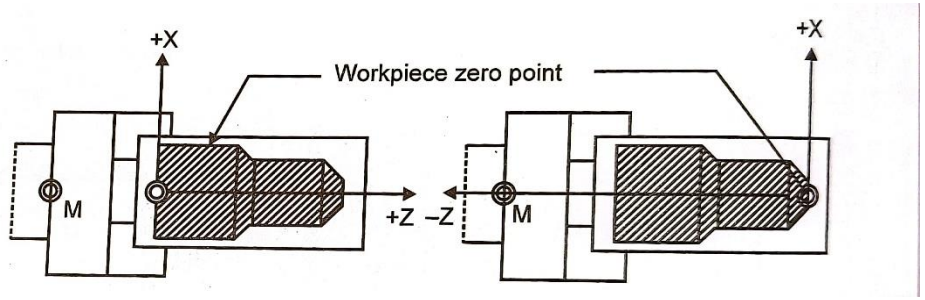
N1 G28 X0 Y0 Z0 (sends spindle to home zero position)

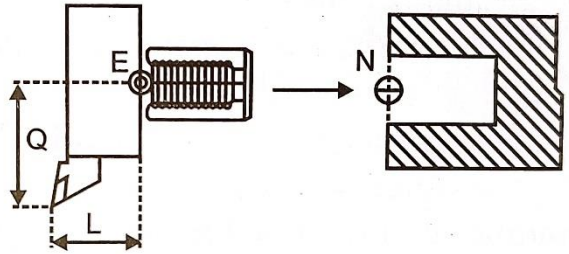
Zero Point

- Machine tool zero point
- Program zero point
- Workpiece zero point
- Tool zero point









Part Programming Format

Tab sequential format (NC only)

N010 G00 X100.00 Y200 Z10 F30 M08

010 > 00 > 100.00 >200 > 10 > 30 > 08

Fixed Format (NC only)

N010	G01	X10	Y20	Z30	F30	S1000
010	01	10	20	30	30	1000

Word address format (NC & CNC)

This format is standardized by EIA and there are no TAB codes used.

N01 G01 X30 Y20 Z10 S500 F80 T01 M01

Compatible format (NC & CNC)

It is similar to word address format, but TAB codes are added in it.

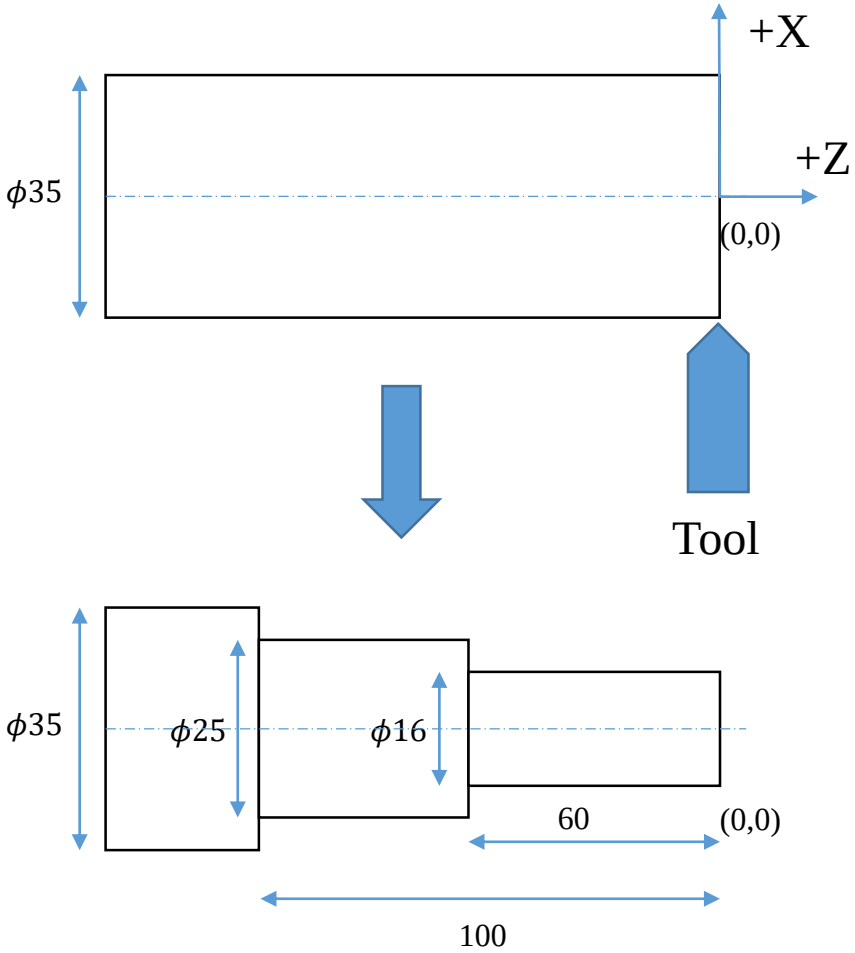
Part Programme Structure

N03	G02	X300	Y200	Z10	I100	J-10	K20	S450	F80	T03	M01#
Block No	Preparatory Code	Location along X-axis	Location along Y-axis	Location along Z-axis	Center position along X-axis Curved paths	Center position along Y-axis Curved paths	Center position along Z-axis Curved paths	Spindle speed	Feed speed	Tool Specification	Miscellaneous code

M-Codes

M00 Program Stop
M01 Program Optional Stop
M02 End the Program
M03 Spindle On Clockwise, Laser, Flame, Power ON
M04 Spindle On Counter Clockwise
M05 Spindle Stop, Laser, Flame, Power OFF
M06 Tool Change
M08 Coolant On
M09 Coolant Off
M10 Reserved for tool height offset
M13 Spindle On, Coolant On
M30 End the Program when macros are used
M91 Readout Display Incremental
M92 Readout Display Absolute
M97 Go to or jump to line number
M98 Jump to macro or subroutine
M99 Return from macro or subroutine
M100 Machine Zero Reset
M199 Mid program start

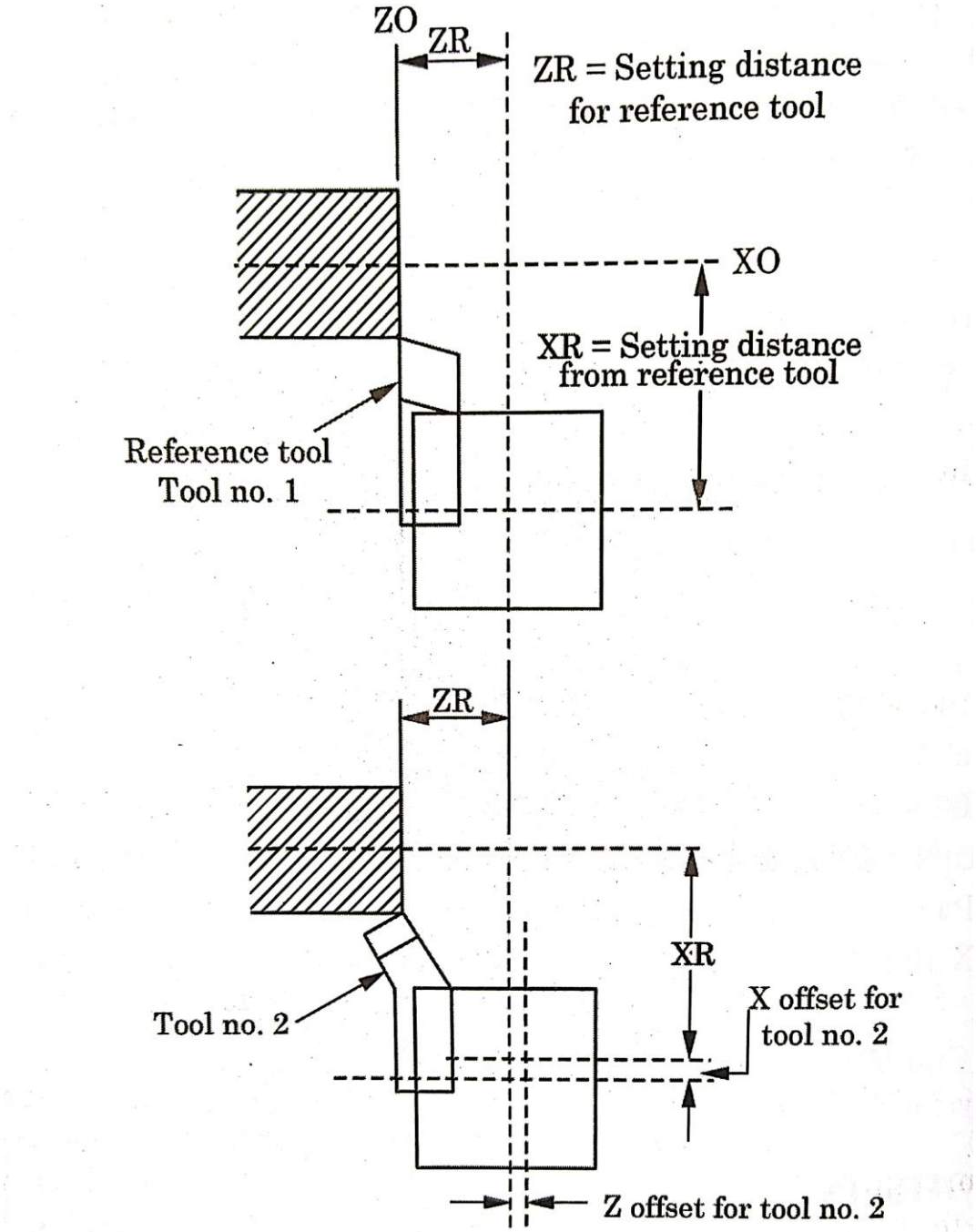
Simple Programming for Rational Components



Part Programme of Turning operation

```
%1000;                                  (Main Programme)
N01 G54 G90 G71 G94 M03 S800;           (Parameter Settings)
N05 G01 X-12.5 Z0 F2;                   (Facing the job)
N10 G00 Z1;                             (Retrieval of tool)
N15 G00 X00;                            (Tool Clearance)
N20 G01 Z-100;                          (Starting cut)
N25 G00 X1 Z1;                          (Clearance position)
N30 G00 X-2;                            (Position of cut)
N35 G01 Z-60;                           (Cutting length)
N40 G00 X-1 Z1;                         (Retrieval of tool)
N45 G00 X-3;                            (Position of cut)
N50 G01 Z-60;                           (Cutting length)
N55 G00 X-2 Z1;                         (Retrieval of tool)
N60 G00 X-4;                            (Position of cut)
N65 G01 Z-60;                           (Cutting length)
N70 G00 X-3 Z1;                         (Retrieval of tool)
N75 G00 X-4.5;                          (Position of cut)
N80 G01 Z-60;                           (cutting length)
N85 G00 X5 Z5;                          (Final position of tool)
N90 M02;                                (End of Programme)
```

Tool Offsets: Correction for dimensions of the tools and movements of the workpiece has to be incorporated to give the exact machining of the component. This is known as tool offset. Normally, it is found that the size of the workpiece is not within the tolerance due to wear of the tool; it is then possible to edit the value of offsets to obtain the correct size, this is known as tool wear compensation.



Tool Compensation

Cutter radius Compensation

This code command allows the programmer to ignore the cutting tool’s radius or diameter during programming

Tool wear compensation

Similar to the cutter radius compensation, tool wear compensation is also used in part programming.

Canned Cycles: - Canned cycle or fixed cycle may be defined as a set of instructions, inbuilt or stored in the system memory, to perform a fixed sequence of operations. A canned cycles defines a series of machining sequence for drilling, boring, tapping etc. The canned cycle G81 to G89 are stored as subroutines L81 to L89. These cycles are used for repetitive and commonly used machining operations.

Sub Routines: - These are also known as subprograms, a very powerful saving method. The subroutines provide the capability of programming certain program that are repeated frequently. They are independent programmes that can be called any time and any number of times.

Do Loops: - The Do loops gives the facility to programmer to jump back to an earlier part of programme and execute the intervening programme and not separately like subroutines. It is given in the main program itself.

Refer to book for more details

Thank You