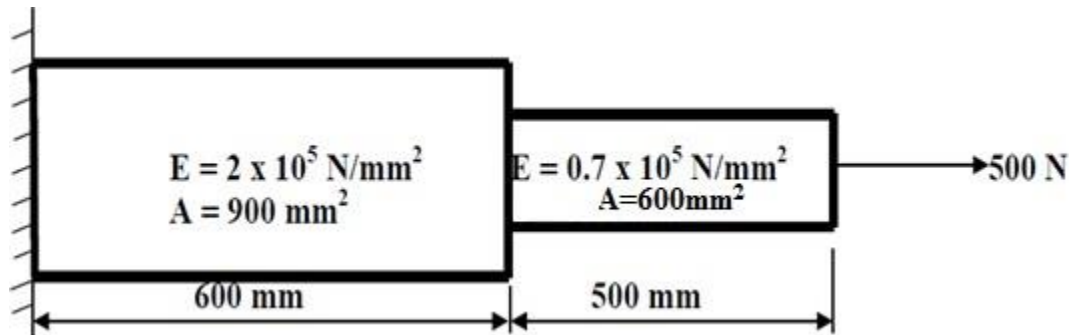


Problem 1.3: Stepped Bar

Consider the stepped bar shown in figure below. Determine the Nodal Displacement, Stress in each element, Reaction forces.



1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – link, 3D Finit stn 180 – ok- close.
3. Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA 1 – 900 – apply-ok
4. Add – ok – real constant set no – 2 – cross-sectional AREA 2 – 600-ok
5. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – $2e5$ –PRXY – 0.3- material- new material-define material id=2- Structural – Linear – Elastic – Isotropic – EX – $0.7e5$ –PRXY – 0.3– ok – close.
6. Modeling – Create – key points– In Active CS, =0, Y=0 – Apply (first key point is created) – location in active CS, X= 600, Y=0, apply (second key point is created) - location in active CS X=1100, Y=0(third key point is created) -ok.
7. Modeling-Create – lines-straight lines-pick key points 1 & 2-ok- pick key points 2 & 3-ok
8. Meshing-mesh attributes-picked lines (pick the lines)-ok-material no= 1, real constants set no = 1, element type no =1, link 1, element section= none defined-pick the other line-ok-material number 2-define material id 2- real constants set no = 2, element type no =2-element section= none defined-ok.
9. Meshing-size controls-manual size-lines-all lines- no of element divisions=10(yes)-ok
10. Meshing-mesh tool-mesh-pick the lines-ok (the color changes to light blue)
11. Loads – Define loads – apply – Structural – Displacement – on key points- pick key point 1 – apply –DOFs to be constrained – ALL DOF, displacement value=0 – ok.

12. Loads – Define loads – apply – Structural – Force/Moment – on key points- pick last key point – apply – direction of For/Mom – FX – Force/Moment value – 500 (+ve value) – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. Element table – Define table – Add –‘Results data item’ – By Sequence num – LS – LS1 – ok.
15. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
16. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. Plot results- nodal solution-ok-DOF solution- x component of displacement-ok.
18. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

Analytical approach:

Calculation:

Displacement:_____

Stress in each element:_____

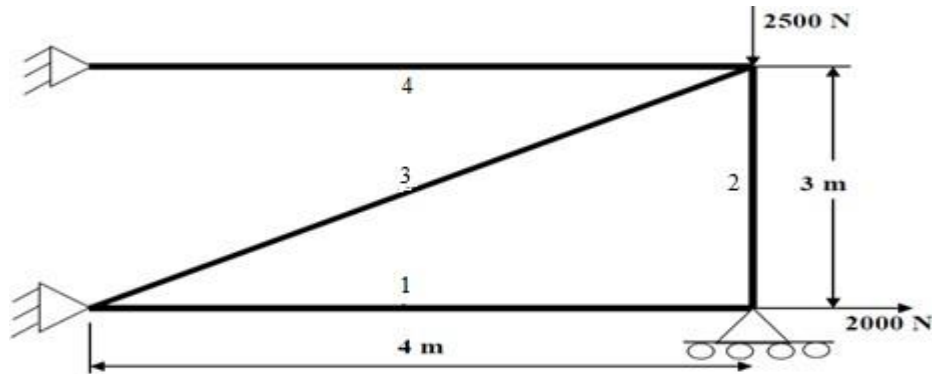
Reaction force: _____

ANSYS Results:

	ANSYS	Theoretical
Deformation		
Stress		
Reaction		

2. TRUSSES

Problem 2.1: Consider the four bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. $E = 210 \text{ GPa}$, $A = 0.1 \text{ m}^2$.



1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– Ok – close.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 4 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 4, 3 (x, y value w.r.t first node) – apply (third node is created) – 0, 3 (x, y value w.r.t first node) – ok (forth node is created).
6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number – 1 – ok
7. 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).
8. 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.
9. 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 – Structural –
10. Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY –

Force/Moment value – -2500 (-ve value) – ok.

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

12. Element table – Define table – Add –‘Results data item’ – By Sequence num – LS – LS1 – ok.

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

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

15. Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.

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

RESULT:

Analytical approach:

Calculation:

Displacement:_____

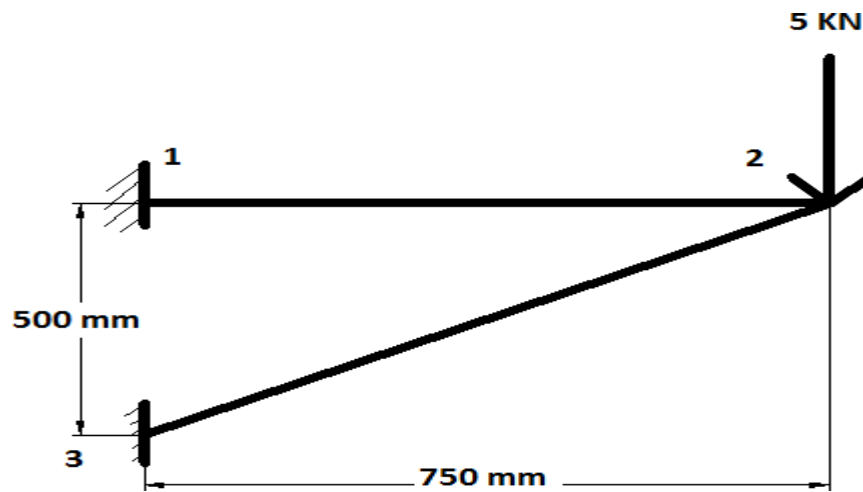
Stress:_____

Reaction force: _____

ANSYS Results:

	ANSYS	Theoretical
Deformation		
Stress		
Reaction		

Problem 2.2: Consider the two bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. $E = 210 \text{ GPa}$, $A = 0.1 \text{ m}^2$.



1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– Ok – close.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 0.75 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS –(0, -0.5),(x, y value w.r.t first node) – ok (third node is created)
6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number – 1 – ok
7. Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3— ok (elements are created through nodes).
8. Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 &3 – apply – DOFs to be constrained – All DOF – ok
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – 5000 (-ve value)
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Element table – Define table – Add –‘Results data item’ – By Sequence num – LS – LS1 – ok.
12. Plot results – contour plot –Element table – item to be plotted LS,1, avg common

nodes- yes average- ok.

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

14. Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.

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

RESULT:

Analytical approach:

Calculation:

Displacement: _____

Stress: _____

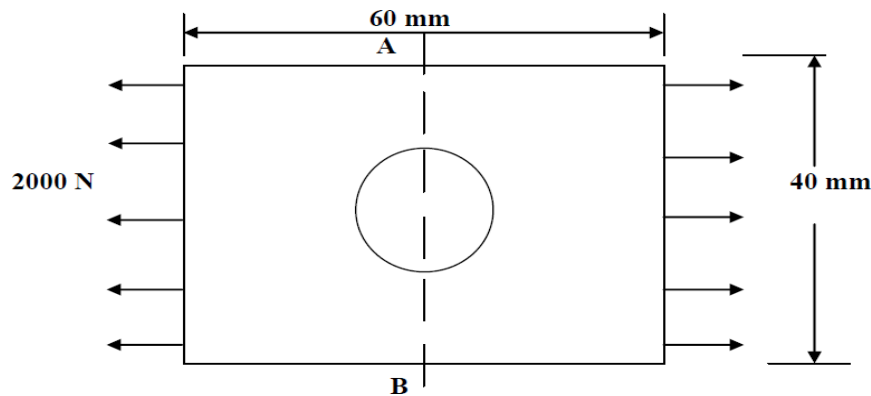
Reaction force: _____

ANSYS Results:

	ANSYS	Theoretical
Deformation		
Stress		
Reaction		

Stress analysis of a rectangular plate with circular hole

Problem 4.1: 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). $E = 210\text{GPa}$, $t = 1$ mm, Poisson's ratio = 0.3, Dia. of the circle = 10 mm, Analysis assumption – plane



stress with thickness is used.

1. Ansys Main Menu – Preferences-Select – STRUCTURAL-h method – ok
2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 42 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – Thickness – 1 – ok.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – $2.1e5$ – PRXY – 0.3 – ok – close.
5. Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 60, 0, 40 – ok.
6. Create – Area – Circle – solid circle – X, Y, radius – 30, 20, 5 – ok.
7. Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply – pick area which is to be deleted (circle) – ok.
8. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.
9. Loads – Define loads – apply – Structural – Displacement – on Nodes – select box – drag the left side of the area – apply – DOFs to be constrained – ALL DOF – ok.
10. Loads – Define loads – apply – Structural – Force/Moment – on Nodes – select box – drag the right side of the area – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok.
11. Solve – current LS – ok (Solution is done is displayed) – close.

12. Deformed shape-Plot Results – Deformed Shape – def+undeformed – ok.

13. Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

RESULT:

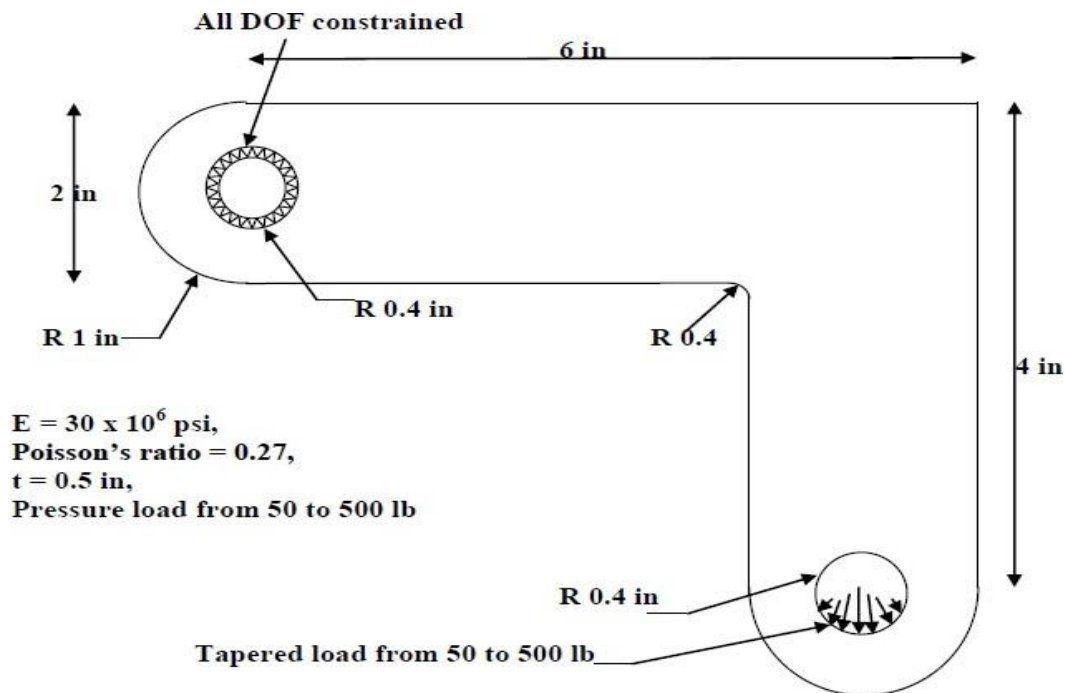
Analytical approach:

Calculation:

ANSYS Results:

	ANSYS	Theoretical
Deformation		
Stress		

Problem 4.2: The corner angle bracket is shown below. The upper left hand pin-hole is constrained around its entire circumference and a tapered pressure load is applied to the bottom of lower right hand pin-hole. Compute Maximum displacement, Von-Mises stress.



1. Ansys Main Menu – Preferencesselect – STRUCTURAL – ok
2. Element type – Add/Edit/Delete – Add – Solid – Quad 8 node – 82 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – Thickness – 0.5 – ok.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 30e6 –PRXY – 0.27 – ok – close.
5. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 6, 0, 2 – apply –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 4, 6, -2, 2 – ok. Create – Area –Circle – solid circle – X, Y, radius – 0, 1, 1 – apply – X, Y, radius – 5, -2, 1 – ok.
6. Operate – Booleans – Add – Areas – pick all.
7. Create – Lines – Line fillet – pick the two lines where fillet is required – apply – fillet radius – 0.4 – ok. Create – Areas – Arbitrary – by lines – pick filleted lines – ok. Operate – Booleans –
8. Add – Areas – pick all. Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 0.4

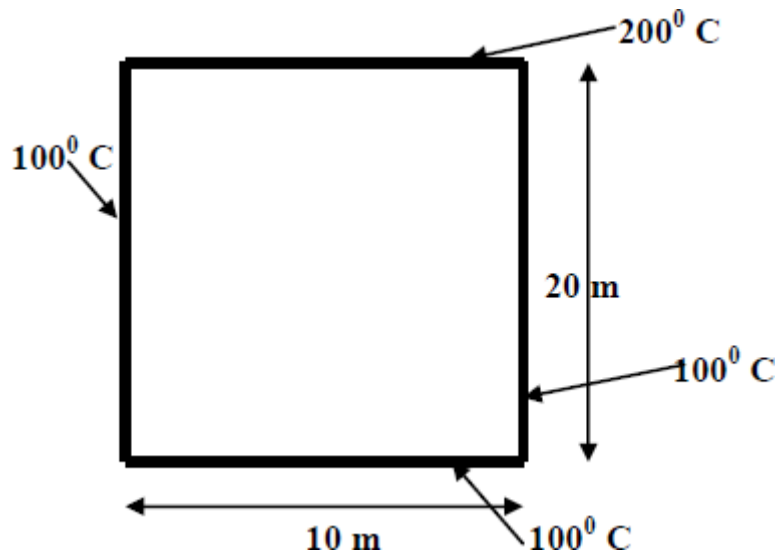
- apply –X, Y, radius – 5, -2, 0.4 – ok.
- 9. Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (bracket)
– apply – pick areas which is to be deleted (pick two circles) – ok.
- 10. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh
Tool – Refine– pick all – Level of refinement – 3 – ok.
- 11. Loads – Define loads – apply – Structural – Displacement – on Lines – select the
inner lines of the upper circle – apply – DOFs to be constrained – ALL DOF – ok.
- 12. Loads – Define loads – apply – Structural – Pressure – on Lines – Pick line defining
bottom left part of the circle – apply – load PRES value – 50 – optional PRES value –
500 – ok. Structural – Pressure – on Lines – Pick line defining bottom right part of the
circle – apply – load PRES value – 500 – optional PRES value – 50 – ok.
- 13. Solve – current LS – ok (Solution is done is displayed) – close.
- 14. Plot Results – Deformed Shape – def+undeformed – ok.
- 15. Plot results – contour plot – Element solu. – Stress – Von Mises Stress – ok (the stress
distribution diagram will be displayed).
- 16. PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

PART B

THERMAL ANALYSIS

Problem 5.1: Solve the 2-D heat conduction problem for the temperature distribution within the rectangular plate. Thermal conductivity of the plate, $K_{XX}=401 \text{ W/(m-K)}$.



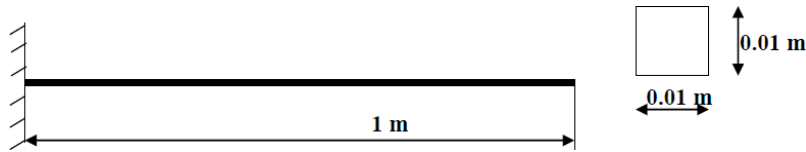
1. Ansys Main Menu – Preferences-select – THERMAL- h method– ok
2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – elementbehavior K3 – Plane stress with thickness – ok – close.
3. Material Properties – material models – Thermal – Conductivity – Isotropic – K_{XX} – 401.
4. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok.
5. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.
6. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 100°C lines – apply – DOFs to be constrained – TEMP – Temp value – 100°C – ok.
7. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 100°C lines –
8. Solve – current LS – ok (Solution is done is displayed) – close.
9. Read results-last set-ok

10. List results-nodal solution-select temperature-ok
11. Observe the nodal solution per node.
12. From the menu bar-plot ctrl-s-style-size and shape-display of the element-click on real constant multiplier=0.2, don't change other values-ok.
13. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok
14. Element table-define table-add-enter user label item=HTRANS, select by sequence no SMISC, 1-ok-close.
15. Element table-list table-select HTRANS-ok

RESULT:

Problem 6.1: Modal Analysis of Cantilever beam for natural frequency determination.

Modulus of elasticity = 200GPa, Density = 7800 Kg/m³.



1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 200e9– PRXY – 0.27 – Density – 7800 – ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=0.01, h=0.01- preview-ok.
5. Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations –1,0 – ok (Keypoints created).
6. Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.
7. Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh– Lines – Pick All – ok.
8. Solution – Analysis Type – New Analysis – Modal – ok.
9. Solution – Analysis Type – Subspace – Analysis options – no of modes to extract – 5 – no of modes to expand – 5 – ok – (use default values) – ok.
10. Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick firstkeypoint – apply – DOFs to be constrained – ALL DOF – ok.
11. Solve – current LS – ok (Solution is done is displayed) – close.
12. Result Summary
13. Read Results – First Set
14. Plot Results – Deformed Shape – def+undeformed – ok.
15. PlotCtrls – Animate – Deformed shape – def+undeformed-ok.
16. Read Results – Next Set
17. Plot Results – Deformed Shape – def+undeformed – ok.
18. PlotCtrls – Animate – Deformed shape – def+undeformed-ok

RESULT:

Analytical solution:

ANSYS Results:

VIVA QUESTIONS & ANSWERS

1. Theories of failures

- a. **Maximum Principal Stress Theory**- A material in complex state of stress fails, when the maximum principal stress in it reaches the value of stress at elastic limit in simple tension.
- b. **Maximum Shear Stress Theory**- A material in complex state of stress fails when the maximum shearing stress in it reaches the value of shearing stress at elastic limit in uniaxial tension test.
- c. **Maximum Principal Strain Theory**-Failure in a complex system occurs when the maximum strain in it reaches the value of the strain in uniaxial stress at elastic limit.
- d. **Maximum Strain Energy Theory**- A material in complex state of stress fails when the maximum strain energy per unit volume at a point reaches the value of strain energy per unit volume at elastic limit in simple tension test.
- e. **Maximum Distortion Energy Theory**-This theory is also known as Von-Mises criteria for failure of elastic bodies. According to this theory part of strain energy causes only changes in volume of the material and rest of it causes distortion. At failure the energy causing distortion per unit volume is equal to the distortion energy per unit volume in uniaxial state of stress at elastic limit.

2. What is factor of safety?

The maximum stress to which any member is designed is much less than the ultimate stress and this stress is called working stress. The ratio of ultimate stress to working stress is called factor of safety.

3. What is Endurance limit?

The max stress at which even a billion reversal of stress cannot cause failure of the material is called endurance limit.

4. Define: Modulus of rigidity, Bulk modulus

Modulus of rigidity: It is defined as the ratio of shearing stress to shearing strain within elastic limit.

Bulk modulus: It is defined as the ratio of identical pressure 'p' acting in three mutually perpendicular directions to corresponding volumetric strain.

5. What is proof resilience?

The maximum strain energy which can be stored by a body without undergoing

permanent deformation is called proof resilience.

6. What is shear force diagram?

A diagram in which ordinate represent shear force and abscissa represents the position of the section is called SFD.

7. What is bending moment diagram?

A diagram in which ordinate represents bending moment and abscissa represents the position of the section is called BMD.

8. Assumptions in simple theory of bending.

- a. The beam is initially straight and every layer of it is free to expand or contract.
- b. The material is homogeneous and isotropic.
- c. Young's modulus is same in tension and compression.
- d. Stresses are within elastic limit.
- e. Plane section remains plane even after bending.
- f. The radius of curvature is large compared to depth of beam.

9. State the three phases of finite element method.

Preprocessing, Analysis & Post processing

10. What are the h and p versions of finite element method?

Both are used to improve the accuracy of the finite element method. In h version, the order of polynomial approximation for all elements is kept constant and the numbers of elements are increased. In p version, the numbers of elements are maintained constant and the order of polynomial approximation of element is increased.

11. What is the difference between static analysis and dynamic analysis?

Static analysis: The solution of the problem does not vary with time is known as static analysis. E.g.: stress analysis on a beam.

Dynamic analysis: The solution of the problem varies with time is known as dynamic analysis. E.g.: vibration analysis problem.

12. What are Global coordinates?

The points in the entire structure are defined using coordinates system is known as global coordinate system.

13. What are natural coordinates? A natural coordinate system is used to define any point

inside the element by a set of dimensionless number whose magnitude never exceeds unity. This system is very useful in assembling of stiffness matrices.

14. What is a CST element?

Three node triangular elements are known as constant strain triangular element. It has 6 unknown degrees of freedom called $u_1, v_1, u_2, v_2, u_3, v_3$. The element is called CST because it has constant strain throughout it.

15. Define shape function.

In finite element method, field variables within an element are generally expressed by the following approximate relation:

$\Phi(x,y) = N_1(x,y) \Phi_1 + N_2(x,y) \Phi_2 + N_3(x,y) \Phi_3 + N_4(x,y) \Phi_4$ where Φ_1, Φ_2, Φ_3 and Φ_4 are the values of the field variables at the nodes and N_1, N_2, N_3 and N_4 are interpolation function. N_1, N_2, N_3, N_4 are called shape functions because they are used to express the geometry or shape of the element.

16. What are the characteristics of shape function?

The characteristics of the shape functions are as follows:

- The shape function has unit value at one nodal point and zero value at the other nodes.
- The sum of shape functions is equal to one.

17. Why polynomials are generally used as shape function?

- Differentiation and integration of polynomials are quite easy.
- The accuracy of the results can be improved by increasing the order of the polynomial.
- It is easy to formulate and computerize the finite element equations.

18. State the properties of a stiffness matrix.

The properties of the stiffness matrix $[K]$ are:

- It is a symmetric matrix.
- The sum of the elements in any column must be equal to zero.
- It is an unstable element, so the determinant is equal to zero.

19. What are the difference between boundary value problem and initial value problem?

The solution of differential equation obtained for physical problems which satisfies some specified conditions known as boundary conditions. If the solution of differential

equation is obtained together with initial conditions then it is known as initial value problem. If the solution of differential equation is obtained together with boundary conditions then it is known as boundary value problem.

20. What is meant by plane stress?

Plane stress is defined as a state of stress in which the normal stress (α) and the shear stress directed perpendicular to plane are zero.

21. Define plane strain.

Plane strain is defined to be a state of strain in which the strain normal to the xy plane and the shear strains are assumed to be zero.

22. Define Quasi-static response.

When the excitations are varying slowly with time then it is called quasi-static response.

23. What is a sub parametric element?

If the number of nodes used for defining the geometry is less than the number of nodes used for defining the displacements is known as sub parametric element.

24. What is a super parametric element?

If the number of nodes used for defining the geometry is more than the number of nodes used for defining the displacements is known as sub parametric element.

25. What is meant by isoparametric element?

If the number of nodes used for defining the geometry is same as number of nodes used for defining the displacements then it is called parametric element.

26. What is the purpose of isoparametric element?

It is difficult to represent the curved boundaries by straight edges finite elements. A large number of finite elements may be used to obtain reasonable resemblance between original body and assemblage. In order to overcome this drawback, iso parametric elements are used i.e for problems involving curved boundaries, a family of elements 'isoparametric elements' are used.

27. What are isotropic and orthotropic materials?

A material is isotropic if its mechanical and thermal properties are the same in all directions. Isotropic materials can have homogeneous or non-homogeneous microscopic structures.

Orthotropic materials: A material is orthotropic if its mechanical or thermal properties are

unique and independent in three mutually perpendicular directions.

28. What is discretization?

Discretization is the process of dividing given problem into several small elements, connected with nodes.

29. Steps in FEM

- Discretization
- Selection of the displacement models
- Deriving element stiffness matrices
- Assembly of overall equations/ matrices
- Solution for unknown displacements
- Computations for the strains/stresses