LAB MANUAL

IV Year B. Tech I - Semester

MECHANICAL ENGINEERING

AY: 2025-26

MECHANICAL MEASUREMENTS & INSTRUMENTATION AND COMPUTER AIDED ENGINEERING LAB

R22A0390

Prepared by:

Dr. B. Sandhya Rani

Mr. K. Bicha

Assoc. Professor

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

DEPARTMENT OF MECHANICAL ENGINEERING

(Autonomous Institution-UGC, Govt. of India)

Secunderabad-500100, Telangana State, India.

www.mrcet.ac.in

MRCET CAMPUS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(AUTONOMOUS INSTITUTION - UGC, GOVT. OF INDIA)

Affiliated to JNTUH; Approved by AICTE, NBA-Tier 1 & NAAC with A-GRADE | ISO 9001:2015

Maisammaguda, Dhulapally, Komaplly, Secunderabad - 500100, Telangana State, India

LABORATORY MANUAL & RECORD

Name:.....

Roll No:..... Branch:.....

Year:..... Sem:.....

MRCET CAMPUS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(AUTONOMOUS INSTITUTION - UGC, GOVT. OF INDIA)

Affiliated to JNTUH, Approved by AICTE, NBA-Tier 1 & NAAC with A-GRADE | ISO 9001:2015

Maisammaguda, Dhulapally, Kompally, Secunderabad - 500100, Telangana State, India

Certificate

Certified that this is the Bonafide Record of the Work Done by

Mr./Ms.....Roll.No.....of

B.Tech year..... Semester for Academic year 2025-2026

in.....Laboratory.

Date: Faculty Incharge HOD

Internal Examiner External Examiner

INDEX

S.No	Date	Name of the Activity/Experiment	Grade/ Marks	Faculty Signature

INDEX

S.No	Date	Name of the Activity/Experiment	Grade/ Marks	Faculty Signature

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

DEPARTMENT OF MECHANICAL ENGINEERING

CONTENTS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

VISION

- To establish a pedestal for the integral innovation, team spirit, originality and competence in the students, expose them to face the global challenges and become technology leaders of Indian vision of modern society.

MISSION

- To become a model institution in the fields of Engineering, Technology and Management.
- To impart holistic education to the students to render them as industry ready engineers.
- To ensure synchronization of MRCET ideologies with challenging demands of International Pioneering Organizations.

QUALITY POLICY

- To implement best practices in Teaching and Learning process for both UG and PG courses meticulously.
- To provide state of art infrastructure and expertise to impart quality education.
- To groom the students to become intellectually creative and professionally competitive.
- To channelize the activities and tune them in heights of commitment and sincerity, the requisites to claim the never – ending ladder of **SUCCESS** year after year.

For more information: www.mrcet.ac.in

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

www.mrcet.ac.in

Department of Mechanical Engineering

VISION

To become an innovative knowledge center in mechanical engineering through state-of-the-art teaching-learning and research practices, promoting creative thinking professionals.

MISSION

The Department of Mechanical Engineering is dedicated for transforming the students into highly competent Mechanical engineers to meet the needs of the industry, in a changing and challenging technical environment, by strongly focusing in the fundamentals of engineering sciences for achieving excellent results in their professional pursuits.

Quality Policy

- ✓ To pursuit global Standards of excellence in all our endeavors namely teaching, research and continuing education and to remain accountable in our core and support functions, through processes of self-evaluation and continuous improvement.
- ✓ To create a midst of excellence for imparting state of art education, industry-oriented training research in the field of technical education.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

www.mrcet.ac.in

Department of Mechanical Engineering

PROGRAM OUTCOMES

Engineering Graduates will be able to:

1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **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.
3. **Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **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.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and teamwork**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

12. **Life-long learning**: Recognize the need for and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

PSO1 Ability to analyze, design and develop Machine learning systems to solve the Engineering problems by integrating design and manufacturing Domains.

PSO2 Ability to succeed in competitive examinations or to pursue higher studies or research.

PSO3 Ability to apply the learned Mechanical Engineering knowledge for the Development of society and self.

Program Educational Objectives (PEOs)

The Program Educational Objectives of the program offered by the department are broadly listed below:

PEO1: PREPARATION

To provide sound foundation in mathematical, scientific and engineering fundamentals necessary to analyze, formulate and solve engineering problems.

PEO2: CORE COMPETANCE

To provide thorough knowledge in Mechanical Engineering subjects including theoretical knowledge and practical training for preparing Artificial models pertaining to Automobile Engineering, Element Analysis, Production Technology, Mechatronics etc.,

PEO3: INVENTION, INNOVATION AND CREATIVITY

To make the students to design, experiment, analyze, interpret in the core field with the help of other inter disciplinary concepts wherever applicable.

PEO4: CAREER DEVELOPMENT

To inculcate the habit of lifelong learning for career development through successful completion of advanced degrees, professional development courses, industrial training etc.

PEO5: PROFESSIONALISM

To impart technical knowledge, ethical values for professional development of the student to solve complex problems and to work in multi-disciplinary ambience, whose solutions lead to significant societal benefits.

CODE OF CONDUCT

1. Students should bring lab Manual/Record for every laboratory session and should enter the readings/observations in the manual while performing the experiment.
2. The group- wise division made in the beginning should be adhered to, and no mix up of students among different groups will be permitted later.
3. The components required pertaining to the experiment should be collected from stores in –charge after duly filling in the requisition form.
4. When the experiment is completed, students should disconnect the setup made by them, and should return all the components/instruments taken for the purpose.
5. Any damage to the apparatus that occurs during the experiment should be brought to the notice of lab in-charge, consequently, the cost of repair or new apparatus should be brought by the students.
6. After completion of the experiment, certification of the concerned staff in –charge in the observation book is necessary.
7. Students should be present in the labs for the total scheduled duration.
8. Students should not carry any food items inside the laboratory.
9. Use of cell phones and IPODs are forbidden.
10. Students should not write on or deface any lab desks, computers, or any equipment provided to them during the experiment.
11. Every student should keep his/her work area properly before leaving the laboratory.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

IV Year B. TECH - I- SEM **L/T/P/C**

-/-/4/2

(R22A0390) Mechanical Measurements & Instrumentation lab and Computer Aided Engineering Laboratory

Course Objectives:

1.To prepare the students for successful career in industry and motivate for higher education.
2.To provide strong foundation in basic science and mathematics necessary to formulate, solve and analyze Control and Instrumentation problems
3.To provide strong foundation in circuit theory, control theory and signal processing concepts.
4.To introduce fundamentals of the analysis software, its features and applications.
5.To learn the basic element types in Finite Element analysis

LIST OF EXPERIMENTS:

CYCLE-1

1.Calibration of Pressure Gauges
2.Study and calibration of LVDT transducer for displacement measurement.
3.Calibration of strain gauge for temperature measurement.
4.Calibration of thermocouple for temperature measurement.
5.Calibration of capacitive transducer for angular displacement.
6.Study and calibration of photo and magnetic speed pickups for the measurement of speed.
7.Calibration of resistance temperature detector for temperature measurement.
8.Study and calibration of McLeod gauge for low pressure

CYCLE-2

2-D stress analysis of bar
1.Plane stress analysis
2.Plain strain analysis
3. Beam analysis
4.Truss analysis 3-D analysis
5.Modal analysis
6.Buckling analysis Ansys, Abaqus

Course Outcomes:

1.At the end of the course, the student will be able to characterize and calibrate measuring devices.
2.Identify and analyze errors in measurement.
3.Analyze measured data using regression analysis.
4.Classify different types of truss and perform static analysis.
5.Perform static analysis on connecting rod with 3D elements.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(An Autonomous Institution – UGC, Govt. of India)

Recognizes under 2(f) and 12(B) of UGC ACT 1956

(Affiliated to JNTUH, Hyderabad, Approved by AICTE –Accredited by NBA & NAAC-"A" Grade-ISO 9001:2015Certified)

CONTENTS

1. CALIBRATION OF PRESSURE CELL

AIM:

Calibrate the given Pressure Cell with Pressure Gauge for its performance

APPARATUS REQUIRED:

Pressure cell

Hydraulic dead weight Pressure gauge Tester to develop the pressure

Digital pressure indicator

Dial type pressure indicator

THEORY BEHIND:

Pressure is defined as force per unit area and is measured in Newton per square meter (Pascal) or in terms of an equivalent head of some standard liquid.

Thus, a pressure gauge is connected to the hydraulic line and the gauge itself stands in atmospheric pressure.

Pressure transducer is basically an electro mechanical devices, especially manufactured and designed or wide range application in pressure measurement.

The pressure transducer comprises of diaphragm and an inputs to facilitate pressure measurement. The strain gauges are bonded directly to the sensing member to provide excellent linearity, low hysteresis and repeatability.

Fluid medium whose parameter has to be measured is allowed to deflect the diaphragm (sensing member), which is a single block material and forms an integral part of the pressure transducer.

Usually, the Pressure Transducers are made up non-magnetic stainless steel and thus has the advantage of avoids the yielding effects and leakage problems. The slight deflection of the diaphragms due to the pressure provides an electrical output.

The material most commonly used for manufacture of diaphragms are steel, phosphor bronze, nickel silver and beryllium copper. The deflection generally follows a linear variation with the diaphragm thickness.

PANEL DETIALS:

MAINS ON INDICATOR: To indicate the Power given to the system.

CONSOLE ON SWITCH: Provided to activate the system.

PRESSURE INDICATOR: To indicate the Pressure in digital format with Zero knob facility.

SOFTWARE: Facilitates to do things in computer format.

PREPARATION OF EQUIPMENT:

1. Connect the pressure cell to the pressure indicator with given cable.
2. Connect the instrument to 1ph, 230V AC supply which is having proper earthing.
3. Adjust the zero pot of the indicator to indicate zero.
4. Close the release valve of pressure gauge tester and apply the 10kg dead weight on flange.

PROCEDURE:

1. Slowly rotate the screw road in clockwise direction with the help of handle until flange lift up (so that pressure is developed up to applied load). Now observed the digital reading. If it is not showing zero then make it zero by rotating ZERO knob. Now instrument is calibrated.
2. Apply the load up to 10Kgs one by one on the flange and give pressure by rotating the screw rod such that the dial gauge reads 1 to 10 with respect to load applied.
3. Note down the readings of dial gauge and pressure indicator, simultaneously in every step.
4. Calculate the error and % error.

TABULATIONS:

Sl. No	Pressure in Dial gauge, P_c kg / cm^2	Pressure in Digital indicator, P_g kg / cm^2	Correction $P_c - P_g$	Error $P_g - P_c$	% Error $(P_g - P_c)/ P_g *100$

CONCLUSIONS OF THE RESULTS TABULATED:

Summarizing the entire operation

Describing the possible error factors

Techniques which can be adopted to minimize the errors in all aspects i.e., from startup to end.

LIMITATIONS

1. Range of Pressure Cell ----- Max. 10kg/cm^2

APPLICATIONS:

1. In compressors
2. In boilers

VIVA-VOCE QUESTIONS:

1. One atmospheric pressure is equivalent to?
2. A barometer is used to measures?
3. Piezometer tube is used to measure?
4. The stagnation(total) pressure at a point is measured by?
5. A U-tube differential manometer is used inverted when pressure difference is ----------
6. A well-type manometer is used in preference to a simple U-tube manometer to obtained-----
7. Which manometer is likely to have the highest sensitivity and accuracy?
8. Mercury is used in barometers because?
9. In a bourdon tube pressure gauge incorrect readings may be encountered due to?
10. Most common material chosen for the fabrication of bellows of a bellows pressure gauge is?
11. A dead weight tester is used for?

2. CALIBRATION OF LVDT

AIM:

Calibrate **Linear Variable Differential Transformer (LVDT)** for the performance using Micrometer.

APPARATUS REQUIRED:

LVDT

Digital LVDT indicator

MICROMETER

THEORY BEHIND:

LVDT is an inductive transducer used to translate the linear motion into electrical signal LVDT consists of a single primary winding 'P' and two secondary windings (S1 & S2) wounds on a cylindrical armature. An AC source is connected to the primary winding. A movable soft iron core attached with an arm placed inside the armature.

The primary winding produces an alternating magnetic field which induces alternating voltage in the secondary windings. Single voltage is obtained by connecting the two secondary windings in series. Thus, the output voltage of the transducer is the difference of the two voltages.

When the core is at null position, the flux linking with both the secondary windings is equal. Since both the secondary winding have equal number of turns, M the induced emf is same in them. The output voltage is the difference of the two emf say E1 & E2. When they are equal, the voltage is zero at null position.

When the core is moved to the left side from null position more flux links with S1. The output voltage is V=E2-E1, is greater, the V value is negative (–ve). Means the voltage is read in terms of mm length on the display board indicates the negative value. When the core is moved to the right side of the null position, more flux links with S2 induces voltages which is +ve. The display board indicates the +ve value in mm of length.

The voltage output is linear and is depending on the position of the core. Hence LVDT can be conveniently used to measure the thickness ranging from fraction of mm to a few cms. Normally LVDT can give better result up to 5mm.

PANEL DETIALS:

MAINS ON INDICATOR: To indicate the power given to the system.

CONSOLE ON SWITCH: Provided to activate the system.

LVDT INDICATOR: To indicate the Distance moved.

SOFTWARE: FACILITATES TO DO THINGS IN COMPUTER FORMAT.

PREPARATION OF EQUIPMENT:

1. Make the Micrometer to Read 10mm on the scale. (ZERO POSITION of LVDT)
2. Connect the instrument to 1ph, 230V AC supply which is having proper earthing.

LIMITATIONS

Range of Pressure LVDT: -10mm to +10mm

PROCEDURE:

1. Slowly rotate the screw head of the micrometer either clockwise or anticlockwise to measure 1mm on it.
2. Note the Reading on the LVDT indicator
3. Repeat step 1 and step 2 until 10mm on either side.
4. Note down the readings of Micrometer, simultaneously in every step.
5. Calculate the error and % error.

Note: Clockwise will give readings in –ve direction and Anticlockwise will give in +ve direction.

TABULATIONS:

Sl. no	Actual Reading, 'R_a' mm	Measured Reading, 'R_m' mm	Error 'E'	% Error

RESULT: ----------

Note: Plot a graph between Ra Vs Rm

APPLICATIONS:

1. In pressure cells
2. In force cells
3. In accelerometers

VIVA-VOCE QUESTIONS:

1. The abbreviation LVDT stands for?
2. What is the difference between active and passive transducers?
3. What information is needed to describe a transducer for a particular measurement?
4. What are the major considerations which govern the selection of an instrument transducer?
5. Define displacement?
6. What is the use of wire wound potentiometer for the measurement of linear and rotary Transducers?

7. What are the advantages of wire wound potentiometer?

8. What are the disadvantages of wire wound potentiometer?

9. What are the advantages of capacitive transducers?

10. What are the disadvantages of capacitive transducers?

3. CALIBRATION OF STRAIN GAUGE

AIM:

To determine the elastic constant (modulus of elasticity) of a **Cantilever beam** subjected to concentrated end load by using **STRAIN GUAGES**.

APPARATUS REQUIRED :

Load Cell with Strain Gauge

Digital Strain indicator

Weights

THEORY BEHIND:

A body subjected to external forces is in a condition of both stress and strain. Stress can be directly measured but its effect i.e., change of shape of the body can be measured. If there is a relationship between stress and strain, stresses occurring in a body can be computed if sufficient strain information is available. The constant connecting the stress and strain in elastic material under the direct stresses is the modulus of elasticity,

i.e., $E=\sigma / \epsilon$

the principle of the electrical resistance strain gauge was discovered by Lord Kelvin, when he observed that a stress applied to a metal wire, besides changing resistance strain gauges are made into two basic forms, bonded wire and bonded foil. Wire gauges are sandwiched between two sheets thin paper and foil gauges are sandwiched between two thin sheets of epoxy.

The resistance factor 'R' of a metal depends on its electrical resistively, ρ, its area, a and the length l, according to the equation $R = \rho l / a$.

Thus, to obtain a high resistance gauge occupying a small area, the metal chosen has a high resistively, a large number of grid loops and a very small cross-sectional area. The most common material for strain gauge is a copper - -nickel alloy known as Advance.

The strain gauge is connected to the material in which it is required to measure the strain, with a thin coat of adhesive. Most common adhesive used is Eastman, Deco Cement, etc. as the test specimens extends or contracts under stress in the direction of windings, the length and cross-

sectional area of the conductor alter, resulting in a corresponding increase or decrease in electrical resistance.

GAUGE FACTOR:

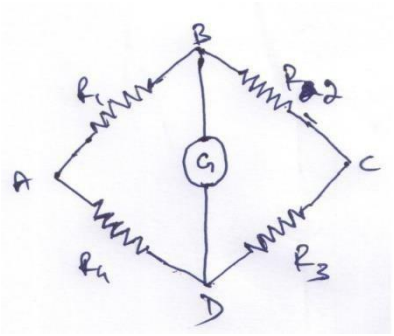
The dimension less relationship between the change in gauge resistance and change in length is called Gauge factor of the strain, which is expressed mathematically,

Gauge Factor, $\delta g = (\Delta R/R) / (\Delta l/l)$

In this relationship R and I represent, respectively the initial resistance and initial length of the strain gauge filament, while ΔR & Δl represents the small change in resistance and length, which occurs as the gauge is strained along with the surface to which it is bonded. This gauge factor of a gauge is a measure of the amount of resistance change for a given strain. The higher the gauge factor greater the electrical output for indication or recording purpose. The gauge factor is supplied by the manufacturer and may range from 1.7 to 4.

The usual method of measuring the change of resistance in a gauge element is by means of Wheatstone bridge as shown in figure. It consists of Galvanometer, 4 resistor & a battery. Resistance R_1 is the strain gauge is used for strain measurement, which is mounted on the specimen. The three resisters R_2, R_3 and R_4 are internal to the device.

Let us assume that the resistance has been adjusted so that the bridge is balanced.



i.e. Voltage $\mathbf{E_{bd} = 0.}$

The most common bridge arrangements are single arm, two arm and four arm mode.

Single Arm Mode (Quarter bridge).

This bridge arrangement consists of a single active gauge in position, say R_1 and three resistor are internal to the device. Temperature compensation is possible only if a self temperature compensating strain gauge is used.

Two Arm Mode (Half bridge).

In this mode, two resistors are internal to the device and the remaining two are strain gauges. One arm of this bridge is commonly labeled as active arm and the other as compensating arm. The bridge is temperature compensated.

Four Arm Mode (Full Bridge).

In this bridge arrangement, four active gauges are placed in the bridge with one gauge in each of the four arms. If the gauges are placed on a beam in bending as shown in fig of the elastic constant by bending test experiment, the single from each of the four gauges will add. This bridge arrangement is temperature compensated.

PANEL DETIALS:

MAINS ON INDICATOR: To indicate the Power given to the system.

CONSOLE ON SWITCH: Provided to activate the system.

STRAIN INDICATOR: To indicate the Distance moved.

SOFTWARE: FACILITATES TO DO THINGS IN COMPUTER FORMAT.

PREPARATION OF EQUIPMENT:

1. Connect the instrument to 1ph, 230V AC supply which is having proper earthing.
2. Switch on the Console
3. Select the BRIDGE Mode
4. Connect the Strain wires accordingly in the table below
5. 5. Make the Indicator to Read Zero. (ZERO POSITION)
6. Prepare the Loads to be added

Sl. No	Bridge Mode	Color	No of Wires	Connect To
1	Quarter	ORANGE	2	A & B
2	Half	ORANGE	2	A & B
		GREEN	2	B & C
3	FULL	WHITE	2	A & D
		ORANGE	2	A & B
		GREEN	2	B & C
		BLACK	2	C & D

LIMITATIONS

Range of Load cell: 10kg

PROCEDURE:

1. For the Bridge selected and connected wires, slowly add the Weights in steps of 1kg.
2. Note the Reading on the Strain indicator
3. Repeat step 1 and step 2 until 10kg is loaded.
4. Note down the weights added, simultaneously in every step.
5. Calculate the error and % error
6. Modulus of Elasticity of the given load cell.

TABULATIONS:

Sl. No	Load Applied W (N)		Strain Indicator Reading Є – micro strain	Measured Strain $Є_m = ϵ*10^{-6}$ / No. of Bridge	Bending stress. $\sigma = 6wl / bh^2$	Modulus of elasticity. $E = \sigma / Є_m$ (N/nm^2)
	W	N				

CONCLUSIONS OF THE RESULTS TABULATED:

Summarizing the entire operation

Describing the possible error factors

Graph Plotting

Techniques which can be adopted to minimize the errors in all aspects i.e, from startup to end.

APPLICATIONS:

1. In the ropes
2. In in the beams

VIVA-VOCE QUESTIONS:

1. Define strain?
2. Define gauge factor of a resistance strain gauge?
3. What are the various types of strain gauges for different applications?
4. What is the difference between bonded and unbounded type of resistance strain gage?
5. What are the advantages of semi-conductor strain gauges?
6. Whar are the requirements of materials for strain gauge?
7. What do you mean by resistance strain gauges?

8. Mechanical strain gauges can measure----------

9. The gauge factor of a resistance strain gauge depends upon--------

10. The wire material of strain gauges should have----------

11. Which gauge can be detached from the test specimen and used again?

4. MEASUREMENT OF ANGULAR DISPLACEMENT

AIM: Measurement of angular displacement using capacitive transducer.

APPARATUS REQUIRED:

CAPACITIVE ANGULAR DISPLACEMENT CELL

Digital ANGULAR indicator

STEPPER MOTOR ASSEMBLY with Controller

THEORY BEHIND:

Capacitance is well known to be a function of effective area of the conductors, separation between them, the dielectric strength of the material in the separation. Capacitive transducers convert the physical quantity to be measured into a change of capacitance which is processed by the measuring circuit of the transducer. The capacitance of a parallel plate capacitor may be changed by varying the separation between the plates, varying the effective area of the plates or varying the dielectric.

Capacitive type transducers are used essentially for displacement or positioning measurements. But they are more susceptible to environmental factors such as dust or moisture in the atmosphere than inductive type transducers.

The meshing area between two stator and rotor plates of the capacitor goes on changing as the shaft capacitor is rotated. The arrangement is used to demonstrate the measurement of angular displacement.

The transducer is mounted on to the face of a protractor which indicates the angle of displacement and the readout display the amount of displacement.

PANEL DETIALS:

MAINS ON INDICATOR: To indicate the Power given to the system.

CONSOLE ON SWITCH: Provided to activate the system.

ANGULAR INDICATOR: To indicate the Distance moved.

STARTER: To Start the Motor and to Select the Mode of Operation

INCREMENTAL: To Give Step input to the Motor according to mode selected.

SOFTWARE: FACILITATES TO DO THINGS IN COMPUTER FORMAT.

PREPARATION OF EQUIPMENT:

1. Connect the instrument to 1ph, 230V AC supply which is having proper earthing
2. Press the STARTER switch and select MANUAL MODE.
3. Set ZERO position on the Manual Indicator by pressing the INCREMENTAL Button.

LIMITATIONS

1. Range of ANGULAR DISPLACEMENT: Up to 360deg
2. In AUTO Mode the Incremental will be in steps of 22.5deg.

PROCEDURE:

1. After the Preparation of equipment, Now press the INCREMENTAL button so the stepper motor sets accordingly with the selected mode of previously.
2. Note the Reading on the Angular Displacement indicator
3. Repeat step 1 and step 2 until required.
4. Note down the readings of Manual Angular, simultaneously in every step.
5. Calculate the error and % error.

Note: Rotation will be in Clockwise direction only.

TABULATIONS:

Sl. no	The displacement protractor reading	Measured displacement meter reading	Deviation

CONCLUSIONS OF THE RESULTS TABULATED:

Summarizing the entire operation

Describing the possible error factors

Graph Plotting

Techniques which can be adopted to minimize the errors in all aspects i.e., from startup to end.

APPLICATIONS:

1. In the rotating shafts

VIVA-VOCE QUESTIONS:

1. What is the difference between speed counter and tachoscope?
2. What is the difference between tachoscope and tachometer?
3. What is the difference between speed counter and tachometer?
4. What are the different type of mechanical tachometers?
5. The average speed measurements are given by--------
6. Tachometers are used to measure-------
7. The speed of a sealed compressor units can be measured by---------
8. What are the advantages of stroboscope?

5. CALIBRATION OF PHOTO SPEED SENSOR

AIM:

Calibrate **PHOTO SPEED SENSOR** for the performance using MAGNETIC Speed Sensor.

APPARATUS REQUIRED:

PHOTO & MAGNETIC SPEED SENSORS

Digital SPEED indicators

MOTOR with Controller.

THEORY BEHIND:

The measurement of rotational velocity is more common. For velocity (speed) measurement the most convenient calibrator scheme uses a combination of toothed wheel, a simple magnetic proximity pickup a photo couple sensor and an electronic indicator to measure the speed. The angular rotation is provided by some adjustable speed drive of adequate stability. The toothed wheel mounted with iron rods while passing under magnetic and photo pickup produces an electric pulse. These pulses are fed to signal conditioner unit and displays reading visually. The stability of the rotational drive is easily checked by observing the variation of display reading.

PANEL DETIALS:

MAINS ON INDICATOR: To indicate the Power given to the system.

CONSOLE ON SWITCH: Provided to activate the system.

SPEED INDICATORS: To indicate the SPEED.

MOTOR CONTROLLER: To run the motor at various speeds.

SOFTWARE: FACILITATES TO DO THINGS IN COMPUTER FORMAT.

PREPARATION OF EQUIPMENT:

1. Keep the Speed controller knob in minimum position.
2. Connect the instrument to 1ph, 230V AC supply which is having proper earthing.
3. Switch on the Console to activate the panel.

LIMITATIONS

Maximum RPM: 2500rpm

PROCEDURE:

1. Once the equipment is prepared, slowly rotate the Speed controller to set the speed desired in range.
2. Note the Reading on the PHOTO SENSOR PICKUP Indicator
3. Repeat step 1 and step 2 until required within the range.
5. Note down the readings of MAGNETIC PICKUP, simultaneously in every step.
6. Calculate the error and % error.

Note: Clockwise to increase the speed and Anticlockwise to reduce the speed.

TABULATIONS:

Sl. no	Actual Reading, 'R_a' mm	Measured Reading, 'R_m' mm	Error E = (Ra- Rm)	% Error {(Ra- Rm)/Ra}*100

RESULT: ----------

Note: Plot a graph between Ra Vs Rm

APPLICATIONS:

1. To measure the electric motors shaft speed.
2. To measure the turbine shaft speed.
3. To measure the engines shaft speed.

VIVA-VOCE QUESTIONS:

1. What is the difference between speed counter and tachoscope?
2. What is the difference between tachoscope and tachometer?
3. What is the difference between speed counter and tachometer?
4. What are the different type of mechanical tachometers?
5. The average speed measurements are given by--------
6. Tachometers are used to measure-------
7. The speed of a sealed compressor units can be measured by---------
8. What are the advantages of stroboscope?
9. What are the different type of electrical tachometer?
10. Tachoscope speed up to----------

6. CALIBRATION OF McLeod GAUGE

AIM:

Calibrate the Vacuum Cell (McLeod Gauge) in comparison with Vacuum Dial Gauge.

APPARATUS REQUIRED :

CYLINDRICAL CELL

Digital Vacuum Gauge

Analog Vacuum Gauge

High-Capacity Vacuum Pump.

THEORY BEHIND:

In everyday usage, vacuum is a volume of space that is essentially empty of matter, such that its gaseous pressure is much less than atmospheric pressure. The word comes from the Latin for “empty”. A perfect vacuum would be one with no particles in it at all, which is impossible to achieve in practice. Physicists often discuss ideal test results that would occur in a perfect vacuum, which they simply call “vacuum” or “free space”, and use the term partial vacuum to refer to real vacuum. The Latin term in vacuum is also used to describe an object as being in what would otherwise be a vacuum.

Vacuum is useful in a variety of processes and devices. Its first widespread use was in the incandescent light bulb to protect the filament from chemical degradation. The chemical inertness produced by a vacuum is also useful for electron beam welding, cold welding, vacuum packing and vacuum frying. Ultra-high vacuum is used in the study of atomically clean substrates, as only a very good vacuum preserves atomic-scale clean surfaces for a reasonably long time (on the order of minutes to days). High to ultra-high vacuum removes the obstruction of air, allowing particle beams to deposit or remove materials without contamination. This is the principle behind chemical vapor deposition, physical vapor deposition, and dry etching which are essential to the fabrication of semi conductors and optical coatings, and to surface science. The reduction of convection provides the thermal insulation of thermos bottles. Deep vacuum lowers the boiling point of liquids and promotes low temperature out gassing which is

used in freeze drying, adhesive preparation, distillation, metallurgy, and process purging. The electrical properties of vacuum make electron microscopes and vacuum tubes possible, including cathode ray tubes. The elimination of air friction is useful for flywheel energy storage and ultracentrifuges.

PANEL DETIALS:

MAINS ON INDICATOR: To indicate the Power given to the system.

CONSOLE ON SWITCH: Provided to activate the system.

VACUUM ON SWITCH: Provided to activate the system.

VACUUM INDICATORS: To indicate the Vacuum inside the cylinder.

PREPARATION OF EQUIPMENT:

1. Make sure the Vacuum On switch is in off condition.
2. Connect the instrument to 1ph, 230V AC supply which is having proper earthing.

LIMITATIONS

Maximum Vacuum Range: 600mm of Hg or 800mbar

PROCEDURE:

1. Switch on the Console to activate the Panel.
2. Close the Valve on the Cylinder.
3. Start the Vacuum pump and allow attaining maximum vacuum.
4. Note down the Readings on Digital and Analog gauges.
5. Now slowly open the Valve on the cylinder and set to the required Vacuum.
6. Once again note down the Digital and Analog readings.
7. Repeat Steps 5 and 6 until zero.
8. Calculate the error and % error.

TABULATIONS:

Sl. no	Actual Reading, 'R_a' mm	Measured Reading, 'R_m' mm	Error E = (Ra- Rm)	% Error {(Ra- Rm)/Ra}*100

RESULT: ----------

Note: Plot a graph between Ra Vs Rm

APPLICATIONS:

1. To clear halls

VIVA-VOCE QUESTIONS:

1. One atmospheric pressure is equivalent to?
2. A barometer is used to measures?
3. A piezometer tube is used to measure?
4. The stagnation(total) pressure at a point is measured by?
5. A U-tube differential manometer is used inverted when pressure difference is ----------
6. A well-type manometer is used in preference to a simple U-tube manometer to obtained-----
7. Which manometer is likely to have the highest sensitivity and accuracy?
8. Mercury is used in barometers because?
9. In a bourdon tube pressure gauge incorrect readings may be encountered due to?
10. Most common material chosen for the fabrication of bellows of a bellows pressure gauge is?
11. A dead weight tester is used for?

7. CALIBRATION OF TEMPERATURE SENSOR

AIM:

Calibrate **TEMPERATURE SENSORS** for the performance using STANDARD water bath.

APPARATUS REQUIRED:

STANDARD WATER BATH with Temperature Controller

Temperature Sensors like RTD, J – Type, K – Type & T – type

Individual Digital Temperature Indicators

THEORY BEHIND:

RESISTANCE TEMPERATURE DETECTOR (RTD)

Resistance thermometers, also called **resistance temperature detectors or resistive thermal devices (RTD)**, are temperature sensors that exploit the predictable change in electrical resistance of some materials with changing temperature. As they are almost invariably made of platinum, they are often called **platinum resistance thermometers (PTR)**. They are slowly replacing the use of thermocouples in many industrial applications below 600°C, due to higher accuracy and repeatability.

There are many categories like carbon resistors, film and wire wound types are the most widely used.

- ***Carbon resistor***s are widely available and are very inexpensive. They have very reproducible results at low temperatures. They are the most reliable from at extremely low temperatures. They generally do not suffer from significant hysteresis or strain gauge effects.
- ***Film thermometer*** have a layer of platinum on a substrate, the layer may be extremely thin, perhaps one micrometer. Advantages of this type are relatively low cost and fast response. Such devices have improved performance although the different expansion rates of the substrate and platinum give "strain gauge" effects and stability problems.
- ***Wire – wound thermometers*** can have greater accuracy, especially for wide temperature ranges.
- ***Coil element*** **has** largely replaced wire wound elements in industry. This design has a wire coil which can expand freely over temperature, held in place by some mechanical support which lets the coil keep its shape.

THERMOCOUPLES

The common electrical method of temperature measurement uses the thermocouple, when two dissimilar metal wires are joined at both ends, an emf will exist between the two junctions, if the two junctions are at different temperatures. This phenomenon is called Setback effect. If the temperature of one junction is known then the temperature of the other junction may be easily calculated using the thermoelectric properties of the materials. The known temperature is called reference temperature and is usually the temperature of ice. Potential (emf) is also obtained if a temperature gradient along the metal wires. This is called Thomson effect and is generally neglected in the temperature measuring process. If two materials are connected to an external circuit in such a way that current is drawn, an emf will be produced. This is called as Peltier effect. In temperature measurement, setback emf is of prime concern since it is dependent on junction temperature.

The thermocouple material must be homogeneous. A list of common Thermocouple materials in decreasing order of emf chrome, iron and copper platinum – 10% rhodium, platinum, alumel and constantan (60% copper and 40% nickel). Each material is thermoelectrically positive with respect to the below it and negatives with respect those above.

The material used in the Thermocouple probe is:

1. Iron – Constantan (Type J)
2. Copper – Constantan (Type T)
3. Chromyl – Alumel (Type K)

PANEL DETIALS:

MAINS ON INDICATOR: To indicate the Power given to the system.

CONSOLE ON SWITCH: Provided to activate the system.

TEMPERATURE INDICATORS: To indicate the Temperatures of various sensors.

WATER BATH SETTINGS: To Set the Temperature of Water bath.

SOFTWARE: FACILITATES TO DO THINGS IN COMPUTER FORMAT.

PREPARATION OF EQUIPMENT:

1. Fill the water into the Water Bath to the required level.
2. Make sure the water bath is in off condition.
3. Connect the instrument to 1ph, 230V AC supply which is having proper earthing.

LIMITATIONS

Maximum Temperature: up to 100deg

PROCEDURE:

1. Switch on the Console to activate the Panel and then switch on the water bath.
2. Set the required Temperature of the bath
3. Select the Sensor under study {however other sensors will also be working condition, this is only done to concentrate on particular sensor type}and Note the readings of the selected Sensor indicator for every two degree rise of water bath temperature.
4. Repeat step 2 and step 3 until required within the range.
5. Repeat the above for different sensors.
6. Calculate the error and % error.

TABULATIONS:

Sl. no	Actual Reading, 'R_a' mm	Measured Reading, 'R_m' mm	Error E = (Ra- Rm)	% Error {(Ra- Rm)/Ra}*100

RESULT: ----------

Note: Plot a graph between Ra Vs Rm

APPLICATIONS:

1. To control of gas flow
2. In electric iron boxes
3. In domestic ovens

VIVA-VOCE QUESTIONS:

1. What are the types of temperature measuring instruments?
2. The lowest temperature limit for mercury-in-glass thermometer is--------
3. Which substance can be used as thermometric substance for temperature measurements Below-40^0C?
4. What should be the shape of an ideal thermometer bulb?
5. Thermocouples are generally used for temperature measurements up to---------
6. Which thermocouple can measure temperature in a comparatively high range?

7. Disappearing filament type optical pyrometer works on the principle of----------

8. In optical pyrometer absorption filter is used to----------

9. Which thermometer is most suitable for the measurement of surface temperature?

10. Which metal/ non-metal has the highest temperature range?

8. ROTAMETER SETUP

i. AIM:

The experiment is conducted to know how to

a. **Calibrate Rotameter** at different flow rate.

ii. PROCEDURE:

1. Fill in the sump tank with clean water.
2. Keep the delivery valve closed.
3. Paste the Log sheet from zero marking on the rotameter to its full height and make marking if necessary.
4. Connect the power cable to 1Ph, 220V, 10 Amps with earth connection.
5. Switch on the pump & open the delivery valve.
6. Adjust the flow through the control valve of the pump.
7. Set the height from the log sheet sticked on the rotameter and note the **height in cm** the **rotameter reading in lpm**.
8. Note down the differential head reading in the Manometer. (Expel if any air is the by opening the drain cocks provided with the Manometer.)
9. Operate the Butterfly valve to note down the collecting tank reading against the known time and keep it open when the readings are not taken.
10. Change the flow rate and repeat the experiment.

iii. OBSERVATIONS:

Sl. No	Time for 'R' cm rise in water 'T' sec	Rotameter Reading, 'Q' Lpm	Height measured on Log scale, cm
1			
2			
3			
4			

iv. CALCULATIONS:

1. Theoretical Discharge, Q_{TR}

$$Q_{TR} = \frac{Q}{1000\,x\,60}\ m^3/s$$

Where,

Q = Rotameter reading in LPM

2. Actual Discharge, Q_A

$$Q_A = \frac{A\,x\,R}{t\,x\,100}\ m^3/s$$

Where,

A = Area of collecting tank = 0.125 m².

R = Rise in water level of the collecting tank, cm.

t = time for 'R' cm rise of water, sec

100 = Conversion from cm to m.

3. Co – efficient of discharge, C_D

$$C_d = \frac{Q_A}{Q_{TH}}$$

Where,

Q_A = Actual Discharge.

Q_{TH} = Theoretical Discharge from Venturi or Rotameter.

v. TABULAR COLUMNS AND GRAPHS:

A. For Rotameter:

Height measured on Log scale, cm	Actual Discharge Q_A m³/sec	Theoretical Discharge Q_{TR} m³/sec	Coefficient of Discharge 'C_D'	Average 'C_D'

B. Graphs

Draw graph of **Actual discharge Vs Height** on Log scale

Draw graph of **Theoretical Discharge Vs Height** on log scale

vi. RESULTS:

For Rotameter:

1. Actual Discharge, Q_A = m^3/s
2. Theoretical Discharge, Q_T = m^3/s
3. Co – efficient of Discharge, C_d =

4. PRECAUTIONS

1) Do not run the pump dry.
2) Clean the tanks regularly, say for every 15days.
3) Do not run the equipment if the voltage is below 180V.
4) Check all the electrical connections before running.
5) Before starting and after finishing the experiment the main control valve should be in close position.
6) Do not attempt to alter the equipment as this may cause damage to the whole system.

Note: For any further clarifications on how to run the equipment or for up gradation, please contact us.

COURSE MATERIAL

IV Year B. Tech I- Semester

MECHANICAL ENGINEERING

AY: 2025-26

COMPUTER AIDED ENGINEERING LAB

R22A0390

Prepared by:
Dr. B. Sandhya Rani
Assoc. Professor

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

DEPARTMENT OF MECHANICAL ENGINEERING

(Autonomous Institution-UGC, Govt. of India)
Secunderabad-500100, Telangana State, India.
www.mrcet.ac.in

MRCET CAMPUS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(AUTONOMOUS INSTITUTION - UGC, GOVT. OF INDIA)

Affiliated to JNTUH; Approved by AICTE, NBA-Tier 1 & NAAC with A-GRADE | ISO 9001:2015

Maisammaguda, Dhulapally, Komaplly, Secunderabad - 500100, Telangana State, India

LABORATORY MANUAL & RECORD

Name:.....

Roll No:..... Branch:.....

Year:.....Sem:.....

MRCET CAMPUS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(AUTONOMOUS INSTITUTION - UGC, GOVT. OF INDIA)

Affiliated to JNTUH, Approved by AICTE, NBA-Tier 1 & NAAC with A-GRADE | ISO 9001:2015

Maisammaguda, Dhulapally, Kompally, Secunderabad - 500100, Telangana State, India

Certificate

Certified that this is the Bonafide Record of the Work Done by

Mr./Ms.....Roll.No.....of

B.Tech year..... Semester for Academic year 2025-2026

in.....Laboratory.

Date: Faculty Incharge HOD

Internal Examiner External Examiner

INDEX

S.No	Date	Name of the Activity/Experiment	Grade/ Marks	Faculty Signature

INDEX

S.No	Date	Name of the Activity/Experiment	Grade/ Marks	Faculty Signature

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

DEPARTMENT OF MECHANICAL ENGINEERING

CONTENTS

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

VISION

- To establish a pedestal for the integral innovation, team spirit, originality and competence in the students, expose them to face the global challenges and become technology leaders of Indian vision of modern society.

MISSION

- To become a model institution in the fields of Engineering, Technology and Management.
- To impart holistic education to the students to render them as industry ready engineers.
- To ensure synchronization of MRCET ideologies with challenging demands of International Pioneering Organizations.

QUALITY POLICY

- To implement best practices in Teaching and Learning process for both UG and PG courses meticulously.
- To provide state of art infrastructure and expertise to impart quality education.
- To groom the students to become intellectually creative and professionally competitive.
- To channelize the activities and tune them in heights of commitment and sincerity, the requisites to claim the never – ending ladder of **SUCCESS** year after year.

For more information: www.mrcet.ac.in

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

www.mrcet.ac.in

Department of Mechanical Engineering

VISION

To become an innovative knowledge center in mechanical engineering through state-of-the-art teaching-learning and research practices, promoting creative thinking professionals.

MISSION

The Department of Mechanical Engineering is dedicated for transforming the students into highly competent Mechanical engineers to meet the needs of the industry, in a changing and challenging technical environment, by strongly focusing in the fundamentals of engineering sciences for achieving excellent results in their professional pursuits.

Quality Policy

- ✓ To pursuit global Standards of excellence in all our endeavors namely teaching, research and continuing education and to remain accountable in our core and support functions, through processes of self-evaluation and continuous improvement.
- ✓ To create a midst of excellence for imparting state of art education, industry-oriented training research in the field of technical education.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

www.mrcet.ac.in

Department of Mechanical Engineering

PROGRAM OUTCOMES

Engineering Graduates will be able to:

1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **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.
3. **Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **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.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and teamwork**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

12. **Life-long learning**: Recognize the need for and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

PSO1 Ability to analyze, design and develop Machine learning systems to solve the Engineering problems by integrating design and manufacturing Domains.

PSO2 Ability to succeed in competitive examinations or to pursue higher studies or research.

PSO3 Ability to apply the learned Mechanical Engineering knowledge for the Development of society and self.

Program Educational Objectives (PEOs)

The Program Educational Objectives of the program offered by the department are broadly listed below:

PEO1: PREPARATION

To provide sound foundation in mathematical, scientific and engineering fundamentals necessary to analyze, formulate and solve engineering problems.

PEO2: CORE COMPETANCE

To provide thorough knowledge in Mechanical Engineering subjects including theoretical knowledge and practical training for preparing Artificial models pertaining to Automobile Engineering, Element Analysis,Production Technology,Mechatronics etc.,

PEO3: INVENTION, INNOVATION AND CREATIVITY

To make the students to design, experiment, analyze, interpret in the core field with the help of other inter disciplinary concepts wherever applicable.

PEO4: CAREER DEVELOPMENT

To inculcate the habit of lifelong learning for career development through successful completion of advanced degrees, professional development courses, industrial training etc.

PEO5: PROFESSIONALISM

To impart technical knowledge, ethical values for professional development of the student to solve complex problems and to work in multi-disciplinary ambience, whose solutions lead to significant societal benefits.

GENERAL LABORATORY INSTRUCTIONS

1. Students are advised to come to the laboratory at-least 5 minutes before (to the starting time), those who come after 5minutes will not be allowed into the lab.
2. Plan your task properly much before to the commencement, come prepared to the lab with the synopsis / program / experiment details.
3. Student should enter into the laboratory with:

   a) Laboratory observation notes with all the details (Problem statement, Aim, Algorithm, Procedure, Program, Expected Output, etc.,) filled in for the lab session.

   b) Laboratory Record updated upto the last session experiments and other utensils (if any) needed in the lab.

   c) Proper Dress code and Identity card.
4. Sign in the laboratory login register, write the TIME-IN, and occupy the computer system allotted to you by the faculty.
5. Execute your task in the laboratory, and record the results/output in the lab observation notebook, and get certified by the concerned faculty.
6. All the students should be polite and cooperative with the laboratory staff, must maintain the discipline and decency in the laboratory.
7. Computer labs are established with sophisticated and high end branded systems, which should be utilized properly.
8. Students / Faculty must keep their mobile phones in SWITCHED OFF mode during the lab sessions. Misuse of the equipment, misbehaviors with the staff and systems etc., will attract severe punishment.
9. Students must take the permission of the faculty in case of any urgency to go out; if anybody found loitering outside the lab / class without permission during working hours will be treated seriously and punished appropriately.
10. Students should LOG OFF/ SHUT DOWN the computer system before he/she leaves the lab after completing the task (experiment) in all aspects. He/she must ensure the system / seat is kept properly.

IV Year B.Tech – ME - I Semester

(R22A0390) COMPUTER AIDED Engineering LAB

COURSE OBJECTIVES:

1. To analyze the various mechanical components in both static conditions

2. To impart the students with necessary computer aided analysis skills.

3. To analyze the various mechanical components in the dynamic conditions.

4. Simulation of mechanical components by visualization software's.

5. To impart the knowledge on program-based simulation for solving the problems.

LIST OF EXPERIMENTS

1. Determination of deflections component and principal and Von-mises stresses in plane stress and Axi symmetric components.

2. Determination of deflections component and principal and Von-mises stresses in plane strain and Axi symmetric components.

3. To determine the stresses acting on a cantilever beam with a point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.

4. To find the displacement, maximum, minimum stresses induced in a given cantilever beam and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.

5. To determine the nodal deflections, reaction forces, and stress of the in determinate truss system when subjected to point loads, E = 210 GPa, A = 0.1 m^2.

6. To perform a Modal Analysis of Cantilever beam for natural frequency determination. Modulus of elasticity = 200GPa, Density = 7800 Kg/m^3, Poison ratio = 0.27

Note: At least 06 experiments are to be conducted.

Any Three Software Packages from the following:

Use of Auto CAD, CATIA, Creo, ANSYS, SCILAB, Open FOAM, Matlab, NISA, CAEFEM, etc.

Course Outcomes

1. Understand the various types of analysis in Ansys.

2. Able to do the Simulation of mechanical components by visualization software

3. Understand the dynamic analysis of the various components and their properties.

4. Understand the concept of simulation using the program-based software.

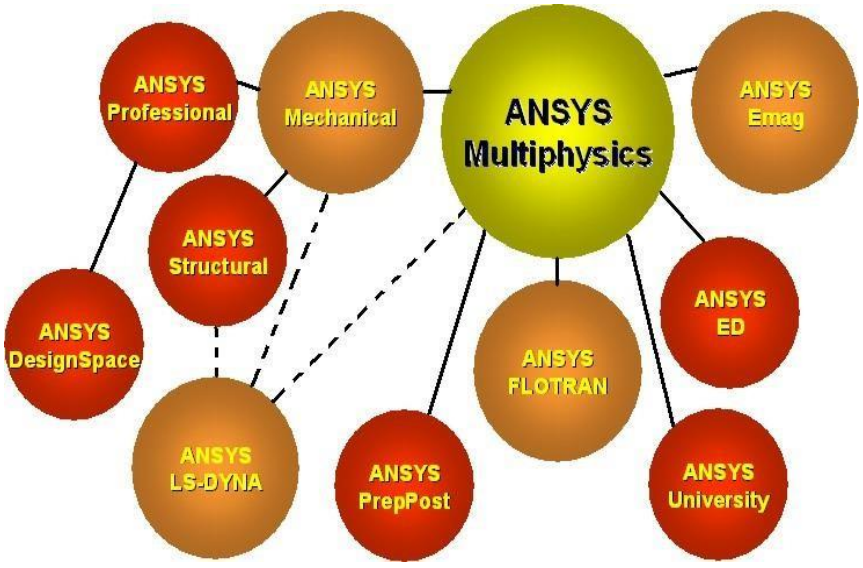
5. Understand the solving and analyzing the mechanical components using empirical equations.

INTRODUCTION TO ANSYS

ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

Why Ansys?

- ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering:
  - ✓ Structural
  - ✓ Thermal
  - ✓ Fluid (CFD, Acoustics, and other fluid analyses)
  - ✓ Low- and High-Frequency Electromagnetics
- A partial list of industries in which ANSYS is used:
  - ✓ Aerospace--- Electronics & Appliances
  - ✓ Automotive--- Heavy Equipment & Machinery
  - ✓ Biomedical--- MEMS - Micro Electromechanical Systems
  - ✓ Bridges & Buildings--- Sporting Goods
- ANSYS Multiphysics is the flagship ANSYS product which includes all capabilities in all engineering disciplines.
  - ✓ ANSYS Classic Environment for exposure to all ANSYS functionality



- There are three main component products derived from ANSYS Multiphysics:
  - ✓ ANSYS Mechanical – structural & thermal capabilities
  - ✓ ANSYS Emag – electromagnetics
  - ✓ ANSYS FLOTRAN – CFD capabilities
- Other product lines:
  - ✓ ANSYS LS-DYNA – for highly nonlinear structural problems
  - ✓ ANSYS Professional – linear structural and thermal analyses, a subset of ANSYS Mechanical capabilities
  - ✓ ANSYSDesign Space – linear structural and steady state thermal analyses, a subset of ANSYS Mechanical capabilities in the Workbench Environment.

Structural analysis: is used to determine deformations, strains, stresses, and Reaction forces.

- **Static analysis:**
  - ✓ Used for static loading conditions.
  - ✓ Nonlinear behavior such as large deflections, large strain, contact, plasticity, hyper elasticity, and creep can be simulated.
- **Dynamic analysis:**
  - ✓ Includes mass and damping effects.
  - ✓ Modal analysis calculates natural frequencies and mode shapes.
  - ✓ Harmonic analysis determines a structure's response to sinusoidal loads of known amplitude and frequency.
  - ✓ Transient Dynamic analysis determines a structure's response to time-varying loads and can include nonlinear behavior.
- **Other structural capabilities**
  - ✓ Spectrum analysis
  - ✓ Random vibrations
  - ✓ Eigen value buckling
  - ✓ Substructuring, submodeling
- **Explicit Dynamics with ANSYS/LS-DYNA**
  - ✓ Intended for very large deformation simulations where inertia forces are dominant.
  - ✓ Used to simulate Impact, crushing, rapid forming, etc.

Thermal analysis: is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux. All three primary **heat transfer modes** can be simulated:

Conduction, convection, radiation.

- Steady-State – Time dependent effects are ignored.
- Transient – To determine temperatures, etc. as a function of time.
  – Allows phase change (melting or freezing) to be simulated.

Electromagnetic analysis: is used to calculate magnetic fields in electromagnetic devices.

Static and low-frequency electromagnetics:

– To simulate devices operating with DC power sources, low-frequency AC, or low-frequency transient signals.
Example: solenoid actuators, motors, transformers

– Quantities of interest include magnetic flux density, field intensity, magnetic forces and torques, impedance, inductance, eddy currents, power loss, and flux leakage.

Computational Fluid Dynamics (CFD): To determine the flow distributions and temperatures in a fluid.

– ANSYS/FLOTRAN can simulate laminar and turbulent flow, compressible and incompressible flow, and multiple species.
Applications: aerospace, electronic packaging, automotive design.

– Typical quantities of interest are velocities, pressures, temperatures, and film coefficients.

Working in ANSYS

1.1. Opening ANSYS SESSION:

Ansys can be opened in Windows Operating System through

- Start>programs>Ansys18>Interactive
- Start>programs>Ansys18>Run Interactive
- Start>programs>Ansys18>Batch

The Interactive Option is used in the very beginning of Ansys Session to set

- Working Directory
- Default File Name
- Graphics driver
- Data Space
- Workspace

- ❖ Menus to be visible
- ❖ Command Line Arguments

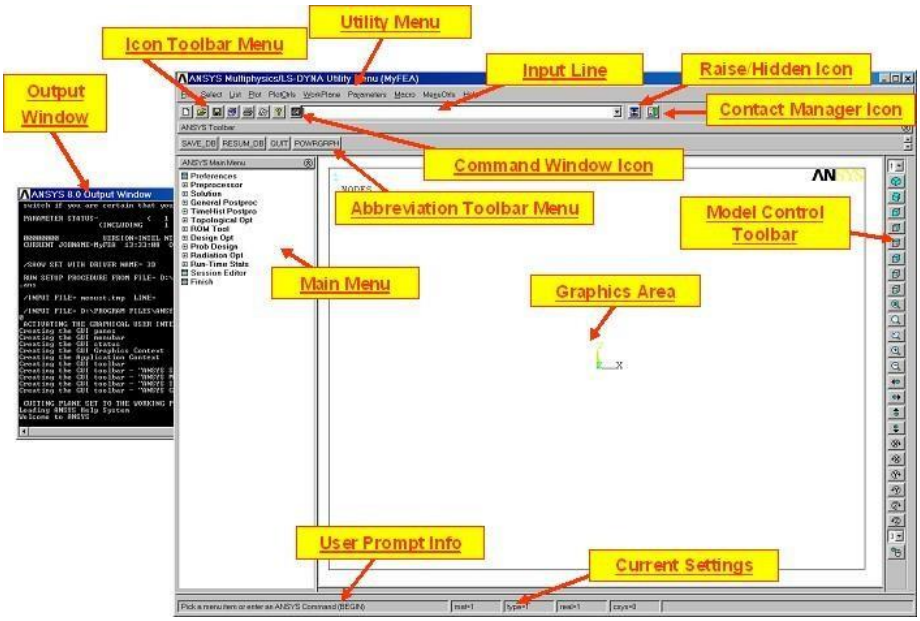
Run-Interactive directly opens the Ansys Graphical user Interface (GUI)

Batch Utility is used to run the Programs Background.

MODELLING APPROACH

- **Bottom-up approach:** Creation of model by defining the geometry of the structure with nodes and elements
- **Top-down approach:** Building a solid model using a 3D CAD program and then dividing the model into nodes and elements

1.2. ANSYS Menu:

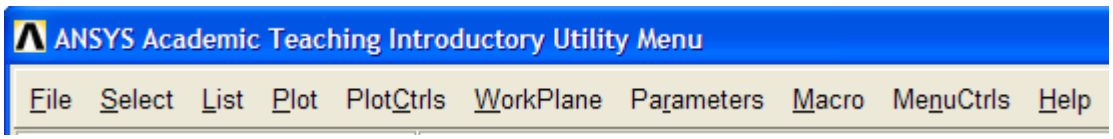


By Default, ANSYS opens 6 Menus. They are

1. Utility Menu
2. Main Menu
3. Input Window
4. Tool Bar
5. Graphics Window
6. Output Window

Utility Menu

This menu contains all important options as follows



A. File: The file contains

- **Clear & Start:** To clear the database &Start a new job
- **Resume from:** To resume the previously stored job
- **Save as:** Save the database as filename.db

- **Read Input from:** if input is taken from Outside programmed file
- **Switch Output:** To external file or by default files in *.iges format is supported without any additional software. BY CATIA, UG, PRO-E you can import the geometry
- **Export:** To export to use in other software's.
- **Exit:** To close the Ansys Session.

B. Select: This is very important option for viewing the results or applying the boundary conditions. The parts of the model can be selected and can manipulate for data. This option contains

- **Entities:** Entities to be selected like key points, lines, nodes, elements, areas, volumes, etc
- **Components:** Naming and grouping the selected components.
- **Everything:** Selecting only that part
- **Everything below:** Selecting the entities below that.

C. List: This option can be used to listing the elements, nodes, volumes, forces, displacements etc.

D. Plot: This option is used to plot the areas, volumes, nodes, elements etc.

E. Plot Controls: This option is very important and contains

- **Pan Zoom Rotate:** It opens another menu through which zooming and rotation of the model is possible.
- **View Setting:** By default Z plane is perpendicular to the viewer. By this view option, view settings can be changed.
- **Numbering:** this is useful for setting on/off the entity numbering
- **Symbols:** to view the applied translations, forces, pressures, etc. this option should be used to set them on.
- **Style:** Sectioning, vector arrow sizing and real structural appearances is possible through this.
- **Window Controls:** Window positioning (Layout) is possible with this.
- **Animate:** Animation can be done for the output data using this.
- **Device Options:** Wireframe models can be observed through this.
- **Hard Copy:** data can be sent either to printer or any external file.

- **Capture Image:** To capture the graphics window output to a *.bmp image.
- **Multiplot Window Layout:** To view the results in more than one window.
-

F. Work plane: By default, Z Plane is perpendicular for data input. For any changes in the global X,Y & Z planes, the work plane should be rotated to create the model or view

G. Parameters: These are the scalar parameters represented with values.

Eg: b=10

H. Macros: These are grouping of Ansys commands to fulfill particular work. These can be taken equivalent to C, C++ & Java Functions.

I. Menu Controls: This can be used to set on/off the menus.

J. Help: For all the help files related to commands and topics

Main Menu: This menu contains

- **Pre-processor:** This sub option can be used to build and mesh the model through proper element selection and boundary conditions.
- **Solution:** this option can be used solve the matrix equation through proper solver.
- **Post Processor:** This option is used to interpret the results.
- **Design Optimization:** This option is used to optimize the structure.
- **Time History Processor:** For dynamic problems, results can be viewed through this option.
- **Run Stats:** This option can be used to find the status of the model, time it
- take s for execution, computer processor capabilities, wave front size etc.

Input Window: This can be used to input commands or named selection.

Tool Bar: This contains options like saving the file, resuming the file database, Quitting the Ansys session and Graphics Type.

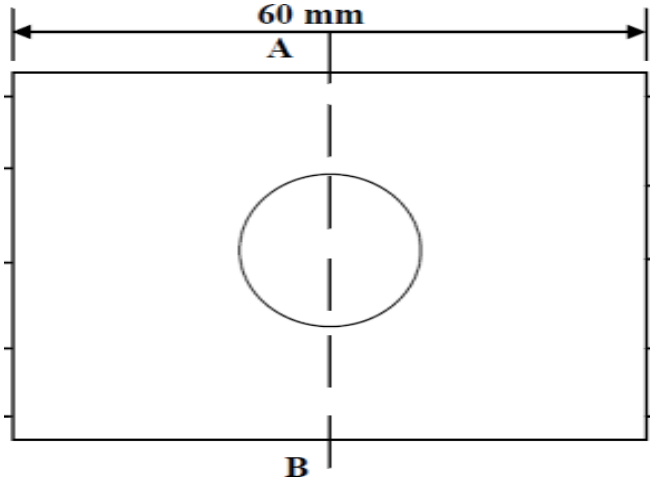
Graphics Window: This is where the model creation and plotting of results carried out.

Output Window: This shows the status of the work being carried out.

EXPERIMENT NO. 1

AIM: Determination of deflections component and principal and Von-mises stresses in plane stress and Axisymmetric components.

In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use t = 1 mm). E = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 20 mm, Density = 7800 Kg/m3. Analysis assumption – plane stress with thickness is used and do with axisymmetric



AIM: To perform a stress analysis of a rectangular plate with circular hole using analysis software ANSYS.

SYSTEM CONFIGURATION:

- Ram : 2 GB
- Processor : Intel CORE i3
- Operating system : Window XP Service Pack 3
- Software : ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – Solid – Quad 4 node182 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

Step 4: Real constants – Add – ok – real constant set no – 1 – Thickness – 1 – ok.

Step 5: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 210e9– PRXY – 0.3– ok – close.

Step 6: Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 30, 0,20 ok.

Step 7: Create – Area – Circle – solid circle – X, Y, radius – 0, 0, 10 – ok.

Step 8: Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply – pick area which is to be deleted (circle) – ok.

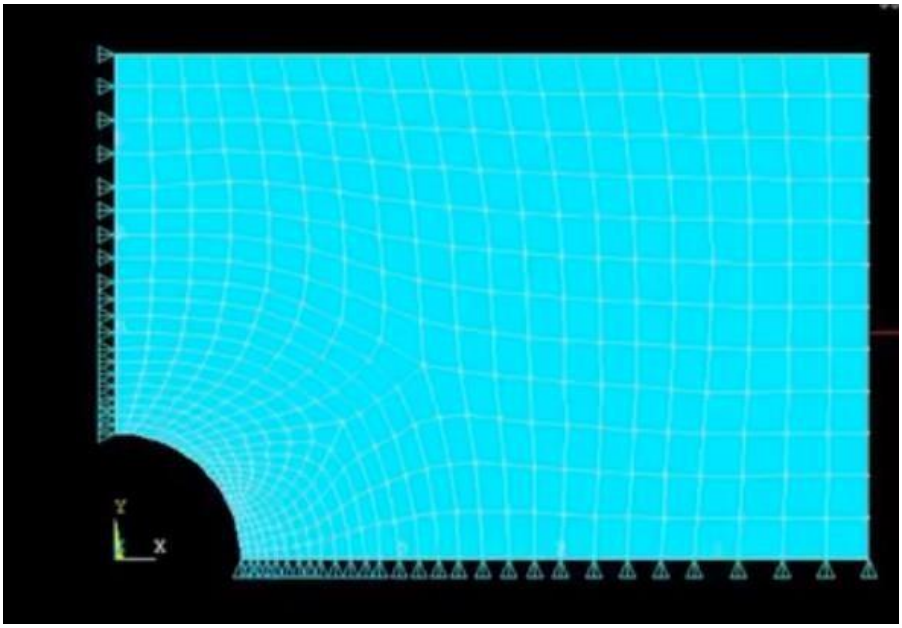
Step 9: Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok.
Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

Solution

Step 10: Solution – Define Loads –loads – apply – Structural – Displacement – Axisymm B.C – on lines – select bottom and left side line – ok.

Step 11: Solution – Define loads – apply – Structural – pressure – on lines – select the right-side line – apply – enter pressure value – 50 (-ve value) –ok.

Step 12: Solve – current LS – ok (Solution is done is displayed) – close.



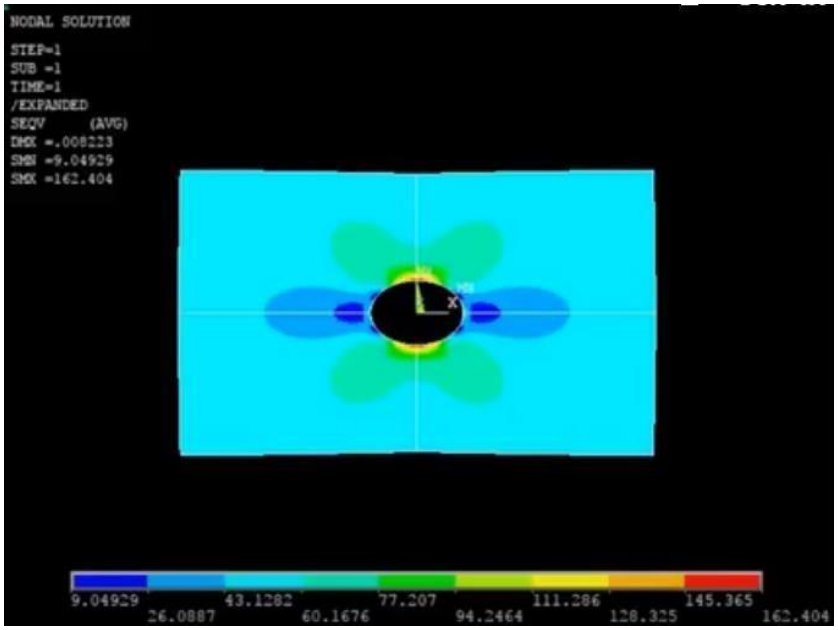
General Post Processor

Step 15: Plot results – contour plot – Element solution – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

Step 16: Plot Ctrls – Animate – Deformed shape – def + undeformed-ok.

Step 17: utility menu – plotctrls – styles- symmetric expansion – periodic/cyclic

symmetry – select (1/4) dihedral sym - ok.



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA QUESTIONS:

1. What are the advantages of axisymmetric elements?
2. Define frontal method for finite element matrices
3. Define mesh plotting
4. Define stress and strain with units
5. Explain the boundary conditions for above problem

CAE LAB

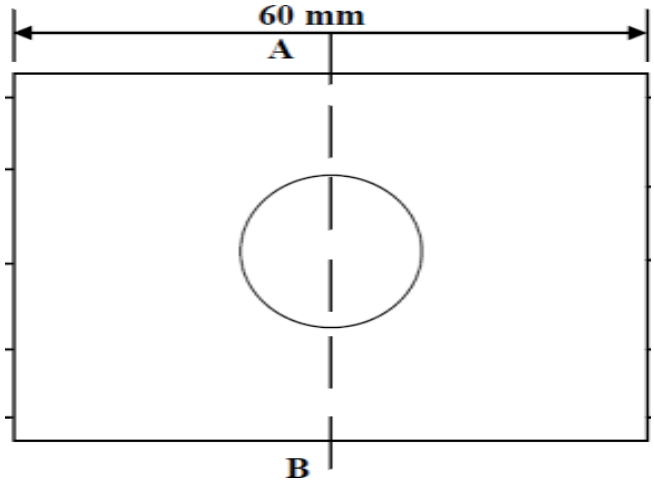
OBSERVATIONS

CAE LAB

EXPERIMENT NO. 2

AIM: Determination of deflections component and principal and Von-mises stresses in plane strain and Axi symmetric components.

In the plate with a hole under plane strain, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use t = 1 mm). E = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 20 mm, Density = 7800 Kg/m3. Analysis assumption – plane strain with thickness is used and do with axisymmetric



AIM: To perform a stress analysis of a rectangular plate with circular hole using analysis software ANSYS.

SYSTEM CONFIGURATION:

- Ram : 2 GB
- Processor : Intel CORE i3
- Operating system : Window XP Service Pack 3
- Software : ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – Solid – Quad 4 node182 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

Step 4: Real constants – Add – ok – real constant set no – 1 – Thickness – 1 – ok.

Step 5: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 210e9– PRXY – 0.3– ok – close.

Step 6: Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 30, 0,20 ok.

Step 7: Create – Area – Circle – solid circle – X, Y, radius – 0, 0, 10 – ok.

Step 8: Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply – pick area which is to be deleted (circle) – ok.

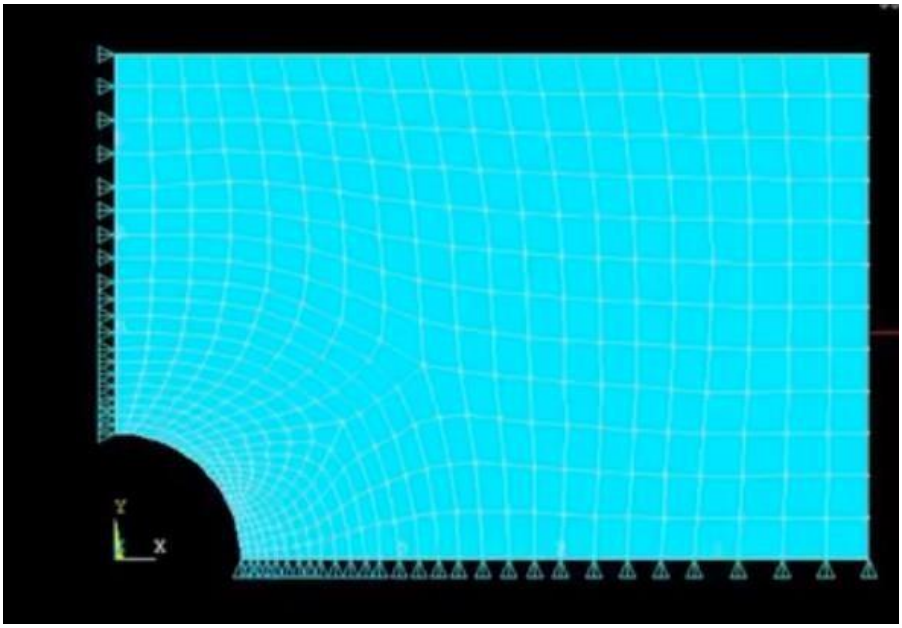
Step 9: Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok.
Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

Solution

Step 10: Solution – Define Loads –loads – apply – Structural – Displacement – Axisymm B.C – on lines – select bottom and left side line – ok.

Step 11: Solution – Define loads – apply – Structural – pressure – on lines – select the right-side line – apply – enter pressure value – 50 (-ve value) –ok.

Step 12: Solve – current LS – ok (Solution is done is displayed) – close.



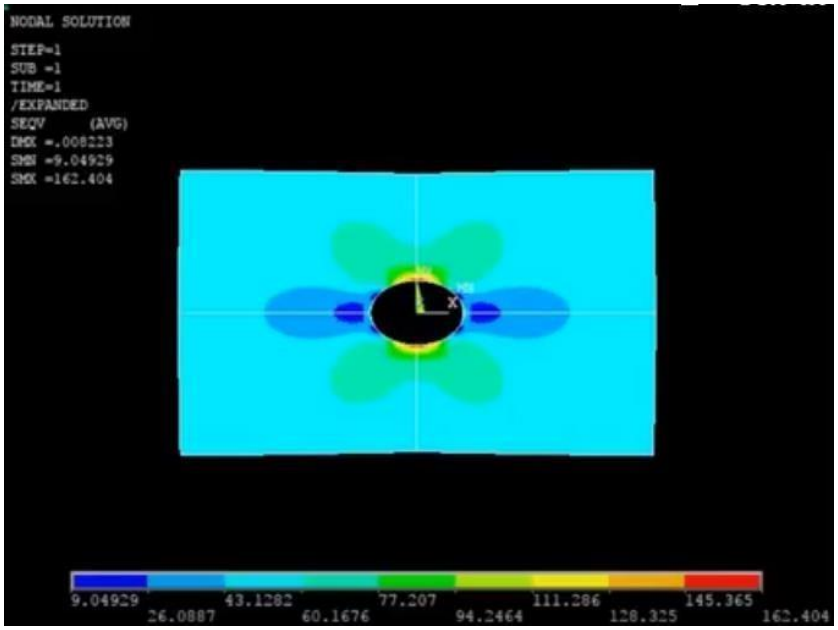
General Post Processor

Step 15: Plot results – contour plot – Element solution – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

Step 16: Plot Ctrls – Animate – Deformed shape – def + undeformed-ok.

Step 17: utility menu – plotctrls – styles- symmetric expansion – periodic/cyclic

symmetry – select (1/4) dihedral sym - ok.



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA QUESTIONS:

6. What are the advantages of axisymmetric elements?
7. Define frontal method for finite element matrices
8. Define mesh plotting
9. Define stress and strain with units
10. Explain the boundary conditions for above problem

CAE LAB

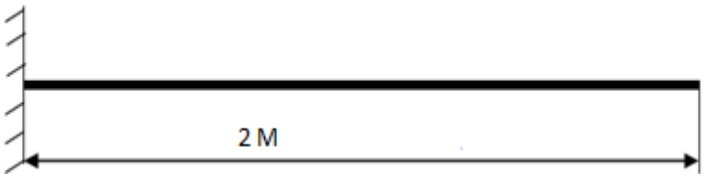
OBSERVATIONS

EXPERIMENT NO. 3-1

Determination of deflection and stresses in 2D beams

Two-Dimensional static linear analysis of a cantilever beam

AIM: To determine the stresses acting on a cantilever beam with a point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.



Young's modulus = 2e5

Poisson's ratio = 0.3

Length of the beam = 2m = 2000mm

Breadth of the beam = 10 cm = 100mm

Height of the beam = 50mm

SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 2e5– PRXY – 0.3 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1St element) - enter B=100, H = 50- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 2000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

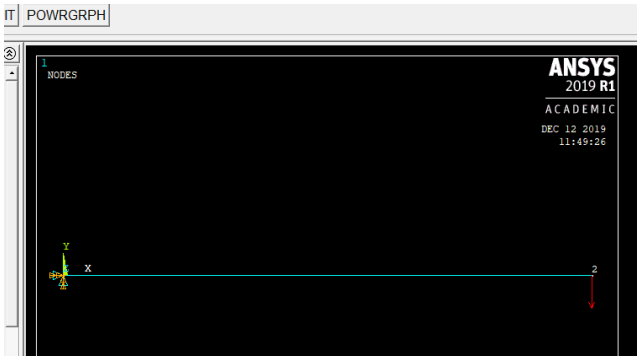
Solution

Step 8: Solution – Analysis Type – New Analysis – Static – ok.

Step 9: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node – apply – DOFs to be constrained – ALL DOF – ok**.**

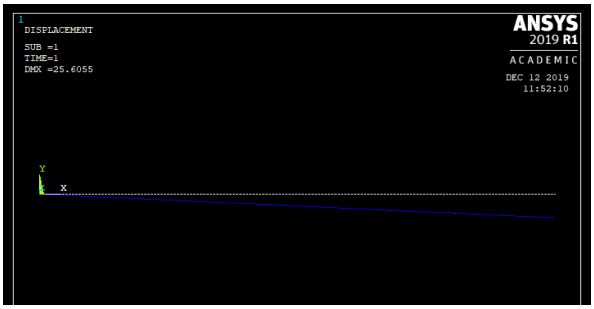
Step 10: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes– Pick 2nd node – apply – direction of force/mom – FY – Real part of force/mom – -10000 – ok.

Step 11: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 12: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'



Step 13: Plot Results – Contour plot – Nodal solution – DOF solution – Y component of displacement – OK

RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA-VOCE QUESTIONS

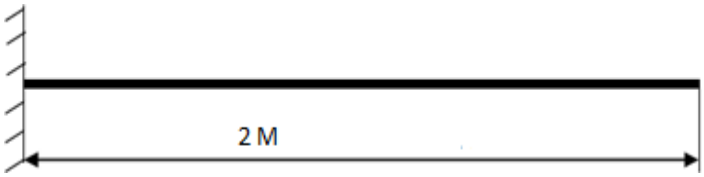
1. What is meant by beam?
2. Explain the types of beams?
3. Define term element
4. Define term node
5. Give example for higher order elements

CAE LAB

OBSERVATIONS

EXPERIMENT NO. 3-2

AIM: To find the displacement, maximum, minimum stresses induced in a given cantilever beam and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



Young's modulus = 2e5

Poisson's ratio = 0.3

Length of the beam = 2m = 2000mm

Breadth of the beam = 10 cm = 100mm

Height of the beam = 50mm

Point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.

SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 2e5– PRXY – 0.3 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1st element) - enter B=100, H = 50- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 2000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

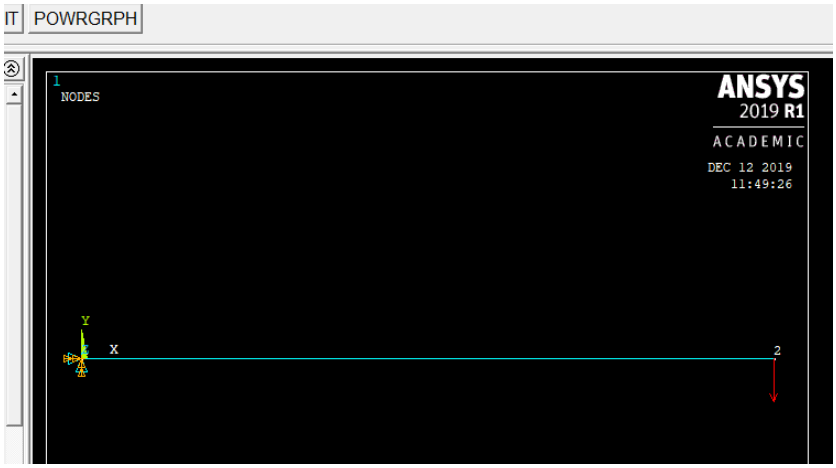
Solution

Step 8: Solution – Analysis Type – New Analysis – Static – ok.

Step 9: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node – apply – DOFs to be constrained – ALL DOF – ok**.**

Step 10: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes– Pick 2nd node – apply – direction of force/mom – FY – Real part of force/mom – -10000 – ok.

Step 11: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 12: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'

Step 13: Element Table – Define Table – Add – By sequence num – SMISC, 2 – Apply

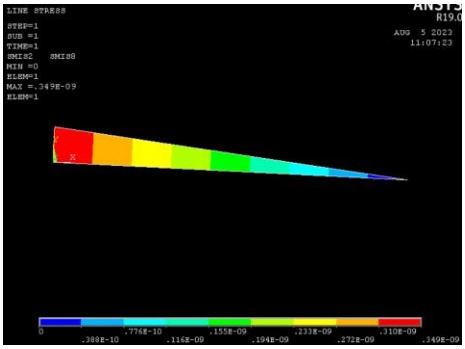
By sequence num – SMISC, 8 – Apply

By sequence num – SMISC, 6 – Apply

By sequence num – SMISC, 12 - ok

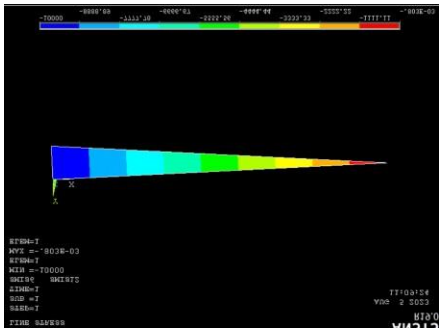
Step 14: Shear Force Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS2, SMIS8



Step 15: Bending Moment Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS6, SMIS12



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA-VOCE QUESTIONS

1. What is meant by beam?
2. Explain the types of beams?
3. What is difference between node and element?
4. What is shear force?
5. What is bending moment?

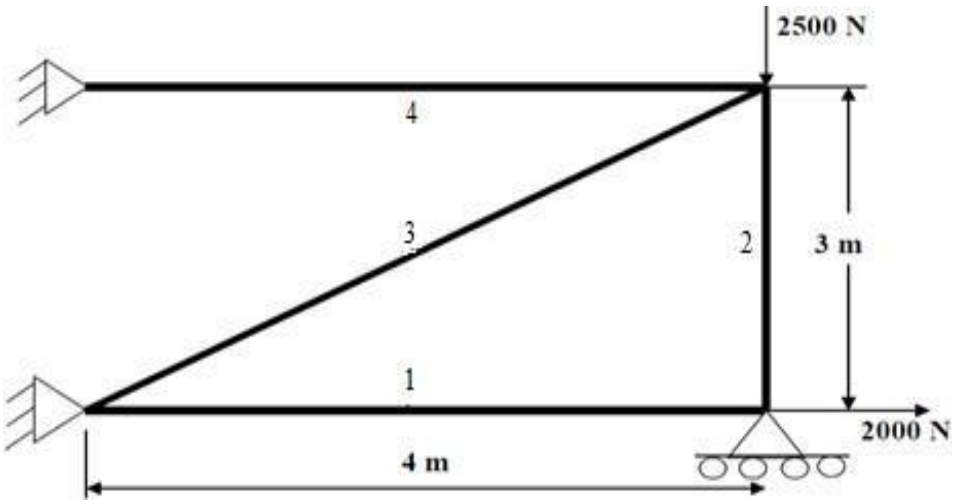
OBSERVATIONS

EXPERIMENT NO. 4

Determination of deflection and stresses in 2D trusses

2-D Static linear analysis of a truss structure

AIM: To determine the nodal deflections, reaction forces, and stress of the indeterminate truss system when subjected to point loads, E = 210 GPa, A = 0.1 m^2



.

SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok- close.

Step 4: Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.

Step 5: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 210e9 – ok – close.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations

= 4,0,0, - ok (second node is created) - Node Number =3 – X, Y, Z Locations = 4,3,0, - ok (Third node is created) - Node Number =4 – X, Y, Z Locations = 0,3,0, - ok (fourth node is created)

Step 7: Create–Elements–Elem Attributes – Material number – 1 – Real constant set number – 1 – ok

Step 8: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply- pick 2 & 3 – apply – pick 3 & 1 – apply –pick 3 & 4 – ok (elements are created through nodes).

Solution

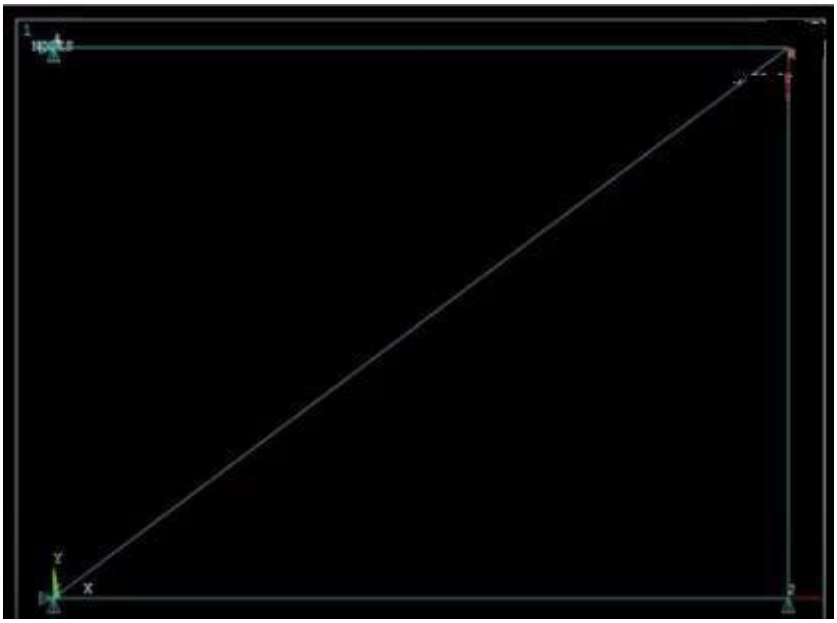
Step 9: Solution – Analysis Type – New Analysis – Static – ok.

Step 10: Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 4 – apply – DOFs to be constrained – All DOF – ok – on Nodes – pick node 2 – apply – DOFs to be constrained – UY – ok.

Step 11: Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok

Step 12: Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.

Step 13: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 14: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'

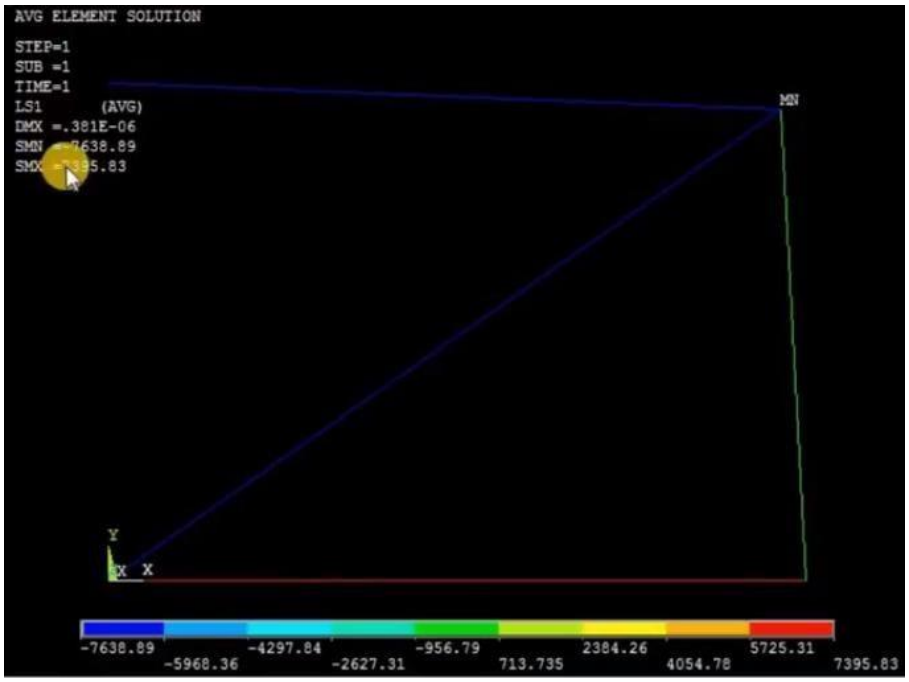
Step 15: Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.

Step 16: Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.

Step 17: Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

Step 18: Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.

Step 19: Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.



RESULT:

Maximum Nodal Displacement (DMX) =

Minimum Stress (SMN) =

VIVA-VOCE QUESTIONS

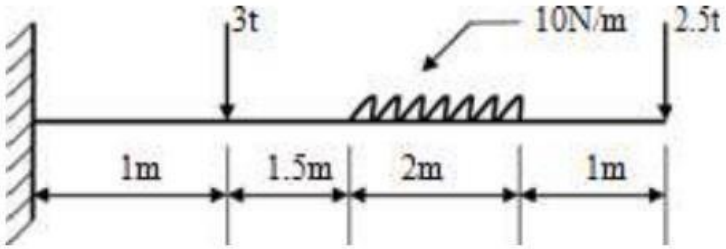
1. What is the difference between beam and truss?
2. Explain shear force and bending moment
3. What is meant by point of contraflexure?
4. Element attributes must be set before meshing the solid model. (T/F)
5. In a plane strain, the strain in the direction of thickness is assumed to be zero.(T/F)

OBSERVATIONS

CAE LAB

EXPERIMENT NO. 4

AIM: To find the displacement, maximum, minimum stresses induced in a given cantilever beam with uniformly distributed load and point loads and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 2e5– PRXY – 0.3 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1st element) - enter B=1, H = 1- preview-ok.

Step 6: Modelling > create > nodes > Inactive CS > (Enter x,y,z values) as shown in the figure:

Node 1**:**0,0,0

Node 2: 1,0,0

Node 3**:**2.5,0,0

Node 4**:**4.5,0,0

Node 5**:**5.5,0,0

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick all the nodes one by one – ok (elements are created through nodes).

Solution

Step 8: Solution – Analysis Type – New Analysis – Static – ok.

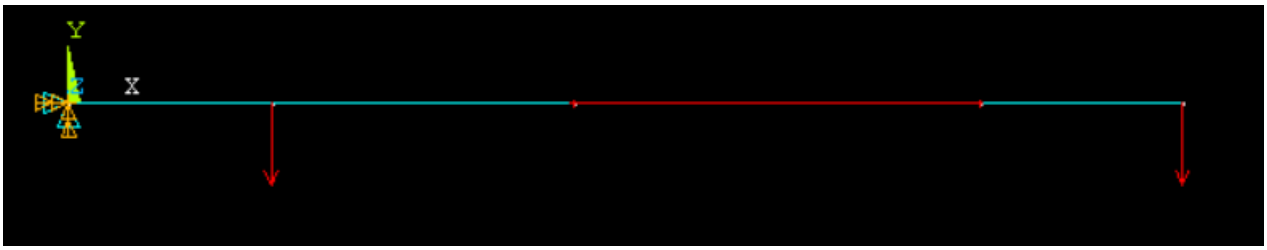
Step 9: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node – apply – DOFs to be constrained – ALL DOF – ok**.**

Step 10: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes– Pick 2nd node – apply – direction of force/mom – FY = -3 – ok.

Step 11: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes– Pick 5th node – apply – direction of force/mom – FY = -2.5 – ok

Step 12: Solution – Define Loads – Apply – Structural – Pressure – On Beam– Select beam between nodes 3, 4 – Enter pressure values as 0.001 at both I & J - ok

Step 13: Solve – current LS – ok (Solution is done is displayed) – close.



General Post Processor

Step 14: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'

Step 15: Element Table – Define Table – Add – By sequence num – SMISC, 2 – Apply

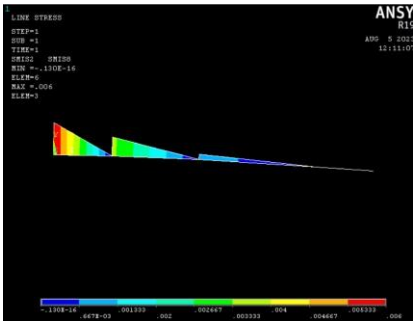
By sequence num – SMISC, 8 – Apply

By sequence num – SMISC, 6 – Apply

By sequence num – SMISC, 12 - ok

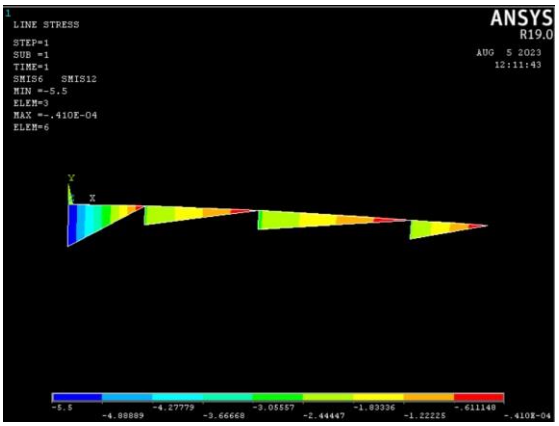
Step 16: Shear Force Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS2, SMIS8



Step 17: Bending Moment Diagram

Plot Results – Contour plot – Line Elem Res – Select SMIS6, SMIS12



RESULT:

Maximum Nodal Displacement (DMX) =

Maximum Stress (SMN) =

VIVA-VOCE QUESTIONS

1. What is nodal solution?
2. What is pre processor?
3. What is post processor?
4. What type of options we use in preferences?
5. What is DOF?

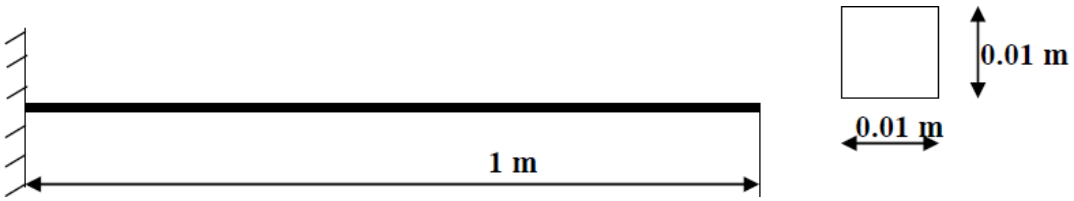
OBSERVATIONS

Experiment 5

Estimation of natural frequencies and mode shapes

AIM: To perform a Modal Analysis of Cantilever beam for natural frequency determination.

Modulus of elasticity = 200GPa, Density = 7800 Kg/m^3, Poison ratio = 0.27



SYSTEM CONFIGURATION:

- Ram: 2 GB
- Processor: Intel CORE i3
- Operating system: Window XP Service Pack 3
- Software: ANSYS (Version19.0)

PROCEDURE:

Step 1: File – clear and start new – do not read file – ok

Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok

Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX– 200e9– PRXY – 0.27– Density – 7800 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1St element) - enter B=10, H = 10- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 1000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Solution

Step 8: Solution – Analysis Type – New Analysis – Modal – ok.

Step 9: Solution – Analysis Type – Analysis options – no of modes to extract – 5 – no of modes to expand – 5 – ok – (use default values) – ok.

Step 10: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node apply – DOFs to be constrained – ALL DOF – ok.

Step 11: Solve – current LS – ok (Solution is done is displayed) – close.

General Post Processor

Step 12: Result Summary – close.

Step 13: Read Results – First Set

Step 14: Plot Results – Deformed Shape – def + undeformed – ok.

Step 15: Plot Ctrls – Animate – Deformed shape – def + undeformed-ok.

Step 16: Read Results – Next Set

Step 17: Plot Results – Deformed Shape – def + undeformed – ok.

Step 18: Plot Ctrls – Animate – Deformed shape – def + undeformed-ok.

RESULT:

Thus, the Model Analysis of Cantilever beam with load for natural frequency is done by using the ANSYS Software.

VIVA QUESTIONS:

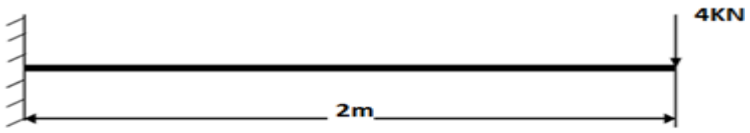
1. What is the total degree of freedom of Ansys commercial package?
2. What are the different menus in ANSYS?
3. What is Work space and Swap space.
4. What is default value of worksp1ace and Swap space?
5. What file format ANSYS can support

OBSERVATIONS

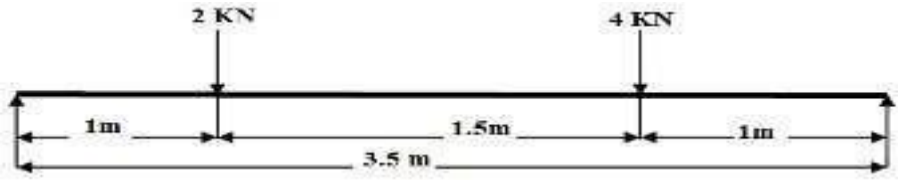
CAE LAB

PRACTICE PROBLEMS

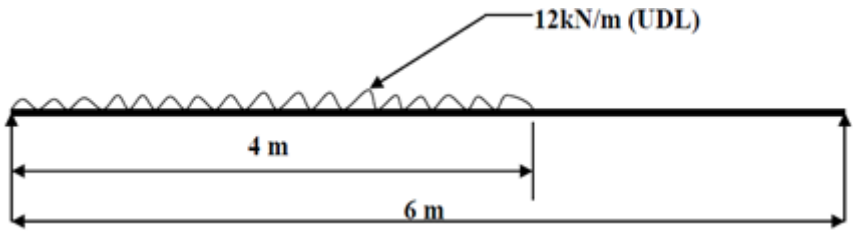
1. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



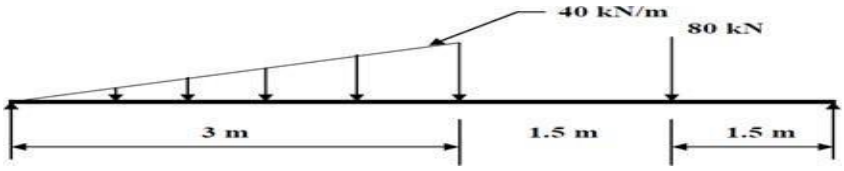
2. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100 mm * 100mm, Young's modulus of210 MPa, Poisson's ratio 0.27.



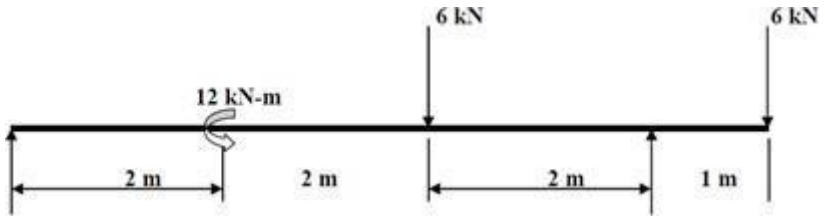
3. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100 mm * 100mm, Young's modulus of210 MPa, Poisson's ratio 0.27.



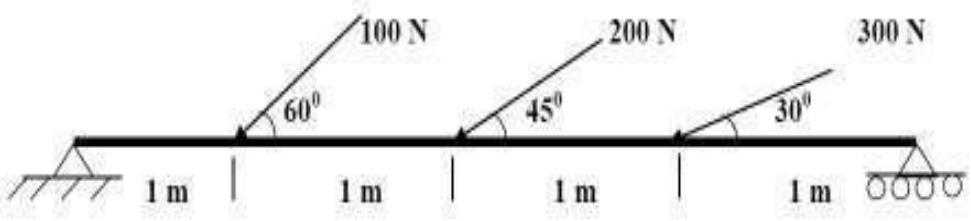
4. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100 mm * 100mm, Young's modulus of210 MPa, Poisson's ratio 0.27.



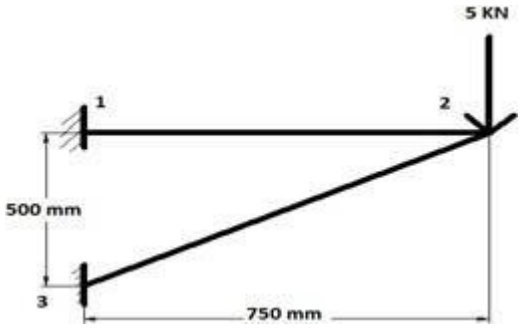
5. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



6. Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



7. Consider the two bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. E = 210 GPa, A = 0.1 m^2



8. In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use t = 1 mm). E = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 10 mm, Analysis assumption – plane stress with thickness is used.