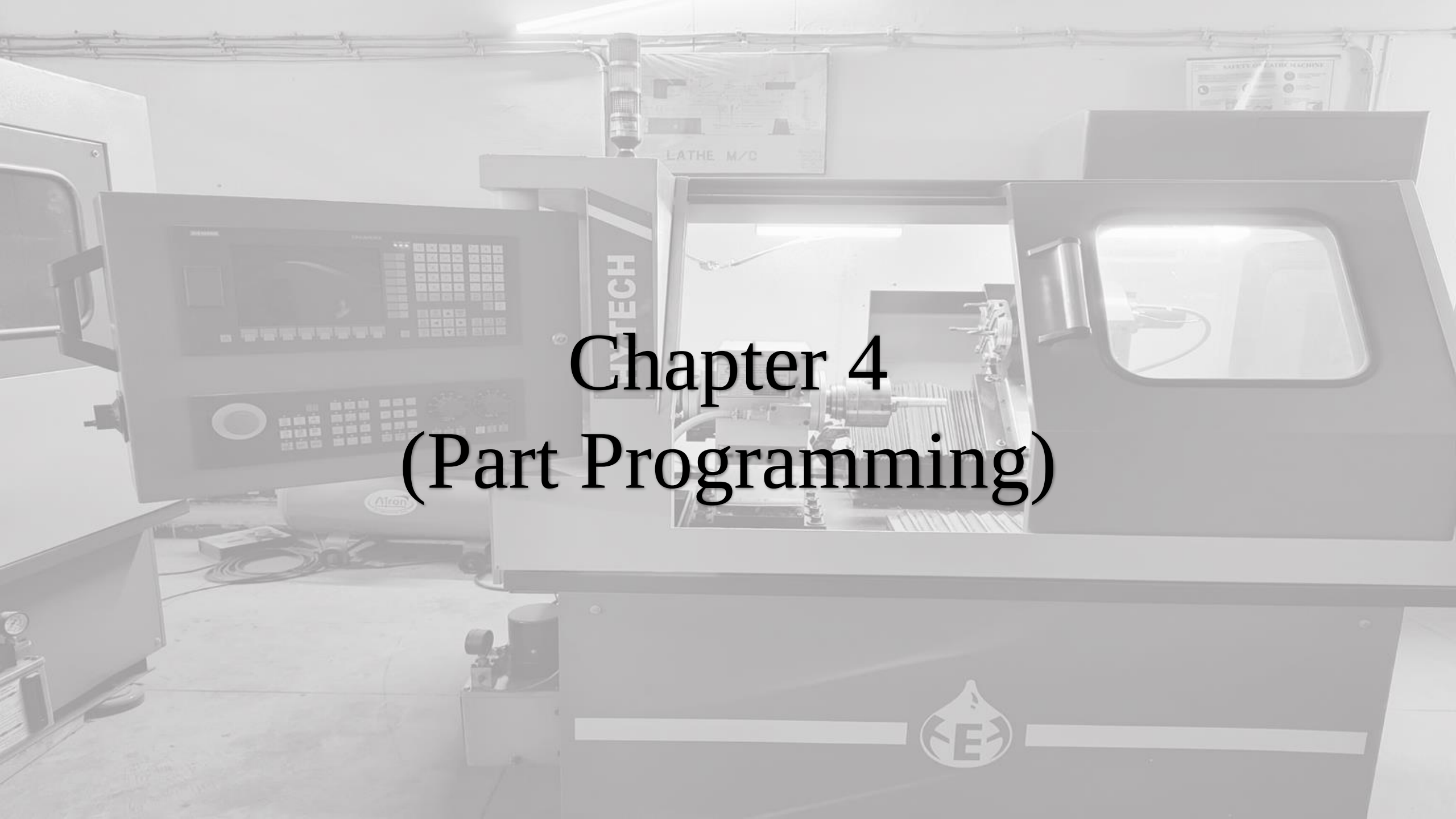
CNC MACHINES AND AUTOMATION

AMIT JANGRA
Lecturer
Mechanical Engineering Department
GP HISAR



Chapter 4
(Part Programming)

Part Programming

The **part program** is a sequence of instructions, which describe the work, which has to be done on a part in the form required by a computer under the control of **numerical control** computer program.

Basic Concept of Part Programming

The coded instruction or commands in the form of numbers, alphabets and symbols listed in logical sequence and are fed to the controller unit of machine tool to perform a series of operations. This set of coded instruction is called part program.

Following steps are needed to perform while performing a part program:

i) Determine the start up procedure, which includes the extraction of dimensional data from part drawings.
ii) Select the tool and determine the tool offset.
iii) Set up the zero position for the work piece.
iv) Select the speed and rotation of the spindle.
v) Set up the tool motions according to the profile required.
vi) Return the cutting tool to the reference point after completion of work.
vii) End the program by stopping the spindle and coolant.

Fundamental of Part Programming

i) Process planning

ii) Axes selection

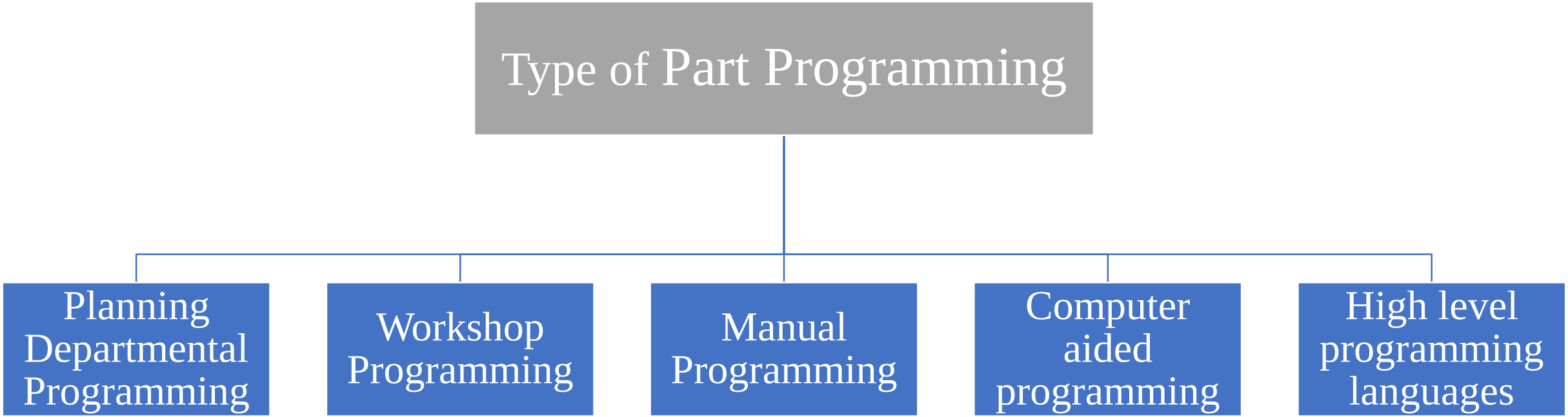
iii) Tool selection

iv) Cutting process parameter

v) Job and tool setup planning

vi) Machining path selection

vii) Part Program writing


Type of Part Programming
Planning Departmental Programming
Workshop Programming
Manual Programming
Computer aided programming
High level programming languages

Basic Terms of Part Programming

i) Part Program
ii) Main Program and structure
iii) Input unit
iv) Coordinate system

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

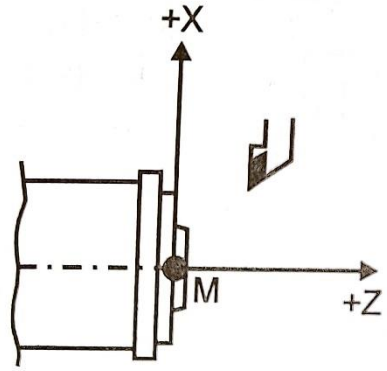
i) **Manual Setting**: - The operator can used MCU controls to locate the spindle over the desired part zero and then set X and Y coordinate registers on the console to zero.

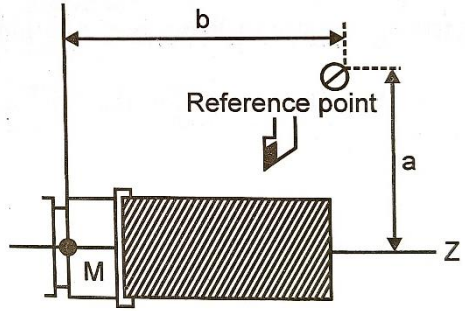
ii) **Absolute zero shift**: - This method can change the position of the coordinate system by a command line in the CNC program.

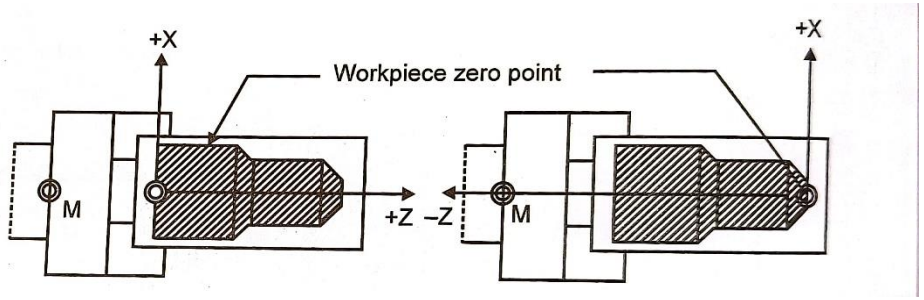
N1 G28 X0 Y0 Z0 (sends spindle to home zero position)

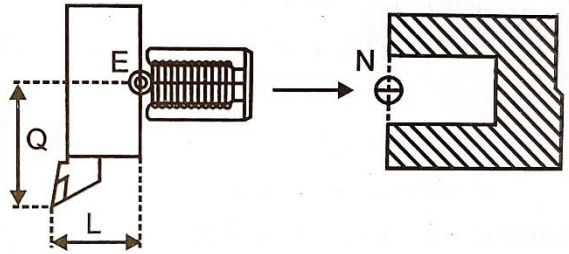
Zero Point

- 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

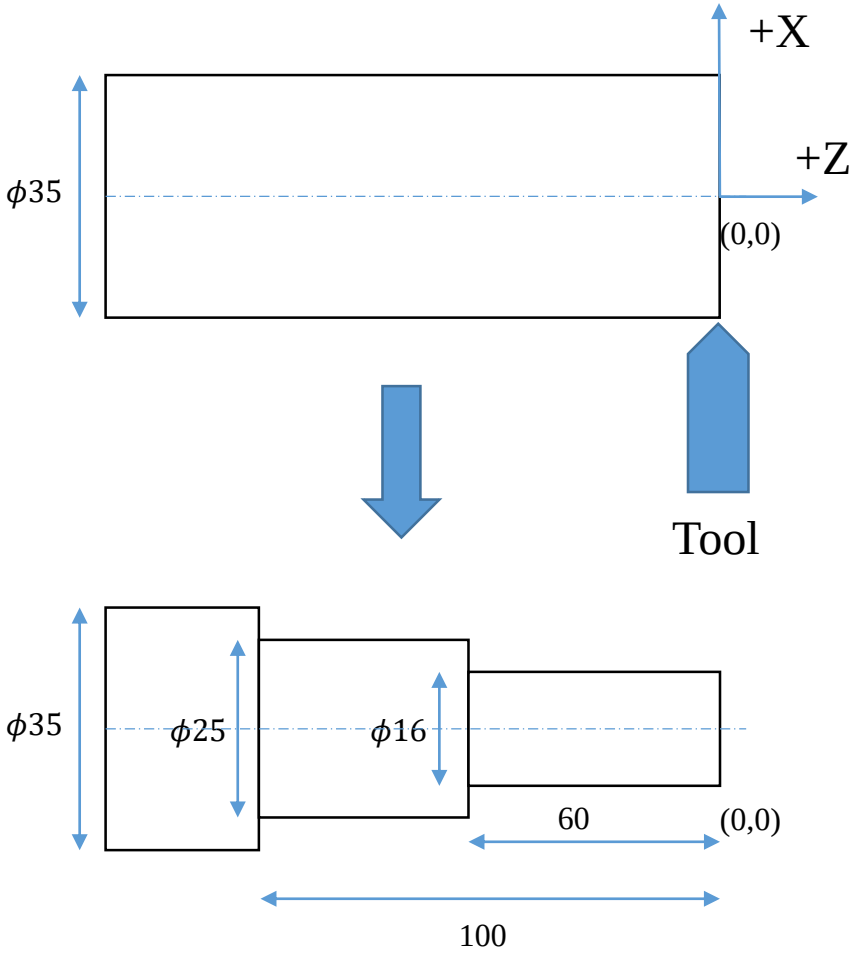
G Codes

Code	Description	Code	Description	Code	Description
G00*	Rapid positioning	G43	Tool length compensation in +Z	G82	Drilling cycle, counter boring
G01	Linear interpolation	G44	Tool length compensation in -Z	G83	Peck drilling cycle
G02	Circular interpolation CW	G49*	Tool length compensation cancel	G84	Tapping cycle
G03	Circular interpolation CCW	G52	Local coordinate system setting	G84.2	Rigid tapping cycle
G04	Dwell, Exact stop	G53	Positioning in machine coordinate	G85	Boring cycle
G09	Exact stop	G54*	Work coordinate system 1 select	G86	Boring cycle
G10	Programmable data input	G55	Work coordinate system 2 select	G87	Back boring cycle
G11*	Programmable data input cancel	G56	Work coordinate system 3 select	G88	Boring cycle
G17*	XY plane selection	G57	Work coordinate system 4 select	G89	Boring cycle
G18	ZX plane selection	G58	Work coordinate system 5 select	G90*	Select absolute command
G19	YZ plane selection	G59	Work coordinate system 6 select	G91	Select incremental command
G20	Select inch unit	G61	Exact stop mode	G92	Programming of absolute zero point
G21	Select metric unit in mm	G64*	Cutting mode	G93	Inverse time feed
G27	Reference point return check	G65	Macro call	G94*	Per minute feed
G28	Return to reference point	G66	Macro modal call	G95	Per revolution feed
G29	Return from reference point	G67	Macro modal call cancel	G96	Constant surface speed control
G30	Return to 2 nd reference point	G73	High speed peck drilling cycle	G97*	Constant surface speed control cancel
G33	Thread Cutting	G74	Counter tapping cycle	G98*	Return to initial point in canned cycle
G40*	Cutter compensation cancel	G76	Fine boring cycle	G99	Return to R point in canned cycle
G41	Cutter compensation left	G80*	Canned cycle cancel		
G42	Cutter compensation right	G81	Drilling cycle, spot boring		

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

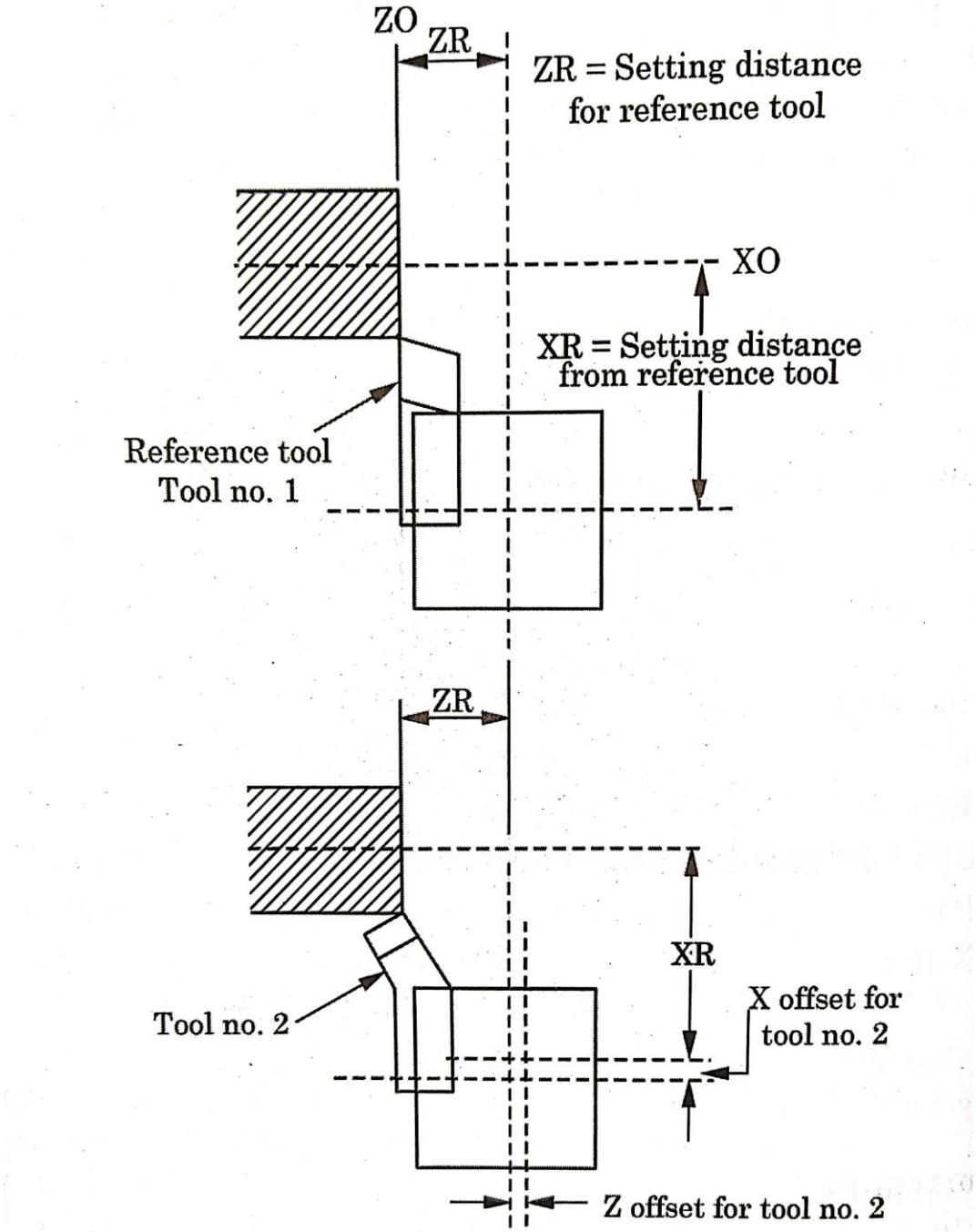
Simple Programming for Rational Components



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