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

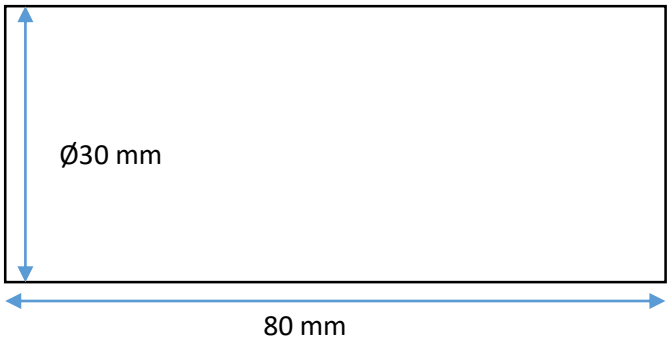

Fig 4.1

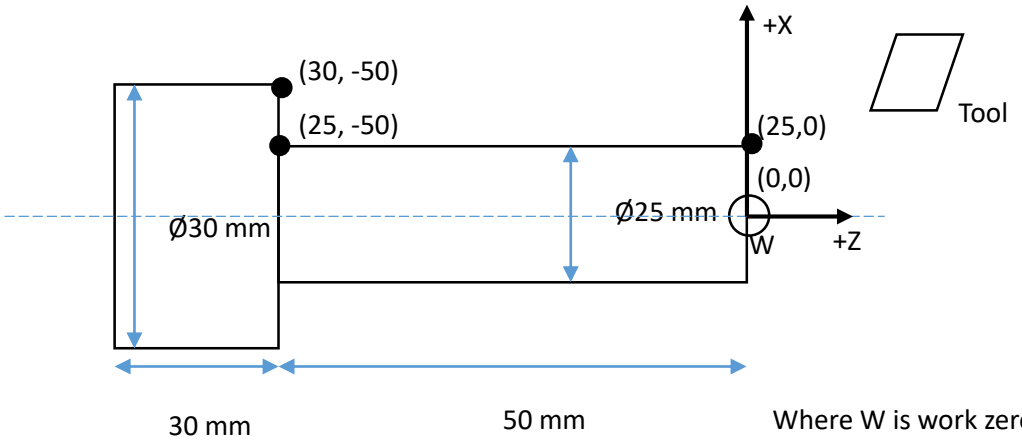

Fig 4.2

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

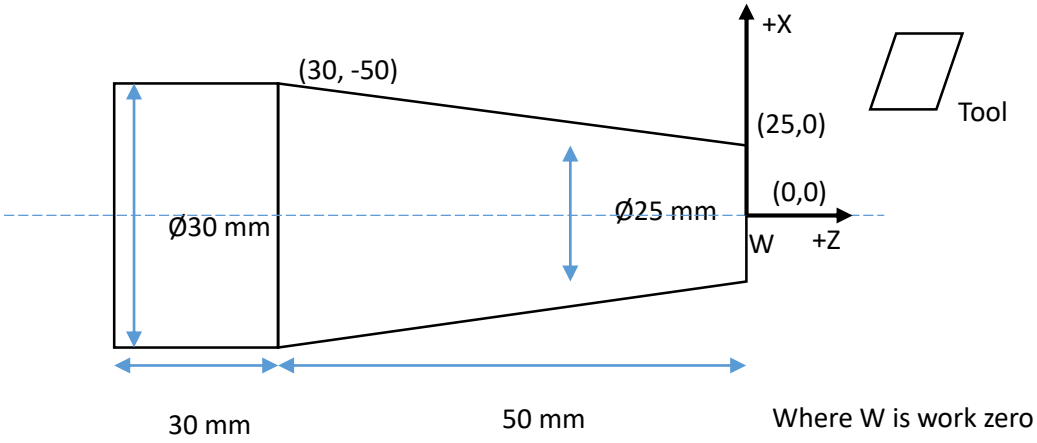

Fig 4.3

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

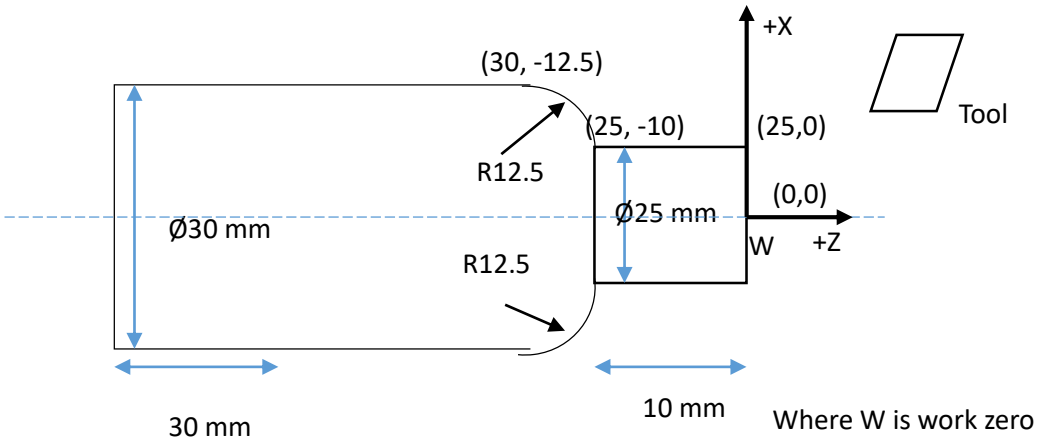

Fig 4.4

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; (Counter 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 of Program)