

## Practical No. 4

AIM: - To develop a part programme for following lathe operations and make the job on CNC lathe and CNC turning centre (for finish pass only). Calculating coordinate points for a cylindrical job by considering sign convention for lathe (Material: Aluminium/Acrylic/Plastic rod)

- Plain turning and facing operations
- Taper turning operations
- Operation along contour using circular interpolation.

### Plain turning and Facing operation

Theory: The part program is a sequence of instruction, which describe the work, which has to be done on a part in the form required by a computer under the control of NC computer program

- i) Part Program for Plain turning and facing operation  
(All dimensions are in mm) Fig. 4.1 and 4.2

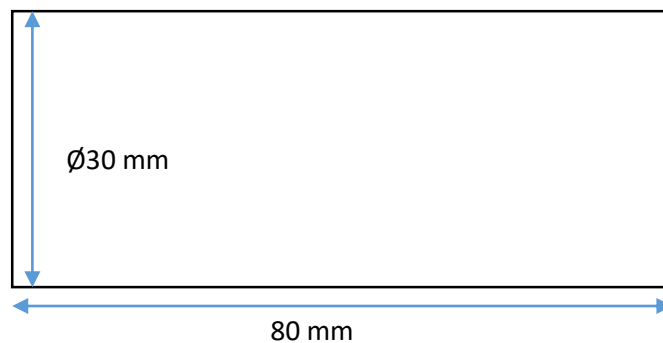


Fig 4.1

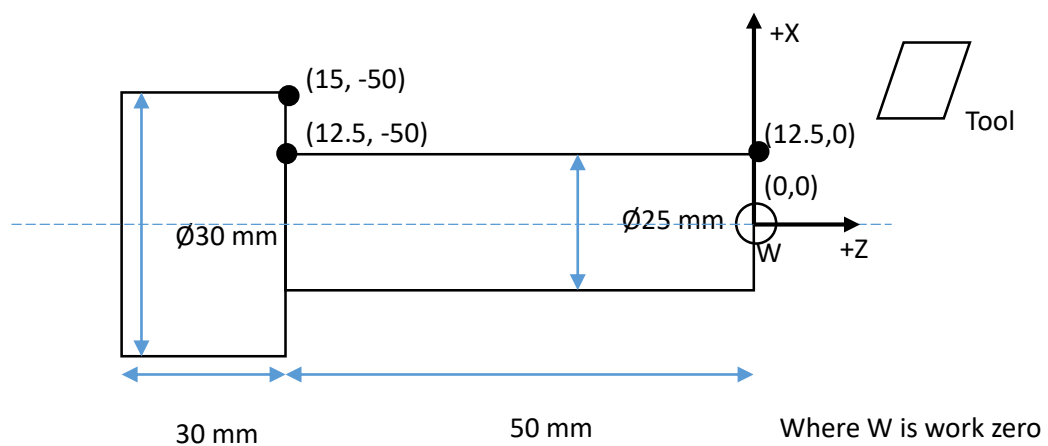


Fig 4.2

Table 1 represent the program for Turning and facing operation of Fig 4.1. First we did the facing operation and then we did the turning operation

*Table 1 Program for Turning and Facing Operation*

Program	Comment
G90G54G71G40;	(Parameter settings)
G74X0.0Z0.0;	(Return tool at home position)
M03S200;	(Spindle Rotate clock wise at 200 rpm)
M07;	(Coolant On)
G00X31.0Z0.0;	(Rapid travel of tool near workpiece at safe position)
Z-1.0;	(depth of cut in z for facing operation)
G01X-1.0F0.5;	(Cutting in X with feed rate in mm/min)
Z0.0;	(Retrieval of tool)
G00X31.0;	(Rapid travel of tool in X)
X28.0;	(Position of cut in X for turning operation)
G01Z-50.0;	(Cutting in Z with feed rate in mm/min)
Z-48.0X29.0;	(retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
X26.0;	(Position of cut in X for turning operation)
G01Z-50.0;	(Cutting in Z with feed rate in mm/min)
Z-48.0X29.0;	(retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
X25.0;	(Position of final cut in X for turning operation)
G01Z-50.0;	(Cutting in Z with feed rate in mm/min)
Z-48.0X29.0;	(retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
G74X0.0Z0.0;	(Return tool at home position)
M09;	(Coolant off)
M05;	(Spindle stop)
M30;	(End Program)

### Taper turning operation

Theory: The part program is a sequence of instruction, which describe the work, which has to be done on a part in the form required by a computer under the control of NC computer program

- ii) Part Program Taper turning operation  
(All dimensions are in mm) Fig. 4.3

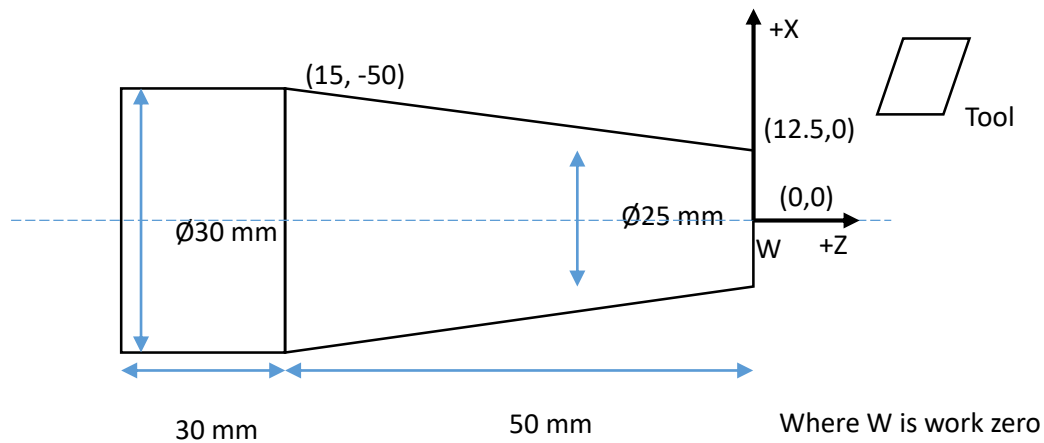


Fig 4.3

Table 2 represent the program for Taper turning operation of Fig 4.3. First we did the facing operation and then we did the taper turning operation

*Table 2 Taper Turning operation*

Program	Comment
G90G54G71G40;	(Parameter settings)
G74X0.0Z0.0;	(Return tool at home position)
M03S200;	(Spindle Rotate clock wise at 200 rpm)
M07;	(Coolant On)
G00X31.0Z0.0;	(Rapid travel of tool near workpiece at safe position)
Z-0.5;	(depth of cut in z for facing operation)
G01X-1.0F0.5;	(Cutting in X with feed rate in mm/min)
Z0.0;	(Retrieval of tool)
G00X31.0;	(Rapid travel of tool in X)
X28.0;	(Position of cut in X for turning operation)
G01X30.0Z-50.0;	(Cutting in Z with feed rate in mm/min)
Z-48.0X31.0;	(Retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
X26.0;	(Position of cut in X for turning operation)
G01X30.0Z-50.0;	(Cutting in Z with feed rate in mm/min)
Z-48.0X31.0;	(retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
X25.0;	(Position of final cut in X for turning operation)
G01X30.0Z-50.0;	(Cutting in Z with feed rate in mm/min)
Z-48.0X31.0;	(retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
G74X0.0Z0.0;	(Return tool at home position)
M09;	(Coolant off)
M05;	(Spindle stop)
M30;	(End Program)

## Operation along contour using circular interpolation

Theory: The part program is a sequence of instruction, which describe the work, which has to be done on a part in the form required by a computer under the control of NC computer program

- iii) Part Program Operation along contour using circular interpolation  
(All dimensions are in mm) Fig. 4.4

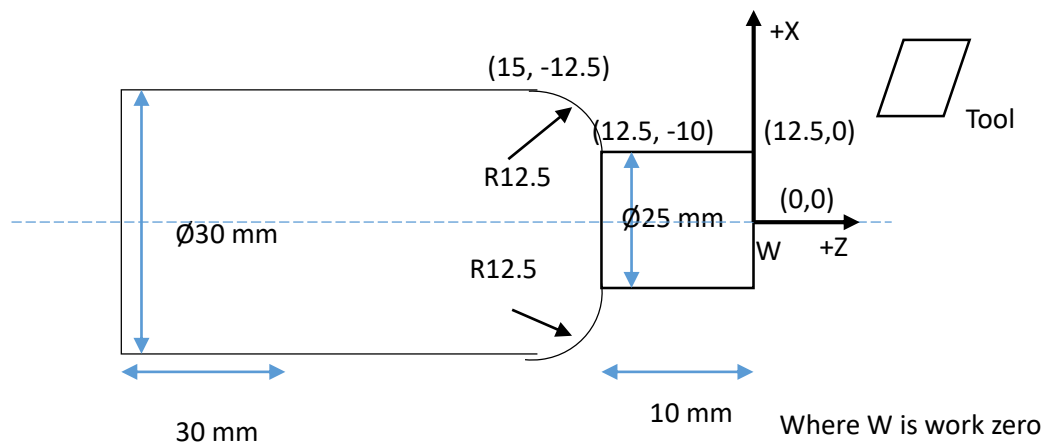


Fig 4.4

Table 3 represent the program for circular interpolation operation of Fig 4.4. First we did the facing operation and then we did the circular interpolation operation along with turning operation

*Table 3 Operation along contour using circular interpolation*

Program	Comment
G90G54G71G40;	(Parameter settings)
G74X0.0Z0.0;	(Return tool at home position)
M03S200;	(Spindle Rotate clock wise at 200 rpm)
M07;	(Coolant On)
G00X31.0Z0.0;	(Rapid travel of tool near workpiece at safe position)
Z-0.5;	(depth of cut in z for facing operation)
G01X-1.0F0.5;	(Cutting in X with feed rate in mm/min)
Z0.0;	(Retrieval of tool)
G00X31.0;	(Rapid travel of tool in X)
X28.0;	(Position of cut in X for turning operation)
G01Z-10.4;	(Cutting in Z with feed rate in mm/min)
Z-8.0X29.0;	(Retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
X26.0;	(Position of cut in X for turning operation)
G01Z-10.0;	(Cutting in Z with feed rate in mm/min)
Z-8.0X29.0;	(retrieval of tool)
G00Z0.0;	(Rapid travel of tool in Z)
X25.0;	(Position of final cut in X for turning operation)
G01Z-10.0;	(Cutting in Z with feed rate in mm/min)
G4F1.0;	(Dwell for 1s)
G03X30.0Z-12.5CR=2.5;	(Clockwise circular interpolation with radius 2.5)
G00X31.0;	(retrieval of tool)
Z0.0;	(Rapid travel of tool in Z)
G74X0.0Z0.0;	(Return tool at home position)
M09;	(Coolant off)
M05;	(Spindle stop)
M30;	(End Program)