# EXPERIMENT NO: 01

# VERIFICATION OF LAMI'S THEOREM

**AIM:** Verify experimentally the law of Lami's theorem.

**Apparatus:** Drawing board, pulleys, thread, scale, weight hangers, etc

**Principle:** It states that "If three forces are acting at a point which are at equilibrium both in magnitude and direction, then each force is directly proportional to sine of angle between other two forces".

**Formula:** If P, Q and R be the three forces and $\alpha, \beta$ $and$ $\gamma$ be the opposite angles respectively, then by the law we have

$$\frac{P}{sin\alpha} = \frac{Q}{sin\beta} = \frac{R}{sin\gamma}$$

## Procedure:

1) Drawing board is kept vertically and the given two smooth and frictionless pulleys are fixed to it as shown in the given diagram.

2) A thread is made to pass over both the pulleys. Two weight hangers P and Q are taken and are attached to the two free ends of the tread. Another short string is tied to the center of the first string at O. A third weight hanger R is attached to the free end of the short thread.

3) The weight hangers P, Q and R are adjusted such that the system is at rest.

4) The point O is in equilibrium under the action of the three forces P,Q and R acting along the threads.

5) A sheet of paper is fixed on drawing board behind the system. Light is made to fall on the system and The common knot O and the directions of OA,OB and OD are marked on the paper and the paper is taken out from the drawing board.

6) The lines on the paper are extended to meet at common point O. Now measure the angles $\alpha, \beta$ $and$ $\gamma$.

7) Then it can be verified that

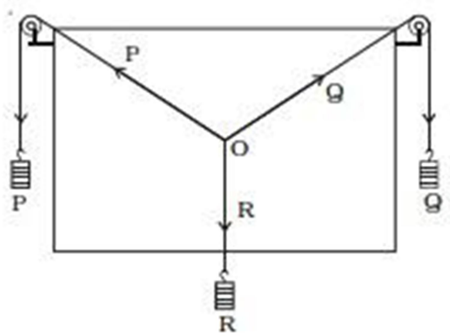
$$\frac{P}{sin\alpha} = \frac{Q}{sin\beta} = \frac{R}{sin\gamma}$$

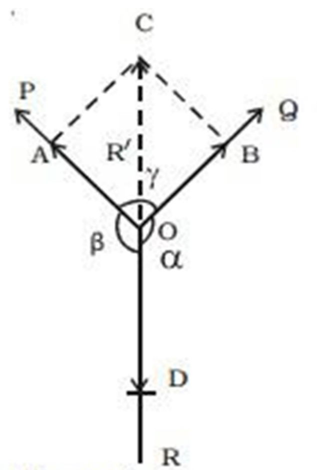
8) Repeat this experiment for different values of P, Q and R and verify the law.

**Result:** Within the given experimental conditions it is observed that

$\frac{\mathbf{P}}{\mathbf{sin\alpha}} = \frac{\mathbf{Q}}{\mathbf{sin\beta}} = \frac{\mathbf{R}}{\mathbf{sin\gamma}}$. Hence Lami's theorem is verified.

## Diagram:




## Tabular Column:

| SI. NO | Forces ( In grams) | | | Angles (In degrees) | | | sinα | sinβ | sinγ | $\frac{P}{\sin\alpha}$ | $\frac{Q}{\sin\beta}$ | $\frac{R}{\sin\gamma}$ |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | P | Q | R | A | B | γ | | | | | | |
| 1 | | | | | | | | | | | | |
| 2 | | | | | | | | | | | | |
| 3 | | | | | | | | | | | | |

## Calculations:

# INTRODUCTION TO ANSYS

ANSYS is a complete FEA (Finite Element Analysis) software package used by engineers worldwide in virtually all fields of engineering.

- Structural
- Thermal
- Fluid including CFD (Computational Fluid Dynamics)
- Electrical / Electrostatics
- Electromagnetics

A partial list of industries in which ANSYS is used

- Aerospace
- Automotive
- Bio-medical
- Construction

ANSYS / Multiphysics is the flagship ANSYS product which include all capabilities in all engineering disciplines. There are three main component products derived from ANSYS

- ANSYS / Mechanical – Structural & Thermal capabilities
- ANSYS / Emag – Electromagnetics
- ANSYS / FLOTRAN – CFD capabilities

**Structural Analysis**

It is used to determine deformations, strains, stresses and reaction forces.

- Static analysis – Used for static loading conditions
- Dynamic analysis – Includes mass and damping effects

***Thermal Analysis***

It is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat gain or lost, thermal gradient and thermal flux. All the three primary modes of heat transfer can be simulated: conduction, convection and radiation.

Steady state – Time dependent effects are ignored

Transient – To determine temperature etc as a function of time

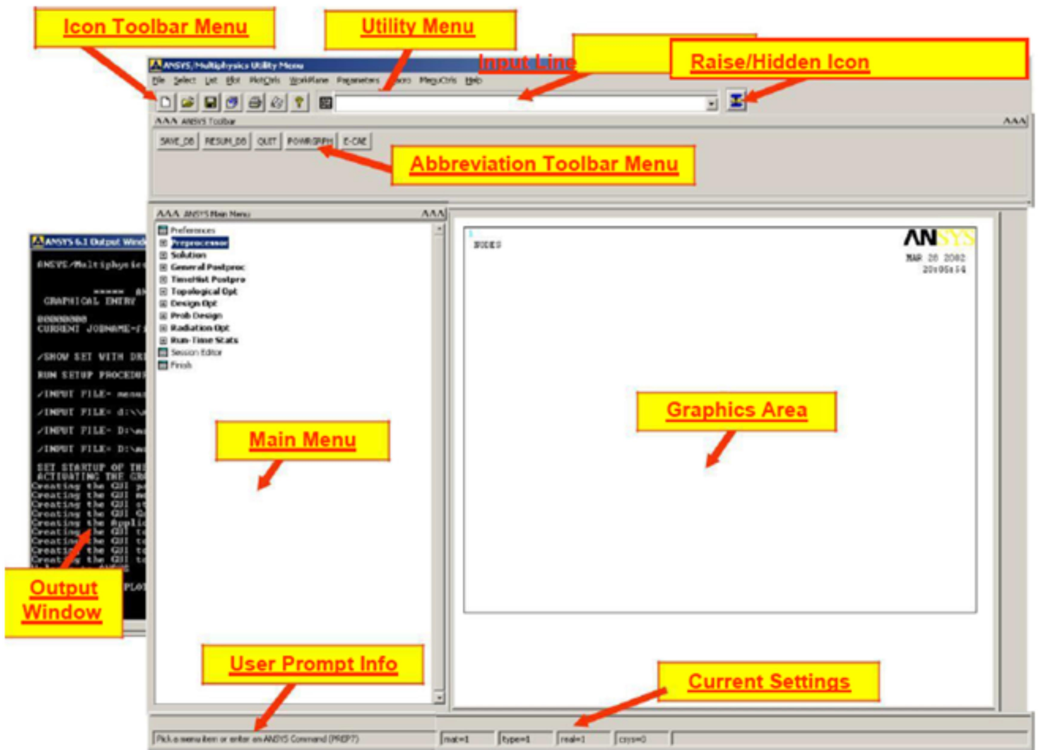
**Computational Fluid Dynamics (CFD)**

It is used to determine the flow distributions and temperatures in a fluid. ANSYS

/ FLOTRAN can simulate laminar and turbulent flow, compressible and incompressible flow and multi species. Typical quantities of interest are velocities, pressure, temperatures and film coefficients.

**Applications**: aerospace, electronic packaging, automotive design.

**The GUI Layout:**




**Utility Menu**

It contains functions which are available throughout the ANSYS session, such as file controls, selecting, graphics controls, parameters and exiting.

**Toolbar Menu**

It contains push buttons for executing commonly used ANSYS commands and functions. Customized buttons can be created.

**Graphics Area**

It displays graphics created in ANSYS or imported into ANSYS.

**Input Line**

It displays program prompt messages and a text field for typing commands. All previously typed commands appear for easy reference and access.

**Main Menu**

It contains the primary ANSYS functions, organized by processors (preprocessor, solution, general postprocessor etc.)

**Output**

It displays text output from the program. It is usually positioned behind the other windows and can be raised to the front when necessary.

**Resume**

This is opening a previously saved database. It is important to know that if you simply resume a database, it doesn't change the job name.

**Plotting**

Contrary to the name, this has nothing to do with sending an image to a plotter or printer. Plotting in ANSYS refers to drawing something in the graphics window. Generally you plot one type of entity (lines, elements etc.) to the screen at a time.

**Plot Controls**

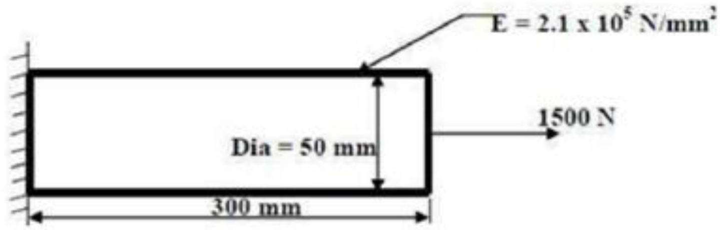
This refers to how you want your plot to look on the screen (shaded, wire frame, entity number on or off, etc). Other plot control functions include sending an image to a graphics file or printer.

# EXPERIMENT NO: 02

## Analysis of Bars of Constant Cross Section Area

**Aim:** To perform displacement and stress analysis for the given bar using ANSYS simulation and analytical expression.

**Problem Description:** A steel rod subjected to tension is modeled by one bar element, as shown in figure. Determine the nodal displacements and the axial stress in each element and reaction forces. E = 2.1 x $10^5$ N/mm2, 0.3 (Poisson's ratio).




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → Link → 3D finitstn 180 → ok → close.
- Real constants → Add/Edit/Delete → Add → ok → Real constant set no → 1 → c/s area →1963.495 → ok → close.
- Material Properties → material Models → Structural → Linear → Elastic → Isotropic →EX → 2.1e5 → PRXY → 0.3 → ok → close.
- Modeling → create → nodes → In Active CS → Apply (first node is created) → x, y, z location in CS → 300, 0, 0 → ok (second node is created).
- Modeling → Create → Elements → Auto numbered → Thru Nodes → pick 1 & 2 → ok (elements are created through nodes)

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Nodes → pick node 1 → Apply → DOFs to be constrained → All DOF → ok.
- Define loads → Apply → Structural →Force/Moment → on Nodes → pick node 2 → Apply →direction of Force/mom → FX → Force/Moment value → 1500 (+ve value) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → LS → and type LS, 1 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- Plot Result → contour plot → Line Element Results → Elem table item at node I → LS1 → Elem table item at node J → LS1 → ok (Line stress diagram will be displayed).
- List Results →Elem table Data → Items to be listed → LS1 → ok. (stress will be displayed with the element numbers)
- List Results→ Reaction Solution → items to be listed → All items → ok (reaction forces will be displayed with the node numbers).
- List Results → Nodal solution → DOF solution → Displacement vector Sum → ok (Deformation will be displayed with the node numbers).

**Step 5**: Plotctrls → Animate → Deformed shape → def+undefromed → ok PlotCtrls → Animate → Deformed results → DOF solution → Displacement Vector sum → ok.

**Comparison between theoretical and ANSYS value:**

| 1 | Theoretical | | ANSYS | |
|---|---|---|---|---|
| DEFORMATION | NODE | mm | NODE | mm |
| | 1 | 0 | 1 | 0 |
| | 2 | $1.0913 \times 10^{-3}$ | 2 | $1.0913 \times 10^{-3}$ |

| 1 | Theoretical | | ANSYS | |
|---|---|---|---|---|
| STRESS | ELEMENT NO | $N/mm^2$ | ELEMENT NO | $N/mm^2$ |
| | 1 | 0.76394 | 1 | 0.76394 |

| 1 | Theoretical | | ANSYS | |
|---|---|---|---|---|
| REACTION | NODE | N | NODE | N |
| | 1 | -1500 | 1 | -1500 |
| | 2 | 1500 | 2 | 1500 |

**Conclusion:** ANSYS simulation and the software results are near to theoretical results.

**THEORETICAL SOLUTION:**

Given,

Diameter, d = 50mm

Young's Modulus, E = $2.1 \times 10^5$ N/mm2

Load, P = 1500 N (reaction at node 2 i.e. R2)

Area of cross section, A = $\frac{\pi d^2}{4} = \frac{\pi x 50^2}{4}$ = 1963.495 mm2

**Deformation:**

At node 1 = 0

At node 2 = $\frac{PL}{AE} = \frac{1500x300}{1963.495x2.1x10^5}$ = $1.0913 \text{ X } 10^{-3}$ mm

**Stress:**

For Element 1, $\sigma 1 = \frac{P}{A} = \frac{1500}{1963.495}$ = 0.76394 N/mm$^2$

**Reaction:**

R1+R2=0

R1 + 1500 = 0

Reaction at node 1, R1 = -1500 N

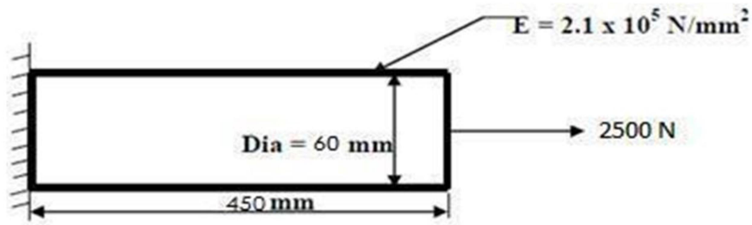
Reaction at node 2, R2 = 1500 N

# EXPERIMENT NO: 03

## Analysis of Bars of Constant Cross Section Area

**Aim:** To perform displacement and stress analysis for the given bar using ANSYS simulation and analytical expression.

**Problem Description:** A steel rod subjected to tension is modeled by one bar element, as shown in figure. Determine the nodal displacements and the axial stress in each element and reaction forces. E = 2.1 x $10^5$ N/mm2, 0.3 (Poisson's ratio).




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → Link → 3D finitstn 180 → ok → close.
- Real constants → Add/Edit/Delete → Add → ok → Real constant set no → 1 → c/s area →2827.433 → ok → close.
- Material Properties → material Models → Structural → Linear → Elastic → Isotropic →EX → 2.1e5 → PRXY → 0.3 → ok → close.
- Modeling → create → nodes → In Active CS → Apply (first node is created) → x, y, z location in CS → 450, 0, 0 → ok (second node is created).
- Modeling → Create → Elements → Auto numbered → Thru Nodes → pick 1 & 2 → ok (elements are created through nodes)

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Nodes → pick node 1 → Apply → DOFs to be constrained → All DOF → ok.
- Define loads → Apply → Structural →Force/Moment → on Nodes → pick node 2 → Apply →direction of Force/mom → FX → Force/Moment value → 2500 (+ve value) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → LS → and type LS, 1 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- Plot Result → contour plot → Line Element Results → Elem table item at node I → LS1 → Elem table item at node J → LS1 → ok (Line stress diagram will be displayed).
- List Results →Elem table Data → Items to be listed → LS1 → ok. (stress will be displayed with the element numbers)
- List Results→ Reaction Solution → items to be listed → All items → ok (reaction forces will be displayed with the node numbers).
- List Results → Nodal solution → DOF solution → Displacement vector Sum → ok (Deformation will be displayed with the node numbers).

**Step 5**: Plotctrls → Animate → Deformed shape → def+undefromed → ok

PlotCtrls → Animate → Deformed results → DOF solution → Displacement Vector sum → ok.

**Comparison between theoretical and ANSYS value:**

| 1 | Theoretical | | ANSYS | |
|---|---|---|---|---|
| DEFORMATION | NODE | mm | NODE | mm |
| | 1 | 0 | 1 | 0 |
| | 2 | $1.8947 \times 10^{-3}$ | 2 | $1.8947 \times 10^{-3}$ |

| 1 | Theoretical | | ANSYS | |
|---|---|---|---|---|
| STRESS | ELEMENT NO | $N/mm^2$ | ELEMENT NO | $N/mm^2$ |
| | 1 | 0.88419 | 1 | 0.88419 |

| 1 | Theoretical | | ANSYS | |
|---|---|---|---|---|
| REACTION | NODE | N | NODE | N |
| | 1 | -2500 | 1 | -2500 |
| | 2 | 2500 | 2 | 2500 |

**Conclusion:** ANSYS simulation and the software results are near to theoretical results.

**THEORETICAL SOLUTION:**

Given,

Diameter, d = 60mm

Young's Modulus, E = 2.1 x $10^5$ N/mm2

Load, P = 2500 N (reaction at node 2 i.e. R2)

Area of cross section, A = $\frac{\pi d^2}{4}$ = $\frac{\pi x 60^2}{4}$ = 2827.433 mm2

**Deformation:**

At node 1 = 0

At node 2 = $\frac{\text{PL}}{AE}$ = $\frac{2500\text{x}450}{2827.433x2.1x10^5}$ = 1.894701 X $10^{-3}$ mm

**Stress:**

For Element 1, σ1= $\frac{P}{A}$ = $\frac{2500}{2827.433}$ = 0.88419 N/mm$^2$

**Reaction:**

R1+R2=0

R1 + 2500 = 0

Reaction at node 1, R1 = -2500 N

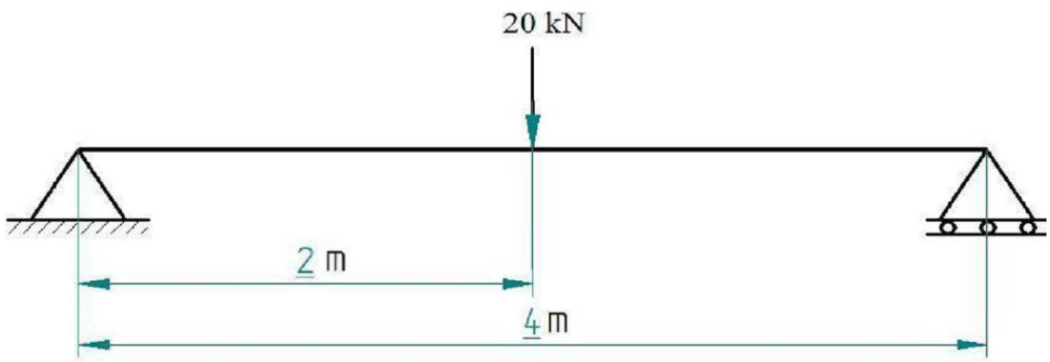
Reaction at node 2, R2 = 2500 N

# EXPERIMENT NO: 04

## Analysis of simply supported Beam with point load

**Aim:** Draw the shear force and bending moment diagram for he given beam due to applied load.

**Problem Description:** Compute the shear force and bending moment diagram for the beam shown. Assume rectangular c/s area of 0.2 m x 0.3 m, Young's Modulus of 210 GPa, Poisson's ratio 0.27




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 210e3 → PRXY → 0.27 → ok → close.
- Sections → Beam → Common section → B = 200, H = 300 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 2000, 0, 0 → Apply (second node is created)→ x, y, z location in CS → 4000, 0, 0 → ok (third node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2, 2 & 3 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → UX, UY, UZ, ROTX, ROTY → Apply → pick keypoint 3 → Apply → DOFs to be constrained → UY → ok.
- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -20e3 (-ve value) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → SMISC → SMISC, 3 → Apply, By Sequence num → SMISC → SMISC, 6 → Apply, By Sequence num → SMISC → SMISC, 16 → Apply, By Sequence num → SMISC → SMISC, 19 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS6 → Elem table item at node J → SMIS19 → ok (Shear force diagram will be displayed).
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS3 → Elem table item at node J → SMIS16 → ok (Bending Moment diagram will be displayed).

**Comparison between Theoretical & ANSYS results**

| Particulars | Theoretical | | ANSYS | |
|---|---|---|---|---|
| | Maximum | Minimum | Maximum | Minimum |
| Shear force (N) | 10000 | -10000 | 10000 | -10000 |
| Bending Moment (N-mm) | $20 \times 10^6$ | - | $20 \times 10^6$ | - |

**Conclusion:** ANSYS simulation and software results are near to the theoretical or analytical results.

**Theoretical Calculation :**

$R_A + R_B$ =20 KN …………….. (1)

Taking moment about A

$\Sigma M_A = 0$

(20 x 2) = $R_B$ x 4

$R_B$ = 10 KN …………….. (2)

By substituting Eqn (2) in Eqn (1) we get

$R_A + R_B = 20$

$R_A$= 20- $R_B$= 20-10 = 10 KN

Hence Reactions

**$R_A$ = 10 KN $R_B$ = 10 KN**

**Shear Force Calculation:**

at A

LHS = 0

RHS = 20 – 10 = 10 KN

at C

LHS = 10 KN

RHS = -10 KN

at B

LHS = 10 – 20 = -10 KN

RHS = 0

**Bending Moment Calculation:**

BM at A

LHS = 0

RHS = (10 x 4) – (20 x 2) = 0

BM at C

LHS = (10 x 2) = 20 KN-m

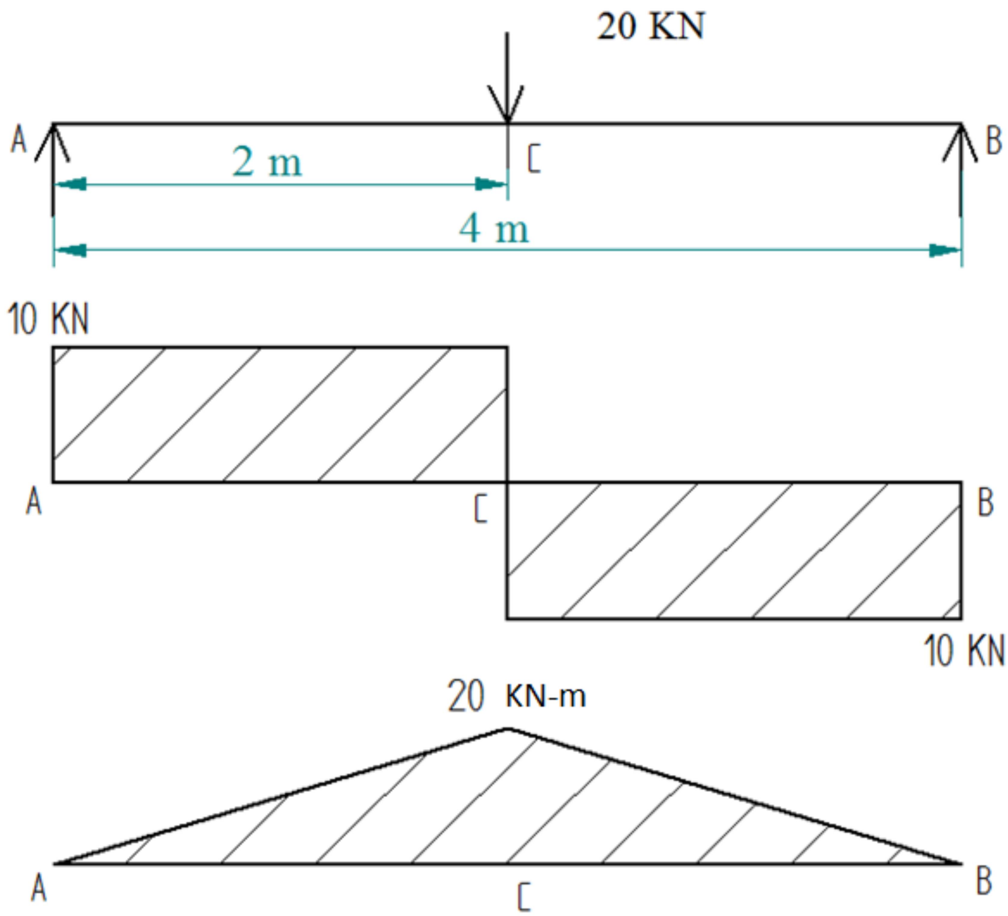
RHS = (10 x 2) = 20 KN-m

BM at B

LHS = (10 x 4) – (20 x 2) = 0

RHS = 0

**SFD & BMD:**




Note:

Shear force calculation:

**LHS= upward +ve downward –ve**

**RHS= upward -ve downward +ve**

Bending moment calculation*:*

**LHS= clockwise moment +ve anticlockwise moment -ve**

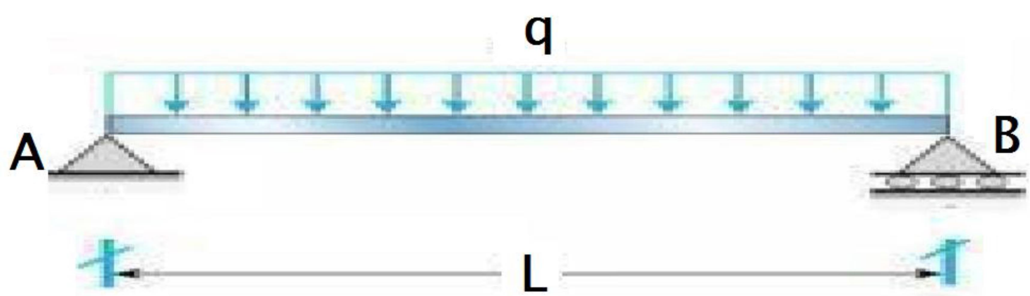
**RHS= clockwise moment -ve anticlockwise moment +ve**

# EXPERIMENT NO: 05

## Analysis of simply supported Beam with UDL

**Aim:** Draw the shear force and bending moment diagram for the given beam due to applied load.

**Problem Description:** Compute the shear force and bending moment diagram for the beam shown. Assume rectangular c/s area of 0.2 m x 0.3 m, Young's Modulus of 200 GPa, Poisson's ratio 0.3, Length (L) = 2m, q = 10 kN/m.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 200e3 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → B = 200, H = 300 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 2000, 0, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 to 2 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → UX, UY, UZ, ROTX, ROTY

→ Apply → pick keypoint2→ Apply → DOFs to be constrained → UY → ok.

- Define loads → Apply → Structural → Pressure → on beams → tick on box → drag and select the beam → apply → LKEY Load key → 2 → Pressure value at node I → 10 → ok → close.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → SMISC → SMISC, 3 → Apply, By Sequence num → SMISC → SMISC, 6 → Apply, By Sequence num → SMISC → SMISC, 16 → Apply, By Sequence num → SMISC → SMISC, 19 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS6 → Elem table item at node J → SMIS19 → ok (Shear force diagram will be displayed).
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS3 → Elem table item at node J → SMIS16 → ok (Bending Moment diagram will be displayed).

**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | | ANSYS | |
|---|---|---|---|---|
| | Maximum | Minimum | Maximum | Minimum |
| Shear force (N) | $10 \times 10^3$ | $-10 \times 10^3$ | $10 \times 10^3$ | $-10 \times 10^3$ |
| Bending Moment (N-mm) | $5 \times 10^6$ | - | $5 \times 10^6$ | - |

**Conclusion:** ANSYS simulation and software results are near to the theoretical or analytical results.

**Theoretical Calculation:**

$R_A + R_B = (10 \times 2) = 20$ KN …………….. (1)

Taking moment about A

$\Sigma M_A = 0$

(20 x 1) = RB x 2

$R_B = 10$ KN …………….. (2)

By substituting Eqn (2) in Eqn (1) we get

$R_A + R_B = 20$

$R_A = 20 - R_B = 20-10 = 10$ KN

Hence Reactions

**$R_A = 10$ KN $R_B = 10$ KN**

**Shear Force Calculation:**

at A

LHS = 0

RHS = 20 – 10 = 10 KN

at B

LHS = 10 – 20 = -10 KN

RHS = 0

**Bending Moment Calculation:**

BM at A

LHS = 0

RHS = (10 x 2 ) – ( 10 x 2 x 1) = 0

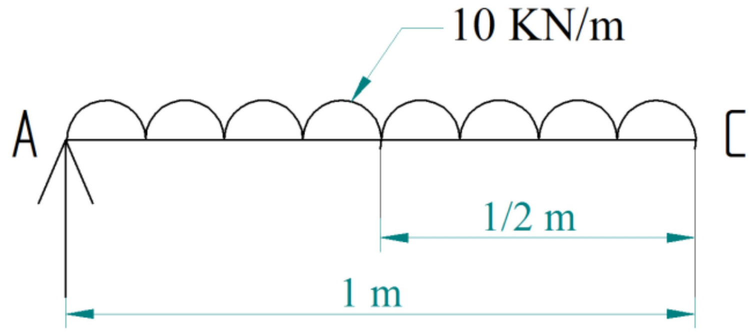
BM at B

LHS = (10 x 2) – (10 x 2 x 1) = 0

RHS = 0

**Maximum Bending Moment calculation:**

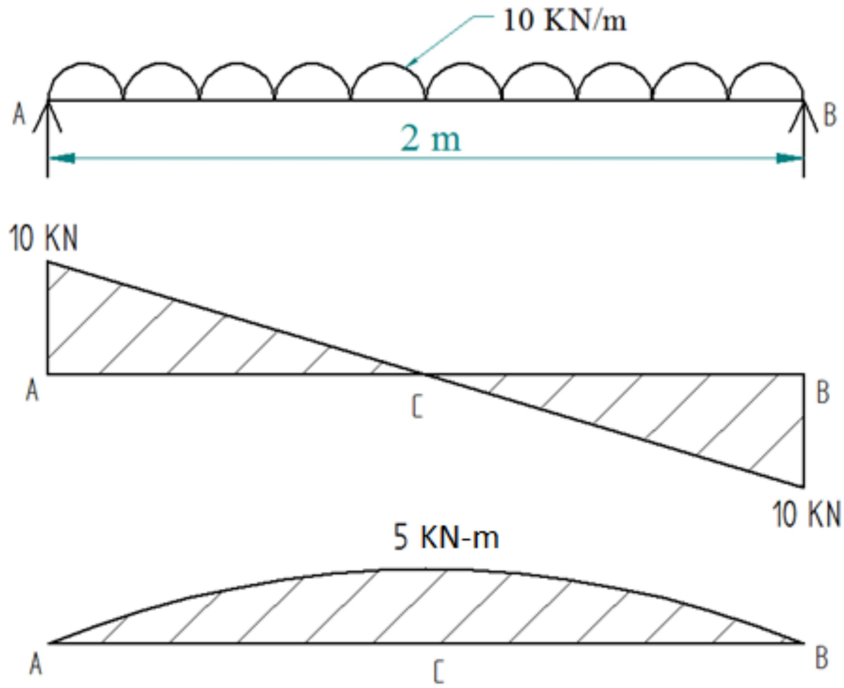
Maximum BM is at point where shear force changes its sign. i.e. at point ‘C’.



Maximum BM at C :-

LHS = (10 x 1) – (10 x 1 x$\frac{1}{2}$) = 5 KN-m

**SFD & BMD:**




Note:

Shear force calculation:

**LHS= upward +ve downward –ve**

**RHS= upward -ve downward +ve**

Bending moment calculation:

**LHS= clockwise moment +ve anticlockwise moment -ve**

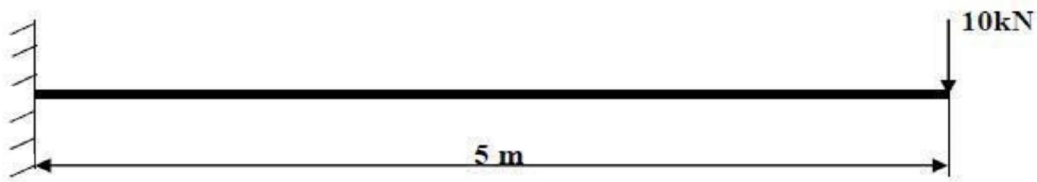
**RHS= clockwise moment -ve anticlockwise moment +ve**

# EXPERIMENT NO: 06

## Analysis of Cantilever Beam with point load

**Aim:** Draw the shear force and bending moment diagram for the given beam due to applied load.

**Problem Description:** Compute the shear force and bending moment diagram for the beam shown. Assume rectangular c/s area of 0.2 m x 0.3 m, Young's Modulus of 210GPa, Poisson's ratio 0.27.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 210e3 → PRXY → 0.27 → ok → close.
- Sections → Beam → Common section → B = 200, H = 300 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 5000, 0, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints→pick node 1 → Apply → DOFs to be constrained → ALL DOF → ok.

- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -10e3 (-ve value) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → SMISC → SMISC, 3 → Apply, By Sequence num → SMISC → SMISC, 6 → Apply, By Sequence num → SMISC → SMISC, 16 → Apply, By Sequence num → SMISC → SMISC, 19 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS6 → Elem table item at node J → SMIS19 → ok (Shear force diagram will be displayed).
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS3 → Elem table item at node J → SMIS16 → ok (Bending Moment diagram will be displayed).

**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | | ANSYS | |
|---|---|---|---|---|
| | Maximum | Minimum | Maximum | Minimum |
| Shear force (N) | 10000 | - | 10000 | - |
| Bending Moment (N-mm) | - | $-50 \times 10^6$ | - | $-50 \times 10^6$ |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Reaction at A

$R_A$ = 10 KN

**Shear Force Calculation:**

at A

LHS = 0

RHS = 10 KN

at B

LHS = 10 KN

RHS = 0

**Bending Moment Calculation:**

BM at A

LHS = 0

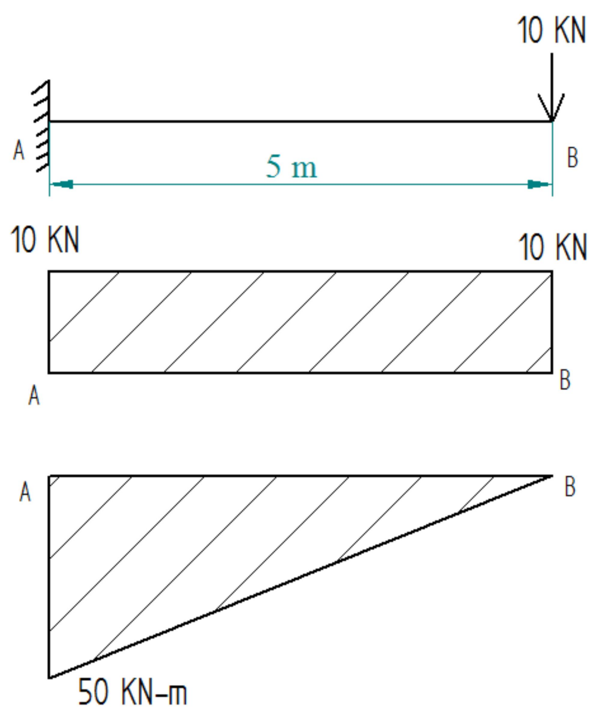
RHS = – 10 x 5 = -50 KN-m

BM at B

LHS = 0

RHS = 0

**SFD & BMD:**




Note:

Shear force calculation:

**LHS= upward +ve downward –ve**

**RHS= upward -ve downward +ve**

Bending moment calculation:

**LHS= clockwise moment +ve anticlockwise moment -ve**

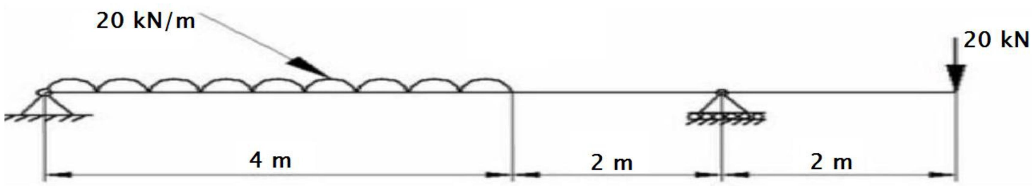
**RHS= clockwise moment -ve anticlockwise moment +ve**

# EXPERIMENT NO: 07

## Analysis of overhanging Beam

**Aim:** Draw the shear force and bending moment diagram for the given beam due to applied load.

**Problem Description:** Draw the shear force and bending moment diagram for the beam loaded as shown in figure. Assume rectangular c/s area of 0.2 m x 0.3 m, E = 200GPa, Poisson's ratio = 0.3




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 200e3 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → B = 200, H = 300 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 4000, 0, 0 → Apply (second node is created)→ x, y, z location in CS → 6000, 0, 0 → Apply (third node is created)→ x, y, z location in CS → 8000, 0, 0 → ok (fourth node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 &2, 2 & 3, 3 & 4 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → UX, UY, UZ, ROTX, ROTY

→ Apply → pick keypoint3 → Apply → DOFs to be constrained → UY → ok.

- Define loads → Apply → Structural → Force/Moment →on keypoint→ pick keypoint 4 → Apply → direction of Force/Moment → FY → Force/Moment value → -20e3 (-ve value) → ok.
- Define loads → Apply → structural → Pressure → on beams → tick on box → drag and select the beam from keypoint 1 to 2 using zoom and fit → apply → LKEY Load key → 2 → Pressure value at node I → 20 → ok → close.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → SMISC → SMISC, 3 → Apply, By Sequence num → SMISC → SMISC, 6 → Apply, By Sequence num → SMISC → SMISC, 16 → Apply, By Sequence num → SMISC → SMISC, 19 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS6 → Elem table item at node J → SMIS19 → ok (Shear force diagram will be displayed).
- Plot Result → contour plot → Line Element Results → Elem table item at node I → SMIS3 → Elem table item at node J → SMIS16 → ok (Bending Moment diagram will be displayed).

**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | | ANSYS | |
|---|---|---|---|---|
| | Maximum | Minimum | Maximum | Minimum |
| Shear force (N) | 46670 | -33330 | 46670 | -33330 |
| Bending Moment (N-mm) | 26640 | - | 26640 | - |

**Conclusion:** ANSYS simulation and software results are near to the theoretical or analytical results.

**Theoretical Calculation:**

$R_A + R_B = (20 \times 4) + 20 = 100$ KN …………….. (1)

Taking moment about A

$\Sigma M_A = 0$

$(20 \times 4 \times 2) + (20 \times 8) = R_B \times 6$

$R_B = 53.33$ KN …………….. (2)

By substituting Eqn (2) in Eqn (1) we get

$R_A + R_B = 100$

$R_A = 100 - R_B = 100 - 53.33 = 46.66$ KN

Hence Reactions

**$R_A = 46.66$ KN $R_B = 53.33$ KN**

**Shear Force Calculation:**

at A,

LHS = 0

RHS= (20 x 4) + 20 - 53.33 = 46.67 KN

at C

LHS = 46.66 – (20 x 4) = -33.34 KN

RHS = 20 – 53.33 = -33.34 KN

at B

LHS = 46.66 – (20 x 4) = -33.34 KN

RHS = 20 KN

at D

LHS = 46.66 + 53.33 – (20 x 4) = 20 KN

RHS = 0

**Bending Moment Calculation:** (LHS = RHS)

BM at A

LHS = 0

RHS = (53.33 x 6) – (20 x 8) – (20 x 4 x 2) = 0

BM at C

LHS = (46.66 x 4) – (20 x 4 x 2) = 26.64 KN-m

RHS = (53.33 x 2) – (20 x 4) = 26.66 KN-m

BM at B

LHS = (46.66 x 6) – (20 x 4 x 4) = - 40 KN-m

RHS = - (20 x 2) = - 40 KN-m

BM at D

LHS = (46.66 x 8) – (20 x 4 x 6) + (53.33 x 2) =0

RHS = 0

**Maximum Bending Moment calculation:**

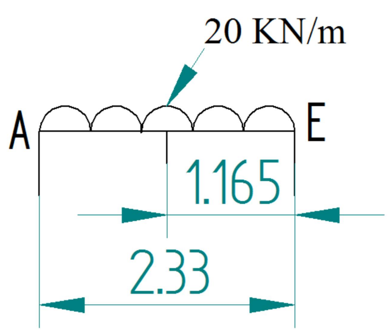
Maximum BM is at point where shear force changes its sign. i.e. at point 'E'.

| Distance in m | Shear Force in N |
|---|---|
| 4 | -33.34 |
| X | 0 |
| 0 | 46.67 |

Interpolation:

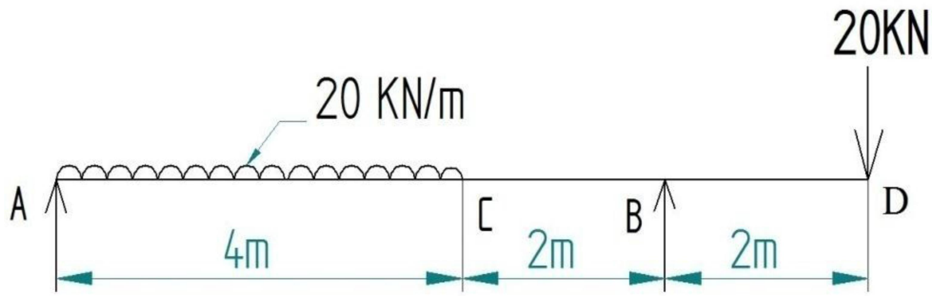
$$\frac{4-0}{x-0} = \frac{-33.34-46.67}{0-46.67}$$

X=2.33m

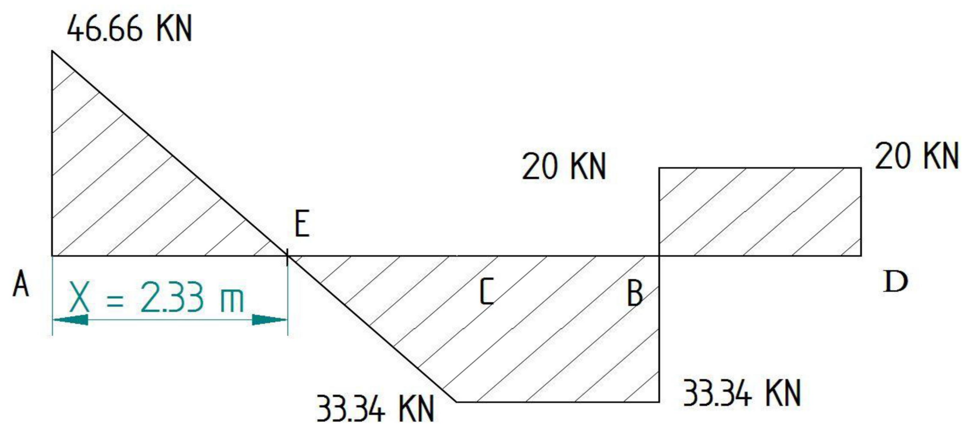



Maximum BM at E:-

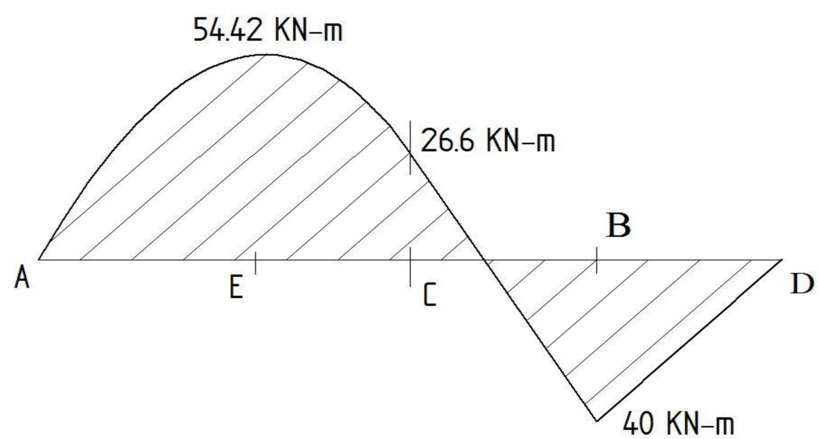
LHS = (46.66 x 2.33) – (20 x 2.33 x 1.165) = 54.428 KN-m




**SFD**




**BMD**




**Note:**

Shear force calculation:

**LHS= upward +ve downward –ve**

**RHS= upward -ve downward +ve**

Bending moment calculation:

**LHS= clockwise moment +ve anticlockwise moment -ve**

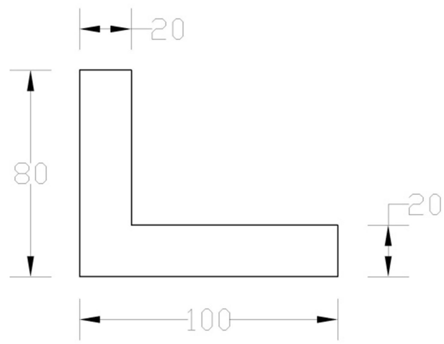
**RHS= clockwise moment -ve anticlockwise moment +ve**

# EXPERIMENT NO: 08

# CENTROID AND MOMENT OF INERTIA OF GIVEN L-SECTION

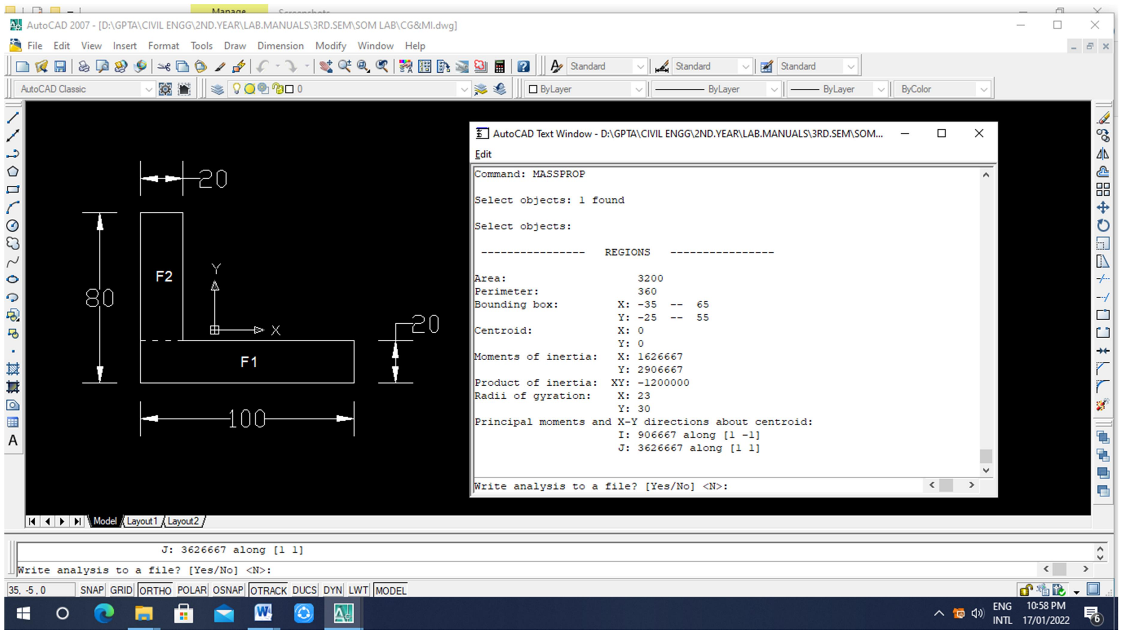
**Aim:** Finding the centroid and moment of inertia of given section by using CADD software and comparison with manual calculation.

**Problem Description:** Find the moment of inertia about centroidal X-X and Y-Y axes of the angle section with measurements 100x80x20mm.




**Procedure:**

1. Open new sheet in AutoCAD.
2. Set the required units [format → units → precision 0, units millimeters].
3. Set out lower left corner 0, 0 and upper right corner depending up on total area required to complete elevation and section of drawing [format → limits].
4. Set the required dimension style [format → dimension style].
5. Save the file with file name as exercise and register number.
6. Use zoom command and choose all option to show working area.
7. Start the drawing of given section by using draw tools.
8. Complete the details by using modify tools.
9. Draw → region → select the whole section → enter.
10. Find the centroid of the section using MASSPROP command or ( tools → inquiry → region / mass properties)
11. Move the section so that its centroid is at 0, 0.
12. Run MASSPROP to find out the moment of inertia along the neutral axes, X and Y.




## Comparison between Theoretical & CADD results:

| Particulars | Theoretical | CADD |
|---|---|---|
| Centroid (mm) | $\bar{X}$=35 | $\bar{X}$=35 |
| | $\bar{Y}$=25 | $\bar{Y}$=25 |
| Moment of inertia ($mm^4$) | $I_{xx}$=1626666.66<br>$I_{yy}$=2906660 | $I_{xx}$=1626667<br>$I_{yy}$=2906667 |

**Conclusion:** CADD software results are near to theoretical results.

**Theoretical Calculation:**

Let $\bar{X}$ and $\bar{Y}$ be the distance of centroid of the section from the bottom face and vertical face of the section respectively.

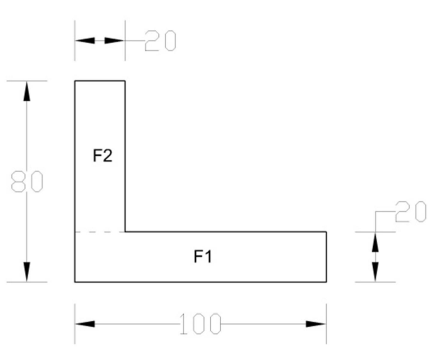


Figure-1:

$a_1$=100x20= 2000mm$^2$

$x_1=\frac{100}{2}$=50mm

$y_1=\frac{20}{2}$=10mm

Figure-2:

$a_2$=60x20= 1200mm$^2$

$x_2=\frac{20}{2}$=10mm

$y_2=20+\frac{60}{2}$=50mm

$\overline{X}=\frac{a1x1+a2x2}{a1+}=\frac{2000x50+1200x1}{2000+1200}$=**35mm**

$\overline{Y}=\frac{a1y1+a2y2}{a1+a2}=\frac{2000x10+1200x50}{2000+1200}$=**25mm**

To find moment of inertia

Figure-1:

$I_{G1x}=\frac{bd3}{12}=\frac{100x202}{12}$=66666.66mm$^4$

$I_{G1y}=\frac{db3}{12}=\frac{20x1003}{12}$=1666.66x10$^3$ mm$^4$

Figure-2:

$I_{G2x}=\frac{bd3}{12}=\frac{20x603}{12}$=360000mm$^4$

$I_{G2y}=\frac{db3}{12}=\frac{60x203}{12}$=40000 mm$^4$

$I_{xx}=I_{G1x}+a_1\,(\overline{y}-y_1)^2+I_{G2x}+a_2\,(\overline{y}-y_2)^2$

$I_{xx}$=66666.66+2000x $(25-10)^2$ + 360000+1200x $(25-50)^2$=**1626666.66 mm$^4$**

$I_{yy}=I_{G1y}+a_1\,(\overline{x}-x_1)^2+I_{G2y}+a_2\,(\overline{x}-x_2)^2$

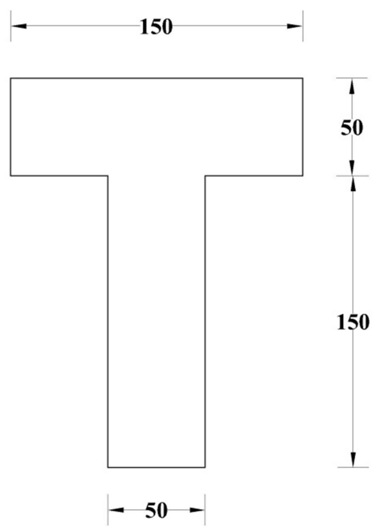
$I_{yy}$=1666.66x10$^3$+2000x $(35-50)^2$ + 40000+1200x $(35-10)^2$=**2906660 mm$^4$**

# EXPERIMENT NO: 09

# CENTROID AND MOMENT OF INERTIA OF GIVEN T-SECTION

**Aim:** Finding the centroid and moment of inertia of given section by using CADD software and comparison with manual calculation.

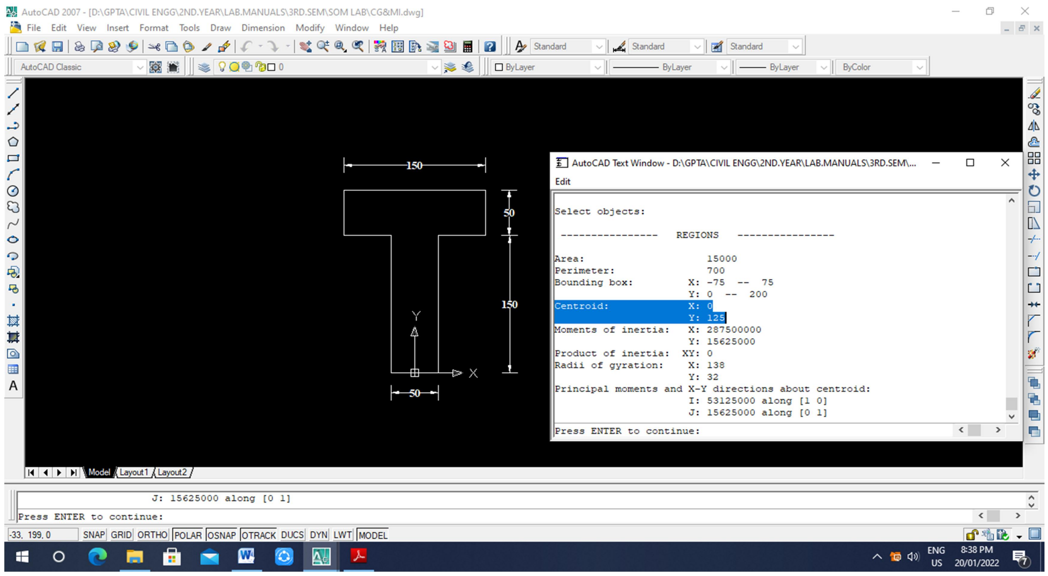
**Problem Description:** Find the moment of inertia about centroidal X-X and Y-Y axes of the T-section with measurements i) Flange -150x50mm ii) Web-50x150mm.

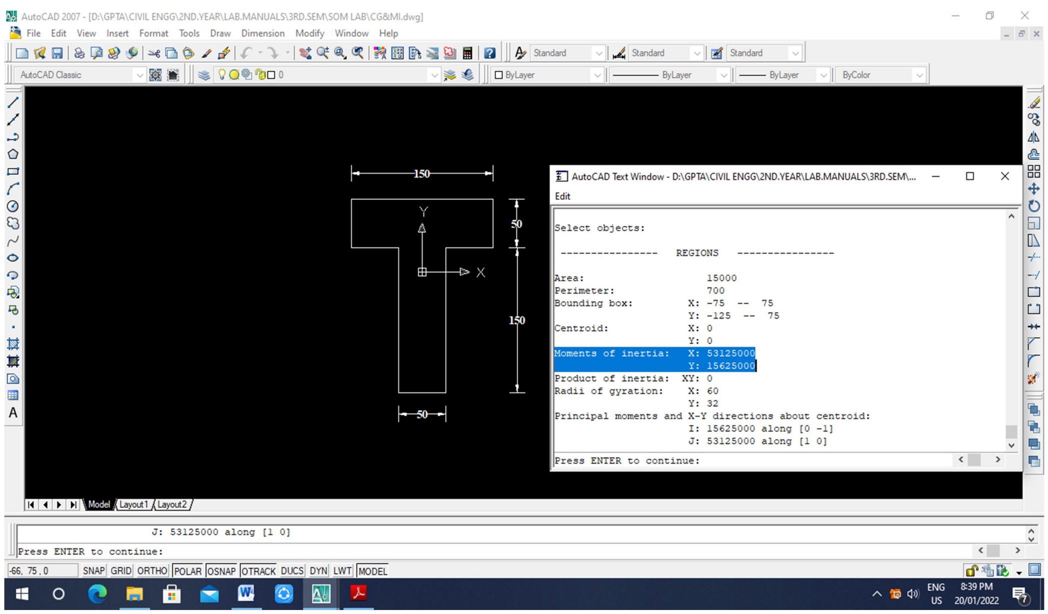



**Procedure:**

1. Open new sheet in AutoCAD.
2. Set the required units [format → units → precision 0, units millimeters].
3. Set out lower left corner 0, 0 and upper right corner depending up on total area required to complete elevation and section of drawing [format → limits].
4. Set the required dimension style [format → dimension style].
5. Save the file with file name as exercise and register number.
6. Use zoom command and choose all option to show working area.
7. Start the drawing of given section by using draw tools.
8. Complete the details by using modify tools.
9. Draw → region → select the whole section → enter.
10. Find the centroid of the section using MASSPROP command or ( tools → inquiry → region / mass properties)
11. Move the section so that its centroid is at 0, 0.

12. Run MASSPROP to find out the moment of inertia along the neutral axes, X and Y.




**Comparison between Theoretical & CADD results:**

| Particulars | Theoretical | CADD |
|---|---|---|
| Centroid (mm) | $\bar{X}=0$ | $\bar{X}=0$ |
| | $\bar{Y}=125$ | $\bar{Y}=125$ |
| Moment of inertia ($mm^4$) | $I_{xx}=53125000$<br>$I_{yy}=15625000$ | $I_{xx}=53125000$<br>$I_{yy}=15625000$ |

**Conclusion:** CADD software results are near to theoretical results.

**Theoretical Calculation:**

Since T-section symmetrical about Y-Y axis, centroid will lie on this axis, Hence $\overline{X}$=0. Divide the section into two rectangles as shown in figure. Let $\overline{Y}$ be the distance of centroid of the section from the bottom of the web.

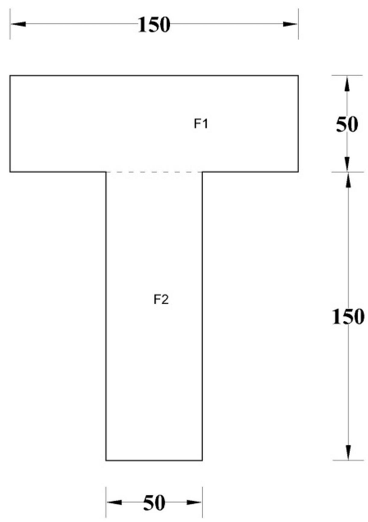



Figure-1:

$a_1$=150x50= 7500mm$^2$

$y_1=\frac{50}{2}+150$=175mm

Figure-2:

$a_2$=150x50= 7500mm$^2$

$y_2=\frac{150}{2}$=75mm

$\overline{Y}=\frac{a1y1+a2y}{a1+a2}=\frac{7500x175+75}{7500+750}$=**125mm**

To find moment of inertia

Figure-1:

$I_{G1x}=\frac{bd3}{12}=\frac{150x503}{12}$=1562500mm$^4$

$I_{G1y}=\frac{db3}{12}=\frac{50x1503}{12}$=14062500 mm$^4$

Figure-2:

$I_{G2x}=\frac{bd3}{12}=\frac{50x1503}{12}$=14062500mm$^4$

$I_{G2y}=\frac{db3}{12}=\frac{150x503}{12}$=1562500 mm$^4$

$I_{xx}=I_{G1x}+a_1\,(\bar{y}-y_1)^2+I_{G2x}+a_2\,(\bar{y}-y_2)^2$

$I_{xx}=1562500+7500\text{x}\,(125-175)^2+14062500+7500\text{x}\,(125-75)^2=$ **53125000 mm⁴**

$I_{yy}=I_{G1y}+I_{G2y}$

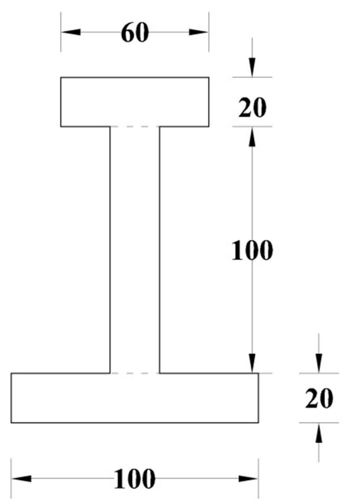
$I_{yy}=14062500+1562500=$ **15625000 mm⁴**

# EXPERIMENT NO: 10

# CENTROID AND MOMENT OF INERTIA OF GIVEN I-SECTION

**Aim:** Finding the centroid and moment of inertia of given section by using CADD software and comparison with manual calculation.

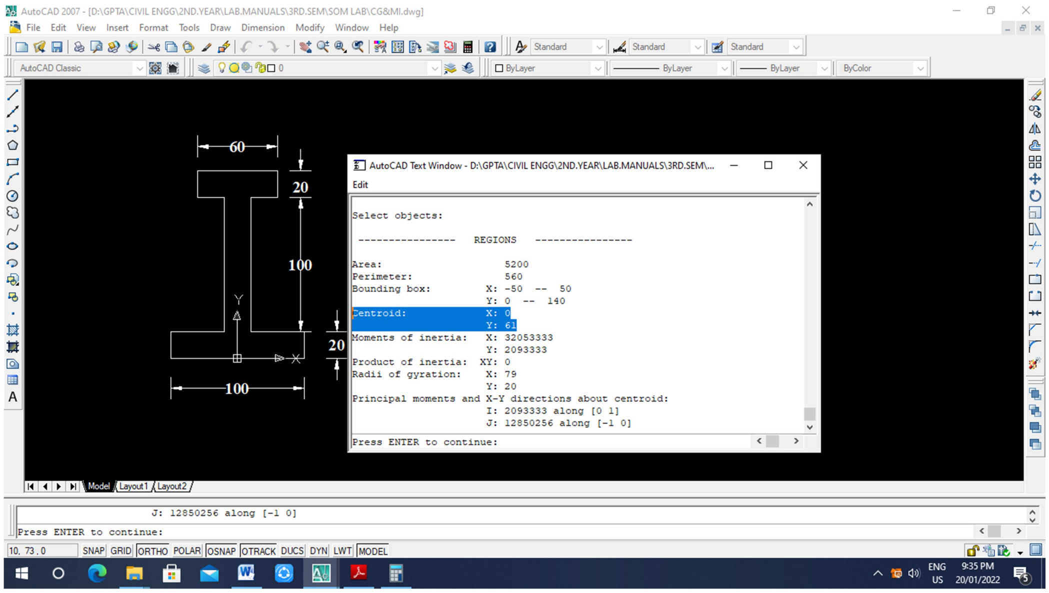
**Problem Description:** Find the moment of inertia about centroidal X-X and Y-Y axes of the I-section with measurements i) Top Flange -60x20mm ii) Bottom Flange -100x20mm iii) Web-20x100mm.

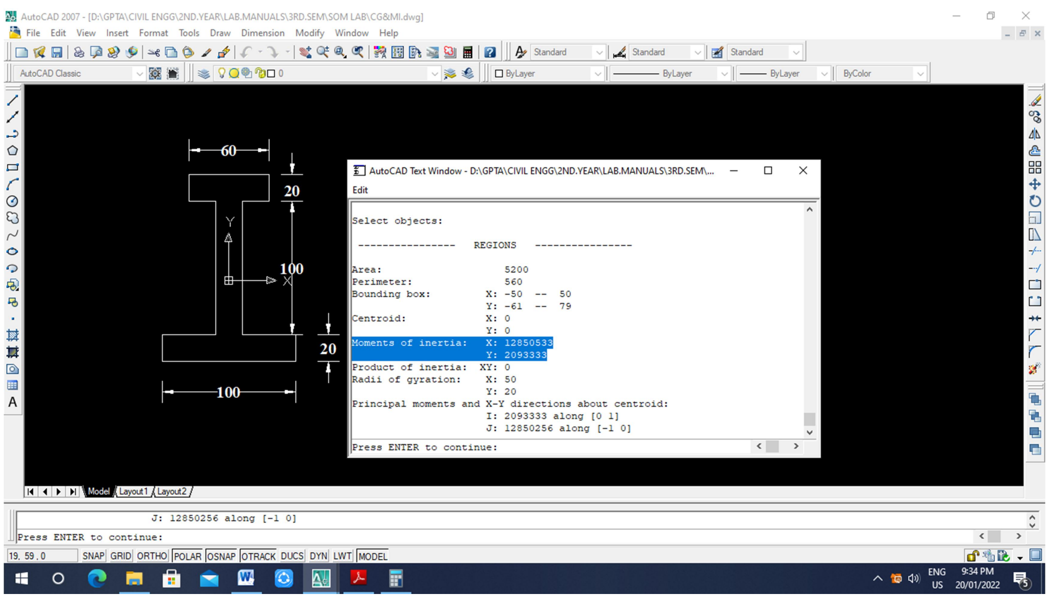



**Procedure:**

1. Open new sheet in AutoCAD.
2. Set the required units [format → units → precision 0, units millimeters].
3. Set out lower left corner 0, 0 and upper right corner depending up on total area required to complete elevation and section of drawing [format → limits].
4. Set the required dimension style [format → dimension style].
5. Save the file with file name as exercise and register number.
6. Use zoom command and choose all option to show working area.
7. Start the drawing of given section by using draw tools.
8. Complete the details by using modify tools.
9. Draw → region → select the whole section → enter.
10. Find the centroid of the section using MASSPROP command or ( tools → inquiry → region / mass properties)
11. Move the section so that its centroid is at 0, 0.

12. Run MASSPROP to find out the moment of inertia along the neutral axes, X and Y.




**Comparison between Theoretical & CADD results:**

| Particulars | Theoretical | CADD |
|---|---|---|
| Centroid (mm) | $\bar{X}=0$ | $\bar{X}=0$ |
| | $\bar{Y}=60.76$ | $\bar{Y}=61$ |
| Moment of inertia ($mm^4$) | $I_{xx}=12850256.84$<br>$I_{yy}=2093333.32$ | $I_{xx}=12850533$<br>$I_{yy}=2093333$ |

**Conclusion:** CADD software results are near to theoretical results.

**Theoretical Calculation:**

Since I-section symmetrical about Y-Y axis, centroid will lie on this axis, Hence $\overline{X}$=0. Divide the section into three rectangles as shown in figure. Let $\overline{Y}$ be the distance of centroid of the I-section from the base of bottom flange.

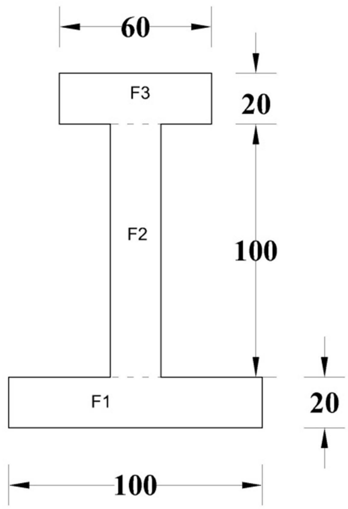



Figure-1:

$a_1$=100x50= 2000mm$^2$

$y_1=\frac{20}{2}$=10mm

Figure-2:

$a_2$=20x100= 2000mm$^2$

$y_2=\frac{100}{2}+20$=70mm

Figure-3:

$a_3$=60x20= 1200mm$^2$

$y_3=\frac{20}{2}+100+20$=130mm

$\overline{Y}=\frac{a1y1+a2y2+}{a1+a2+a3}=\frac{2000x10+2000x70+1200x130}{2000+2000+120}$=**60.76mm**

To find moment of inertia

Figure-1:

$I_{G1x}=\frac{bd3}{12}=\frac{100x203}{12}$=66666.66mm$^4$

$I_{G1y}=\frac{db3}{12}=\frac{20x1003}{12}$=1666666.66 mm$^4$

Figure-2:

$I_{G2x}=\frac{bd3}{12}=\frac{20x1003}{12}$=1666666.66 mm$^4$

$I_{G2y}=\frac{db3}{12}=\frac{100x203}{12}$=66666.66 mm$^4$

Figure-3:

$I_{G3x}=\frac{bd3}{12}=\frac{60x203}{12}$=40000 mm$^4$

$I_{G3y}=\frac{db3}{12}=\frac{20x603}{12}$=360000 mm$^4$

$I_{xx}=I_{G1x}+a_1\,(\bar{y}-y_1)^2+I_{G2x}+a_2\,(\bar{y}-y_2)^2+I_{G3x}+a_3\,(\bar{y}-y_3)^2$

$I_{xx}$=66666.66+2000x (60.76-10)$^2$ +1666666.66 +2000x (60.76-70)$^2$+40000 +1200x (60.76-130)$^2$=**12850256.84 mm$^4$**

$I_{yy}=I_{G1y}+I_{G2y}+I_{G3y}$

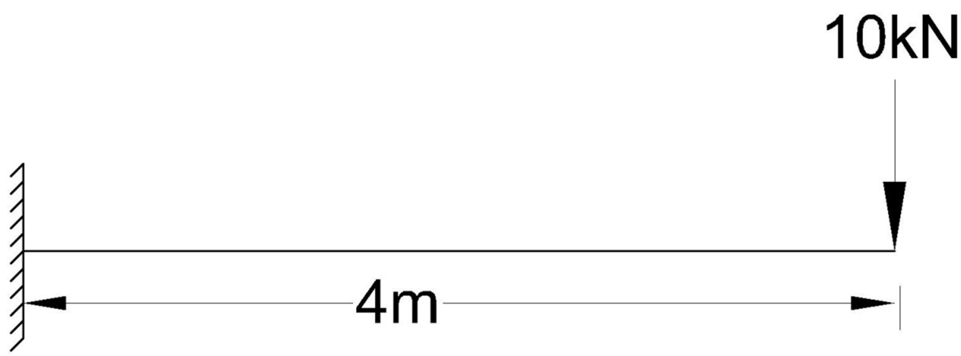
$I_{yy}$=1666666.66 +66666.66+360000=**2093333.32 mm$^4$**

# EXPERIMENT NO: 11

## Slope and deflection of Cantilever Beam with point load

**Aim:** To find the slope and deflection for the given cantilever beam due to applied load both manually and using ANSYS software.

**Problem Description:** Compute the slope and deflection for the beam shown. Take rectangular c/s area of 0.1 m x 0.2 m, Young's Modulus of 200GPa, Poisson's ratio 0.3.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

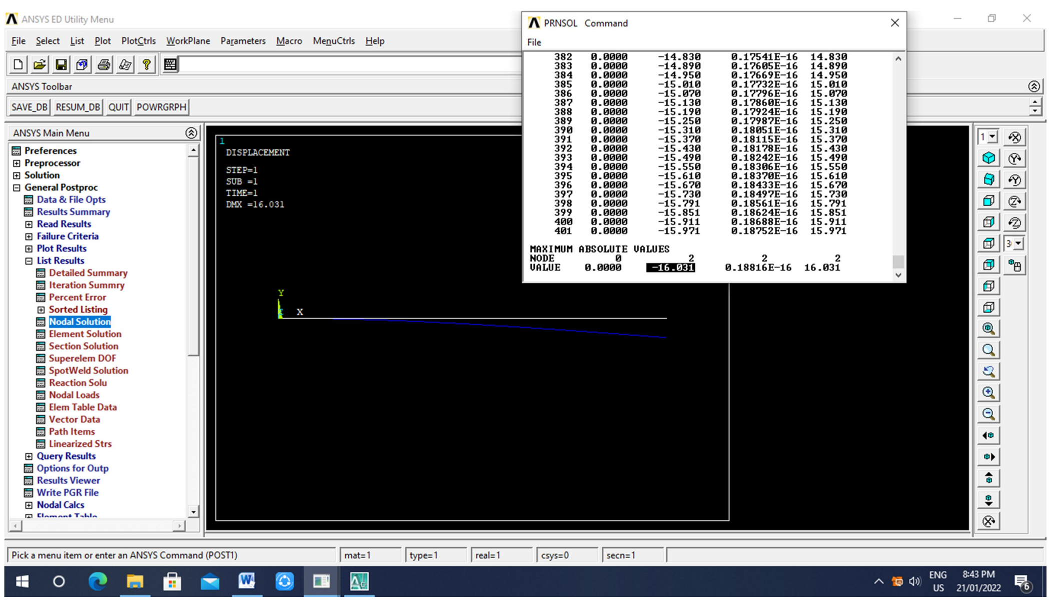
- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → B = 200, H = 100 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 4000, 0, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

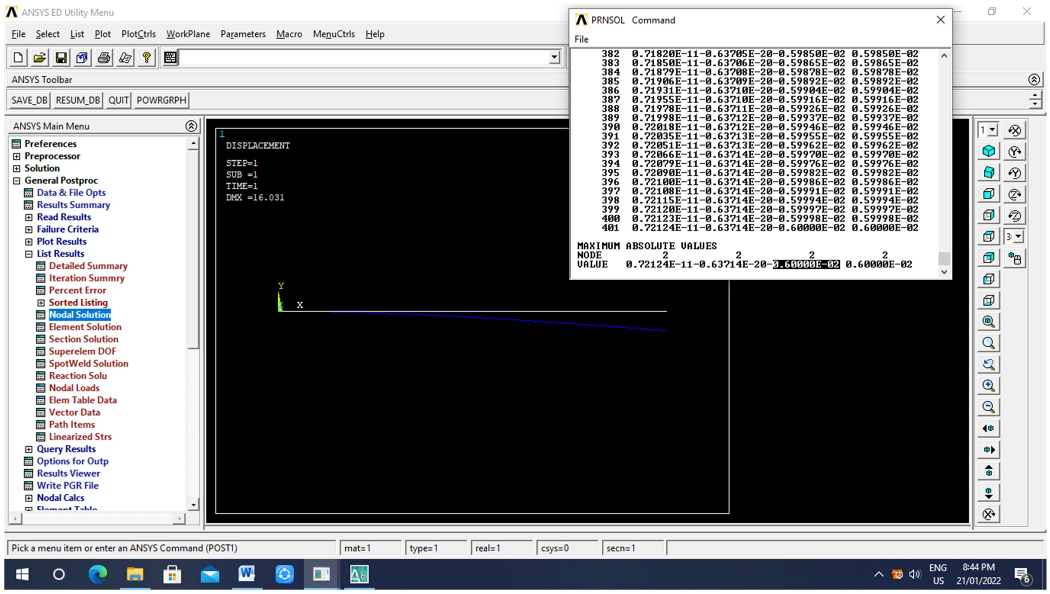
**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints→pick node 1 → Apply → DOFs to be constrained → ALL DOF → ok.
- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -10e3 (-ve value) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → LS → and type LS, 1 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Nodal solution → DOF solution → Displacement vector Sum → ok (Deflection will be displayed with the node numbers).
- List Results → Nodal solution → DOF solution → Rotation vector Sum → ok (slope will be displayed with the node numbers).






**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Slope (radians) | 0.006 | 0.006 |
| Deflection (mm) | 16 | 16.031 |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: l=4m=4000mm, W=10kN=10000N, b=100mm, d=200mm, E=2x$10^5$ N/$mm^2$

Moment of inertia for rectangular section about horizontal centroidal axis

$$I = \frac{bd3}{12} = \frac{100x2003}{12} = 66.67x106\ mm4$$

**Slope at free end,**

$\Theta = Wl^2/2EI = 10000x4000^2/(2x2x10^5x66.67x10^6)$ =**0.006 radians**

**Deflection at free end,**

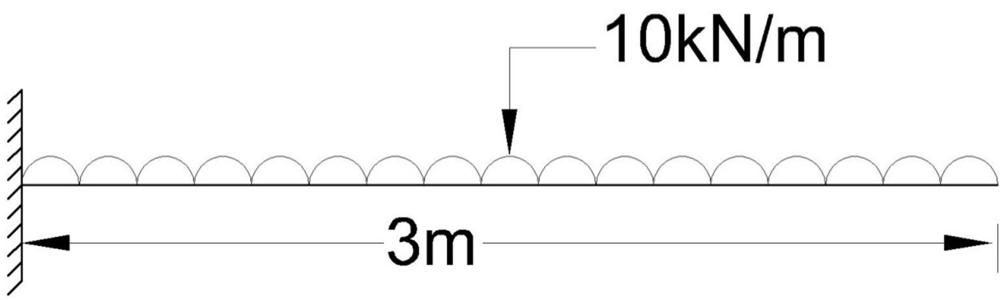
$y = Wl^3/3EI = 10000x4000^3/(3x2x10^5x66.64x10^6)$ = **16mm**

# EXPERIMENT NO: 12

## Slope and deflection of Cantilever Beam with UDL

**Aim:** To find the slope and deflection for the given cantilever beam due to applied load both manually and using ANSYS software.

**Problem Description:** Compute the slope and deflection for the beam shown. Take rectangular c/s area of 0.1 m x 0.15 m, Young's Modulus of 200GPa, Poisson's ratio 0.3.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → B = 150, H = 100 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 3000, 0, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

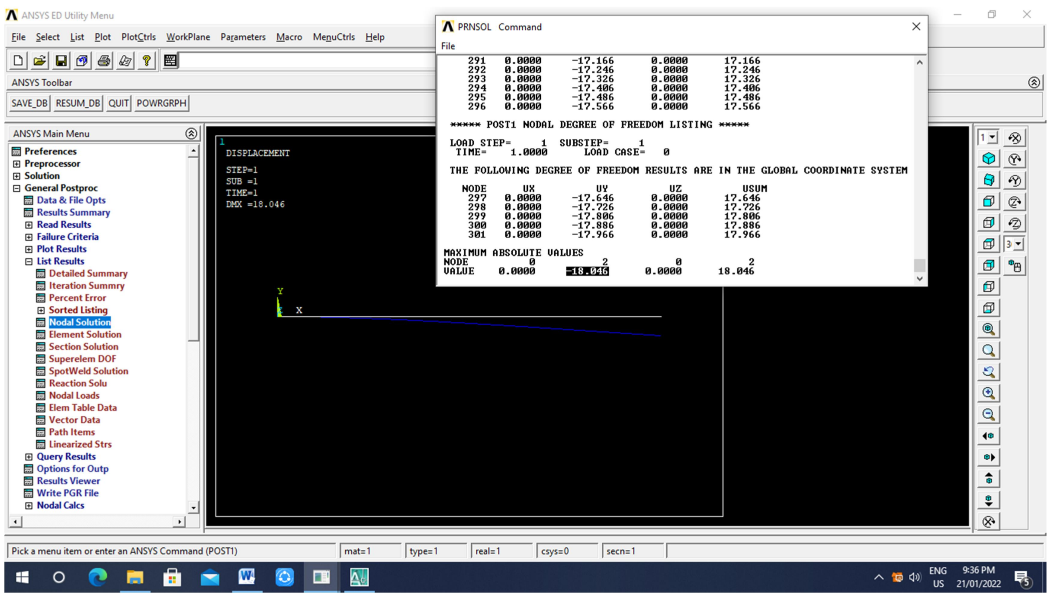
**Step 3**: Solution

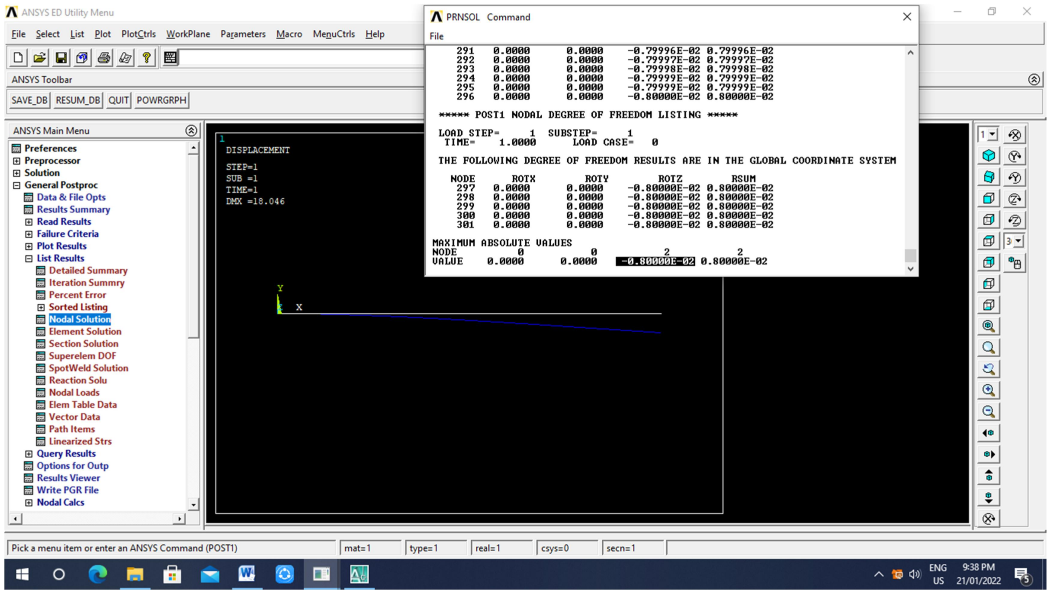
- Define loads → Apply → Structural → Displacement → on Keypoints→pick node 1 → Apply → DOFs to be constrained → ALL DOF → ok.

- Define loads → Apply → Structural → Pressure → on beams → tick on box → drag and select the beam → apply → LKEY Load key → 2 → Pressure value at node I → 10 → ok → close.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → LS → and type LS, 1 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Nodal solution → DOF solution → Displacement vector Sum → ok (Deflection will be displayed with the node numbers).
- List Results → Nodal solution → DOF solution → Rotation vector Sum → ok (slope will be displayed with the node numbers).






**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Slope (radians) | 0.008 | 0.008 |
| Deflection (mm) | 18 | 18.046 |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: l=3m=3000mm, w=10kN/m=10N/mm, b=100mm, d=150mm, E=2x$10^5$ N/$mm^2$

Moment of inertia for rectangular section about horizontal centroidal axis

$$I = \frac{bd3}{12} = \frac{100x1503}{12} = 28.13x106\ mm4$$

**Slope at free end,**

$\Theta = wl^3/6EI = 10x3000^3/(6x2x10^5x28.13x10^6)$ =**0.008 radians**

**Deflection at free end,**

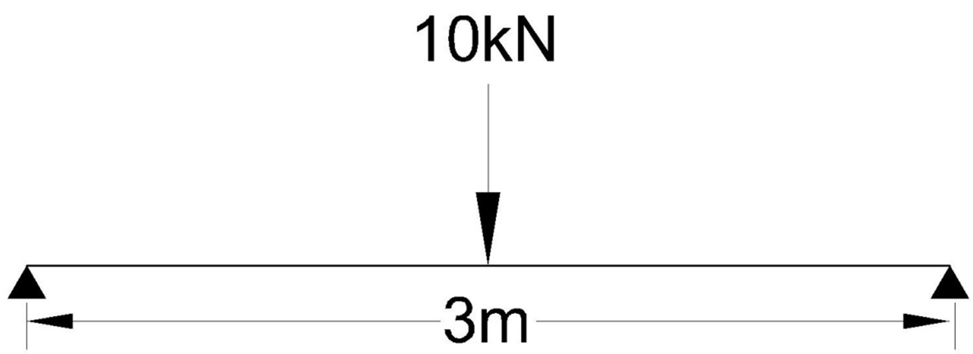
$y = wl^4/8EI = 10x3000^4/(8x2x10^5x28.13x10^6)$ = **18mm**

# EXPERIMENT NO: 13

## Slope and deflection of simply supported Beam with point load

**Aim:** To find the slope and deflection for the given simply supported beam due to applied load both manually and using ANSYS software.

**Problem Description:** Compute the slope and deflection for the beam shown. Take rectangular c/s area of 0.1 m x 0.2 m, Young's Modulus of 200GPa, Poisson's ratio 0.3.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → B = 200, H = 100 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 1500, 0, 0 → Apply (second node is created) → x, y, z location in CS → 3000, 0, 0 → Apply (third node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2, 2 & 3 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok

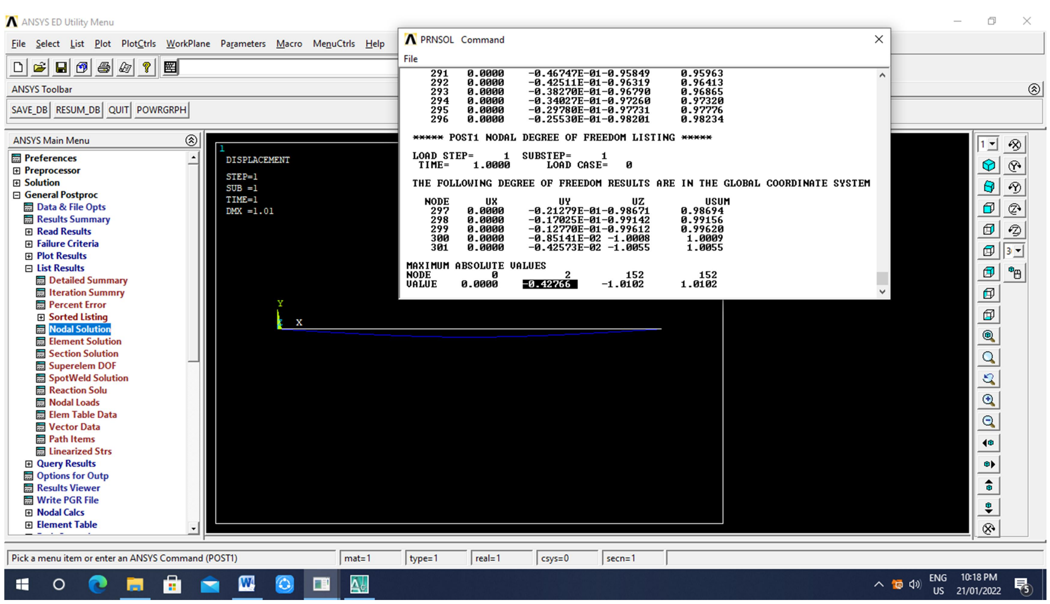
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

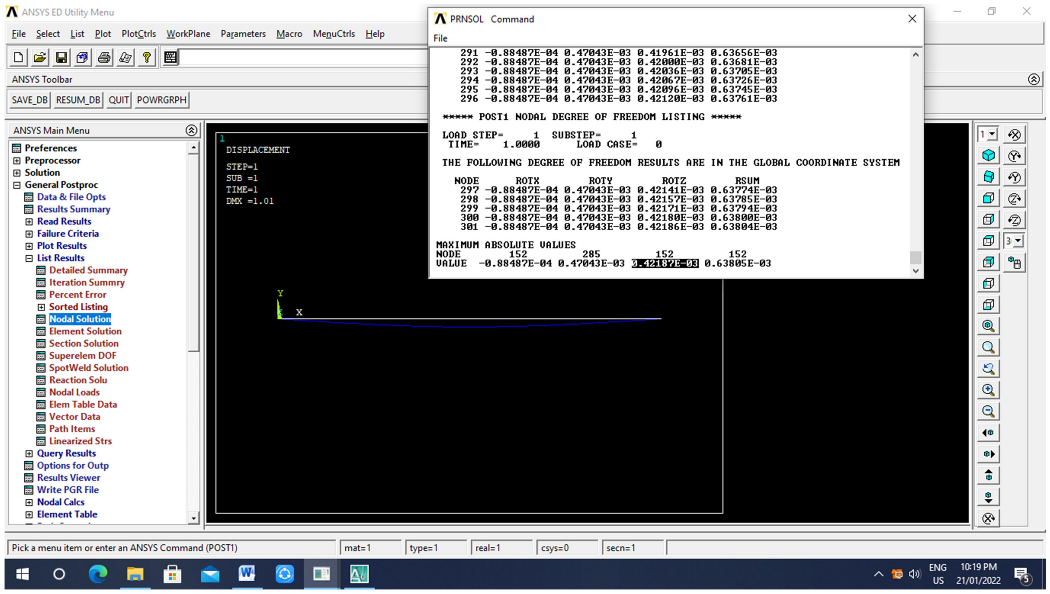
**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → UY → Apply → pick keypoint 3 → Apply → DOFs to be constrained → UY → ok.
- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -10e3 (-ve value) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → LS → and type LS, 1 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Nodal solution → DOF solution → Displacement vector Sum → ok (Deflection will be displayed with the node numbers).
- List Results → Nodal solution → DOF solution → Rotation vector Sum → ok (slope will be displayed with the node numbers).






**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Slope (radians) | 0.00042 | 0.00042187 |
| Deflection (mm) | 0.42 | 0.42766 |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: l=3m=3000mm, W=10kN=10000N, b=100mm, d=200mm, E=2x$10^5$ N/$mm^2$

Moment of inertia for rectangular section about horizontal centroidal axis

$$I = \frac{bd3}{12} = \frac{100x2003}{12} = 66.67x106\ mm4$$

**Slope at free end,**

ϴ = $Wl^2$/16EI = $10000x3000^2$/ (16x2x$10^5$x66.67x$10^6$) =**0.00042 radians**

**Deflection at free end,**

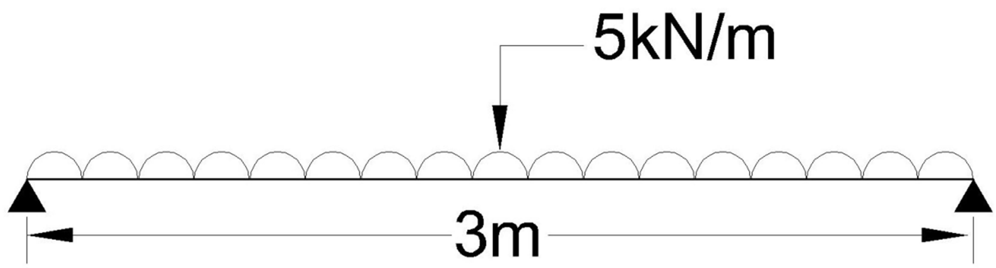
y = $Wl^3$/48EI = $10000x3000^3$/ (48x2x$10^5$x66.64x$10^6$) = **0.42mm**

# EXPERIMENT NO: 14

## Slope and deflection of simply supported Beam with UDL

**Aim:** To find the slope and deflection for the given simply supported beam due to applied load both manually and using ANSYS software.

**Problem Description:** Compute the slope and deflection for the beam shown. Take rectangular c/s area of 0.1 m x 0.15 m, Young's Modulus of 200GPa, Poisson's ratio 0.3.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → B = 150, H = 100 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 3000, 0, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2 → ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.
- Plot → Lines

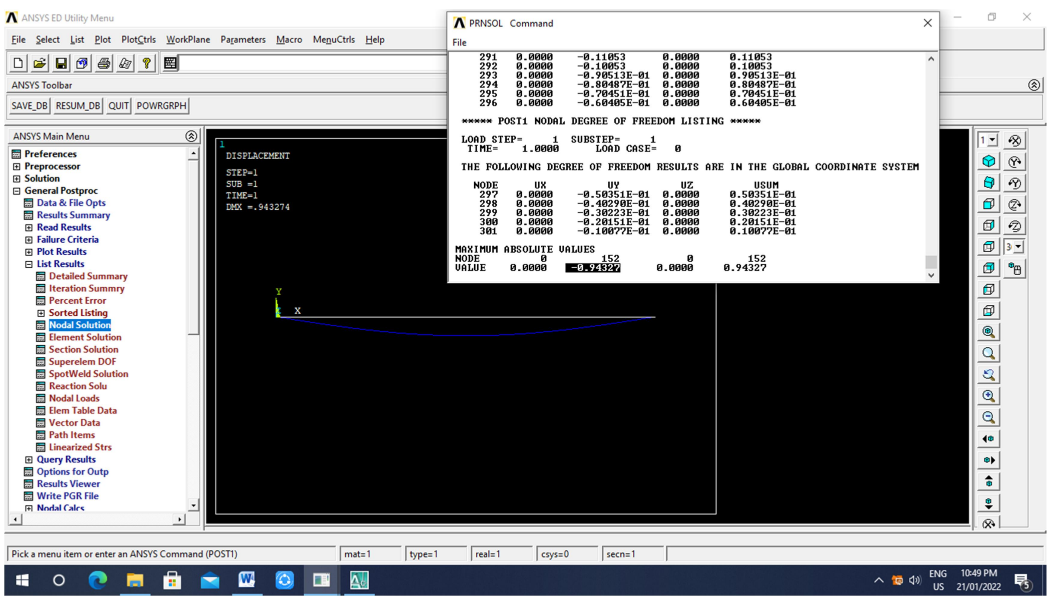
**Step 3**: Solution

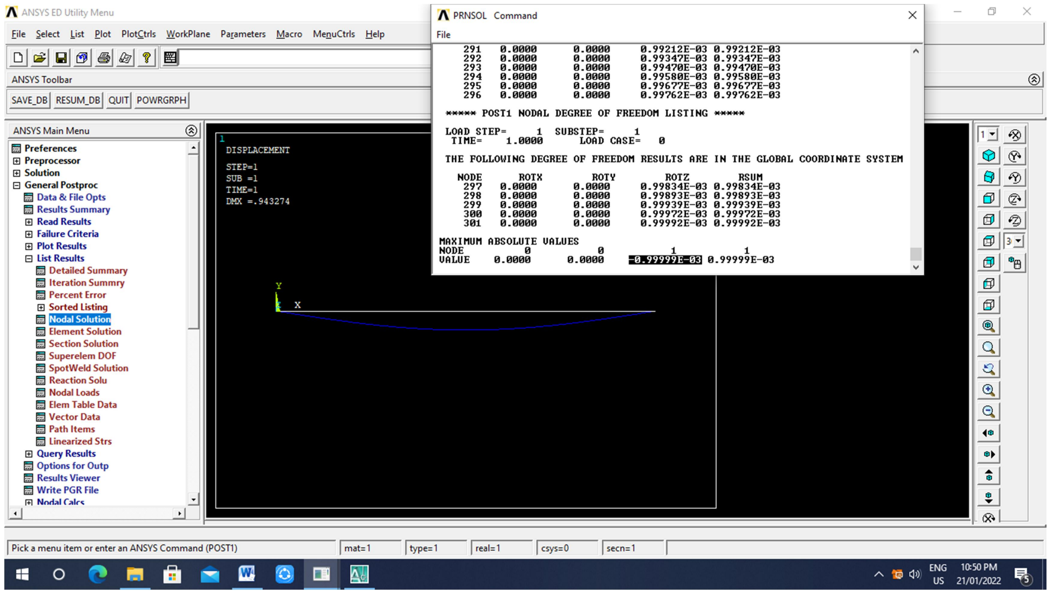
- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → UY → Apply → pick keypoint 2 → Apply → DOFs to be constrained → UY → ok.

- Define loads → Apply → Structural → Pressure → on beams → tick on box → drag and select the beam → apply → LKEY Load key → 2 → Pressure value at node I → 5 → ok → close.
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Element table → Define table → Add → Result data item → By data Sequence item num → LS → and type LS, 1 → ok → close.
- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Nodal solution → DOF solution → Displacement vector Sum → ok (Deflection will be displayed with the node numbers).
- List Results → Nodal solution → DOF solution → Rotation vector Sum → ok (slope will be displayed with the node numbers).






**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Slope (radians) | $9.99 \times 10^{-4}$ | $9.99 \times 10^{-4}$ |
| Deflection (mm) | 0.937 | 0.94327 |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: l=3m=3000mm, w=5kN/m=5N/mm, b=100mm, d=150mm, E=$2 \times 10^5$ N/mm$^2$

Moment of inertia for rectangular section about horizontal centroidal axis

$$I = \frac{bd3}{12} = \frac{100x1503}{12} = 28.13x106\ mm4$$

**Slope at free end,**

$\Theta = wl^3/24EI = 5 \times 3000^3/ (24 \times 2 \times 10^5 \times 28.13 \times 10^6)$ =**$9.99 \times 10^{-4}$ radians**

**Deflection at free end,**

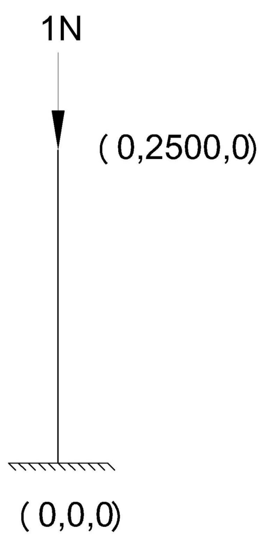
$y = 5wl^4/384EI = 5 \times 5 \times 3000^4/ (384 \times 2 \times 10^5 \times 28.13 \times 10^6)$ = **0.937mm**

# EXPERIMENT NO: 15

## Analysis of column (one end fixed, other end free)

**Aim:** To find the buckling load for the given column manually and using ANSYS software.

**Problem Description:** A solid round bar 60mm in diameter and 2.5m long is used as strut. One end of the strut is fixed, while its other end is free. Find the buckling load for this strut. Take $E=2x10^5$ $N/mm^2$.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → R=30 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 0, 2500, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2→ ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.

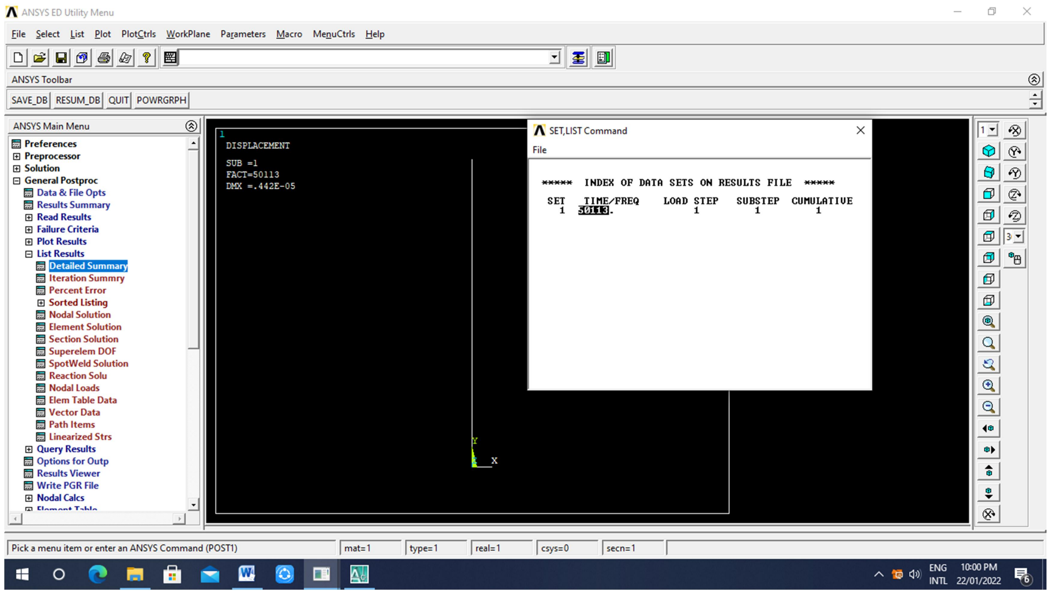
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → all DOF → ok.
- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -1 (-ve value, Dummy load) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → new analysis → eigen buckling → ok
- Analysis type → analysis options → buckling analysis options → block lanczos → No. of modes to extrac = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → expansion pass → expansion pass = on → ok → No. of modes to expand = 1 → ok
- Load step options → expansion pass → single expand → expand modes → No. of modes to expand = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Detailed summary (Buckling load will be displayed).




**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Buckling load (N) | 50230 | 50113 |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: d=60mm, l=2.5m=2500mm, E=2x$10^5$ N/$mm^2$

Moment of inertia

$$I = \frac{\pi d4}{64} = \frac{\pi x604}{64} = 636173\ mm4$$

**Buckling load,** for one end fixed, other end free

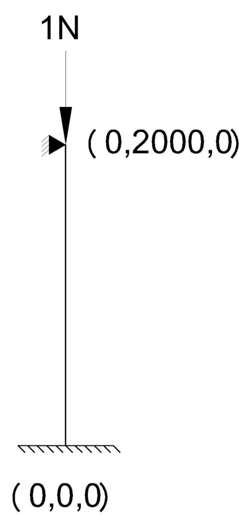
P = $\pi^2$EI/4$l^2$ = $\pi^2$x2x$10^5$x636173/ (4x$2500^2$) = **50230 N**

# EXPERIMENT NO: 16

## Analysis of column (one end fixed, other end hinged)

**Aim:** To find the buckling load for the given column manually and using ANSYS software.

**Problem Description:** A solid round bar 50mm in diameter and 2m long is used as strut. One end of the strut is fixed, while its other end is hinged. Find the buckling load for this strut. Take E=2x$10^5$ N/$mm^2$.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → R=25 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 0, 2000, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2→ ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.

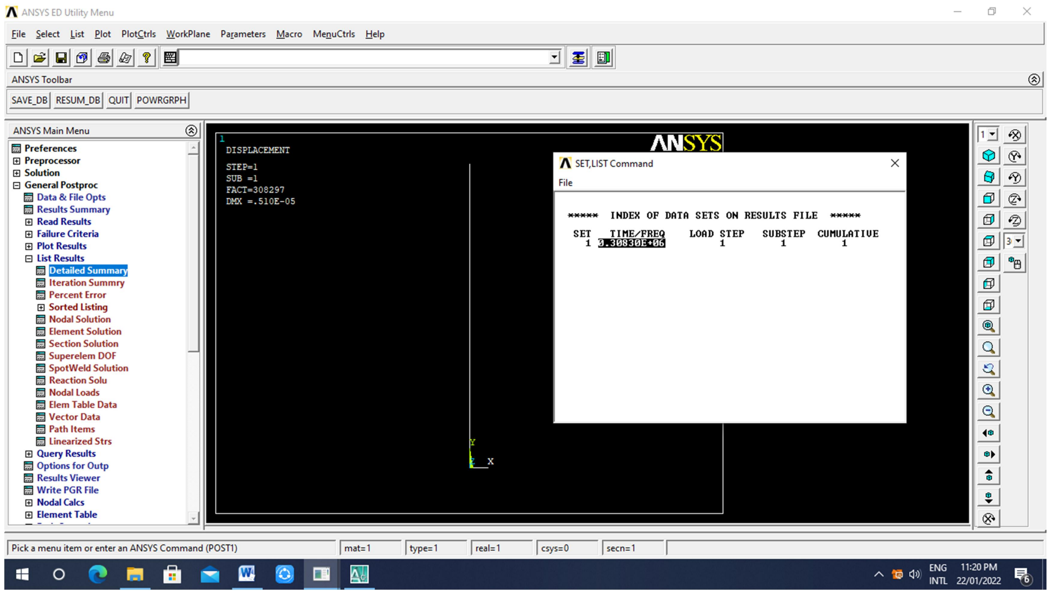
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → all DOF → Apply → pick node 2 → Apply DOFs to be constrained → UX, UZ, ROTX, ROTY → ok.
- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -1 (-ve value, Dummy load) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → new analysis → eigen buckling → ok
- Analysis type → analysis options → mode extraction method → block lanczos → No. of modes to extrac = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → expansion pass → expansion pass = on → ok → No. of modes to expand = 1 → ok
- Load step options → expansion pass → single expand → expand modes → No. of modes to expand = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Detailed summary (Buckling load will be displayed).




**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Buckling load (N) | $30.28 \times 10^4$ | $30.83 \times 10^4$ |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: d=50mm, l=2m=2000mm, E=$2 \times 10^5$ N/mm$^2$

Moment of inertia

$$I = \frac{\pi d4}{64} = \frac{\pi x504}{64} = 306796mm4$$

**Buckling load,** for one end fixed, other end hinged

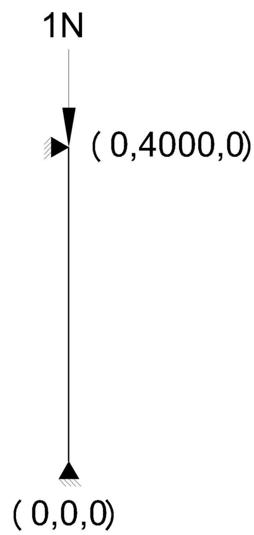
P = $2\pi^2 EI/l^2 = 2x\pi^2 x2x10^5 x306796/(2000^2)$ = **$30.28 \times 10^4$ N**

# EXPERIMENT NO: 17

## Analysis of column (both ends hinged)

**Aim:** To find the buckling load for the given column manually and using ANSYS software.

**Problem Description:** a mild steel tube 4m long and 30mm internal diameter and 38mm external diameter is used as a strut with both ends hinged. Find the buckling load. Take $E=2.1 \times 10^5$ N/mm$^2$.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2.1e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → $R_i$=15, $R_o$=19 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 0, 4000, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2→ ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.

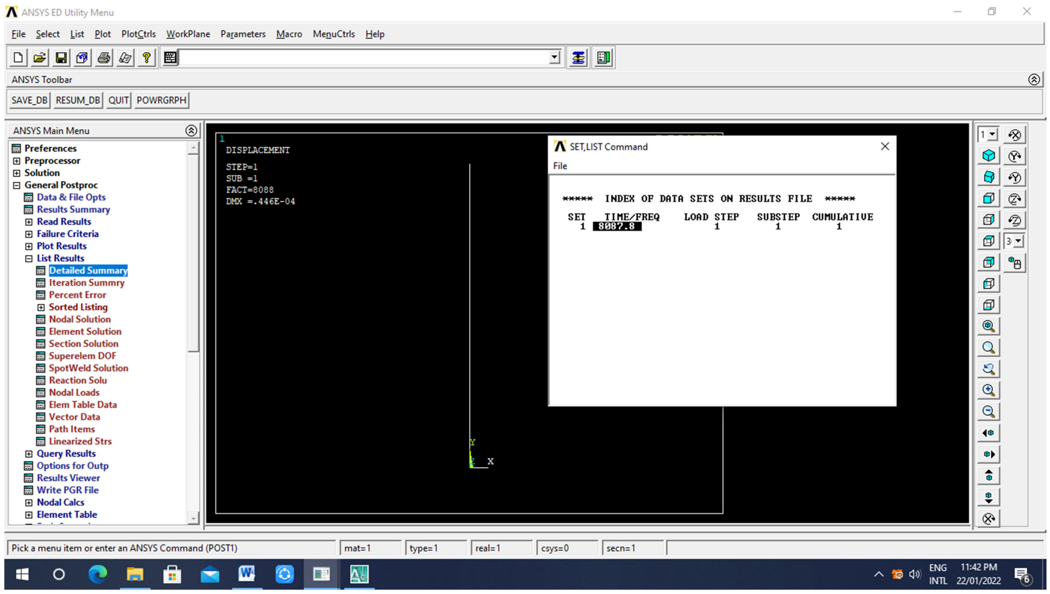
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → UX, UY, UZ, ROTX, ROTY → Apply → pick node 2 → Apply DOFs to be constrained → UX, UZ, ROTX, ROTY → ok.
- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -1 (-ve value, Dummy load) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → new analysis → eigen buckling → ok
- Analysis type → analysis options → mode extraction method → block lanczos → No. of modes to extrac = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → expansion pass → expansion pass = on → ok → No. of modes to expand = 1 → ok
- Load step options → expansion pass → single expand → expand modes → No. of modes to expand = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Detailed summary (Buckling load will be displayed).




**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Buckling load (N) | 8108.2 | 8087.8 |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: l=4m=4000mm, d=30mm, D=38mm, E=2.1x10$^5$ N/mm$^2$

Moment of inertia

$$I = \frac{\pi(D4 - \mathrm{d}4)}{64} = \frac{\pi x(384 - 304)}{64} = 62593mm4$$

**Buckling load,** for both ends hinged

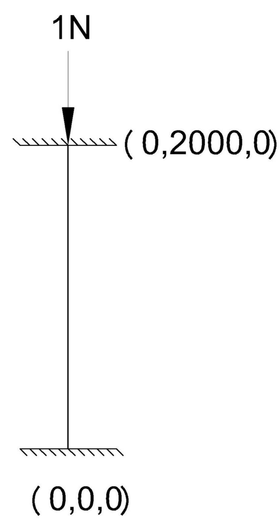
P = $\pi^2$EI/l$^2$ = $\pi^2$x2.1x10$^5$x62593/ (4000$^2$) = **8108.2 N**

# EXPERIMENT NO: 18

## Analysis of column (both ends fixed)

**Aim:** To find the buckling load for the given column manually and using ANSYS software.

**Problem Description:** a mild steel tube 2m long and 40mm internal diameter and 50mm external diameter is used as a strut with both ends fixed. Find the buckling load. Take $E=2x10^5$ N/mm$^2$.




**Procedure:**

**Step 1**: ANSYS Main Menu → Preferences → select → STRUCTURAL → OK

**Step 2**: Preprocessor

- Element type → Add/Edit/Delete → Add → BEAM → 2 node 188 → ok → close.
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → EX → 2e5 → PRXY → 0.3 → ok → close.
- Sections → Beam → Common section → $R_i$=20, $R_o$=25 → ok.
- Modeling → create → key Points → In Active CS → Apply (first node is created) → x, y, z location in CS → 0, 2000, 0 → Apply (second node is created)
- Modeling → Create → Lines → Lines → Straight line → pick 1 & 2→ ok
- Meshing → Size Cntrls → Manual size → Global → size → SIZE Element edge length = 10 → NDIV No. of element divisions = 100 → ok
- Meshing → Mesh → Lines → Pick all.

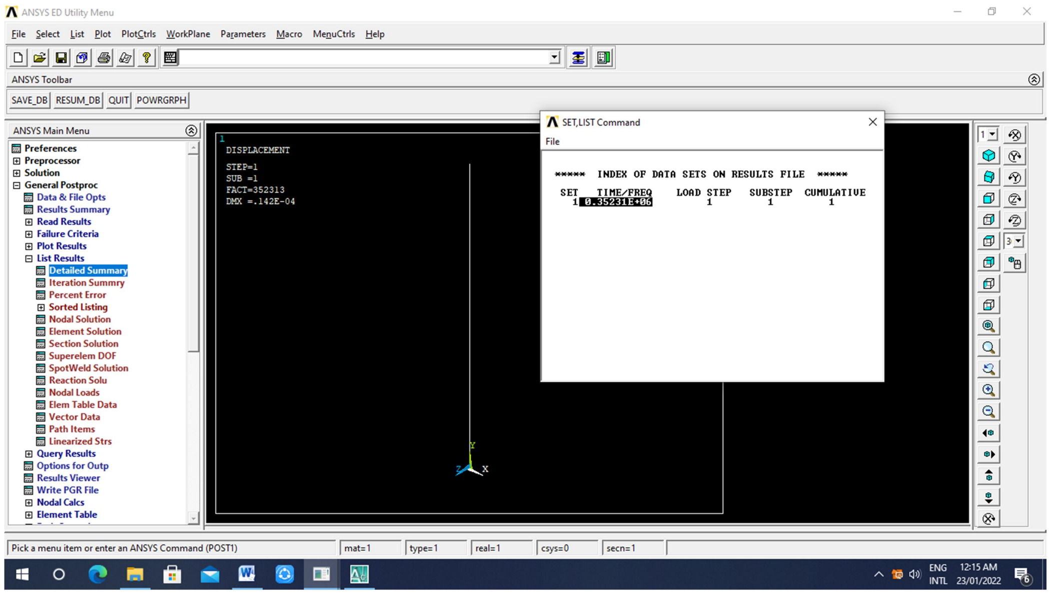
- Plot → Lines

**Step 3**: Solution

- Define loads → Apply → Structural → Displacement → on Keypoints → pick node 1 → Apply → DOFs to be constrained → all DOF → Apply → pick node 2 → Apply DOFs to be constrained → UX, UZ, ROTX, ROTY, ROTZ → ok.
- Define loads → Apply → Structural →Force/Moment → on Key Points → pick node 2 → Apply → direction of Force/mom → FY → Force/Moment value → -1 (-ve value, Dummy load) → ok.
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → new analysis → eigen buckling → ok
- Analysis type → analysis options → mode extraction method → block lanczos → No. of modes to extrac = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.
- Finish
- Analysis type → expansion pass → expansion pass = on → ok → No. of modes to expand = 1 → ok
- Load step options → expansion pass → single expand → expand modes → No. of modes to expand = 1 → ok
- Solve → current LS → ok (solution is done is displayed) → close → close.

**Step 4**: General Post Processor

- Plot Result → Deformed shape → def+undeformed → ok.
- List Results → Detailed summary (Buckling load will be displayed).




**Comparison between Theoretical & ANSYS results:**

| Particulars | Theoretical | ANSYS |
|---|---|---|
| Buckling load (N) | $35.75 x10^4$ | $35.231 x10^4$ |

**Conclusion**: ANSYS simulation and software results are near to the theoretical or analytical results.

**Calculation:**

Given: l=2m=2000mm, d=20mm, D=25mm, E=$2x10^5$ N/$mm^2$

Moment of inertia

$$I = \frac{\pi(D4 - d4)}{64} = \frac{\pi x(504 - 404)}{64} = 181132.45mm4$$

**Buckling load,** for both ends fixed

P = $4\pi^2 EI/l^2 = 4x\pi^2 x2x10^5 x181132.45/ (2000^2)$ = **$35.75x10^4$ N**