COMPUTER INTEGRATED MANUFACTURING LABORATORY

LAB CODE: 6ME4-21

FOR III B.TECH- VI SEM –MECHANICAL ENGINEERING

श्रेष्ठतमाय कर्मणे

DEPARTMENT OF MECHANICAL ENGINEERING

BHARTIYA INSTITUTE OF ENGINEERING & TECHNOLOGY

NEAR SANWALI CIRCLE, JAIPUR-BIKANER BYE-PASS, NH-11 PO-BAJAJ GRAM SANWALI,
SIKAR, RAJASTHAN 332001

CONTENTS

GENERAL INSTRUCTIONS

1. Students should wear the uniform and closed foot wear. Students inappropriately dressed for lab, at the instructor's discretion, are denied access.

2. Eating, drinking and smoking are prohibited in the laboratory at all times.

3. Never work in the laboratory without proper supervision by an instructor.

4. Never carry out unauthorized experiments. Come to the laboratory prepared. If you are unsure about what to do, please ask the instructor.

5. Except the scientific calculator, any other electronic devices are not permitted to use inside the Laboratory.

6. Any damage to any of the equipment/instrument/machine caused due to carelessness, the cost will be fully recovered from the individual (or) group of students.

LIST OF EXPERIMENTS

CNC LATHE OPERATIONS

FACING CYCLE

Experiment No: **Date:**

AIM   **:** To write a program to obtain the facing cycle in the CNC lathe.

EQUIPMENT REQUIRED: CNC Lathe

PROGRAM :

[BILLET X25 Z70] **(Size of the Specimen)**

G21 G98;

G28 U0W0;

M06 T1;**(Facing Tool)**

M03 S1200;

G00 X26 Z0;

G94 X0 Z-0.5 F50;

Z-1.0

Z-1.5

Z-2.0

Z-2.5

Z-3.0

Z-3.5
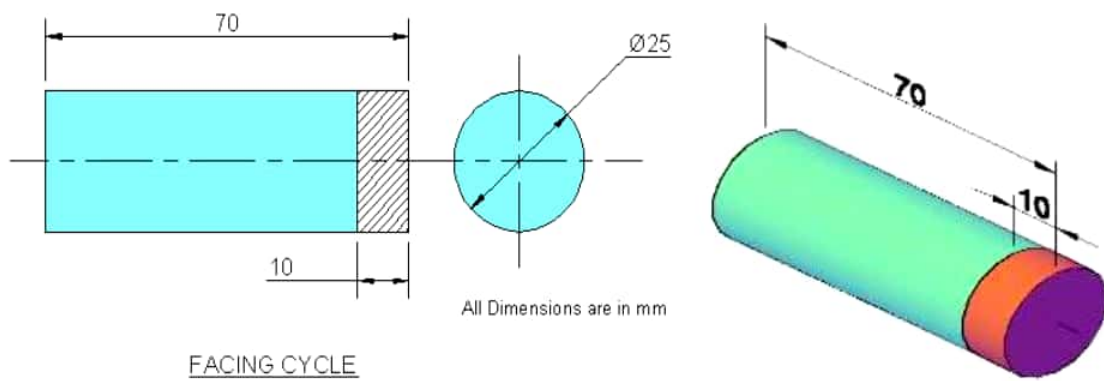
Z-4.0

Z-4.5

Z-5.0

Z-5.5

Z-6.0

Z-6.5

Z-7.0

Z-7.5

Z-8.0

Z-8.5

Z-9.0

Z-9.5

Z-10.0

G28 U0W0;

M05;

M30;

SKETCH   :

All Dimensions are in mm

FACING CYCLE

RESULT:

TURNING CYCLE

Experiment No: **Date:**

AIM: To write a program to obtain the turning cycle in the CNC lathe.

SOFTWARE REQUIRED: CNC Lathe Software with FANUC Language.

PROGRAM:

```
[BILLET X28 Z70] (Size of the Specimen)
G21 G98;
G28 U0W0;
M06 T1; (Facing Tool)
M03 S1000;
G00 X25 Z1;
G90 X24 Z-45 F50;
X23;
X22;
X21;
X20;
G28 U0W0;
M05;
M30;
```

SKETCH   :

All Dimensions are in mm

TURNING CYCLE

RESULT:

STEP TURNING

Experiment No: **Date:**

AIM: To write a program to obtain the step turning cycle in the CNC lathe.

SOFTWARE REQUIRED: CNC Lathe Software with FANUC Language.

PROGRAM:

    [BILLET X28 Z70] **(Size of the Specimen)**

    G21 G98;

    G28 U0W0;

    M06 T1; **(Facing Tool)**

    M03 S1000;

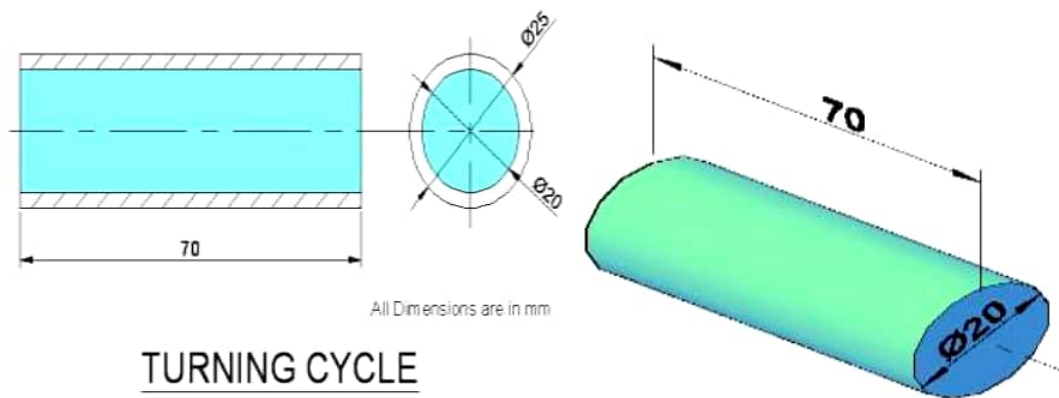   G00 X25 Z1;

    G90 X24 Z-45 F50;

    X23;

    X22;

    X21;

    X20;

    X19 Z-40;

    X18;

    X17;

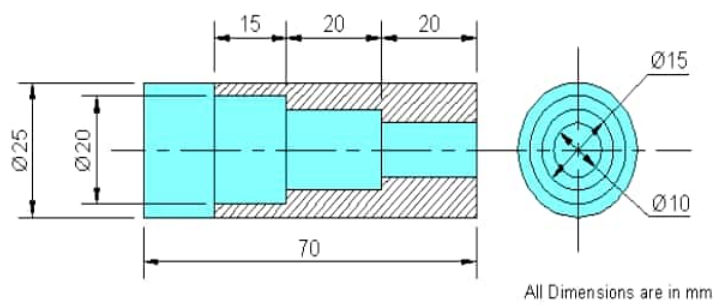   X16;

    X15;
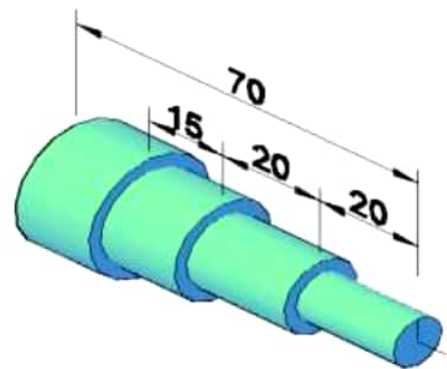
    X14 Z-20;

    X13;

    X12;

    X11;

    X10;

G28 U0W0;

M05;

M30;

SKETCH:

STEP TURNING CYCLE

All Dimensions are in mm

RESULT:

TAPER TURNING

Experiment No: **Date:**

AIM: To write a program to obtain the taper turning cycle in the CNC lathe.

SOFTWARE REQUIRED: CNC Lathe Software with FANUC Language.

PROGRAM:

[BILLET X28 Z70] **(Size of the Specimen)**

G21 G98;

G28 U0W0;

M06 T1; **(Turning Tool)**
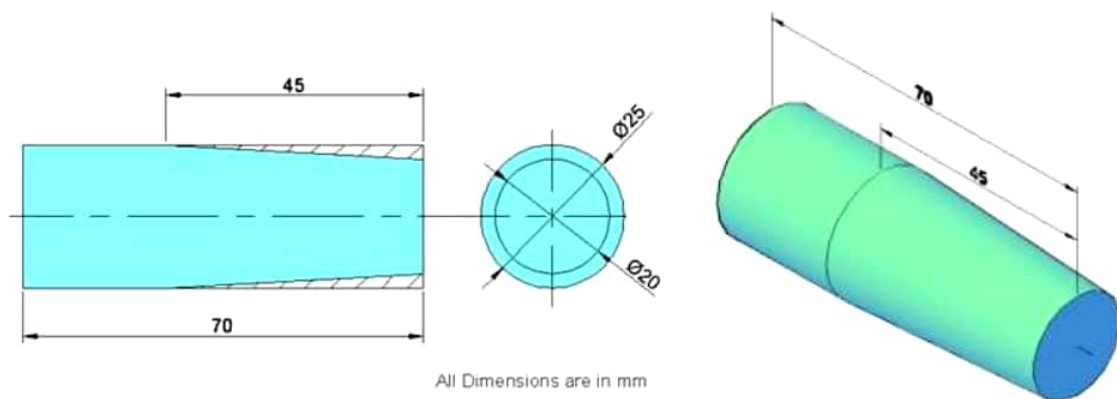
M03 S1000;

G00 X25 Z1;

G90 X24 Z-45 F50;

 X20 Z-45 F50;

G28 U0W0;

M05;

M30;

SKETCH:

45

70

Ø25

Ø20

70

45

All Dimensions are in mm

TAPER TURNING CYCLE

RESULT:

TURNING - CIRCULAR INTERPOLATION

Experiment No: **Date:**

AIM: To write a program to obtain the Circular Interpolation in the CNC lathe.

SOFTWARE REQUIRED: CNC Lathe Software with FANUC Language.

PROGRAM:

[BILLET X62 Z70] **(Size of the Specimen)**

G21 G98;

G28 U0W0;

M06 T1;**(Facing Tool)**

G00 X13 Z1;

G94 X0 Z-0.5 F50;

Z-1.0;

G71 U0.5 R1;

G71 P10 Q20 U0.1 W0.1 F50;

N10 G01 X05;

G01 Z-1.0;

G01 X15 Z-2;

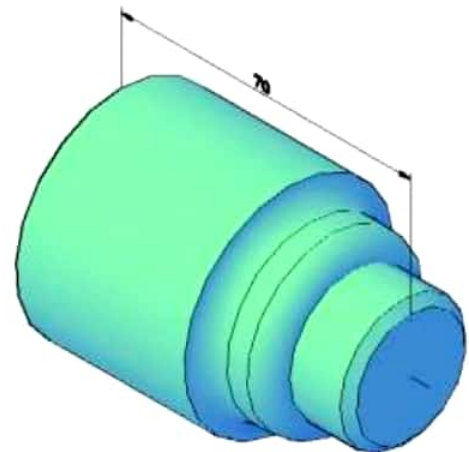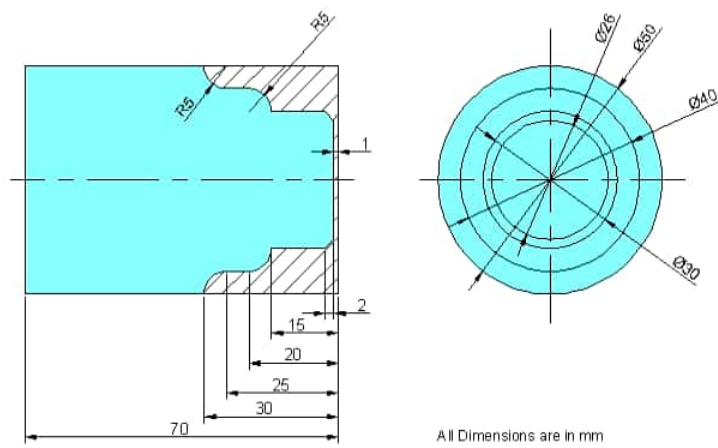G01 X15 Z-12;

G03 X20 Z-17 R5;

G01 X20 Z-22;

N20 G02 X25 Z-27 R5;

G28 U0W0;

M05;

M30;

SKETCH:

All Dimensions are in mm

CIRCULAR INTERPOLATION

RESULT:

DRILLING CYCLE

Experiment No: **Date:**

AIM: To write a program to obtain the Drilling Cycle in the CNC lathe.

SOFTWARE REQUIRED: CNC Lathe Software with FANUC Language.

PROGRAM:

[BILLET X25 Z70] **(Size of the Specimen)**

G21 G98;

G28 U0W0;

M06 T2;**(Center Drill Dia. 3 mm)**

M03 S1000;

G00 X0 Z1;

G74 R1;

G74 X0 Z-25 Q500 F30;

G28 U0 W0;

M06 T4; **(Drill Dia. 10 mm)**

G00 X0 Z1;

G74 R1;

G74 X0 Z-20 Q500 F30;

G28 U0 W0;

M06 T6; **(Drill Dia. 12 mm)**

G00 X0 Z1;

G74 R1;

G74 X0 Z-10 Q500 F30;

G28 U0W0;

M05;

M30;

SKETCH:

DRILLING CYCLE

All Dimensions are in mm
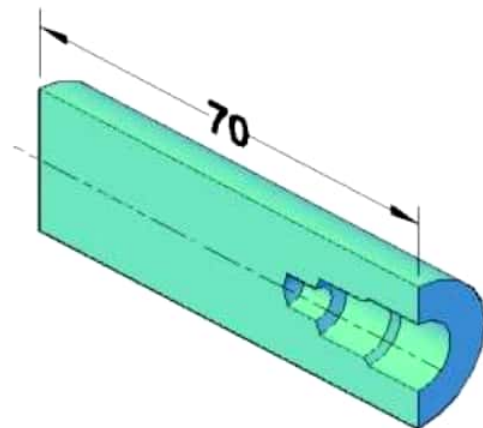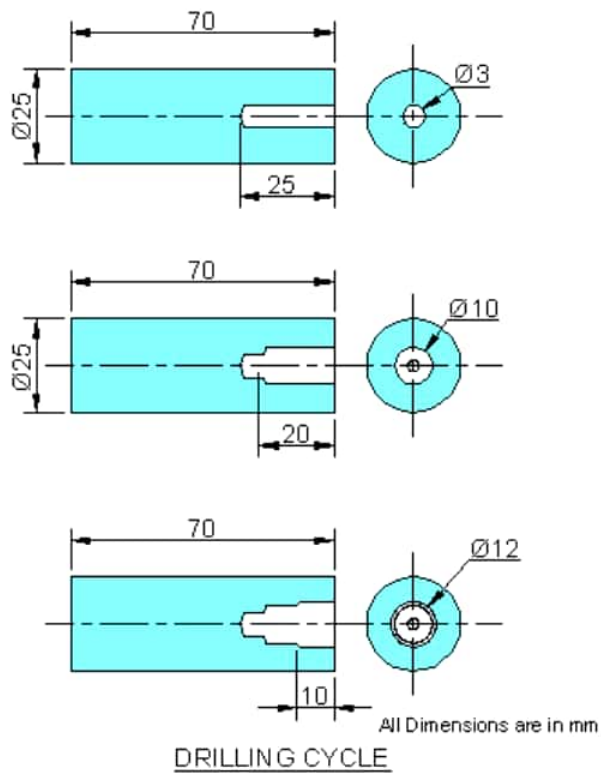
RESULT:

THREADING CYCLE

Experiment No: **Date:**

AIM: To write a program to obtain the Threading Cycle in the CNC lathe.

SOFTWARE REQUIRED: CNC Lathe Software with FANUC Language.

PROGRAM:

[BILLET X25 Z70] **(Size of the Specimen)**

G21 G98;

G28 U0W0;

M06 T1; **(Threading Tool)**

M03 S600;

G00 X26 Z0;

G76 P021560 Q50 R0.1;

G76 X23.774 Z-25 P613 Q100 F1;

G28 U0W0;

M05;

M30;

SKETCH:

All Dimensions are in mm

THREADING CYCLE

RESULT:

GROOVING CYCLE

Experiment No: **Date:**

AIM: To write a program to obtain the Grooving Cycle in the CNC lathe.

SOFTWARE REQUIRED: CNC Lathe Software with FANUC Language.

PROGRAM:

[BILLET X25 Z70] **(Size of the Specimen)**

G21 G98;

G28 U0W0;

M06 T1;(**Facing Cycle**)

M03 S1000;

G00 X26 Z0;

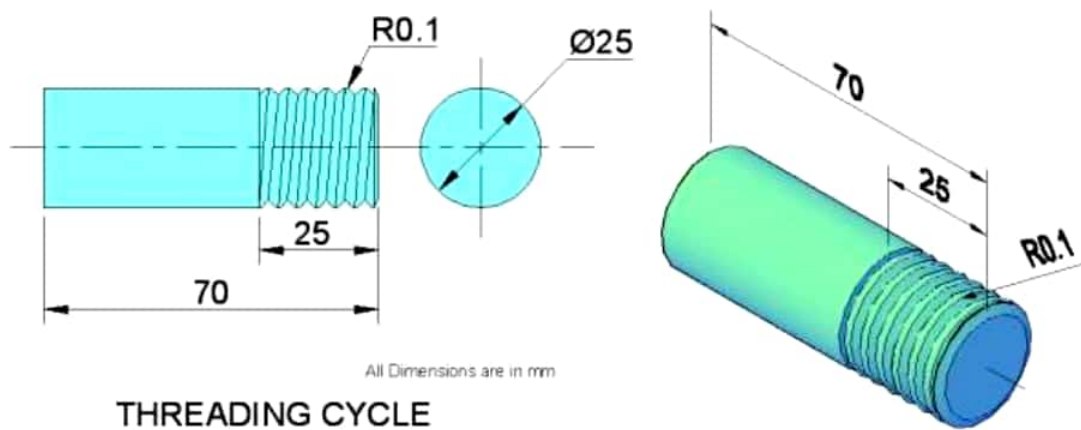G94 X0 Z-0.5 F50;

 Z-1.0;

Z-1.5;

G28 U0W0;

G00 X26Z0;

G90 X26Z0;

G90 X25 Z-36.5 F30;

X24;

X23;

G28 U0W0;

M06 T3; **(Grooving Tool)**

G00 X24 Z-13.5;

G75 R1;

G75 X70 Z-31.5 P200 Q1000 F30;

G28 U0W0;

M05;

M30;

SKETCH:

GROOVING CYCLE
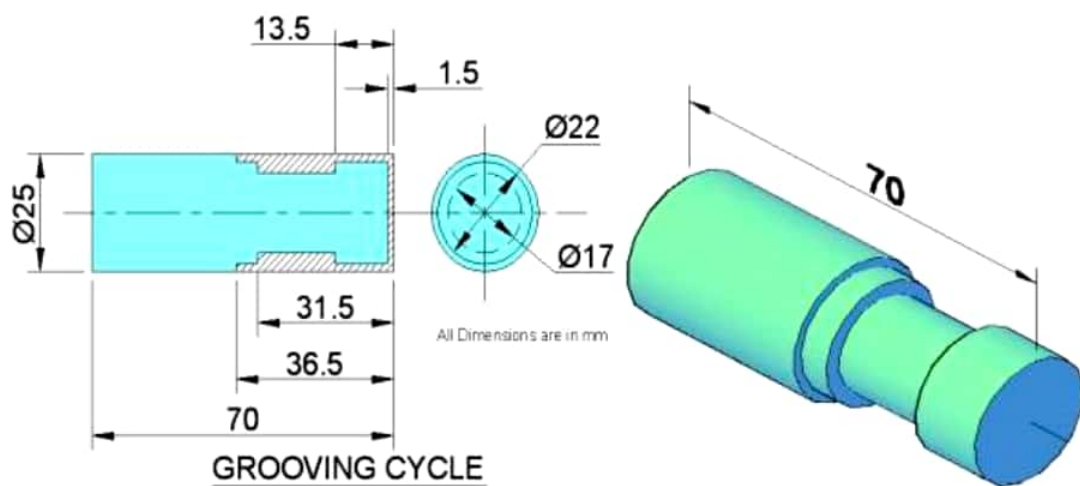
RESULT:

CNC MILLING OPERATIONS

LINEAR AND CIRCULAR INTERPOLATION

Experiment No: **Date:**

AIM: To write a program to obtain linear and circular interpolation on the given work piece.

SOFTWARE REQUIRED: CNC XMILL Software with FANUC Language.

PROGRAM:

```
G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T06
M03 S1300
G90 G00 X0 Y0 Z5
G90 G01 X0 Y0
X30
G03 X54 R12
G01 X82
G02 X108 R13
G01 X123
X80 Y45
```

24

X40

Y75

G03 X35 Y80 R5

G01 X20

G03 X0 Y80 R10

G01 Y0

M30

SKETCH:

LINEAR AND CIRCULAR INTERPOLATION

RESULT:

ENGRAVING OF LETTERS

Experiment No: **Date:**

AIM: To write a program to engrave the letters "SVCET" on the given work piece.
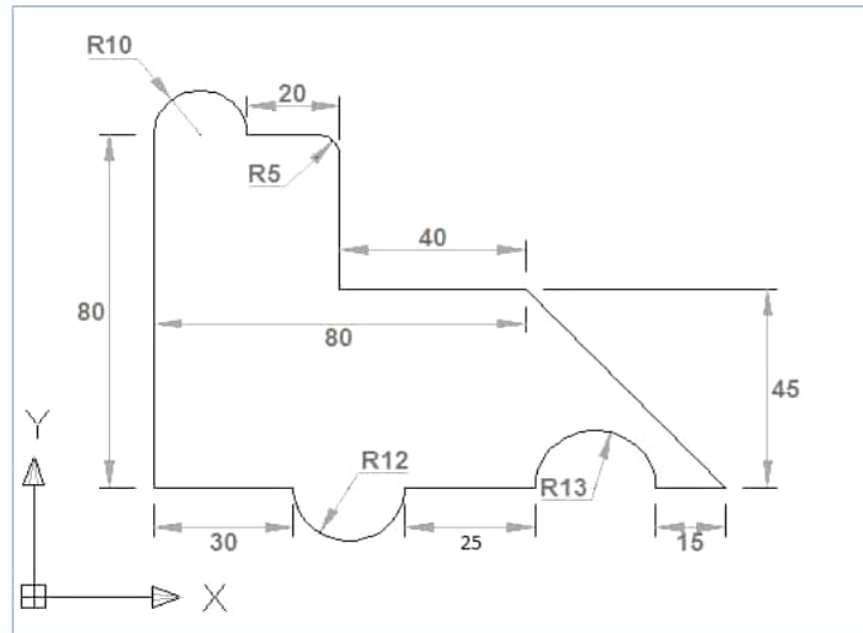
SOFTWARE REQUIRED: CNC XMILL Software with FANUC Language.

PROGRAM:

```
G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T06
M03 S1300
G90 G00 X0 Y0 Z5
        [S]
G00 X2 Y30
G01 Z-1 F60
G01 X10 Y30
G03 X15 Y35 R5
G01 X15 Y 37.5
G03 X10 Y42.5 R5
```

G01 X07 Y42.5

G02 X2 Y47.5 R5

G01 X2 Y50

G02 X7 Y55 R5

G01 X15 Y55

G00 Z2

       [V]

G00 X20 Y55

G01 Z-1 F60

G01 X27.5 Y30

G01 X33 Y55

G00 Z2

       [C]

G00 X51 Y55

G01 Z-1 F60

G01 X43 Y55

G03 X38 Y50 R5

G01 X38 Y35

G03 X43 Y30 R5

G01 X51 Y30

G00 Z2

       [E]

G00 X69 Y55

G01 Z-1F60

G01 X56 Y55

G01 X56 Y42.5

G01 X69 Y42.5

G01 X56 Y42.5

G01 X56 Y30

G01 X69 Y30

G00 Z2

       [T]

G00 X81.5 Y30

G01 Z-1 F60

G01 X81.5 Y55

G01 X74 Y55

G01 X87 Y55

G00 Z2

G91 G28 Z0

G28 X0Y0

M05

M30

SKETCH:
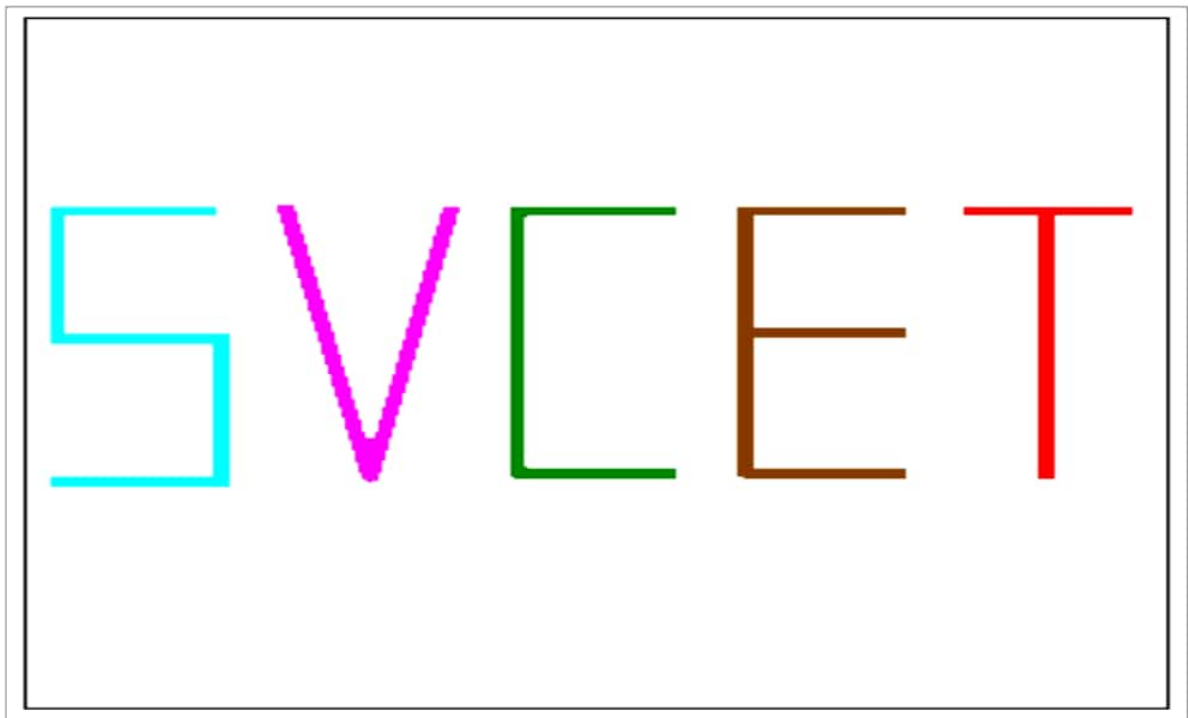
ENGRAVING OF LETTERS

RESULT:

MIRRORING

Experiment No: **Date:**

AIM: To write a program to perform the mirroring operation.

SOFTWARE REQUIRED: CNC XMILL Software with FANUC Language.

PROGRAM:

```
G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T06
M03 S1500
G90 G00 X0 Y0 Z5
M98 P4646
M70
M98 P4646
M80
M70
M71
M98 P4646
```

M80

M81

M71

M98 P4646

M81

G91 G28 Z0

G28 X0 Y0

M05

M30

O4646

G00 X10 Y10

G01 Z-1.5 F80

G01 X30 Y10

G03 X10 Y30 R20

G01 X10 Y10

G00 Z5

M99

SKETCH:

MIRRORING

RESULT:

ROTATION

Experiment No: **Date:**

AIM: To write a program to perform the Rotation operation on the given work piece.

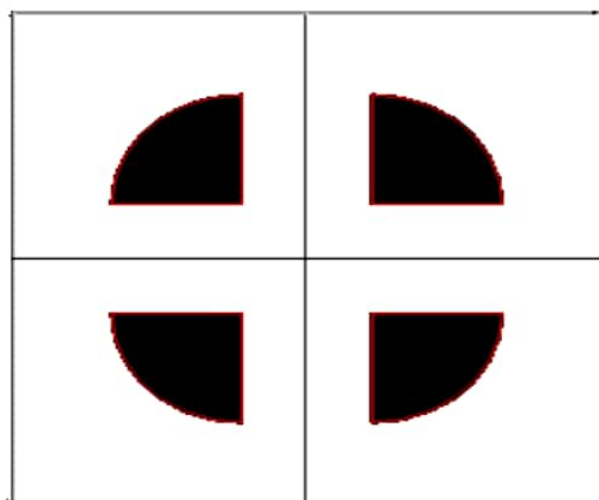**SOFTWARE REQUIRED:** CNC XMILL Software with FANUC Language.

PROGRAM:

```
G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T06
M03 S1300
G90 G00 X0Y0 Z5
M98 P1234
G68 X0Y0 R90
M98 P1234
G68 X0Y0 R180
M98 P1234
G68 X0Y0 R270
```

G69

G68 X0Y0 R360

G69

G91 G28 Z0

G28 X0Y0

M05

M30

O1234

G00 X0Y0

G01 Z-1 F60

G01 X20 Y-10

G01 X40 Y0

G01 X20 Y10

G01 X0 Y0

G00 Z5

M99

SKETCH:

ROTATION

RESULT:

CIRCULAR POCKETING

Experiment No: **Date:**

AIM: To write a program to perform the Circular pocketing operation on the given work piece.

SOFTWARE REQUIRED: CNC XMILL Software with FANUC Language.

PROGRAM:

G21 G94

G91 G28 Z0

G28 X0Y0

M06 T05

M03 S1200

G90 G00X0 Y0 Z5

G170 R0 P0 Q1 X0 Y0 Z-10 I0.5 J0.1 K20

G171 P50 S1500 R60 F60 B1800 J100

G170 R0 P1 Q1 X0 Y0 Z-10 I0 J0 K20

G171 P50 S1500 R60 F60 B1800 J100

G00 Z5

M05

G91 G28 Z0

G28 X0 Y0

M30

SKETCH:

CIRCULAR POCKETING

RESULT:

RECTANGULAR POCKETING

Experiment No: **Date:**

AIM: To write a program to perform the Rectangular pocketing operation on the given work piece.

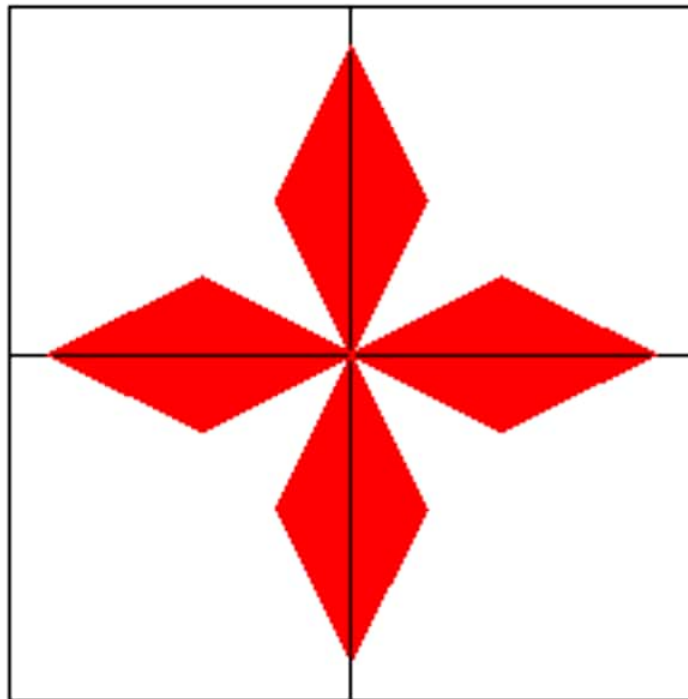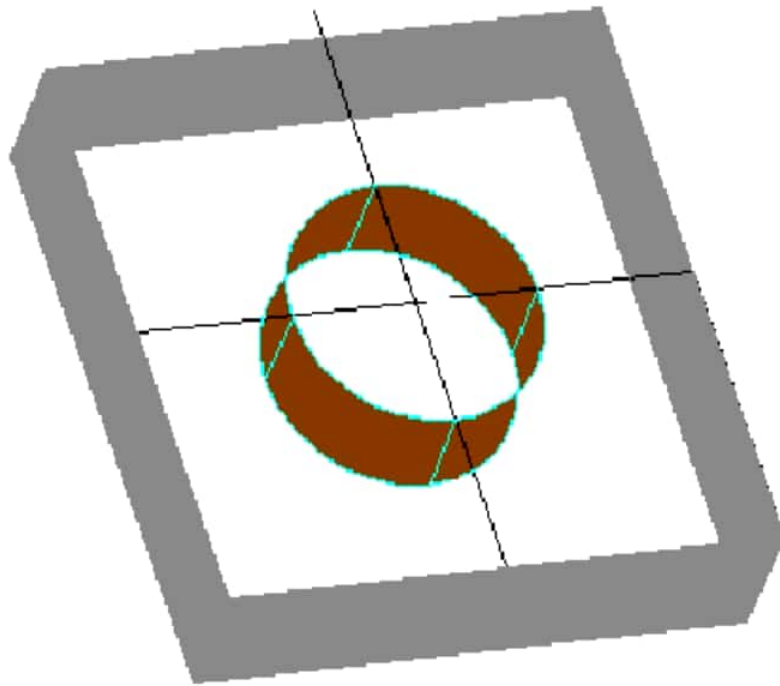**SOFTWARE REQUIRED:** CNC XMILL Software with FANUC Language.

PROGRAM:

G21 G94

G91 G28 Z0

G28 X0 Y0

M06 T04

M03 S1200

G90 G00 X0 Y0 Z5

G172 I30 J30 K0 P0 Q1 R0 X-15 Y-15 Z-10

G173 I0.1 K0.1 P50 S1500 R60 F50 B1800 J100

G172 I30 J30 K0 P1 Q1 R0 X-15 Y-15 Z-10

G173 I0 K0 P50 S1500 R60 F50 B1800 J100

G00 Z5

M05

G91 G28 Z0

G28 X0 Y0

M30

SKETCH:
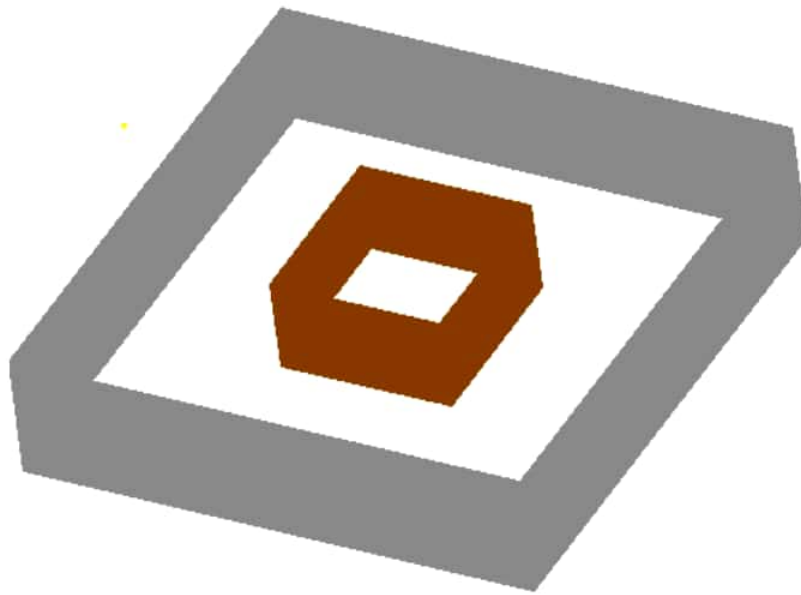
RECTANGULAR POCKETING

RESULT:

ROBOT PROGRAMING

ROBOT PROGRAMING 1

Experiment No: **Date:**

AIM: To write and execute a robot program to perform a pick and place operation.

SOFTWARE REQUIRED: Pick and place robot integrated with system

PROGRAM:

```
SPEED 80
GRIPPER OPEN
LABEL A
LET A = 0
LABEL MTAB
SPEED 80
JOINT A1= 156.25 A2= 90 A3= 90 A4= 95 A5= 0
JOINT A1= 156.25 A2= 61.79 A3= 119.67 A4= 95 A5= 0
JOINT A1= 156.25 A2= 37.77 A3= 111.6 A4= 153.91 A5= 0
JOINT A1= 156.25 A2= 37.77 A3= 111.6 A4= 153.91 A5= 177.27
GRIPPER CLOSE
JOINT A1= 156.25 A2= 46.65 A3= 110.9 A4= 185.56 A5= 177.27
JOINT A1= 156.25 A2= 68.06 A3= 91.41 A4= 103.9 A5= 177.27
JOINT A1= 0 A2= 90 A3= 90 A4= 95 A5= 0
ADD A = A + 2
IF A < 3 JUMP MTAB
```

RESULTS

ROBOT PROGRAMING 2

Experiment No: **Date:**

AIM: To write and execute a robot program to perform a repetitive pick & place operation.

SOFTWARE REQUIRED: Pick and place robot integrated with system

PROGRAM:

```
SPEED 80
GRIPPER OPEN
LABEL A
LET A = 0
LABEL MTAB
SPEED 80
JOINT A1= 156.25 A2= 90 A3= 90 A4= 95 A5= 0
JOINT A1= 156.25 A2= 61.79 A3= 119.67 A4= 95 A5= 0
JOINT A1= 156.25 A2= 37.77 A3= 111.6 A4= 153.91 A5= 0
 JOINT A1= 156.25 A2= 37.77 A3= 111.6 A4= 153.91 A5= 177.27
GRIPPER CLOSE
JOINT A1= 156.25 A2= 46.65 A3= 110.9 A4= 185.56 A5= 177.27
JOINT A1= 156.25 A2= 68.06 A3= 91.41 A4= 103.9 A5= 177.27
 JOINT A1= 0 A2= 90 A3= 90 A4= 95 A5= 0
ADD A = A + 2
IF A < 3 JUMP MTAB
```

RESULT:

APPENDIX

1.   **WORD DETAILS**

Although the control will, in general, accept part programming words in any sequence, it is recommended that the following word order for each block is used.

N; G; X or U; Z or W; I; K; F; S; T;

O: PROGRAM NUMBER

The "O" followed by a 4 digit numeral value is used to assign a program number.

Example: O1002

N: SEQUENCE NUMBER

The N word may be omitted. When programmed, the sequence number following the N address is a four digit numerical value and is used to identify a complete block of information. Although ascending, descending, or duplicate numbering is allowed, it is best to program in ascending order in increments of 10. This allows for future editing and simplified sequence number search.

G: PREPARATORY COMMAND:

The two digit G command is programmed to set up the control to perform an automatic machine operation. A full list of G codes are given, one G word from each modal group and one non modal G word can be programmed on the same block.

Example:

     Valid N 100 G00 <u>G40 G41</u> G90 G95

    *G40 & G41 are from the same group.

A retained G word (Modal) from one group remains active until another G word from the same group is programmed.

One-shot G word (Non-Modal) must be programmed in every block when required.

2.   **AXIS DEFINITION**

Z AXIS

The Z axis is along a line between the spindle and the tailstock, or the center line of rotation of the spindle. Minus (-) movements of the tool are left toward the head stock; positive (+) movements are right towards the tailstock.

X AXIS

The X axis is 90 degrees from the Z axis (perpendicular to the Z axis). Minus (-) movements of the tool are toward the center-line of rotation, and positive (+) movements are away from the center –line of rotation.

X: X AXIS COMMAND

The X word is programmed as a diameter which is used to command a change in position perpendicular to the spindle center-line.

U: X AXIS COMMAND

The U word is an incremental distance (diameter value) which is used to command a change in position perpendicular to the spindle center-line. The movement is the programmed value.

Z: Z AXIS COMMAND

The Z word is an absolute dimension which is used is used to command a change in position parallel to the spindle center-line.

W: Z AXIS COMMAND

The W word is an incremental distance which is used to command a change of position parallel to the spindle center-line.

Do not program X & U or Z & W in the same block. If an X axis command calls for no movement it may be omitted.

X, U or P: DWELL

The X word is used with G04 to command a dwell in seconds.

The P word is used with G04 to command a dwell in milliseconds.

I WORD

For arc programming (G02 or G03) , the K Value (with sign) is programmed to define the incremental distance parallel to the Z axis, between the start of the arc and the arc center.

K WORD

For arc programming (G02 or G03), the K value (with sign) is programmed to define the incremental distance parallel to the Z axis, between the start of the arc and the arc center.

The maximum arc for I & K programming is limited to the quadrant. If I or K is zero, it must be omitted.

F WORD

a)   In G99 mode the F word is used to command feed/rev.

b)   In G98 mode the F word is used to command feed/min.

c)   In G32 mode the F word specifies the lead (pitch) of the thread.

P WORD

a)   Used in automatic cycles to define the first block of a contour.

b)   Used with M98 to define a subroutine number.

Q WORD

Q words are used in automatic cycles to define the last block of a contour.

R WORD

For circular interpolation (G02 or G03) the R word defines the arc radius from the center of the tool nose radius (G40 active) - or the actual radius required (G41/ G42 active).

S WORD

a) In the constant surface speed mode (G96) the four digit S word is used to command the required surface speed in either feet or meters per minute.

b) In the direct R.P.M mode (G97), the four digit S word is used to command the spindle speeds incrementally, in R.P.M between the ranges available for the machine.

c) Prior to entering constant surface speed mode (G96) the S word is used to specify a speed constraint, the maximum speed you wish the spindle to run at. To set this restraint the S word is programmed in conjunction with the G50 word.

T WORD

The T words are used in conjunction with "M06". Those are used to call up the required tool on an automatic indexing turret machine, and to activate its tool offsets.

M WORD

An M word is used to initiate auxiliary functions particular to the machine. One M code can be programmed with in one program block together with other part program information.

3. G-CODES LISTING FOR DENFORD FANUC LATHES

G00   Positioning (Rapid Traverse)
G01   Liner Interpolation (Feed)
G02   Circular Interpolation CW
G03   Circular Interpolation CW
G04   Dwell
G10   Offset Value Setting By Program
G20   Inch Data Input
G21   Metric Data Input
G22   Stored Stroke Check On
G23   Stored Stroke Check Off
G27   Reference Point Return Check
G28   Reference Point Return
G29   Return from Reference Point
G30   Return to 2nd Reference Point
G31   Skip Function
G32   Thread Cutting
G34   Variable Lead Thread Cutting
G36   Automatic Tool Compensation X
G37   Automatic Tool Compensation Z
G40   Tool Nose Radius Compensation cancels
G41   Tool Nose Radius Compensation Left

G42   Tool Nose Radius Compensation Right

G50   Work Co-ord. Change/Max. Spindle Speed setting

G65   Macro call

G66   Macro Modal Call Cancel

G67   Macro Modal Call Cancel

G70   Finishing Cycle

G71   Stock Removal in Turning

G72   Stock Removal in Turning

G73   Pattern Repeating

G74   Peck Drilling in Z Axis

G75   Grooving in X Axis

G76   Thread Cutting Cycle

G90   Cutting Cycle A

G92   Thread Cutting Cycle

G94   Cutting Cycle B

G96   Constant surface Speed Control

G97   Constant Surface Speed Control Cancel

G98   Feed per Minute

G99   Feed per Revolution

Note: - Not All G Codes Apply To Each Machine.

4.   M- CODE LIST FOR DENFORD FANUC LATHES

All M Codes marked with an asterisk will be executed at the end of a block (i.e., after the axis movement).

M00   PROGRAM STOP

M01   OPTIONAL STOP

M02   PROGRAM RESET

M03   SPINDLE FORWARD

M04   SPINDLE REVERSE

M05   SPINDLE STOP

M06   AUTO TOOL CHANGE

M07   COOLANT "B" ON

M08   COOLANT "A" ON

M09   COOLANT OFF

M10   CHUCK OPEN

M11   CHUCK CLOSE

M13   SPINDLE FORWARD & COOLANT ON

M14   SPINDLE REVERSE & COOLANT ON

M15   PROGRAM INPUT USING."MIN P" (SPECIAL FUNCTION)

M16   SPECIALTOOL CALL (TOOL CALL IGNORES TURRET)

M19	SPINDLE ORIENTATE
M20	SPINDLE INDEX A
M21	SPINDLE INDEX 2A
M22	SPINDLE INDEX 3A
M23	SPINDLE INDEX 4A
M25	QUILL EXTEND
M26	QUILL RETRACT
M29	SELECT "DNC" MODE
M30	PROGRAM RESET & REWIND
M31	INCREMENT PARTS COUNTER
M37	DOOR OPEN TO STOP
M38	DOOR OPEN
M39	DOOR CLOSE
M40	PARTS CATCHER EXTEND
M41	PARTS CATCHER RETRACT
M43	SWARF CONVEYOR FORWARD
M44	SWARF CONVEYOR REVERSE
M45	SWARF CONVEYOR STOP
M48	LOCK % FEED AND % SPEED AT 100%
M49	CANCEL M48 (DEFAILT)
M50	WAIT FOR AXIS IN POSITION SIGNAL (CANCELS CONTINUOUS PATH)
M51	CANCEL M50 (DEFAILT)
M52	PULL-OUT IN THREADING = 90 DEGRESS (DEFAILT)
M53	CANCEL M52
M54	DISABLE SPINDLE FLUCTUATION TESTING (DEFAULT)
M56	SELECT INTERNAL CHUCKING (FROM PLC EDITION "F")
M57	SELECT EXTERNAL CHUCKING (FROM PLC EDITION "F")
M62	AUX.1 ON
M63	AUX.2 ON
M64	AUX.1 OFF
M65	AUX.2 OFF
M98	SUB PROGRAM CALL
M99	SUB PROGRAM END