COMPUTER AIDED NUMERICAL CONTROL LABORATORY

LAB MANUAL

Subject Code	**:**	**AME115**
Regulations	**:**	**IARE-R16**
Class	**:**	**IV Year I Semester (ME)**

Prepared by
Mr. M Prashanth Reddy
Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING
INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal – 500 043, Hyderabad

Program Outcomes	
PO1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences
PO4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
Program Specific Outcomes	
PSO1	**Professional Skills:** To produce engineering professional capable of synthesizing and analyzing mechanical systems including allied engineering streams.
PSO2	**Software Engineering Practices:** An ability to adopt and integrate current technologies in the design and manufacturing domain to enhance the employability.
PSO3	**Successful Career and Entrepreneurship:** To build the nation, by imparting technological inputs and managerial skills to become technocrats.

INDEX

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES

Exp. No.	Experiment	Program Outcomes Attained	Program Specific Outcomes Attained
1	Introduction To Computer Numerical Control I	PO1, PO2	PSO1, PSO3
2	Introduction To Computer Numerical Control II	PO1, PO2	PSO1, PSO3
3	CNC Milling I	PO1, PO2, PO4	PSO1, PSO3
4	CNC Milling II	PO1, PO2, PO4	PSO1, PSO3
5	CNC Milling III	PO1, PO2, PO4	PSO1, PSO3
6	CAM Software I	PO1, PO2, PO4	PSO2
7	CAM Software II	PO1, PO2, PO4	PSO2
8	CNC Turning I	PO1, PO2, PO4	PSO1, PSO3
9	CNC Turning II	PO1, PO2, PO4	PSO1, PSO3
10	CAM Software III	PO1, PO2, PO4	PSO2
11	3D Printing	PO1, PO2, PO4	PSO1, PSO3
12	Industry–Institute Interaction	PO1, PO2	PSO1, PSO3

COMPUTER AIDED NUMERICAL CONTROL LABORATORY

Course Overview:

In this laboratory the students learn the fundamentals of numerical control (NC) technology, programming of computer numerical control (CNC) machines in NC codes and APT language and with CAM systems. Students also gain experience in NC postprocessors and distributed numerical control, operation of CNC lathe and milling machines, and programming and machining complex engineering parts.

Course Out Comes:

CO 1	Understand the features and specifications of CNC and 3D printing machines.
CO 2	Develop the process planning sheets and tool layouts
CO 3	Use the CAM software and prepare CNC part programs.
CO 4	Execute the part program and machine the component as per the production drawing.

INSTRUCTIONS TO THE STUDENTS

1. Students are required to attend all labs.
2. Students should work individually in the hardware and software laboratories.
3. Students have to bring the lab manual cum observation book, record etc along with them whenever they come for lab work.
4. Should take only the lab manual, calculator (if needed) and a pen or pencil to the work area.
5. Should learn the prelab questions. Read through the lab experiment to familiarize themselves with the components and assembly sequence.
6. Should utilize 3 hour's time properly to perform the experiment and to record the readings. Do the calculations, draw the graphs and take signature from the instructor.
7. If the experiment is not completed in the stipulated time, the pending work has to be carried out in the leisure hours or extended hours.
8. Should submit the completed record book according to the deadlines set up by the instructor.
9. For practical subjects there shall be a continuous evaluation during the semester for 30 sessional marks and 70 end examination marks.
10. Out of 30 internal marks, 20 marks shall be awarded for day-to-day work and 10 marks to be awarded by conducting an internal laboratory test.

EXPERIMENT I

INTRODUCTION TO COMPUTER NUMERICAL CONTROL I

Aim: Know the basic functions of a machine tool, concept of numerical control, historical development, definition, advantages of CNC machine tools

Computer-aided Manufacturing (CAM) is the term used to describe the use of computerized systems to control the operations at a manufacturing plant. These computerized systems assist manufacturers in various operations such as planning, transportation, management, and storage. CAM helps manufacturers improve their time to market capabilities, and create precise dimensions.

The History of Computer Numerical Control (CNC)

Computer numerical control is a modern concept in the manufacturing and production industries. However, the concept of CNC harkens back to the basic idea of NC, or numerical control.

The idea of numerical control started when the automation of machine tools originally incorporated specific concepts of programmable logic. In the beginning, the first NC machines were built back in the 1940s. Slightly more advanced machines came along in the 1950s. These manufacturing machines were constructed based on existing tools that were modified with motors designed to move the controls of the machine. These controls followed specific points that were fed into the machine on punched tape. These early mechanisms were soon improved with both analog and digital computers. The introduction of computer technology into the concept of numerical control led to what we now know as computer numerical control.

Brief Introduction to Computer-aided Manufacturing

As a process, CAM is used after Computer-aided Design (CAD) or Computer-aided Engineering (CAE). The model designed using CAD is sometimes used as the CAM input. This is why it is referred as CAD-CAM. The functions of this combination are divided into two main categories:

Manufacturing Planning: In this process, the computer delivers information for production planning as well as management. This may include:

- Computer Aided Process Planning (CAPP)
- Computer Assisted NC Part Programming
- Computerized Machinability Data System
- Work Standards Development
- Inventory and Production Planning

Manufacturing Control: In the process, the computer is used to manage and control the physical operations of the manufacturing plant. These may include:

- Shop Floor Controlling
- Process Monitoring and Controlling
- Inventory Controlling

- Production Delivery Controlling

Benefits of CAD-CAM in Manufacturing

Today, every manufacturing plant uses at least a type of CAD-CAM system to control their operations. Here are the various advantages they can avail by using these software applications.

Improves Machining Capabilities: By using a CAD-CAM system, manufacturers can improve their machining capabilities. For example, when a manufacturer takes up a complex 3-axis machining task, they rely on the combination software to create a tool path for machining projects such as molding. The CAM system automates the process, and makes it easier for manufacturers to complete the project in time.

Improves Client Accessibility: The CAD-CAM software allows manufacturers to receive CAD files from their customers. After receiving these files, they can set up the machining tool path, and perform simulations, which help them calculate the machining cycle times. The software allows manufacturers to minimize errors, execute projects easily, and deliver products to the market within a shorter turnaround time.

Improve Productivity of CNC Machines: Most CAM-CAD systems provide high–speed machine tool paths, which help manufacturers minimize their cycle times, reduce tool and machine wear. High-speed tool paths enable manufacturers to improve their cutting quality and accuracy. This type of high- speed machining helps improve the productivity of the CNC machine by more than 50%.

Reduce Material Wastage: As CAM–CAD software feature simulation features, it helps a manufacturer to visually inspect the process of machining. This allows him to capture tool gouges, and collisions at an early phase. This feature contributes to the overall productivity of a manufacturing set up. This also helps them eliminate mistakes, as well as reduce material wastage.

Viva Questions:

1. What is CAM? Explain.
2. What are the advantages of CAM?
3. What is the role of CAM in mass production?
4. What is NC machine? Explain the programming methods in NC machine?

EXPERIMENT II

INTRODUCTION TO COMPUTER NUMERICAL CONTROL II

AIM: To Know Evolution of CNC, advantages of CNC, limitations of CNC, features of CNC, machine control unit (MCU) for CNC, classification of CNC machine tools; CNC machining centers: classification, features of CNC machining centers

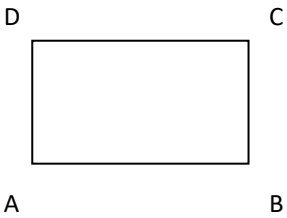
Co-ordinate system:

In order for the part programmer to plan the sequence of positions, moments, the cutting tool. Machine to the WIP, it is memory to establish a standard axis system by which the relative positions can be specified. Two axes "X & Y" are defined in the plane of the table, the 'z' axis in perpendicular. In this plane of the table the vertical motion of the spindle controls the 'z' direction. The positive and negative directions motion of the tool.

Programming methods

1) Incremental method

2) Absolute method



1) Incremental Method:

In this method, every point is considered as origin from this point; the values are calculated, for example

Point A = (0, 0)

Point B = (20, 0)

Point C = (0, 10)

Point D = (-20, 0)

2) Absolute method:

In this absolute system, the set point is considered as a reference point as from that point, all the values are calculated, for example

Point A = (0, 0)

Point B = (20, 0)

Point C = (20, 10)

Point D = (0, 10)

Programming methods:

In CNC machines program are programmed by two methods.

1) Manual part programming

2) Computer assisted part programming

1) Manual part programming:

To prepare a part program using the manual method, the programmer writes the machining instruction is must be hence, menu script the instruction is must be prepared in a very precise manner because the typist prepares the NC type directory from the Manu script some in various form expending on the machine tool and tape format used.

2) Computer assisted part programming:

In the more complicated point and in contour application using manual part programming because an extremely tedious basic and subject to errors. It is must more appropriate to employ the high speed digital computer to assist the part programming languages system have been developed to perform automatically most of the calculation which the programmer would otherwise be forced to do

PREPARATORY FUNCTIONS (G-CODE):

Preparatory functions are used for cutting operations like facing, turning, thread cutting, drilling, etc.,

MISCELLANEOUS FUNCTIONS (M-CODE):

Miscellaneous functions are used for other than cutting operations like spindle ON/OFF, coolant ON/OFF, tool change, etc

Preparatory Functions (G-Codes):

G00 - Positioning at rapid speed; Mill and Lathe

G01 - Linear interpolation (machining a straight line); Mill and Lathe

G02 - Circular interpolation clockwise (machining arcs); Mill and Lathe

G03 - Circular interpolation, counter clockwise; Mill and Lathe

G04 - Mill and Lathe, Dwell

G09 - Mill and Lathe, Exact stop

G10 - Setting offsets in the program; Mill and Lathe

G12 - Circular pocket milling, clockwise; Mill

G13 - Circular pocket milling, counterclockwise; Mill

G17 - X-Y plane for arc machining; Mill and Lathe with live tooling

G18 - Z-X plane for arc machining; Mill and Lathe with live tooling

G19 - Z-Y plane for arc machining; Mill and Lathe with live tooling

G20 - Inch units; Mill and Lathe

G21 - Metric units; Mill and Lathe

G27 - Reference return check; Mill and Lathe

G28 - Automatic return through reference point; Mill and Lathe

G29 - Move to location through reference point; Mill and Lathe

G31 - Skip function; Mill and Lathe

G32 - Thread cutting; Lathe

G33 - Thread cutting; Mill

G40 - Cancel diameter offset; Mill. Cancel tool nose offset; Lathe

G41 - Cutter compensation left; Mill. Tool nose radius compensation left; Lathe

G42 - Cutter compensation right; Mill. Tool nose radius compensation right; Lathe

G43 - Tool length compensation; Mill

G44 - Tool length compensation cancel; Mill (sometimes G49)

G50 - Set coordinate system and maximum RPM; Lathe

G52 - Local coordinate system setting; Mill and Lathe

G53 - Machine coordinate system setting; Mill and Lathe

G54~G59 - Work piece coordinate system settings #1 to #6; Mill and Lathe

G61 - Exact stop check; Mill and Lathe

G65 - Custom macro call; Mill and Lathe

G70 - Finish cycle; Lathe

G71 - Rough turning cycle; Lathe

G72 - Rough facing cycle; Lathe

G73 - Irregular rough turning cycle; Lathe

G73 - Chip break drilling cycle; Mill

G74 - Left hand tapping; Mill

G74 - Face grooving or chip break drilling; Lathe

G75 - OD groove pecking; Lathe

G76 - Fine boring cycle; Mill

G76 - Threading cycle; Lathe

G80 - Cancel cycles; Mill and Lathe

G81 - Drill cycle; Mill and Lathe

G82 - Drill cycle with dwell; Mill

G83 - Peck drilling cycle; Mill

G84 - Tapping cycle; Mill and Lathe

G85 - Bore in, bore out; Mill and Lathe

G86 - Bore in, rapid out; Mill and Lathe

G87 - Back boring cycle; Mill

G90 - Absolute programming

G91 - Incremental programming

G92 - Reposition origin point; Mill

G92 - Thread cutting cycle; Lathe

G94 - Per minute feed; Mill

G95 - Per revolution feed; Mill

G96 - Constant surface speed control; Lathe

G97 - Constant surface speed cancel

G98 - Per minute feed; Lathe

G99 - Per revolution feed; Lathe

Miscellaneous Functions (M-Code):

M00 - Program stop; Mill and Lathe

M01 - Optional program stop; Lathe and Mill

M02 - Program end; Lathe and Mill

M03 - Spindle on clockwise; Lathe and Mill

M04 - Spindle on counterclockwise; Lathe and Mill

M05 - Spindle off; Lathe and Mill

M06 - Tool change; Mill

M08 - Coolant on; Lathe and Mill

M09 - Coolant off; Lathe and Mill

M30 - Program end, return to start; Lathe and Mill

M97 - Local sub-routine call; Lathe and Mill

M98 - Sub-program call; Lathe and Mill

M99 - End of sub program; Lathe and Mill

M00 - program stop

M01 - optional stop using stop button

M02 - end of program

M03 - spindle on CW

M04 - spindle on CCW

M05 - spindle off

M06 - tool change

M07 - flood with coolant

M08 - mist with coolant

M09 - coolant off

M17 - subroutine end

M20 - tailstock back

M21 - tailstock forward

M22 - Write current position to data file

M25 - open chuck

M30 - end of tape

M71 - puff blowing on

M72 - puff blowing off

M96 - compensate for rounded external curves

M97 - compensate for sharp external curves

M98 - subprogram call

M99 - return from subprogram, jump instruction

M101 - move x-axis home

M102 - move y-axis home

M103 - move z-axis home

Viva Questions:

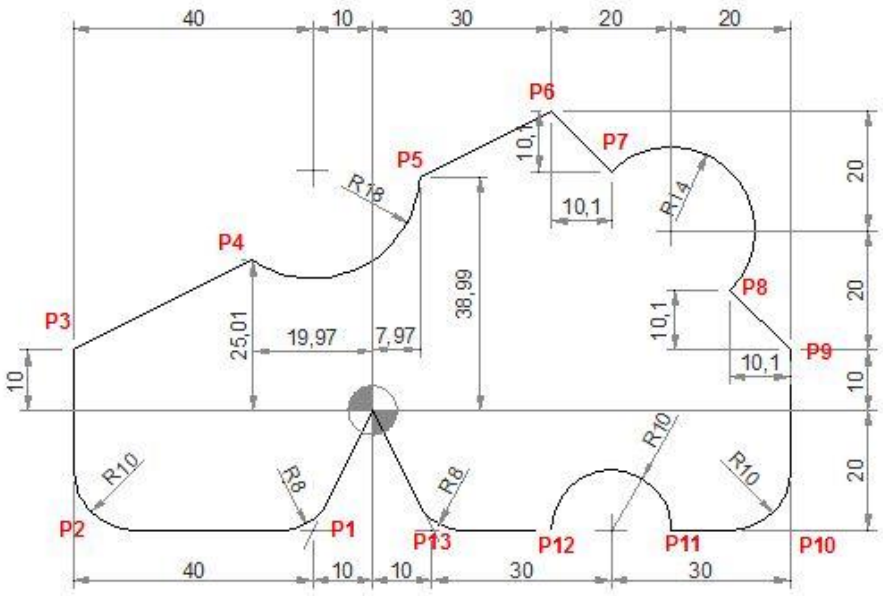
1. Explain Few G codes and functioning?
2. Explain Few M codes and functioning?
3. Explain the methods of programming
4. Explain the advantages of incremental programming over absolute programming.

EXPERIMENT III

CNC MILLING I

Aim: learn the Basic fundamentals of CNC milling, familiarization of machine control panel.

Diagram:



CNC Manual Coding:

N40 G90 G00 X0 Y0

N50 G01 X-10 Y-20 R8 (P1)

N60 G01 X-50 R10 (P2)

N70 Y10 (P3)

N80 X-19.97 Y25.01 (P4)

N90 G03 X7.97 Y38.99 R18 (P5)

N100 G01 X30 Y50 (P6)

N110 G91 X10.1 Y-10.1 (P7)

N120 G90 G02 X59.9 Y20.1 R14 (P8)

N130 G01 X70 Y10 (P9)

N140 Y-20 R10 (P10)

N150 X50 (P11)

N160 G03 X30 R10 (P12)

N170 G01 X10 R8 (P13)

Viva Questions:

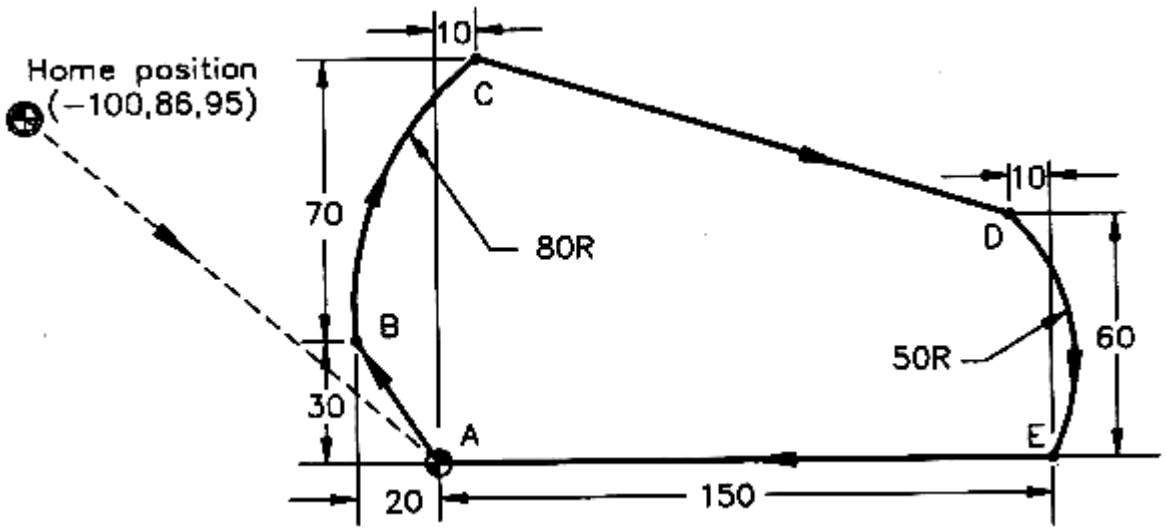
1. What are the basic operations can perform in milling machine?
2. List the various operating systems available in programming?
3. What is the operating system available in present system?
4. Explain G90 and G91 with example.

EXPERIMENT IV

CNC MILLING II

Aim: Fundamentals of CNC programming, Part programming and interpolation techniques.

Diagram:



CNC Manual Coding:

N5 G90 G71

N10 T1 M6

N15 G92 X-100 Y86 Z95

N20 G0 X0 Y0 S2500 M3

N25 Z12.5

N30 G1 Z-12.5 F150

N35 X-20 Y30

N40 G2 X10 Y100 R80

N45 G1 X140 Y60

N50 G2 X150 Y0 R50

N55 G1 X0 Y0

N60 G0 Z12.5

N65 G91 G28 Z0 M5

N70 G91 G28 X0 Y0

N75 M30

Code Explanation

N5 absolute positioning, metric unit

N10 tool change to T1

N15 define work zero point at A

N20 rapid traverse to A, spindle on (2500 RPM, CW)

N25 rapid plunge to 12.5 mm above Z0

N30 feed to Z-12.5, feed rate 150 MMPM

N35 cut line AB to B

N40 cut arc BC to C

N45 cut line CD to D

N50 cut arc DE to E

N55 cut line EA to A

N60 rapid retract to Z12.5

N65 reference point return in Z direction, spindle off

N70 reference point return in X and Y directions

N75 end of program

Viva Questions:

1. What are G02 and G03? Differentiate each.
2. What are the liner motion codes?
3. What is the drilling code?
4. What is meaning of peck drilling?

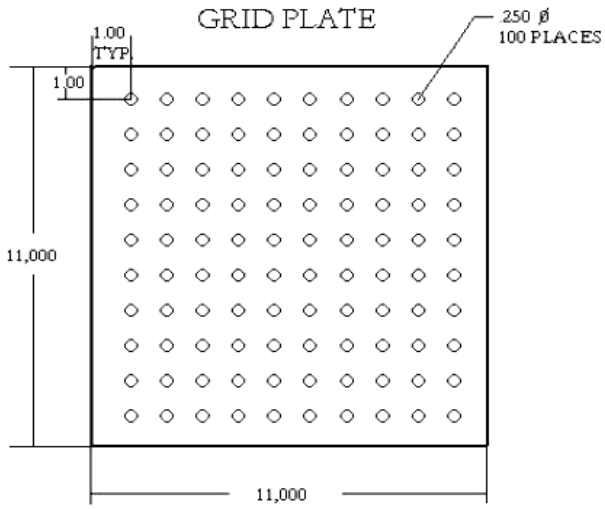
EXPERIMENT V

CNC MILLING III

Aim: Machining practice on CNC machine for drilling

Example I

Diagram:



CNC Part Programming:

03400 (Drilling grid plate)

T1 M06

G00 G90 G54 X1.0 Y-1.0 S2500 M03

G43 H01 Z.1 M08

G81 Z-1.5 F15. R.1

G91 X1.0 L9

G90 Y-2.0 (Or stay in G91 and repeat Y-1.0)

G91 X-1.0 L9

G90 Y-3.0

G91 X1.0 L9

G90 Y-4.0

G91 X-1.0 L9

G90 Y-5.0

G91 X1.0 L9

G90 Y-6.0

G91 X-1.0 L9

G90 Y-7.0

G91 X1.0 L9

G90 Y-8.0

G91 X-1.0 L9

G90 Y-9.0

G91 X1.0 L9

G90 Y-10.0

G91 X-1.0 L9

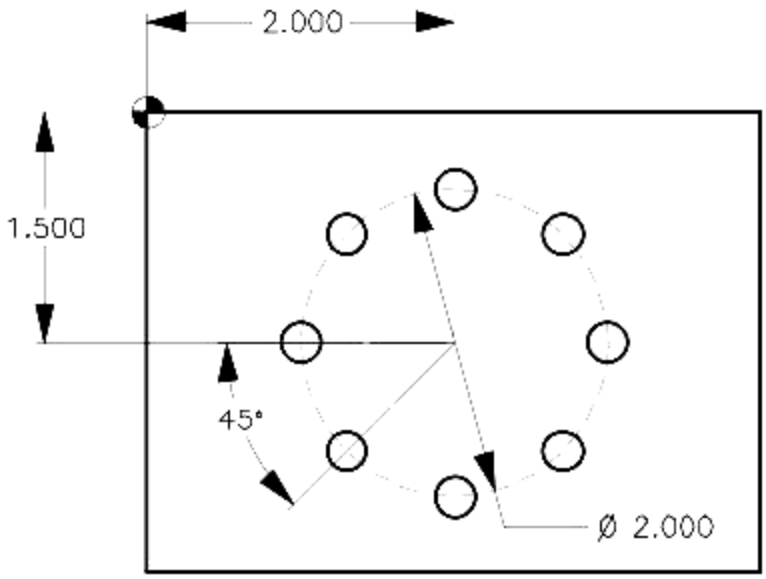
G00 G90 G80 Z1.0 M09

G28 G91 Y0 Z0

M30

Example II

Diagram:



CNC Part programming:

O0009

N1 T1 M06

N2 G90 G54 G00 X2. Y-1.5 (Center position of bolt hole circle)

N3 S1451 M03

N4 G43 H01 Z1. M08

N5 G81 G99 Z-0.45 R0.1 F8. L0

N6 G70 I1. J0. L8

N7 G80 G00 Z1. M09

N8 G53 G49 Z0. M05

N9 M30

Viva Questions:

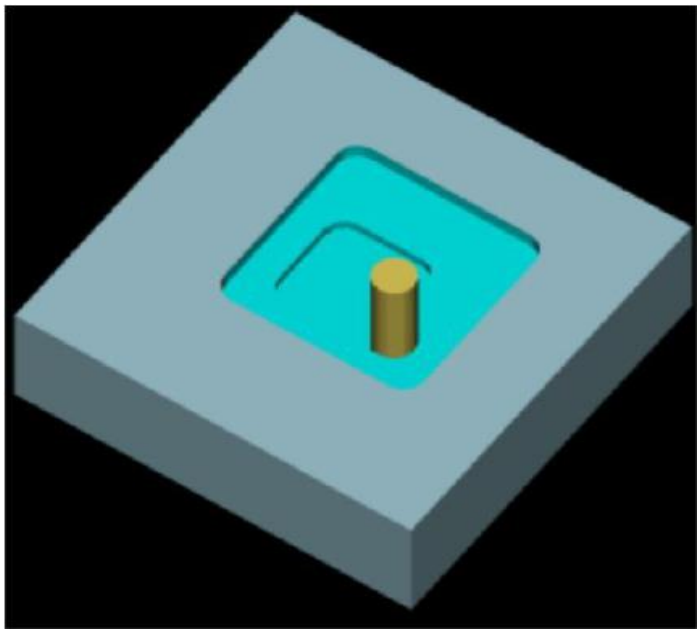
1. What is mean by tool compensation?
2. What is tool compensation code?
3. What is work offset? Explain.
4. What is mean by plane selection?

EXPERIMENT VI

CAM SOFTWARE I

Aim: Generation of part programming through CAM software package.

Diagram:



CADEM Programming:

O1234

G17 G90 G21 G80 G40 G54 G94 G49

N10 T01 M06 (10.0 MM END MILL-ROUGH-3 FLUTE)

(POCKET MILLING)

G90 G00 G54 X0.0 Y0. S1000 M03

G43 H01 Z5.

G0 Z1

G0 G90 X50.0 Y50.0

M98 P10055

G0 G90 X-50 Y50

M98 P10055

G0 G90 X-50 Y-50

M98 P10055

G0 G90 X50.0 Y-50.0

```
M98 P10055
G90 M09
M05
Z100.
G0 G91 G28 Y0.0
M30
O55 (SUB PROGRAM)
G01 Z-5.0 F100
G91 G03 X-4. Y4. I-4. J0.
F200.535 X-4. Y-4. I0. J-4.
X4. Y-4. I4. J0.
X4. Y4. I0. J4.
G01 X8.
G03 X-12. Y12. I-12. J0.
X-12. Y-12. I0. J-12.
X12. Y-12. I12. J0.
X12. Y12. I0. J12.
G01 X8.
G03 X-20. Y20. I-20. J0.
X-20. Y-20. I0. J-20.
X20. Y-20. I20. J0.
X20. Y20. I0. J20.
G0 Z5.0
M99
```

Viva Questions:

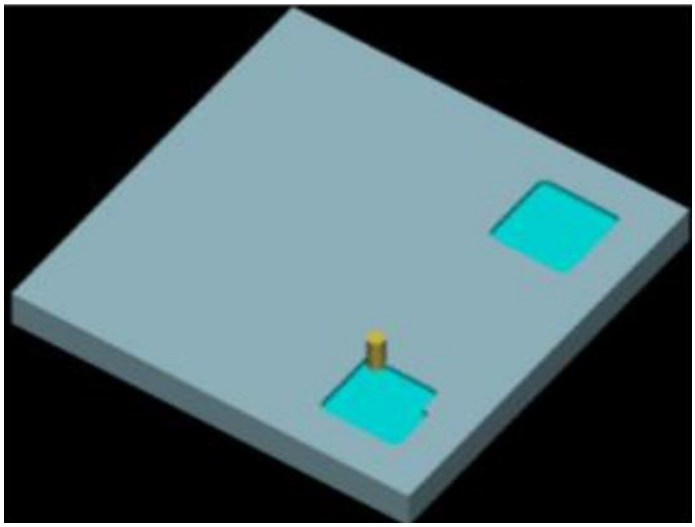
1. What is CAM software explain barfly.
2. What CAM software is using in laboratory?
3. What are the advantages of CAM software?
4. What are the methods of connecting software to machine?

EXPERIMENT VII

CAM SOFTWARE II

Aim: CAM-CNC programming and execution.

Diagram:



CADEM Programming:

Mirroring ab out X axis

O1234

G17 G90 G21 G80 G40 G54 G94 G49

N10 T01 M06 (10.0 MM END MILL-ROUGH-3

FLUTE)

(POCKET MILLING)

G90 G00 G54 X0.0 Y0. S1000 M03

G43 H01 Z5.

Z1

M98 P10055

M70

M98 P10055

M80

M05

Z100.

G0 G91 G28 Y0.0

M30

O55 (SUB PROGRAM)

G0 X75 Y75

G01 Z-5.0 F100

Viva Questions:

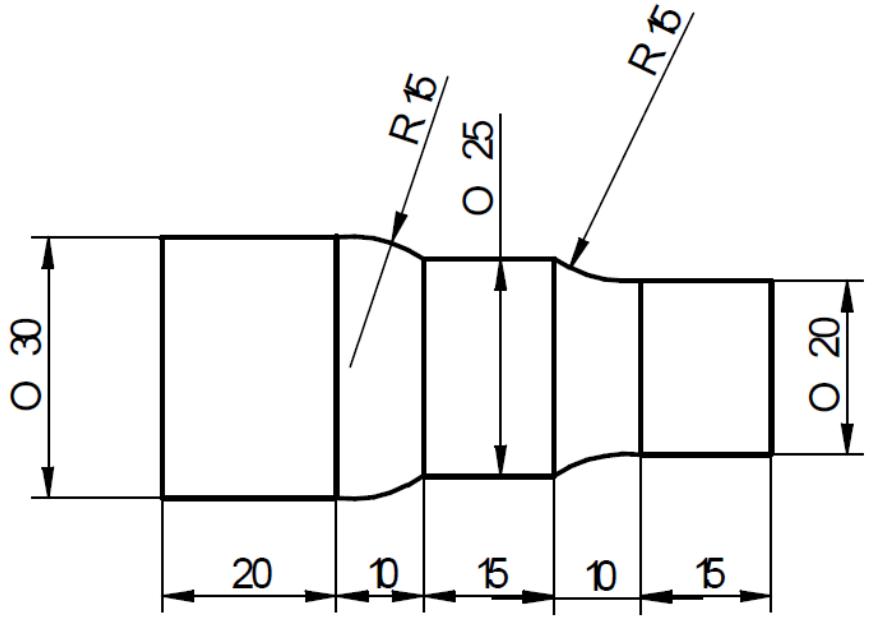
1. Explain the work setting method in CAM software.
2. How you select the tool in CAM software.
3. What is work offset in CAM software?
4. Explain the process flow diagram in CAM software.

EXPERIMENT VIII

CNC TURNING I

Aim: Work piece setting methods, tool setting methods.

Diagram:



CNC Part Programming:

N1 F0.5 S1200 T0101 M06 M03

N10 G00 X35 Z2

N30 G71 U0.5 R1

N35 G71 P36 Q90 U0.05 W0.05

N36 G01 X20 Z0

N50 G01 X20 Z-15

N60 G02 X25 Z-25 R15

N70 G01 X25 Z-40

N80 G03 X30 Z-50 R15

N90 G01 X30 Z-70

N100 G28 U0 W0

N110 M05 M30

Viva Questions:

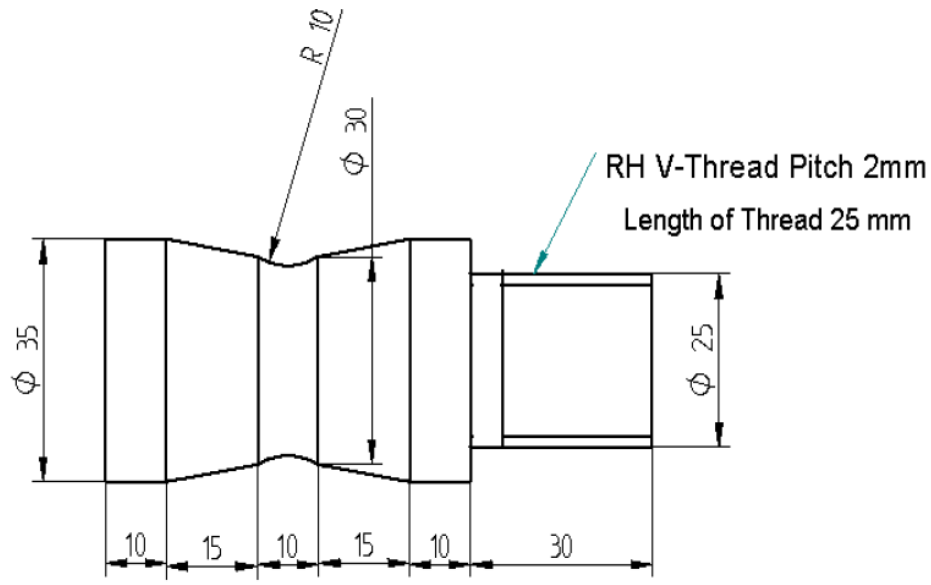
1. What is Lathe? Explain.
2. What are the machining operations can perform in CNC lathe machine:
3. What are the process parameters? Explain the process parameters in brief.
4. What is MRR?

EXPERIMENT IX

CNC TURNING II

Aim: Practice on CNC turning and exercises on machine.

Diagram:



CNC Part Programming:

N1 F0.2 S1200 T0101 M06 M03

N10 G00 X38 Z2

N30 G73 U5 R10

N40 G73 P50 Q130 U0.05 W0.05

N50 G01 X25 Z0

N60 G01 X25 Z-30

N70 G01 X35 Z-30

N80 G01 X35 Z-40

N90 G01 X25 Z-55

N100 G01 X25 Z-65

N110 G01 X35 Z-80

N130 G01 X35 Z-90

N140 G28 U0 W0

N150 S400 T0202 M06

N160 G00 X26 Z2

N170 G76 P010160 Q10

N180 G76 X23.44 Z-25 P1280 Q100 F2

N190 G28 U0 W0

N200 M05 M30

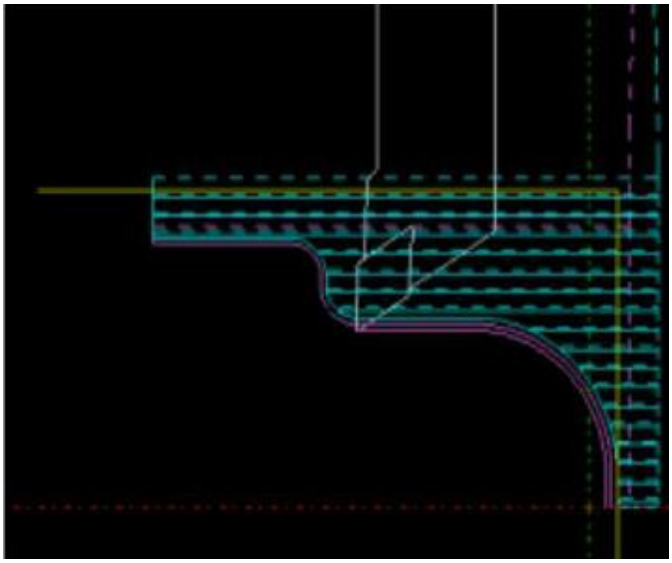
Viva Questions:

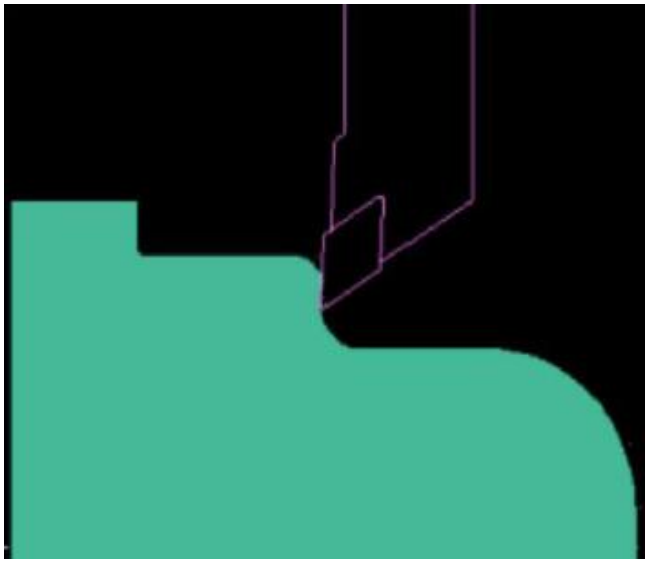
1. What is capacity of CNC lathe machine
2. What is maximum stroke length available in machine?
3. What is the spin over bed explain?
4. Explain few G codes specifically used for lathe operation.

EXPERIMENT X

CAM SOFTWARE III

Aim: Generation of part programming through the CAM software package, CAM-CNC programming and execution on milling and turning machines.





CADEM Software Programming:

G71 – Turning cycle

O1234

G0 G99 G21

G0 X300. Z50.

T0101

G97 S700 M03

X100. Z2. M08

X84.

N20 G71 U3. R0.5

N25 G71 P30 Q75 U1. W0.5 F0.3

N30 G00 X46.

N35 G01 X46. Z0.

N40 X50. Z-2.

N45 Z-45.

N50 G02 X60. Z-50. I5. K0.

N55 G01 X71.

N60 G03 X75. Z-52. I0. K-2.

N65 G01 Z-75.

N70 X80.

N75 X84. Z-75.

G0 X100.

M05

M09

G0 G28 U0.0 W0.0

M30

Viva Questions:

1. Explain the process of selecting the tool in CAM software.
2. Explain the basic feature of CAPSmill, CAPSturn.
3. What is he software prefer to use in laboratory.
4. List the various CAM softwares.

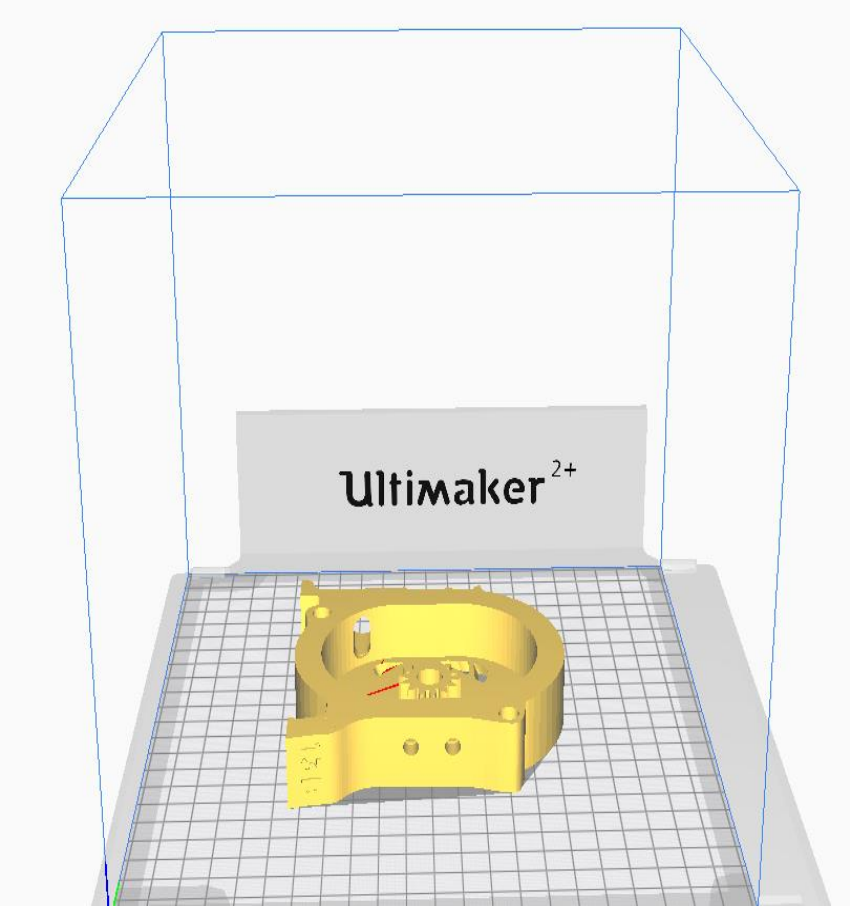
EXPERIMENT XI

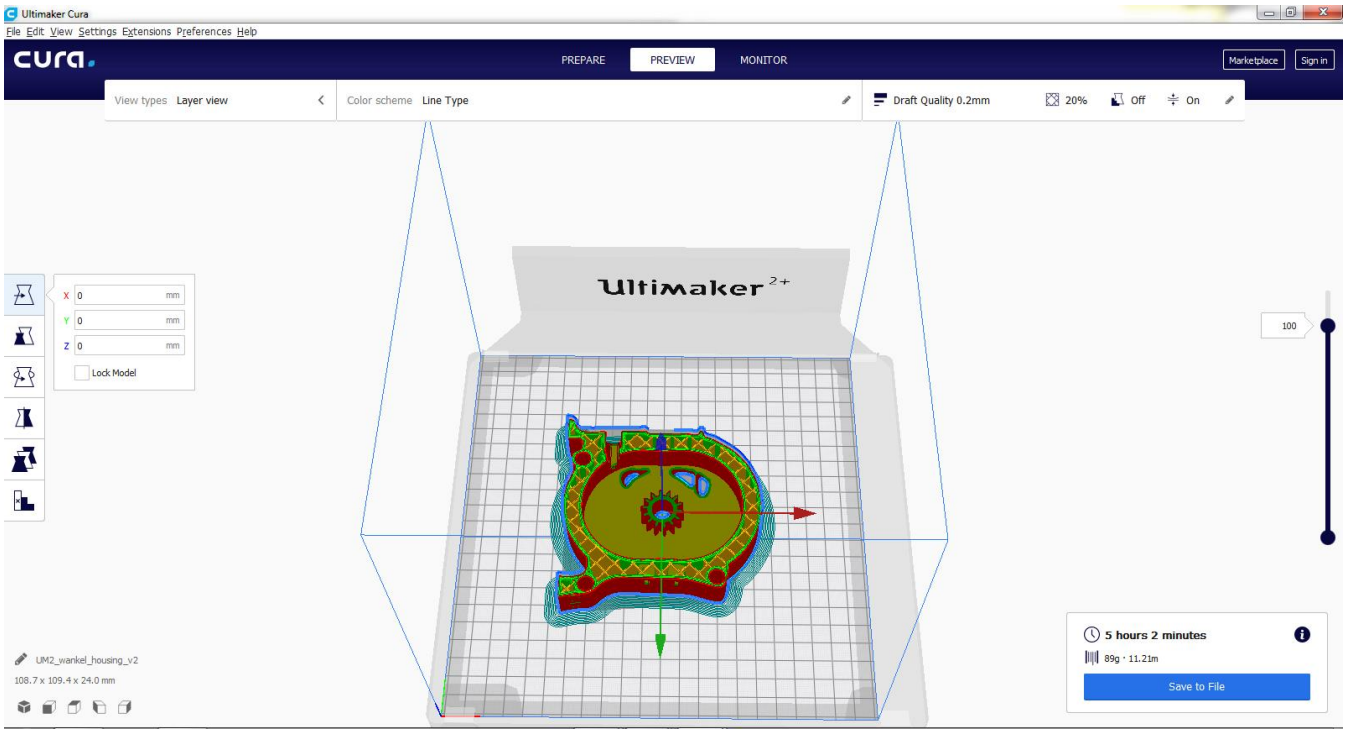
3D PRINTING

Aim: Prepare simple prototype models.

Process diagram:

1. Design the product
2. Save in the form of '.stl' format
3. Open the Cura 4.0 3D software
4. Load the diagram
5. Select the required properties like Nozzle size, material, infill density
6. Slice the part and save the file in '.gcode'
7. Load the '.gcode' in 3D printer and print the part





Machine Codes:

;FLAVOR:UltiGCode

;TIME:18125

;MATERIAL:71506

;MATERIAL2:0

;NOZZLE_DIAMETER:0.8

;Generated with Cura_SteamEngine 4.0.0

M82 ;absolute extrusion mode

G92 E0

G10

;LAYER_COUNT:119

;LAYER:0

M107

G0 F5400 X51.531 Y51.064 Z0.53

;TYPE:SKIRT

G11

G1 F1800 X52.132 Y50.533 E0.29753

G1 X52.783 Y50.064 E0.5952

G1 X53.478 Y49.663 E0.89289

G1 X54.209 Y49.333 E1.19044

G1 X54.969 Y49.077 E1.48797

G1 X55.751 Y48.898 E1.7856

G1 X56.547 Y48.798 E2.08323

G1 X57.349 Y48.777 E2.38088

G1 X58.149 Y48.837 E2.67851

G1 X58.751 Y48.935 E2.90479

G1 X66.318 Y50.468 E5.76918

G1 X67.944 Y50.683 E6.37768

G1 X71.833 Y50.891 E7.82256

G1 X72.587 Y50.967 E8.10371

G1 X73.751 Y51.14 E8.5403

G1 X74.885 Y51.394 E8.97144

G1 X76.452 Y51.865 E9.57849

G1 X77.208 Y52.134 E9.87619

G1 X77.934 Y52.477 E10.17408

G1 X79.391 Y53.257 E10.78722

Continuing …….

Viva Questions:

1. What is additive manufacturing? Explain.
2. What is software used for slice conversion.
3. What is machine understanding format?
4. List the software understanding languages

EXPERIMENT XII

INDUSTRY–INSTITUTE INTERACTION

Aim: Practice session at industry

Students has to write their own experience in industry's

Viva Questions:

1. Explain the flow chart of manufacturing in industry?
2. Explain various production operations in industry.
3. Briefly explain your experience in industry?
4. What are the safety norms followed by industries.