

FINITE ELEMENT METHOD IN DESIGN ANALYSIS**1.1 INTRODUCTION**

The finite element method has been a powerful tool for the numerical solution of a wide range of engineering problems. Application range from deformation and stress analysis of automotive, aircraft, building, defense, missile, and bridge structures to the field analysis of dynamics, stability, fracture mechanics, heat flux, fluid flow, magnetic flux, seepage and other flow problems. With the advances in computer technology and CAD systems Complex problems can be modeled with relative ease. Several alternate configurations can be tried out on a computer before the first prototype is built. The basics in engineering field is must to idealize the given structure for the required behavior. The proven knowledge in the typical problem area, modeling techniques, data transfer and integration, computational aspects of the finite element method is essential.

Most often it is not possible to ascertain the behavior of complex continuous system without some sort of approximation. For simple members like uniform beams, plates etc,. Classical solutions can be sought by forming differential and/or integral equations through structures like machine tool frames. Pressure vessels, automobile bodies, ships, aircraft structures, domes etc, need some approximate treatment to arrive at their behavior, be it static deformation, dynamic properties or heat conducting property. Indeed these are continuous systems with their mass and elasticity being continuously distributed. The classical differential equation solution approach leads to intractability. To overcome this engineers and mathematicians have from time to time proposed complex structure is defined using a finite number of well-defined components. Such systems are then regarded as discrete systems. The discretization method could be finite difference approximation, various residual procedures etc.

Finite element method comes under this category of discretization methods. R.W. Clough appears to be the first to use this term of finite element since 1960s there has been much progress in the method. The method requires a large number of computations requiring a fast computer. In fact digital computer advances have been responsible for the expanding usage of the finite element method. The finite element method was initially developed to solve structural problems. Its use of late, has been rapidly extended to various fields. The delivery of applications of the method can be

seen from the following table 2.1, which still by no means can be claimed as complete since fields of usage are being continuously diversified.

1.2 METHODS OF ENGINEERING ANALYSIS

The general methods for solving the practical problems can be classified as shown in figure:

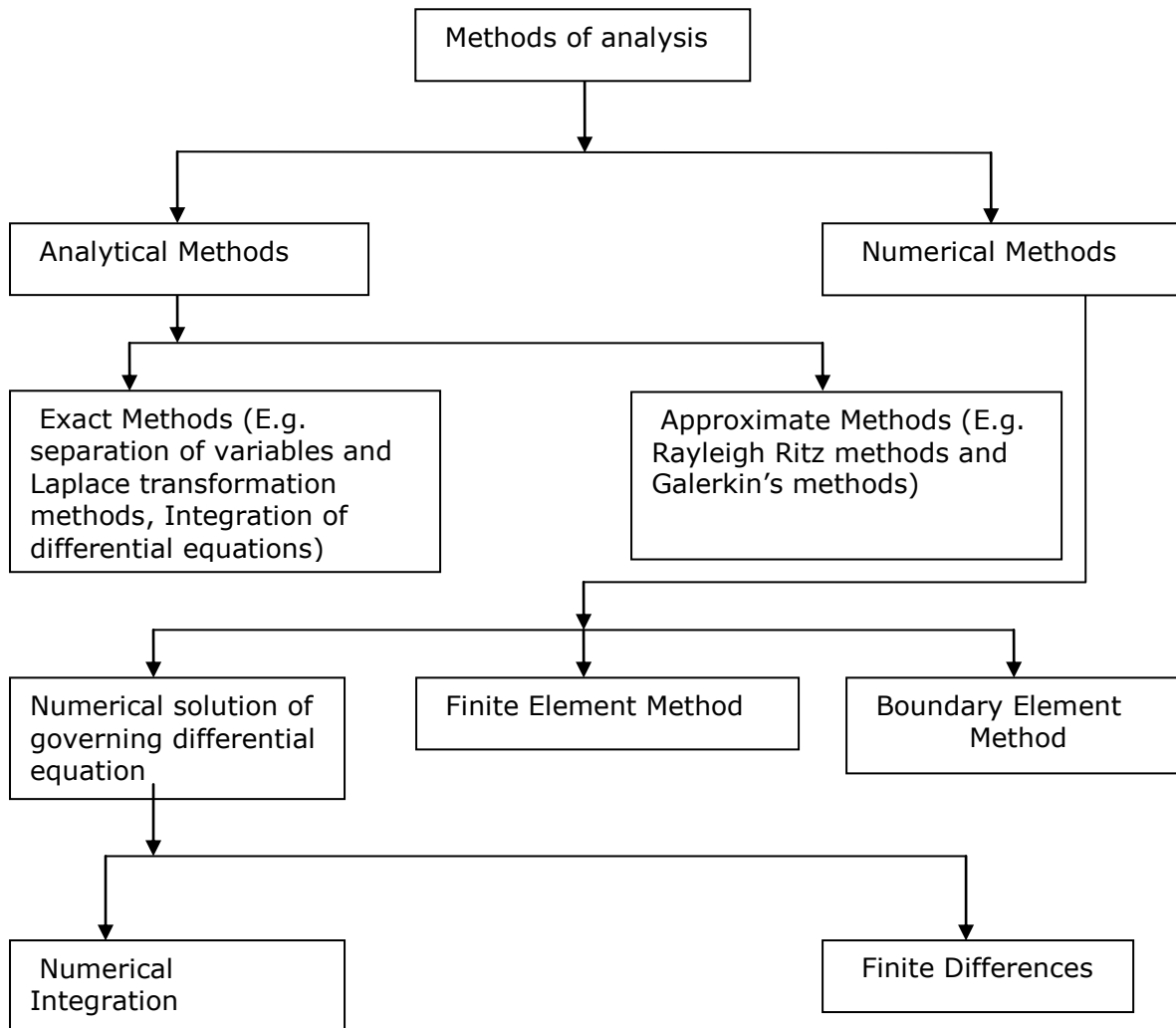


Fig.1.1. Block diagram of engineering analysis

- **Exact method** Provides exact solution to the problems, but the limitation of this method is that all practical problems cannot be solved and even if they can be solved, they may have very complex solution.
- **Approximate analytical methods** are alternative to the exact methods, in which certain functions are assumed to satisfy the geometric boundary conditions, but not necessary the governing equilibrium equations. These assumed functions, which are simpler, are then solved by any conventional method available. The solutions obtained from these methods have limited range of values of variables for which the approximate solution is nearer to the exact solution.
- **Numerical method** has been developed to solve almost all types of practical problems. The two common numerical methods used to solve the governing equations of practical problems are numerical integration and finite element technique.

-
-
- **Numerical integration methods** such as Runge-Kutta, Miline's method etc. adopt averaging of slopes of given function at the given initial values. These methods yield better solutions over the entire field width but sometimes laborious.
 - **Finite difference method** the differential equations are approximated by finite difference equation. Thus the given governing equation is converted to a set of algebraic equations. These simultaneous equations can be solved by any simple method such as Gauss Elimination, gauss-Seidal iteration method, Crout's method etc. the method of finite difference yield fairly good results and are relatively easy to program. Hence, they are popular in solving heat transfer and fluid flow problems. However, it is not suitable for problems with irregular geometry and not suitable for problems of rapidly changing variables such as stress concentration problems.
 - **Finite Element Method (F.E.M)** has emerged out to be powerful method for all kinds of practical problems. In this method the solution region is considered to be built up of many small-interconnected sub regions, called finite elements. These elements are applied with an interpolation model, which is simplified version of substitute to the governing equation of the material continuum property. The stiffness matrices obtained for these elements are assembled together and the boundary conditions of the actual problems are satisfied to obtain the solution all over the body or region FEM is well suited for computer programming.
 - **Boundary element method (B.E.M)** like finite method is being used in all engineering fields. In this approach, the governing differential equations are transformed into integral identities applicable over the surface or boundary. These integral identities are integrated over the boundary, which is divided into small boundary segments. As in finite element method provided that the boundary conditions are satisfied, set of linear algebraic equations emerges, for which a unique solution is obtained.

1.3 INTRODUCTION TO FEM

In the field of engineering, it is necessary to obtain the unknown field parameters like displacement or forces, which can describe the behavior of structures. These unknowns can be obtained by using exact analytical solutions or energy methods, which are formulated with governing equilibrium equations of analytical solutions on the basis of solid mechanics. However in practice the real structure viz., aircraft/automobiles, civil, naval architecture, offshore and nuclear engineering are large in size with irregular shapes, intermediate in nature, possessing geometrical material non-linearity and having complicated loading or boundary conditions.

In these cases, to reduce the continuous system to a discrete one by conventional method, a lot of effort and assumption must be made, which are

cumbersome and error prone. The advent of high-speed electronic computer has given tremendous opportunity to the numerical method approach. The most commonly used numerical technique in solving engineering problems is FEM, because

1. Algebraic work corresponding to the solid mechanics is programmed in systematic matrix calculations and formulated into mathematical model. User need not work with complicated algebra but only to deal with the discredited geometry of the physical structural.
2. Ability to consider alternate loading and bending condition for the same geometry.

-
-
3. Accuracy of data mainly depends on the number of approximations. Since in FEM we can make lesser number of assumptions, degree of accuracy will be high with a minimum percentage or error from the exact value.
 4. Faster and automatic calculations, which are repetitive in nature, simultaneous display of modification and part processing results.
 5. Accurate prediction with adequate details for identifying critical areas of interest like highly stressed regions.

1.4 FINITE ELEMENT METHOD

The given system or structure is divided into small elements of units interconnected at grid points. Each element is solved variable, inherently making use of classical procedures and analytical. This process of breaking down of continuum of finite discrete elements and solving for the variable (velocity, displacement, stress and loads etc) is called finite element analysis.

The two methods used for the analysis of a problem in FEM are:

- (a) **Force method:** Here the forces are unknown (minimum complimentary energy method)
- (b) **Displacement Method:** Here the displacements are unknown (minimum potential energy method)

The analysis by FEM involves the following steps.

1. Pre-processing: It is the act of preparation of data such as nodal locations, element connectivity co-ordinates imposing boundary conditions applications of load and providing material information to element etc. one can decide in this stage regarding the number of nodes, elements, types with their order and the pattern of FE mesh.

2. Processing: This stage involves stiffness generation, strain energy calculation and solutions of equations resulting in the evolution of nodal variables, induced elemental forces, strains and stresses.

3. Post-Processing: The geometric deformation and distribution of forces, stresses and strains in the structure, here the aspect of plotting a displaced plotting nodal data in the form of contour plots, such as isotherms and isobars, and conversion of element oriented data into best fitting nodal values is discussed.

Preprocessing and post processing are integral parts of finite element analysis. The general purpose mesh generation scheme can model a variety of complex regions.

The node numbering gives spars matrices and in many cases should given minimum bandwidth proper block representation.

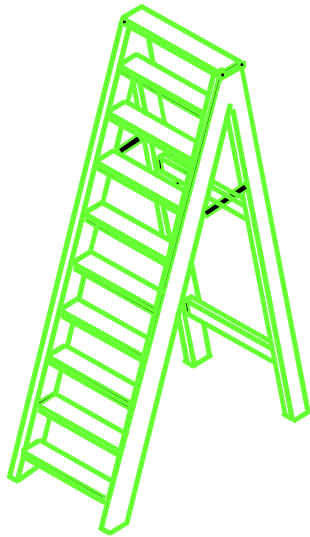
Mesh plotting shows the element layout.

1.5 WHAT IS FEA?

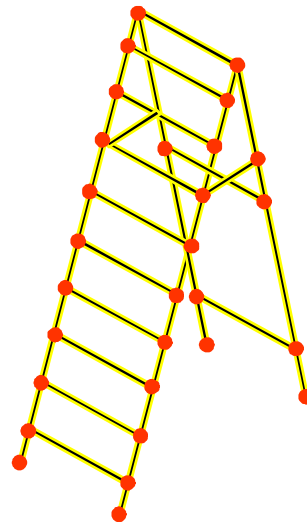
Finite Element Analysis is a way to simulate loading conditions on a design and determine the design's response to those conditions.

- ❖ The design is modeled using discrete building blocks called elements.
 - Each element has exact equations that describe how it responds to a certain load.

- The “sum” of the response of all elements in the model gives the total response of the design.
 - The elements have a finite number of unknowns, hence the name finite elements.
- ❖ The **finite element model**, which has a finite number of unknowns, can only approximate the response of the physical system, which has infinite unknowns. So the question arises: How good is the approximation? Unfortunately, there is no easy answer to this question. It depends entirely on what you are simulating and the tools you use for the simulation. We will, however, attempt to give you guidelines throughout this training course.



Physical System

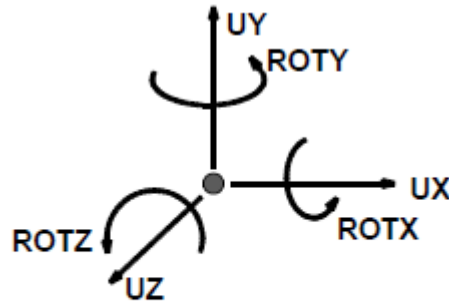


F.E. Model

1.6 DEGREES OF FREEDOM

Degrees of freedom (DOFs) characterize the response of a field.

| Discipline | DOF |
|------------|--------------------|
| Structural | Displacement |
| Thermal | Temperature |
| Electrical | Voltage |
| Fluid | Pressure |
| Magnetic | Magnetic Potential |



Structural DOFs

1.7 BASIC CONCEPTS OF FINITE ELEMENT METHOD

- Full field is divided into finite number of elements.
- Knowing the behavior of one/many elements one can predict the assemblage.
- It is to predict the behavior of individual elements under load.
- Minimum potential energy principle is used to evaluate element stiffness.
- Independent of the problem to be solved, stiffness of the element can be obtained.

1.8 TERMINOLOGY IN THE FINITE ELEMENT METHOD

The commonly used terminologies in FE analysis are as follows.

1. Discretization

The physical structure has to be represented by a mathematical model in such way that it simplify the construction and easy to analyze. In modeling the given global domain is divided into sub-domains called elements. This process of modeling is termed as discrimination, which includes proper selection of type and density of the elements, the location of the nodal points for applying loads/boundary conditions, to extract the results from the areas of interest.

2. Grids

The point in space used for defining the geometry and connecting the elements. These points are useful to specify the displacement, force and boundary conditions of the structure.

3. Displacement

Displacement defines the deformation or the movement caused by an external force. There are six displacements for an element in space. These are three translations vectors u ; v and w and are three rotational vectors with respect to x , y and z . co-ordinate system. These nodal variables are referred to as degrees of freedom.

4. Co-ordinate System

A co-ordinate system is the reference, which can specify the position of a point or an element in a space. The co-ordinate systems used are.

- Global co-ordinate system (Cartesian i.e. x . y . z .)
- Local co-ordinate system.
- Polar co-ordinate system(r . θ)
- Cylindrical co-ordinate system(r . θ . z)

-
-
- Spherical co-ordinate system(r, 0)
 - Can be Cartesian also (x, y, z)

5. Boundary Conditions/Constraints

- Resistance (reaction) offered at grid points due to the motion of the structure being constrained by the applied forces.
These reactions can be obtained with respect to the specified co-ordinate System.
- To prevent the rigid body motion there by reducing the continuous structure to discrete system(avoiding the phenomena of mechanism)
- These constraints can be zero (u, v, w, x, y, z = 0) or non-Zero and specified enforced displacement at grid points.

1.9 STEPS INVOLVED IN FEM

The basic steps involved in finite element method are as follows.

1. Modeling

The basic idea of finite element modeling is to discretize the region and expresses the displacement in terms of values at discrete points.

2. Discretization

Discretization means division of a given structure into finite number elements of better approximations. This forms the first step in the analysis of complicated structural system.

3. Formulation of elemental equation and convergence test

The properties of the elements are formulated and combined to obtain the solution for the entire body or structure. For e.g. in the displacement formulation widely adopted in FE analysis, simple functions known as shape functions are chosen to approximate the variation of the displacement at the nodes of the element.

Convergence test is done to know that how much the approximate solution converges to the exact value and the errors are calculated.

4. Determination of

- a. Elemental stiffness matrix $[K_e]$
- b. Elemental nodal displacement vector $[Q_e]$
- c. Elemental load vector $[F_e]$

5. Equilibrium equations are imposed so that each of the elements is in a state of Equilibrium

$$[K_e] [Q_e] = [F_e]$$

6. Assembly of

- a. Stiffness matrix of all the elements $[K]$
- b. Nodal displacements of all the elements $[Q]$
- c. Nodal forces of all elements $[f]$
- d. Overall stiffness matrix is formed by global addition of element stiffness matrices.

7. Assembled Equilibrium Equation

The equation of equilibrium for the entire structure of body is then obtained by combining the equilibrium equation of each element such that the continuity of displacement is ensured at each node where the elements are connected.

$$[K] [Q] = [F]$$

8. Application of boundary conditions to the overall equilibrium equation.

The necessary conditions are imposed and the equations of equilibrium are solved for the nodal displacements. There are two approaches of applying the boundary conditions.

a. Elimination technique

In this technique after imposing the boundary conditions, whichever displacement is Zero, the corresponding row and column of the overall equilibrium is eliminated.

b. Penalty approach

A second approach for handling boundary condition is the penalty approach. This approach easily implemented in a computer program and retains its simplicity even when considering general boundary conditions where a large value of C is added to the highest diagonal elements. On applying the boundary conditions, the overall equilibrium equation gets modified.

c. Solution of the modified overall equilibrium equation by gauss elimination method

The value of nodal displacements will be obtained in this step. Thus knowing of all the nodal displacements, strain of each element will be computed. From these values elemental stresses are obtained.

Computation of Strains:

For each element strain calculated is given by the overall equation

$$[\varepsilon] = \{b\} \{q\}$$

Where $[\varepsilon]$ = Strain Matrix

$\{b\}$ = Strain Displacement Matrix where be is unique for each element

$\{q\}$ = Elemental Nodal Displacement Vector.

Computation of Stress:

Knowing the strain, the stress is calculated as follows

i.e., $[\sigma] = \{D\} \{\varepsilon\}$ Where $[\sigma]$ = Stress Matrix

$\{D\}$ = Constitutive Matrix for plane stress

$\{\varepsilon\}$ = Strain Matrix

These are the main steps carried out in structural problems where the analysis is simple. It becomes complicated thereafter for each and every problem, which includes dynamic analysis where the forces and all the conditions vary with time. In thermal, it is transient analysis and in fluid flow situations, it is turbulent flow analysis. The finite element method was initially developed on a physical basis for the analysis of problems in structural mechanics; however, it was soon recognized that the method can be applied equally well to the solution of many other classes or problems.

1.10 MERITS OF FEM

The merits of finite element method are:

- The systematic generality of finite procedure makes it a powerful and versatile tool for a wide range of problems. Thus flexible, general-purpose computer programs can be developed and can be applied to various problems with little or no modification.
- FEM can be easily interpreted in physical terms and it has a strong mathematical base. Hence, finite element method can be easily applied to any problem with a proper

knowledge of the physical system under consideration and can be solved to a greater accuracy by the application of proper mathematical tool.

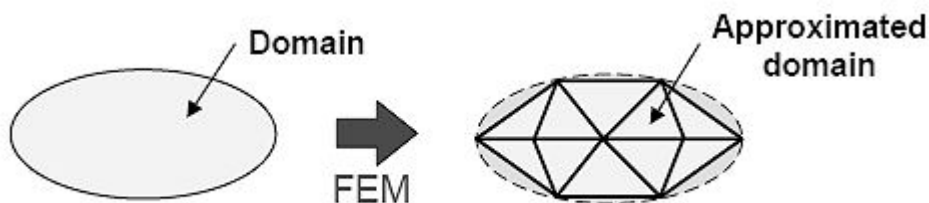
- Non-homogeneous continuum can also be dealt with by merely assigning different properties to different elements. It is even possible to vary the properties within an element according to the polynomial applied.
- Finite element method accommodates complex geometry with ease and is capable of handling non-linear and time dependent system also.
- In finite element method, since boundary conditions are introduced in the assembled equations, it requires only to specify the geometric boundary conditions without regarding its effects on interior elements. Since the boundary conditions do not enter into the individual finite element equations, the field variable models need not be changed, when the boundary conditions change.
- Finite element method considers the multidimensional continuity of the body. Hence it does not require separate interpolation process to extend the approximate solution to every point with in the continuum. Also it does not require the trial solutions that must all apply to the entire multidimensional continuum.

1.11 LIMITATIONS OF FEM

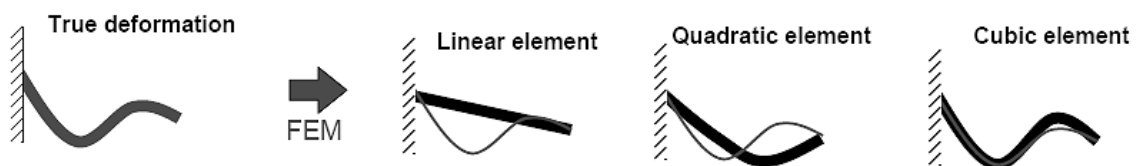
- FEM has been developed as a very high solution technique. However, the solution obtained from FEM can be realistic if and only if the material properties are known precisely.
- The major draw back of FEM is sensitivity of the solution on the geometry of the element such as type, shape and orientation of elements used.
- The compute programs yield a large amount of data as results it is very difficult to separate out the required results from the pile of numbers.

1.12 ERRORS INHERENT IN FEM FORMULATION

1. Geometry is simplified.



2. Field quantity is assumed to be a polynomial over an element. (Which is not true)



3. The computer carries only a finite number of digits.

-
-
4. The computer carries only a finite number of digits.
e.g: $\sqrt{2} = 1.41421356$ $\pi = 3.14159265$

Mistakes by Users

1. Elements are of the wrong type: e.g.) Shell elements are used where solid elements are needed
2. Distorted elements
3. Supports are insufficient to prevent all rigid-body motions
4. Inconsistent units (e.g. E=200 GPa, Force = 100 lbs)

Element Properties: The discretization of the structure or body into Finite Elements forms the basic first step in the analysis of a complicated structural system.

Rules For Discretization Of The Structure Into Elements: Sub-division of a body or structure into Finite Elements should satisfy the following requirements:

1. Two distinct elements can have common points only on their common boundaries if such boundaries exist. No overlapping is allowed. Common boundaries can be points, lines or surfaces.
2. The assembled element should leave no holes within the two elements and approximate the geometry of the real body or structure as closely as possible to do.

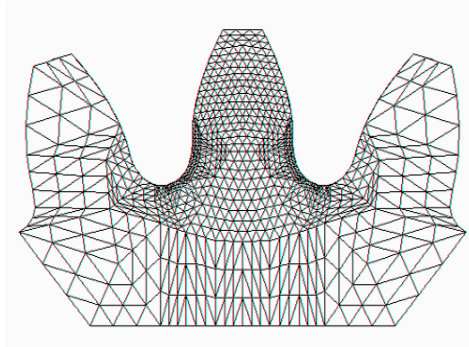
When the boundary of a structure or body cannot be exactly represented by the elements selected, an error cannot be avoided. Such error is called Geometric Discretization Error and it can be decreased by reducing the size of the elements or by using elements allowing boundaries to become curved.

Mesh Generators: The finite element models for practical analysis can contain tens of thousands or even hundreds of thousands degrees of freedom. It is not possible to create such meshes manually. Mesh generator is a software tool, which divides the solution domain into many Sub domains – finite elements. Mesh generators can be of different types.

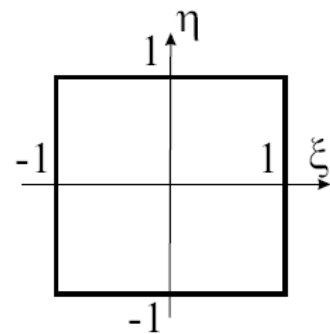
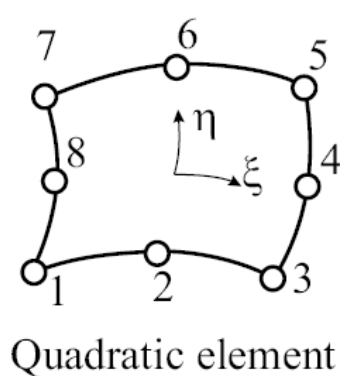
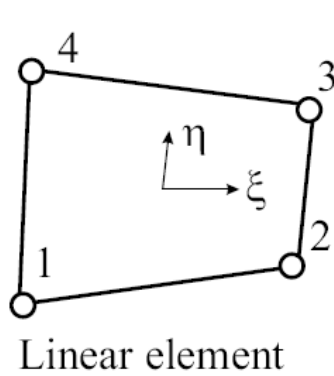
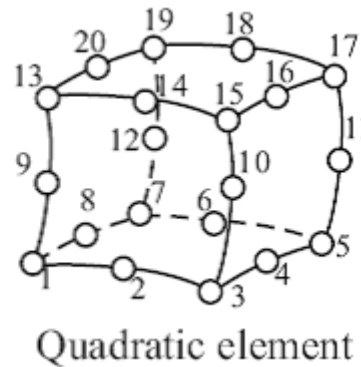
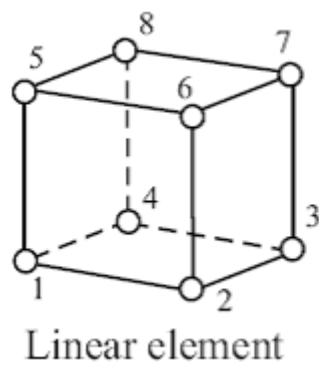
For two-dimensional problems, we want to mention two types: block mesh generators and triangulates.

Block Mesh Generators require some initial form of gross partitioning. The solution domain is partitioned in some relatively small number of blocks. Each block should have some standard form The mesh inside block is usually generated by mapping technique.

Triangulators typically generate irregular mesh inside arbitrary domains. Voronoi polygons and Delaunay triangulation are widely used to generate mesh. Later triangular mesh can be transformed to the mesh consisting of quadrilateral elements. Delaney triangulation can be generalized for three dimensional domains.



Finite Element Model of Gear tooth



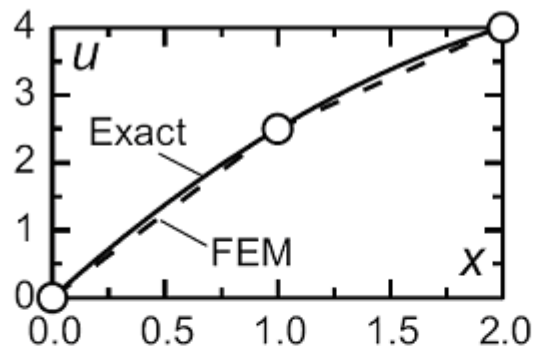
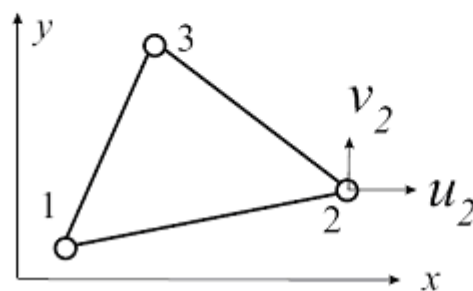
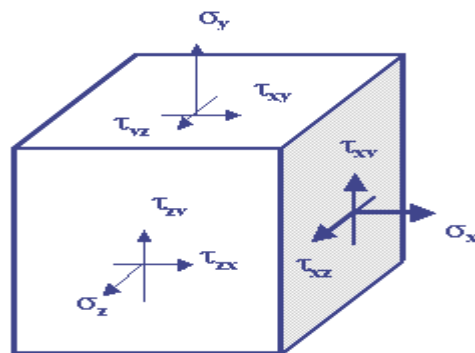


Fig. 1.2: Comparison of Finite Element Solution & Exact Solution



Triangular Finite Element is the Simplest Two - Dimensional Element



Three-dimensional stress on an element.

PICTORIAL SUMMARY OF ELEMENTS TYPES

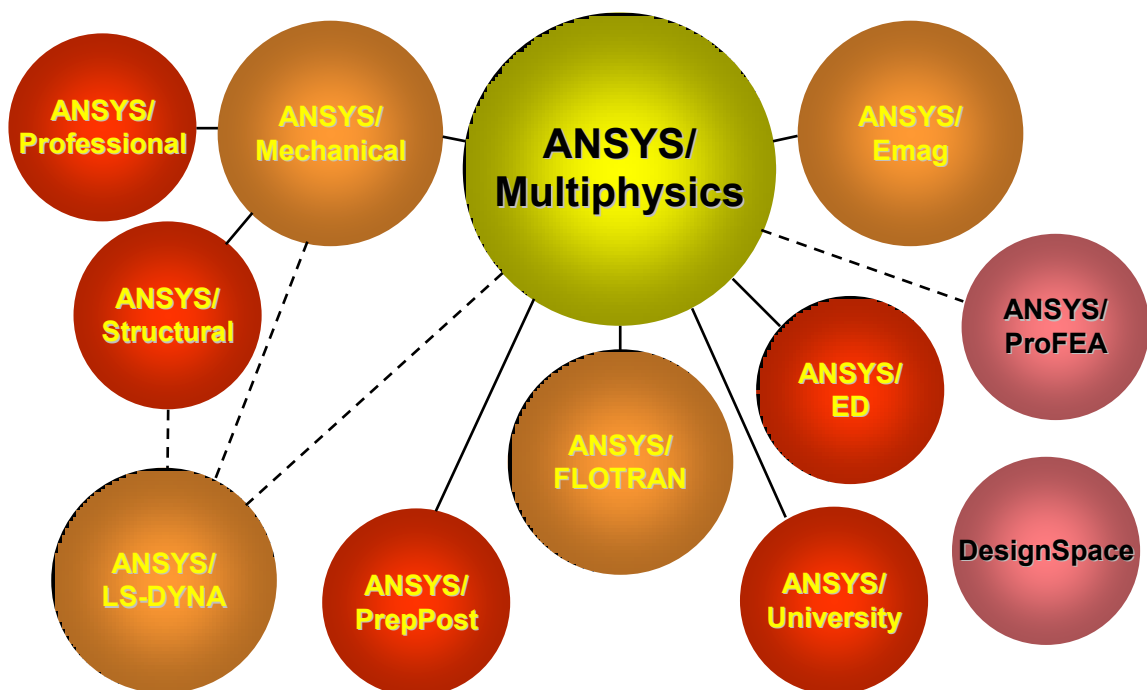
Refer ANSYS Help

COMMERICAL FEM SOFTWARE PACKAGES AVAILABLE

- ANSYS – (General Purpose, PC and workstations).
- SDRC/I-DEAS-- (Complete CAD/CAM/CAE Package).
- Nastran – (General purpose FEA on main frames)
- ABAQUS—(Non linear and dynamic analyses)
- ALGOR – (PC & Workstations).
- DYNA-3D – (Crash/Impact analysis).

INTRODUCTION TO ANSYS

- ❖ ANSYS is a complete FEA 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
 - Biomedical
 - Bridges & Buildings
- ❖ ANSYS/Multiphysics is the flagship ANSYS product which includes all capabilities in all engineering disciplines.



- ❖ There are three main component products derived from ANSYS/Multiphysics:
 - ANSYS/Mechanical - structural & thermal capabilities
 - ANSYS/Emag - electromagnetics
 - ANSYS/FLOTRAN - CFD capabilities
- ❖ Other product lines:
 - ANSYS/LS-DYNA - for highly nonlinear structural problems
 - DesignSpace - an easy-to-use design and analysis tool meant for quick analysis within the CAD environment
 - ANSYS/ProFEA - for ANSYS analysis & design optimization within Pro/ENGINEER

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

 - ❖ **Thermal analysis:** is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux. All three primary heat transfer modes can be simulated: conduction, convection, radiation.
 - Steady-State
 - Time-dependent effects are ignored.
 - Transient
 - To determine temperatures, etc. as a function of time.
 - Allows phase change (melting or freezing) to be simulated.

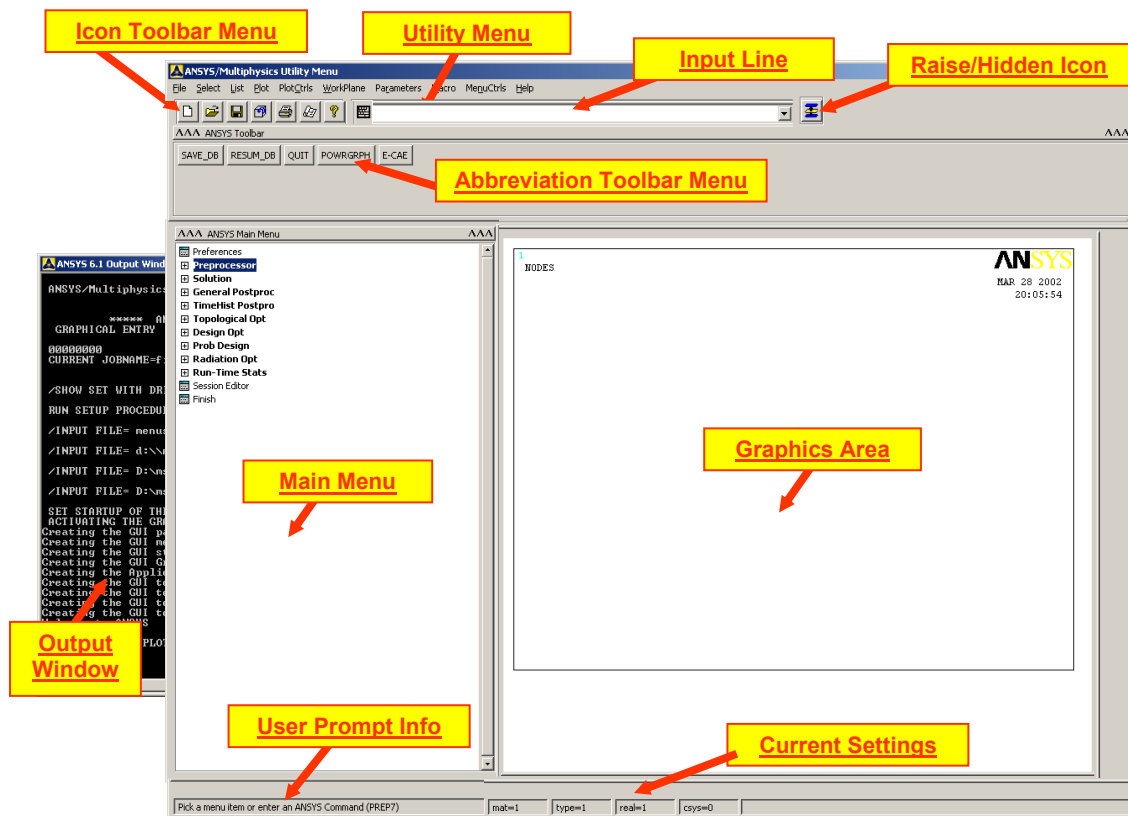
 - Electromagnetic analysis is used to calculate magnetic fields in electromagnetic devices.

 - Static and low-frequency electromagnetics
 - To simulate devices operating with DC power sources, low-frequency AC, or low-frequency transient signals.

 - Computational Fluid Dynamics (CFD)
 - To determine the flow distributions and temperatures in a fluid.
 - ANSYS/FLOTRAN can simulate laminar and turbulent flow, compressible and incompressible flow, and multiple species.
 - Applications: aerospace, electronic packaging, automotive design

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

2.2 THE GUI LAYOUT



Utility Menu

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

Toolbar Menu

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

Graphics Area

Displays graphics created in ANSYS or imported into ANSYS.

Input Line

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

Main Menu

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

Output

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. For example: You start ANSYS with a job name of "file". Then you resume my model.db, do some work, then save. That save is done to file.db! Avoid this issue by always resuming using the icon on the toolbar. If you open

mymodel.db using this method, it resumes the model and automatically changes the job name to my model.

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. If you want to plot more than one kind of entity use, “Plot → Multiplot”, which by default will plot everything in your model at once.

Plot Controls:

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

Creating Geometry:

Geometry in ANSYS is created from “Main Menu → Preprocessor → Modeling → Create” and has the following terminology,

KEYPOINTS: These are points, locations in 3D space.

LINES: This includes straight lines, curves, circles, spline curves, etc. Lines are typically defined using existing keypoints.

AREAS: This is a surface. When you create an area, it’s associated lines and keypoints are automatically created to border it.

VOLUMES: This is a solid. When you create a volume, it’s associated areas, lines and keypoints are automatically created.

SOLID MODEL: In most packages this would refer to the volumes only, but in ANSYS this refers to your geometry. Any geometry. A line is considered a “solid model”.

You can’t delete a child entity without deleting its parent, in other words you can’t delete a line if it’s part of an area, can’t delete a key point if it’s the end point of a line, etc.

Boolean Operations:

Top Down style modeling can be a very convenient way to work. Instead of first creating keypoints, then lines from those keypoints, then areas from the lines

and so on (bottom up modeling), start with volumes of basic shapes and use Boolean operations to add them, subtract them, divide them etc. Even if you are creating a shell model, for example a box, you could create the box as a volume (a single command) and then delete the volume keeping the existing areas, lines and keypoints.

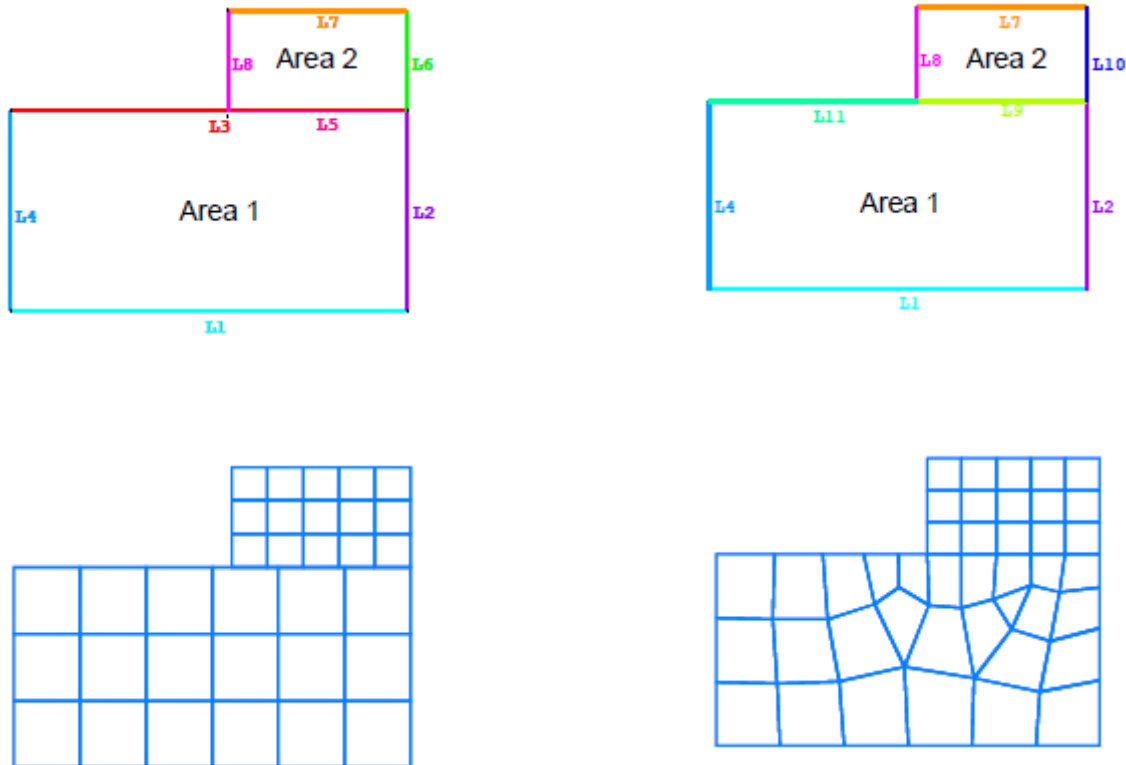
These kinds of operations are found under “Main Menu → Preprocessor → Modeling → Operate → Booleans” with some common ones being:

Add: Take two entities that overlap (or are at least touching) and make them one.

Subtract: Subtract one entity from another. To make a hole in a plate, create the plate (area

of volume) then create a circular area or cylinder and subtract it from the plate.

Glue: Take two entities that are touching and make them contiguous or congruent so that when meshed they will share common nodes. For example, using default mesh parameters,



Meshing without gluing areas.

Meshing after gluing areas.

Note: In case of Meshing after gluing areas. The coincident nodes on the common line between the two areas will be automatically merged. You don't have to manually equivalence them like in some other codes.

The Working Plane:

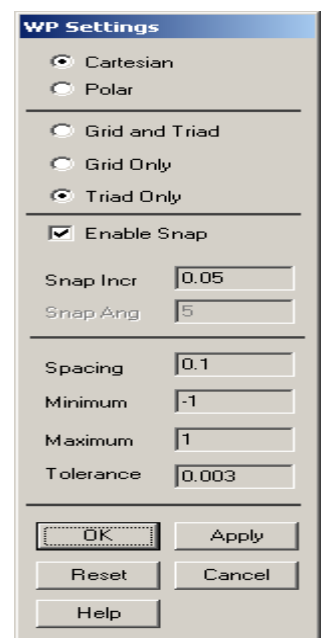
All geometry is created with respect to the working plane, which by default is aligned with the global Cartesian coordinate system. The "Working Plane" is actually the XY plane of the working coordinate system. The working coordinate system ID is coordinate system 4 in ANSYS. Global Cartesian is ID 0, Global Cylindrical is ID 1, and Global Spherical is ID 2.

Working Plane Hints:

Turn on the working plane so you can see it with, "Utility Menu → Work Plane → Display Working Plane".

Change the way the working plane looks or adjust the snap settings under "Utility Menu → Work Plane → WP Settings...".

Move the working plane around using



“Utility Menu → Work Plane → Offset WP to...”.

Align the working plane with various parts of the model using

“Utility Menu → Work Plane → Align WP with...”.

If you select more than one node or keypoint to offset the working plane to, it will go to the average location of the selected entities. VERY handy!

Use the working plane to slice and dice your model. For example to cut an area in pieces use

“Main Menu → Modeling → Operate → Booleans → Divide → Area by WrkPlane”. Do this for lines and volumes as well.

Select Logic:

Selecting is an important and fundamental concept in ANSYS. Selected entities are your active entities. All operations (including Solving) are performed on the selected set. In many operations you select items “on the fly”; ANSYS prompts for what volumes to mesh for example, you pick them with the mouse, and ANSYS does the meshing.

However there are many times when you need to select things in more sophisticated ways. Also, in an ANSYS input file or batch file you can’t select things with the mouse!

Examples where this would be useful:

- You have many different areas at $Z = 0$ you want to constrain. You could select them all one by one when applying the constraint, or select “By Location” beforehand, then say “Pick All” in the picking dialog.
- You have a structure with many fastener holes that you want to constrain. Again, you could select them all one by one when applying the constraint, or select lines “By Length/Radius”, type in the radius of the holes to select all of them in one shot, then “Pick All” in the picking dialog when applying the constraint.

After working with the selected set,

“Utility Menu → Select → Everything” to make the whole model active again.

Select Entities Dialog Box Terminology:

From Full: Select from the entire set of entities in the model.

Reselect: Select a subset from the currently selected entities.

Also Select: Select in addition to (from the whole model) the set you have currently selected.

Unselect: Remove items from the selection set.

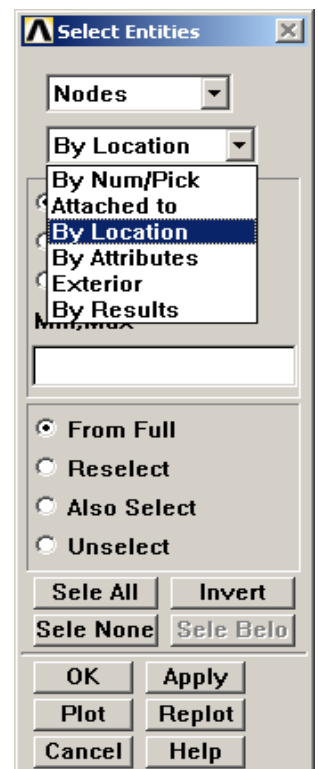
Select All: This is not the same as “Utility Menu → Select → Everything”. This selects all of whatever entity you have specified at the top of the dialog.

Invert: Reverses the selected and unselected entities (just the entities specified at the top of the dialog).

OK: This does the select operation (or brings up a picker dialog so that you can pick with the mouse) and then dismisses the dialog.

Apply: This does the operation but keeps the dialog box. Typically use this so the dialog stays active.

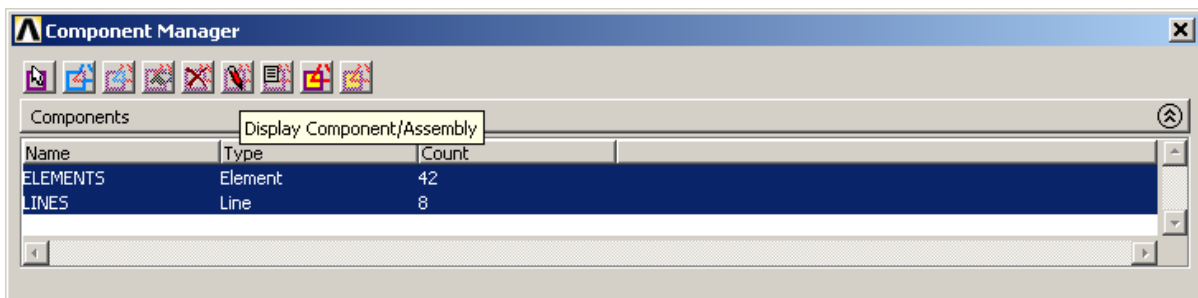
Replot: Replots whatever is active in the graphics window.



Plot: Plots only the entity specified at the top of the dialog.

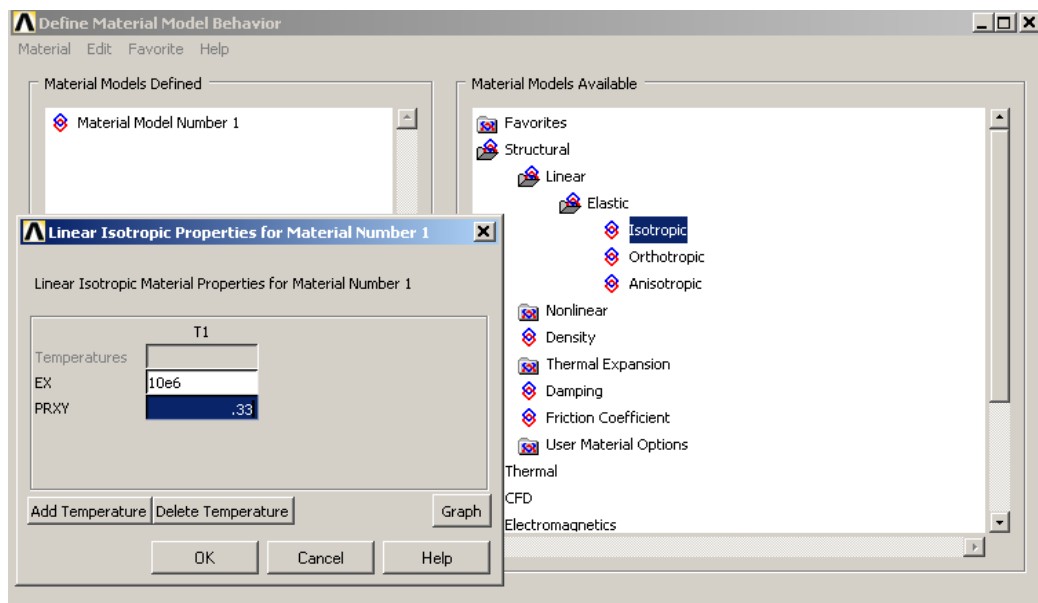
Organizing Your Model Using Components:

If you select a group of entities and think that you might want to use that selection set again, create a component out of it. Components are groups of entities but hold only one kind of entity at a time. Components can themselves be grouped into Assemblies, so this is how you group different types of entities together. Use “Utility Menu → Select → Comp/Assembly → Create Component...” to create a component. The new Component Manager in Release 8.0 makes it very easy to manage and manipulate groups and select/plot what you want to see to the screen. This is found under “Utility Menu → Select → Component Manager”



Creating a Material:

Create the material properties for your model in “Main Menu → Preprocessor → Material Props → Material Models”. This gives you this dialog box where all materials can be created,

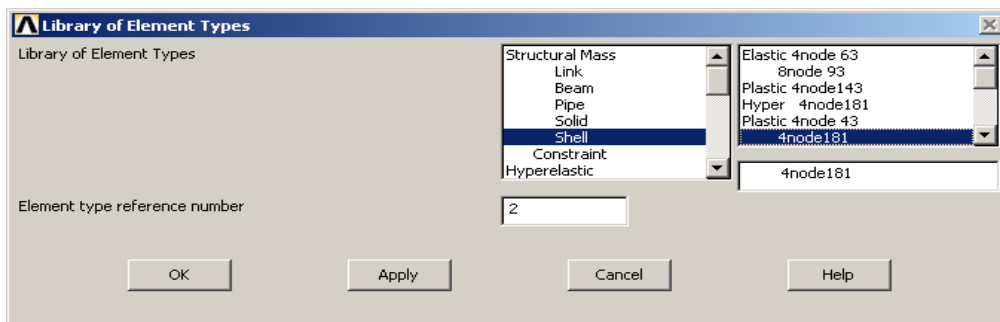


Double click on items in the right hand pane of this window to get to the type of material model you want to create. All properties can be temperature dependant. Click OK to create the material and it will appear in the left hand pane. Create as many different materials as you need for your analysis.

Selecting an Element Type:

ANSYS has a large library of element types. Why so many? Elements are organized into groups of similar characteristics. These group names make up the first part of the element name (BEAM, SOLID, SHELL, etc). The second part of the element name is a number that is more or less (but not exactly) chronological. As elements have been created over the past 30 years the element numbers have simply been incremented. The earliest and simplest elements have the lowest numbers (LINK1, BEAM3, etc), the more recently developed ones have higher numbers. The “18x” series of elements (SHELL181, SOLID187, etc) are the newest and most modern in the ANSYS element library.

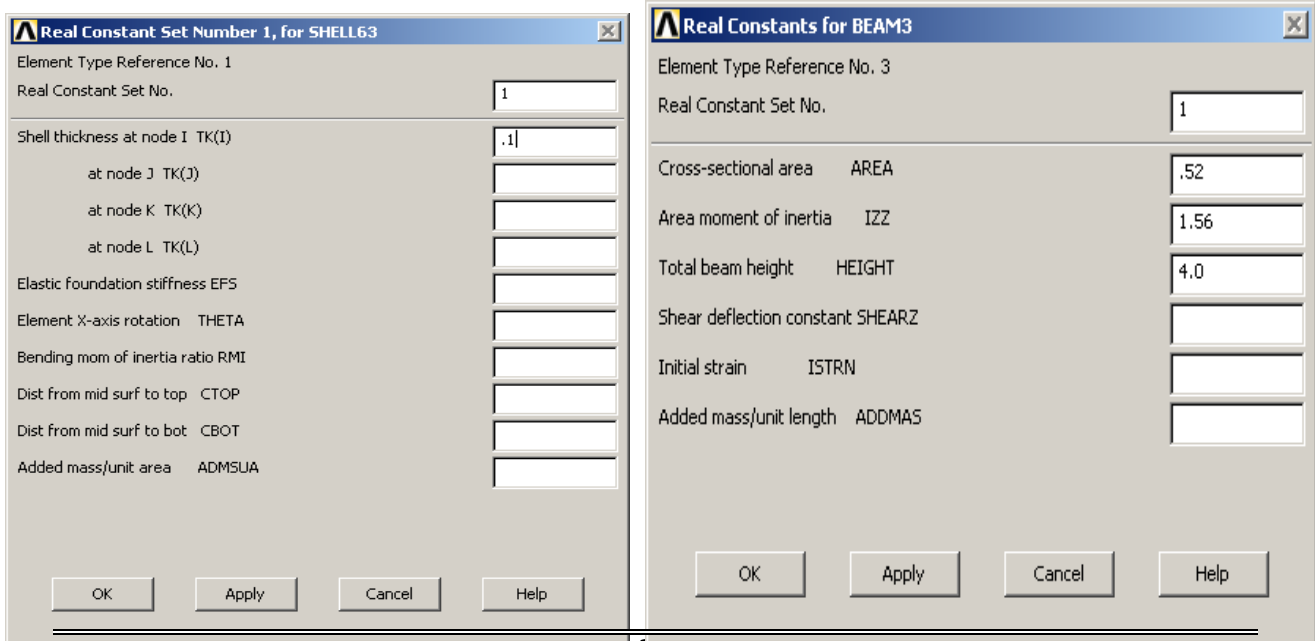
Tell ANSYS what elements you are going to use in your model using “Main Menu → Element Type → Add/Edit/Delete”



Later, when meshing or creating elements manually you will need to tell ANSYS what type of elements you want to create.

Creating Properties

A solid element (brick or tet) knows its thickness, length, volume, etc by virtue of its geometry, since it is defined in 3D space. Shell, beam and link (truss) elements do not know this information since they are a geometric idealization or engineering abstraction. Properties in ANSYS are called Real Constants. Define real constants using “Main Menu → Real Constants → Add/Edit/Delete”.



Creating the Finite Elements Model - Meshing:

If you are just starting out in FEA, it is important to realize that your geometry (called the solid model in ANSYS) is not your finite element model. In the finite element method we take an arbitrarily complex domain, impossible to describe fully with a classical equation, and break it down into small pieces that we can describe with an equation. These small pieces are called finite elements. We essentially sum up the response of all these little pieces into the response of our entire structure. The solver works with the elements. The geometry we create is simply a vehicle used to tell ANSYS where we want our nodes and elements to go. While you can create nodes and elements one by one in a manual fashion (called direct generation in ANSYS) most people mesh geometry because it is much another very good reason we mesh geometry is that we assign materials and properties to that geometry.

Then any element created on or in that geometric entity gets those attributes. If we don't like the mesh we can clear it and re-mesh, without having to re-assign the attributes.

Steps for Creating the Finite Elements:

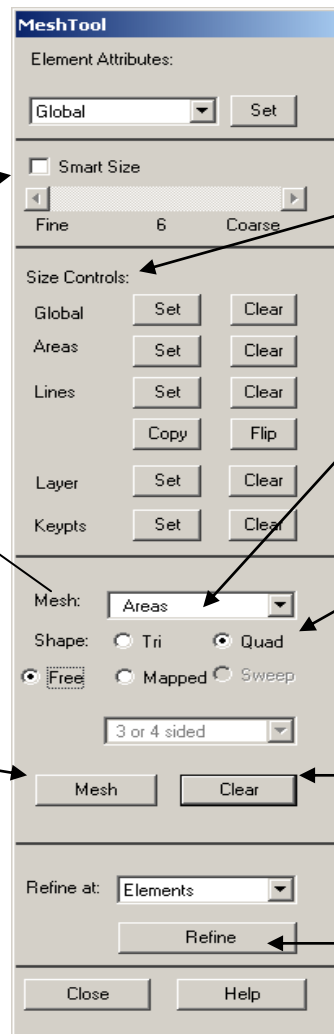
- Assign Attributes to Geometry (materials, real constants, etc).
- Specify Mesh Controls on the Geometry (element sizes you wa
- Mesh.

Most of the meshing operations can be done within the MeshTool, so that will be examined in some detail now. Start it from “Main Menu → Preprocessor → Meshing → MeshTool”.

Smart Size is used to automatically pick the element edge length depending on the sizes of features in the geometry. It makes a finer mesh around smaller Features in order to capture them adequately.

For example in the mesh below no sizes were specified at all except a Smart Size level of 4.

A mapped mesh generates a very regular grid of elements. This can only be used on rectangular shaped areas or volumes. A free mesh will mesh any entity regardless of shape.



If you want to set a specific element edge length on an entity or tell ANSYS to put some specific number of elements along a line for example, use this.

Pick what you want to mesh.

Shape of the element you want to create. For Volumes you would have the choice of Hex or Tet. You can create beam elements by meshing lines.

Brings up a picker dialog. You pick the entities to be cleared, press OK, and ANSYS removes the nodes and elements from that geometry.

Brings up a picker dialog. Pick the entities to be meshed (or Pick All if you have made some selection using select logic), press OK, and ANSYS generates the nodes and elements on/in that geometry.

Refine the mesh (make more nodes and elements locally) at a specific location (elements, nodes, keypoints, lines, etc.).

Applying Loads and Boundary Conditions:

Loads and boundary conditions can be applied in both the Preprocessor (“Main Menu → Preprocessor → Loads → Define Loads → Apply”), and the Solution processor (“Main Menu → Solution → Define Loads → Apply”).

1. Select the kind of constraint you want to apply.
2. Select the geometric entity where you want it applied.
3. Enter the value and direction for it.

There is no “modify” command for loads and B.C.’s. If you make a mistake simply apply it again with a new value (the old one will be replaced if it’s on the same entity), or delete it and reapply it.

Loads: Forces, pressures, moments, heat flows, heat fluxes, etc.

Constraints: Fixities, enforced displacements, symmetry and anti-symmetry conditions, temperatures, convections, etc.

Although you can apply loads and boundary conditions to nodes or elements, it’s generally better to apply all B.C.’s to your geometry. When the solve command is issued, they will be automatically transferred to the underlying nodes and elements. If B.C.’s are put on the geometry, you can re-mesh that geometry without having to reapply them

Solving:

Solution is the term given to the actual simultaneous equation solving of the mathematical model. The details of how this is done internally is beyond the scope of this guideline but is addressed in a separate “ANSYS Tips” white paper. For the moment, it is sufficient to say that the basic equation of the finite element method that we are solving is, $[K]\{u\}=\{F\}$

where $[K]$ is the assembled stiffness matrix of the structure,

$\{u\}$ is the vector of displacements at each node, and

$\{F\}$ is the applied load vector. This is analogous to a simple spring and is the essence of small deflection theory.

To submit your model to ANSYS for solving, go to “**Main Menu** → **Solution** → **Solve** → **Current LS**”. **LS** stands for load step. A load step is a loading “condition”.

This is a single set of defined loads and boundary conditions (And their associated solution results. More on this in the next section). Within an interactive session the first solve you do is load step 1, the next solution is load step 2, etc.

If you leave the solution processor after solving to do post-processing for example, the load step counter gets set back to one. You can also define and solve multiple load steps all at once.

There are several solvers in ANSYS that differ in the way that the system of equations is solved for the unknown displacements. The two main solvers are the sparse solver and the PCG solver. If the choice of solvers is left to “program chosen” then generally ANSYS will use the sparse solver. The PCG (preconditioned conjugate gradient) solver works well for models using all solid elements. From a practical perspective one thing to consider is that the sparse solver doesn’t require a lot of RAM but swaps out to the disk a lot. Disk I/O is very slow. If you have a solid model and lots of RAM the PCG solver could be significantly faster since the solution runs mostly in core memory

Postprocessing:

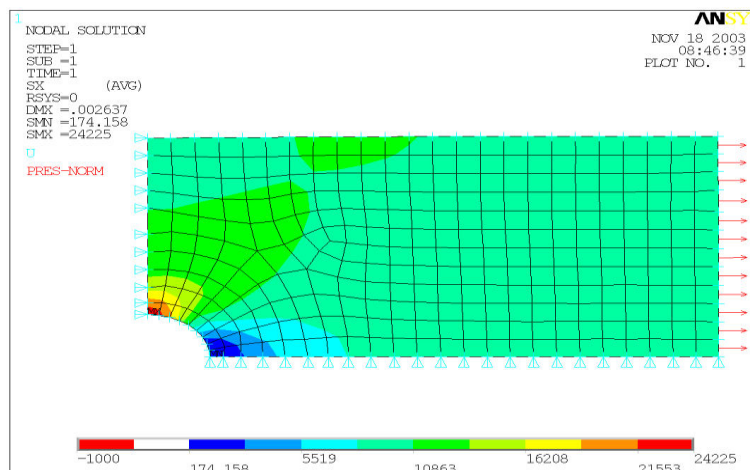
The General Postprocessor is used to look at the results over the whole model at one point in time. This is the final objective of everything we have discussed so far; finding the stresses, deflections, temperature distributions, pressures, etc. These results can then be compared to some criteria to make an objective evaluation of the performance of your design. The solution results will be stored in the results file as result “sets”. For a linear static analysis like we are talking about, the correlation between Load Step numbers and Results Set numbers will be one to one as shown below. Only one set of results can be stored in the

database at a time, so when you want to look at a particular set, you have to read it in from the results file. Reading it in clears the previous results set from active memory.

The screenshot shows a window titled "SET,LIST Command" with a "File" menu. The main content is a table titled "***** INDEX OF DATA SETS ON RESULTS FILE *****".

| SET | TIME/FREQ | LOAD STEP | SUBSTEP | CUMULATIVE |
|-----|-----------|-----------|---------|------------|
| 1 | 1.0000 | 1 | 1 | 1 |
| 2 | 2.0000 | 2 | 1 | 2 |
| 3 | 3.0000 | 3 | 1 | 3 |
| 4 | 4.0000 | 4 | 1 | 4 |
| 5 | 5.0000 | 5 | 1 | 5 |
| 6 | 6.0000 | 6 | 1 | 6 |
| 7 | 7.0000 | 7 | 1 | 7 |
| 8 | 8.0000 | 8 | 1 | 8 |
| 9 | 9.0000 | 9 | 1 | 9 |
| 10 | 10.000 | 10 | 1 | 10 |
| 11 | 11.000 | 11 | 1 | 11 |
| 12 | 12.000 | 12 | 1 | 12 |

To read in a results set from the results file (not needed if you have run only a single load step) use “Main Menu → General Postproc → Read Results → First Set, or By Pick”. Most results are displayed as a contour plot as shown below. To generate a plot of stresses use “Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution”, then pick the stresses you want to see



- There are many, many other ways to look at your results data including:
- Listing them to a file.
 - Querying with the mouse to find a result at a particular node.
 - Graphing results along a path.
 - Combining different load cases.
 - Summing forces at a point.
 - Extracting data and storing it an APDL array that you can do further operations with.

Animate any result on the deformed shape with “Utility Menu → PlotCtrls → Animate”. This is very helpful for understanding if your model is behaving in a reasonable way.

Chapter 3

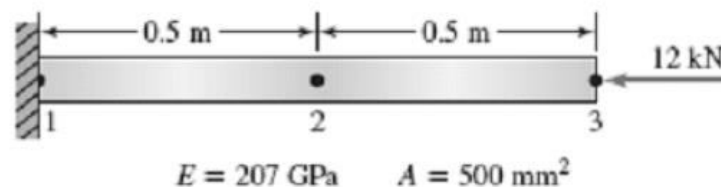
ANSYS EXERCISE (PART – A)

PART A.1:STRESS ANALYSIS OF BARS OF CONSTANT CROSS SECTION AREA

Problem 1

A steel rod subjected to compression is modeled by two bar elements, as shown in Figure. Determine the nodal displacements and the axial stress in each Element.

$E=207 \text{ GPa}$, $A = 500 \text{ mm}^2$



Procedure:

1. **Utility Menu > File > Change Jobname >** Enter 'Bar Analysis'
2. **Utility Menu > File > Change Title >** Enter Bar Analysis.
3. **Preprocessor> Element type>Add/Edit/Delete>Add> Link1**
4. **Preprocessor> Real constant> Add/Edit/Delete>Add>Ok> Area>500**
5. **Preprocessor>Material Property> Young's Modulus > Enter Ex = 2e5.**
6. **Preprocessor > Modeling > Create>node > In active CS > Create three nodes at Location (0, 0, 0) (500, 0, 0) and (1000, 0, 0).**
7. **Preprocessor > Modeling > Create > Element > Auto numbered > Thru node >** (Pick nodes (1, 2), (2, 3) and create element).
8. **Preprocessor > Load > Define load > Apply > Structural > Displacement> Select** Node number 1 and apply 'ALL DOF'.
9. **Preprocessor > Load > Define load > Apply> Structural > Force / moment on nodes >** Select node number 3 and Enter value $F_x = -12000 \text{ N}$.
10. **Solution > Solve > Current LS**
11. **General postprocessor > Plot Results > Deform shape**
12. **Postprocessor> Contour plot> Nodal solution> Displacement vector sum.**
13. **Postprocessor>Element Table>Define Table>Add>By sequence num>LS 1**
14. **Postprocessor>Element Table>List Elem table>LS 1>ok**

Comparison between theoretical and Ansys values

| | Ansys | Theoretical |
|-------------|-------|-------------|
| Deformation | | |
| Stress | | |
| Reaction | | |

Problem 2:

For the simple bar shown in the figure determine the displacements, stress and the reaction. The cross section of the bar is 500 mm^2 , length is 1000mm and the Young's modulus is $E= 2 \times 10^5 \text{ N/mm}^2$. Take load = 1000N



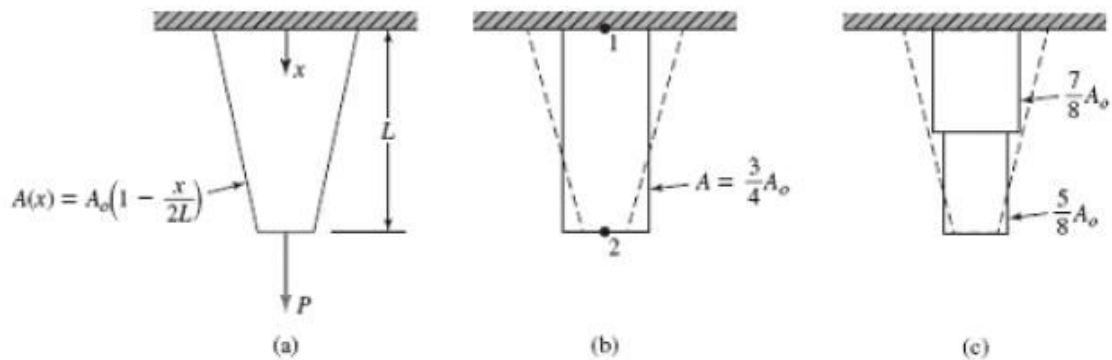
Comparison between Ansys and Theoretical Results

| | Ansys | Theoretical |
|-------------|-------|-------------|
| Deformation | | |
| Stress | | |
| Reaction | | |

PART A. 2: STRESS ANALYSIS OF BARS OF TAPERED CROSS SECTION AREA

Problem 3

Obtain nodal displacements for the tapered bar. Use the following numerical values: $E = 10 \times 10^6 \text{ lb/in}^2$, $A_0 = 4 \text{ in}^2$, $L = 20 \text{ in}$, $P = 4000 \text{ lb}$.



Procedure:

1. **Utility Menu > File > Change Jobname**
2. **Utility Menu > File > Change Title**
3. **Preprocessor > Element type > Add/Edit/Delete > Add > Link1.**
4. **Preprocessor > Real constant > Add/Edit/Delete > Add > Ok > Area >**
(create 2 set of Real Constant) > $A_1 = 3.5 \text{ in}^2$ and $A_2 = 2.5 \text{ in}^2$.
5. **Preprocessor > Material Property > Young's Modulus > Enter $E_x = 10 \times 10^6$.**
6. **Preprocessor > modeling > Create > Node > In active CS > Create 3 Nodes >**
(0, 0, 0),
(0, -10, 0) and (0, -20, 0).
7. **Preprocessor > Modeling > Create > Element > Element Attributes**
(you can set Real Constant number set to 1).
8. **Preprocessor > Modeling > Create > Element > Auto numbered > Thru Node >**
(Pick nodes and create element between nodes 1 and 2)
9. Repeat the procedures given in 7 and 8 set Real constant number to 2 and create element between nodes 2 and 3.
10. **Preprocessor > Load > Define load > Apply > Structural > Displacement >** on nodes and
select node 1 and **Apply 'ALL DOF'** in the wizard.
11. **Preprocessor > Load > Define load > Apply > Structural > Force / moments > on Nodes >**
Select node number '3' and **Apply a force $F_y = -4000$.**
12. **Solution > Solve > Current LS.**
13. **General postprocessor > Plot Result > Deform shape**
14. **Postprocessor > Contour plot > Nodal solution > Displacement vector sum.**

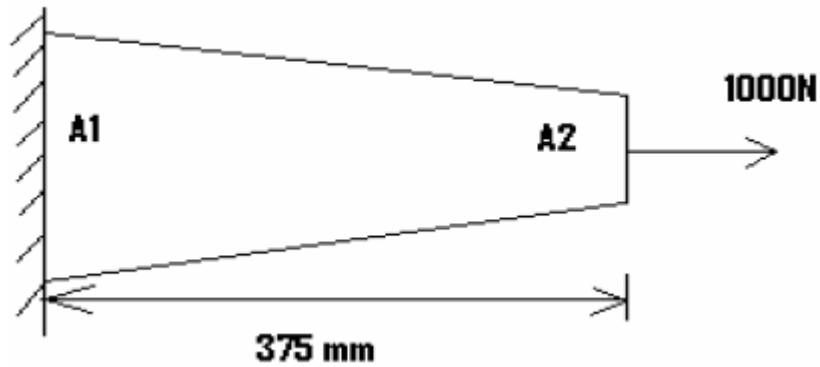
Comparison between theoretical and Ansys values

| | Ansys | Theoretical |
|-------------|-------|-------------|
| Deformation | | |
| Stress | | |
| Reaction | | |

Problem 4

For the tapered bar shown in the figure determine the displacement, stress and reaction in the bar.

Given $A_1 = 1000 \text{ mm}^2$ and $A_2 = 500 \text{ mm}^2$, $E = 2 \times 10^5 \text{ N/mm}^2$

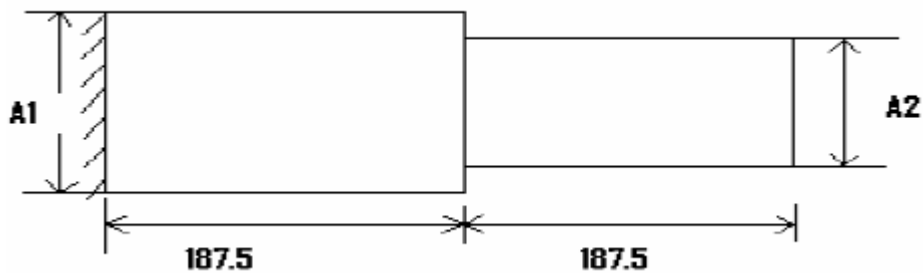


The Tapered bar is modified into 2 elements as shown below with modified area of cross section

$$(1000+500)/2 = 750 \text{ mm}^2$$

$$A_1 = (1000+750)/2 = 875 \text{ mm}^2$$

$$A_2 = (750+500)/2 = 625 \text{ mm}^2$$



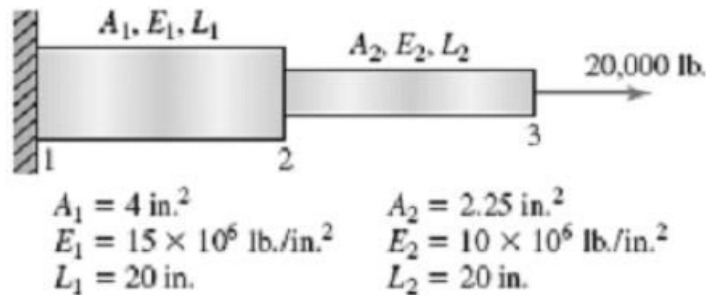
Comparison between Ansys and Theoretical Results

| | Ansys | Theoretical |
|-------------|-------|-------------|
| Deformation | | |
| Stress | | |
| Reaction | | |

PART A.3: STRESS ANALYSIS OF A STEPPED BAR

Problem 5

Determine the nodal displacements and Reaction forces for the Stepped bar shown below.



Procedure:

1. **Utility Menu > File > Change Job name**
2. **Utility Menu > File > Change Title**
3. **Preprocessor > Element type > Add/Edit/Delete > Add > Link1.**
4. **Preprocessor > Real constant > Add/Edit/Delete > Add > Ok > Area >**
(create 2 set of real constants).
5. **Preprocessor > Material Property > Material Models > Young's Modulus > Enter**
Ex = 15e6 Under **Material Models > Material > New Model** (define the second
property) Enter **Ex = 10e6**.
6. **Preprocessor > Modeling > Create > Node > In active CS > Create 3 Nodes**
(0, 0, 0), (20, 0, 0) and (40, 0, 0).
7. **Preprocessor > Modeling > Create > Element > Element Attributes**
(you can see 2 real Constant sets and 2 material properties select the real constant
number 1 and create element).
8. **Preprocessor > Modeling > Create > Element > Auto numbered > Thru Node >**
(pick nodes 1 and 2 and create element).
9. Repeat the procedure given in 7 and 8 by choosing material number 2 and real
Constant number 2 and create element between nodes 2 and 3.
10. **Preprocessor > Load > Define load > Apply > Structural > Displacement > on**
Nodes > Pick node number 1 and apply 'All Dof'.
11. **Preprocessor > Load > Define load > Apply > Structural > Force > on Nodes >**
Apply
Fx = 2000.
12. **Solution > Solve > Current LS**
13. **General postprocessor > Plot Result > Deform shape.**
14. **Postprocessor > Contour plot > Nodal solution > Displacement vector sum.**
15. **General Post processor > List Result > Reaction solu > All structural force 'F'.**

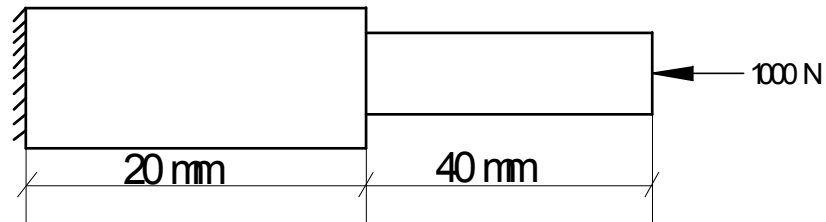
Comparison between theoretical and Ansys values

| | Ansys | Theoretical |
|-------------|-------|-------------|
| Deformation | | |
| Stress | | |
| Reaction | | |

Problem 6:

A Stepped bar is subjected to axial load as shown in the figure find the Stress Reactions for the following following problem using Link 1 Element.

Given Data: $A_1 = 40 \text{ mm}^2$, $A_2 = 20 \text{ mm}^2$

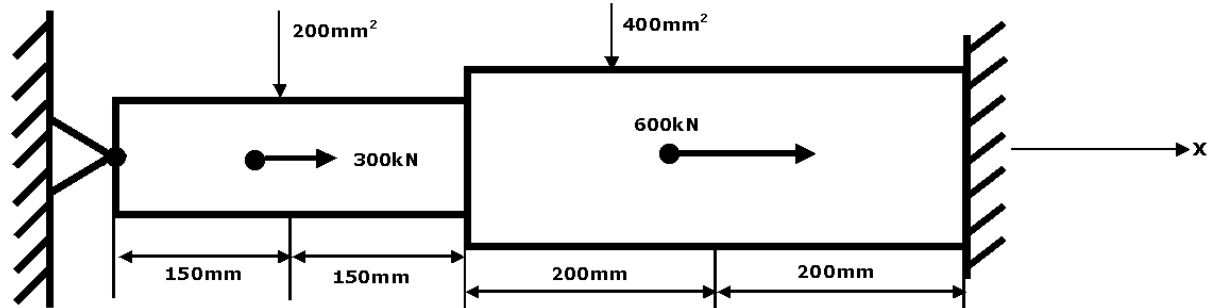
**Comparison between Ansys and Theoretical Results**

| | Ansys | Theoretical |
|--------------------|--------------|--------------------|
| Deformation | | |
| Stress | | |
| Reaction | | |

Problem 7:

Consider the bar in figure. Determine the nodal displacements, element stresses & support reactions. The Temperature is then raised to 60°C $E = 200 \times 10^9 \text{ N/m}^2$

Solution:



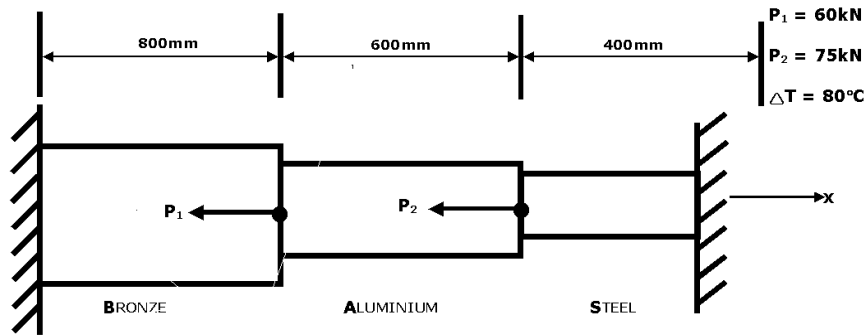
| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

When temperature raised to 60°C.

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

Problem 8:

- a. Determine the displacements, stresses & Support reactions
- b. In the structure is subjected to an increase in Temperature of 80°C.
Determine the Displacements, Support Reactions & Stresses



| Bronze | Aluminium | Steel |
|------------------------------------|---|---|
| A = 2400mm ² | A ₁ = 1200mm ² | A ₂ = 600mm ² |
| E = 83 Gpa | E ₁ = 70 Gpa | E ₂ = 200 Gpa |
| α = 18.9 × 10 ⁻⁶ per °C | α ₁ = 23 × 10 ⁻⁶ per °C | α ₂ = 11.7 × 10 ⁻⁶ per °C |

Solution:

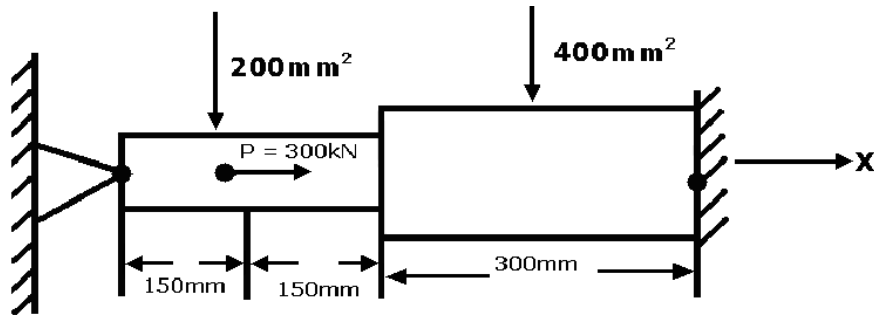
| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm ² | |
|---------|------------|------------|-------|---------------------|-------|--------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

When temperature raised to 80°C.

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm ² | |
|---------|------------|------------|-------|---------------------|-------|--------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

Problem 9:

An Axial load $P = 300 \times 10^3 \text{ N}$ is applied at 20°C to the rod as shown in figure. Determine the nodal Displacements, Reaction & Stresses.



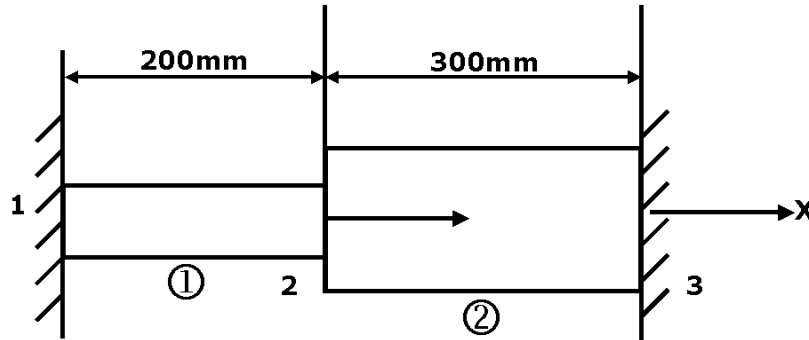
Assume $E = 200 \times 10^9 \text{ N/mm}^2$

Solution:

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

Problem 10:

- a. An axial load of $P = 300 \times 10^3 \text{ N}$ is applied at 20°C to the rod as shown in Figure. Determine the nodal Displacements, reactions & Stresses
 Figure. Determine the nodal Displacements, reactions & Stresses
 b. In the temperature is raised to 60°C . Determine the Displacements, Support Reactions & Stresses



| Aluminium | Steel |
|--|--|
| $A_1 = 900\text{mm}^2$ | $A_2 = 1200\text{mm}^2$ |
| $E_1 = 70 \times 10^9 \text{ N/m}^2$ | $E_2 = 200 \times 10^9 \text{ N/m}^2$ |
| $\alpha_1 = 23 \times 10^{-8} \text{ per}^\circ\text{C}$ | $\alpha_2 = 11.7 \times 10^{-8} \text{ per}^\circ\text{C}$ |

Solution:

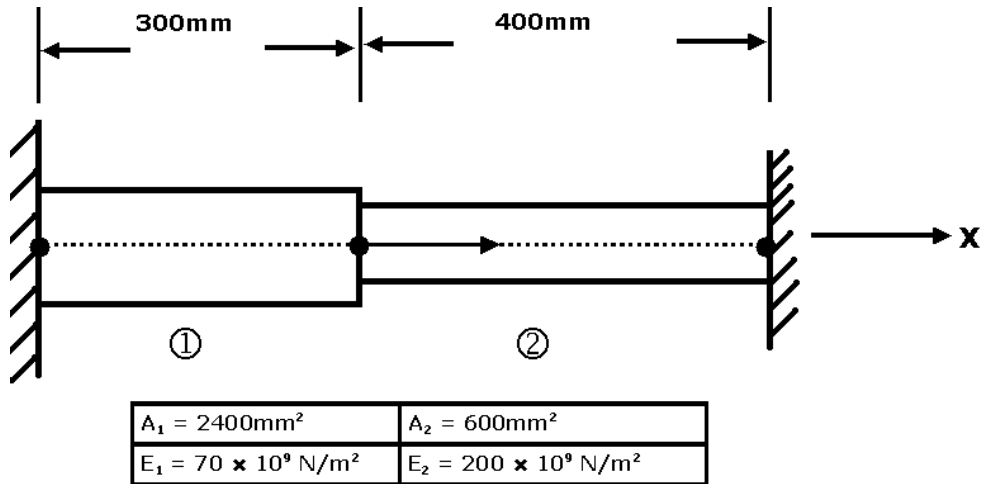
| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

In the temperature is raised to 60°C .

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

Problem 11:

Consider the bar shown in figure. An axial load $p = 200 \times 10^9 \text{ N}$ is applied as shown. Determine the nodal displacements and the reaction forces

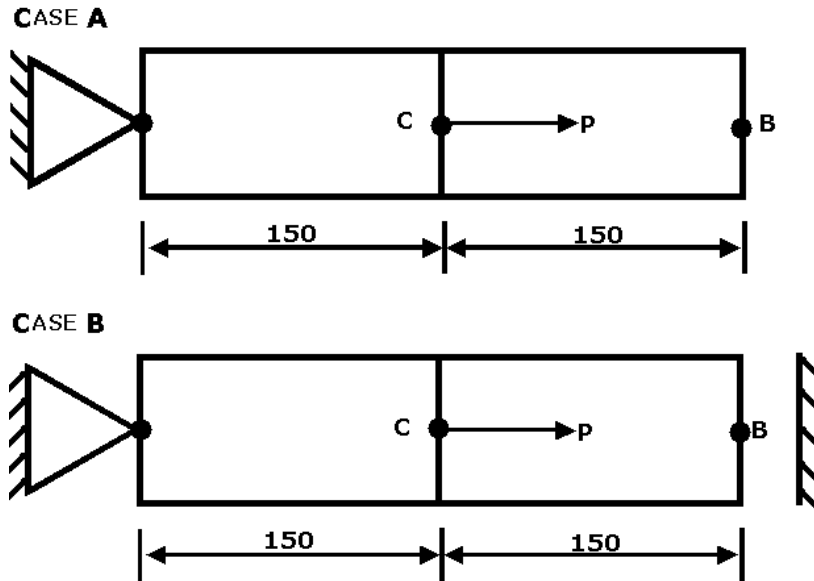


Solution:

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

Problem 12:

Determine the displacement field, support reactions and stress in the body shown as loaded. Use 1D bar element $E= 20\text{Gpa}$, $A= 250\text{mm}^2$, $P = 60\text{KN}$



CASE A

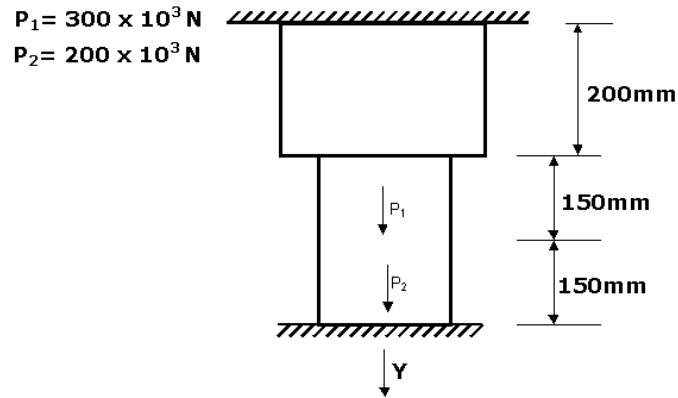
| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

CASE B: When the displacement $g = 1.2\text{mm}$ is given

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

Problem 13:

An axial load $P = 500 \times 10^3 \text{ N}$ is applied at 40°C to the rod as shown in figure. Determine the nodal displacements, reaction and stresses.



| Aluminium | Steel |
|---|---|
| $A_1 = 1800 \text{ mm}^2$ | $A_2 = 1200 \text{ mm}^2$ |
| $E_1 = 70 \times 10^9 \text{ N/m}^2$ | $E_2 = 83 \times 10^9 \text{ N/m}^2$ |
| $\alpha = 20 \times 10^{-6} / ^\circ\text{C}$ | $\alpha = 18 \times 10^{-6} / ^\circ\text{C}$ |

Solution:

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

Problem 14:

Determine the Nodal displacement and Maximum stress in a 1D stepped bar made of steel and aluminum subjected to point load and thermal load with the boundary conditions as shown.

$T_o = 20^{\circ}\text{C}$ and $T_f = 80^{\circ}\text{C}$,

$E_{\text{steel}} = 2e5 \text{ N/mm}^2$

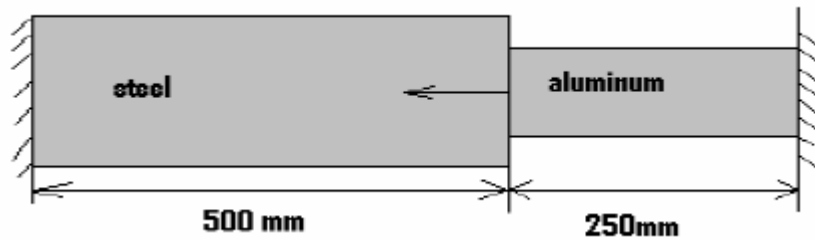
$E_{\text{Al}} = 0.7e5 \text{ N/mm}^2$

Area of steel, $A_{\text{steel}} = 1000\text{mm}^2$

Area of aluminum $A_{\text{al}} = 500\text{mm}^2$

$\alpha_{\text{steel}} = 11.7e-6/^{\circ}\text{C}$

$\alpha_{\text{aluminum}} = 23e-6/^{\circ}\text{C}$



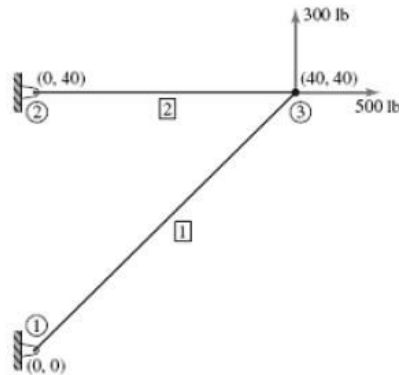
Solution:

| Node No | Element No | Deflection | | Load/ Reaction (KN) | | Stress N/mm^2 | |
|---------|------------|------------|-------|---------------------|-------|------------------------|-------|
| | | Theory | Ansys | Theory | Ansys | Theory | Ansys |
| 1 | | | | | | | |
| 2 | | | | | | | |
| 3 | | | | | | | |
| 4 | | | | | | | |

PART A. 4: ANALYSIS OF TRUSSES

Problem 15

The two-element truss in Figure is subjected to external loading as shown. Using the same node and element numbering as in Figure, determine the displacement components of node 3, the reaction force components at nodes 1 and 2, and the element displacements, stresses and forces. The elements have modulus of elasticity $E_1 = E_2 = 10 \times 10^6 \text{ lb/in}^2$ and cross-sectional areas $A_1 = A_2 = 1.5 \text{ in}^2$.



Procedure:

1. **Utility Menu > File > Change Job name**
2. **Utility Menu > File > Change Title**
3. **Preprocessor > Element type > Add/Edit/Delete > Add > Link1.**
4. **Preprocessor > Real constant > Add/Edit/Delete > Add > Ok > Area > 1.5.**
5. **Preprocessor > Material Property > Young's Modulus > Enter 10e6.**
6. **Preprocessor > Modeling > Create > Node > In Active CS > Create 3 nodes**
Node 1 = (0, 0, 0), Node 2 (0, 40, 0) and Node 3 (40, 40, 0).
7. **Preprocessor > modeling > create > element > auto numbered > thru node >**
(pick nodes and create element between nodes (1, 3) and (2, 3)).
8. **Preprocessor > Load > Define load > Apply > Structural > Displacement > on nodes >**
Select nodes 1 and 2.
9. **Preprocessor > Load > Define load > Apply > Structural > Force/ moment on nodes >**
Select node number 3 and Apply a force 'Fy' = 300
10. **Solution > Solve > Current LS**
11. **General postprocessor > plot Result > Deform shape**
12. **Postprocessor > Contour plot > Nodal solution > Displacement vector sum.**
13. **General Postprocessor > List Result > Reaction Solu > All 'struc forcF'.**

Results

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

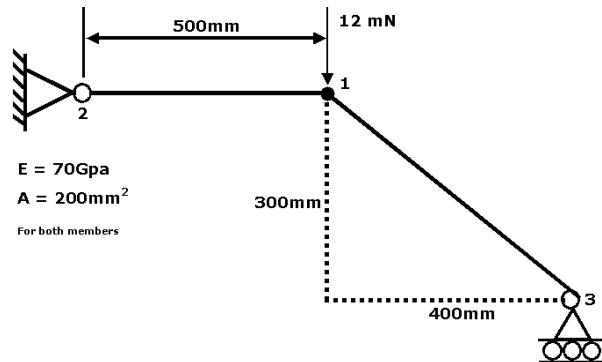
| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 16:

For the two bar truss shown in figure, determine the reaction forces, nodal displacement and element Stress.



Results

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

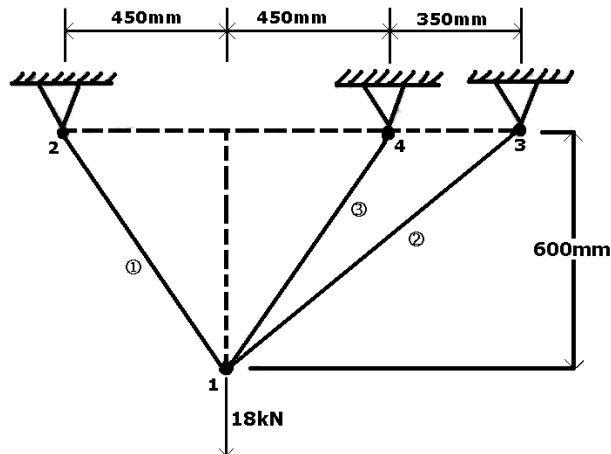
| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 17:

For the two bar truss shown in figure, determine the reaction forces, nodal displacement and element Stress.



Results

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

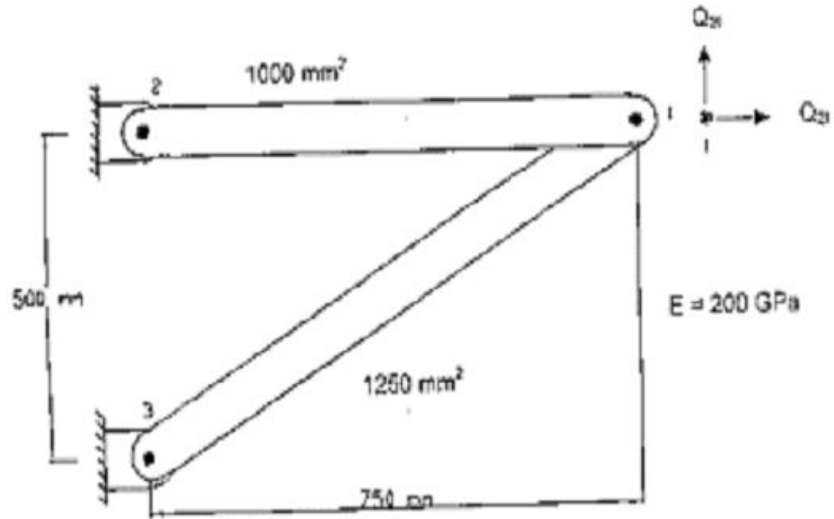
3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 18:

For the Pin jointed configuration shown in figure

- a. Calculate the reaction forces
- b. Nodal Displacements
- c. Element Stresses.



Results

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

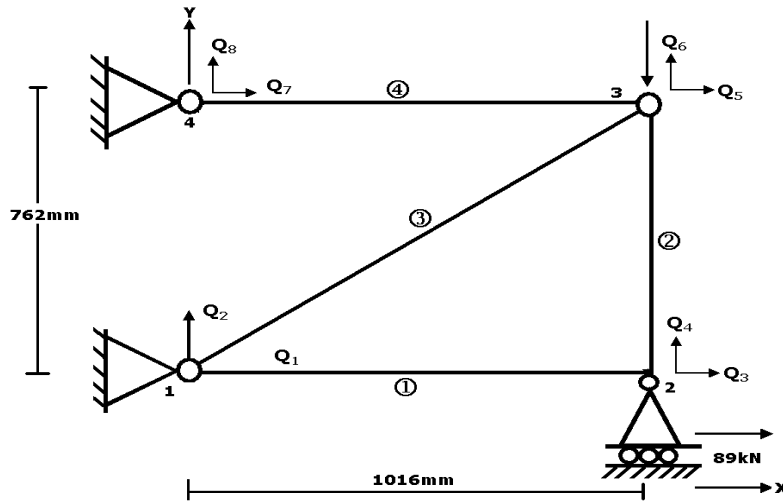
| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 19:

Consider the four bar truss shown in figure. It is given that $E= 200 \text{ Gpa}$ and $\text{Area}= 625 \text{ mm}^2$ for all elements. Calculate the reaction forces, Nodal displacement and element stresses.



Results

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

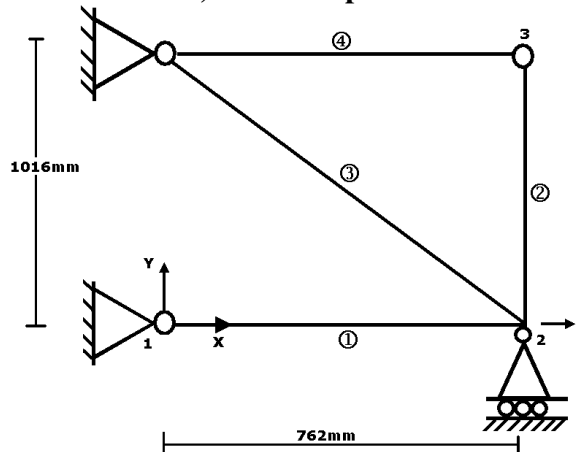
| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 20:

For truss shown in figure an horizontal load of $P= 18\text{KN}$ is applied in the direction at Node 2. Calculate the reaction forces, Nodal displacement and element stresses.



Results

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

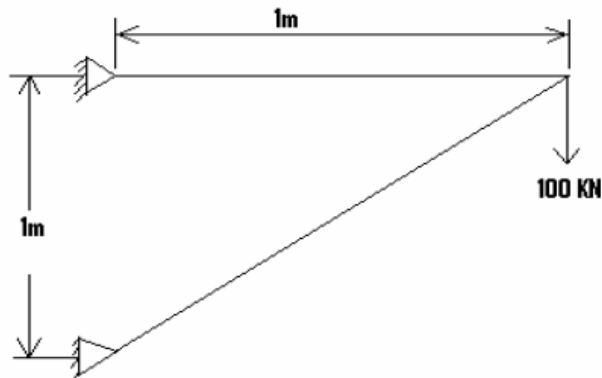
3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 21:

Two bar truss shown in fig. Determine the nodal displacements and the stress in each member.

Take $E = 210 \times 10^9$ Pa, Area $A = 0.01\text{m}^2$

**Results**

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

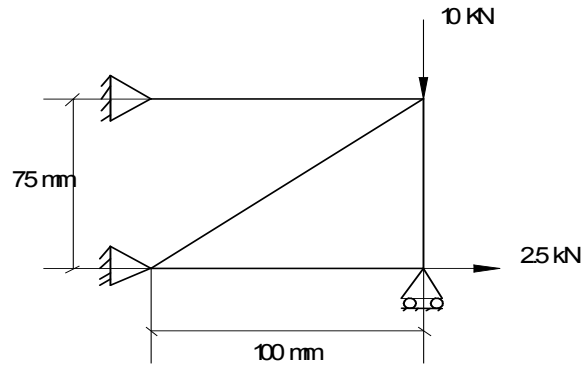
3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 22:

Consider a 4 Bar Truss as shown in the below figure. And solve the problem for displacement stress and Reaction Forces

Data: $A=100 \text{ mm}^2$, $E=2 \times 10^5 \text{ N/mm}^2$, Poisson ratio is 0.3.

**Results**

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|-------------|-------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

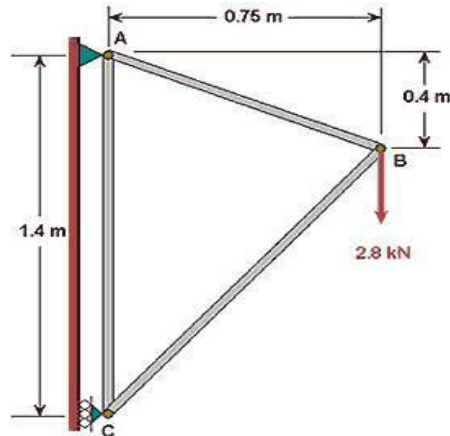
| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

Problem 23:

Consider a 3 Bar Truss as shown in the below figure. And solve the problem for displacement stress and Reaction Forces The cross-sectional area of each member is 0.01 m^2 , the Young's modulus is $200 \times 10^9 \text{ N/m}^2$ and Poisson ratio is 0.3.

**Results**

1. Nodal solution and Displacements

| Node No | Ux (Theory) | Ux (ANSYS) | Uy (Theory) | Uy (ANSYS) |
|---------|---------------|--------------|---------------|--------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

2. Elemental Solution and Displacement

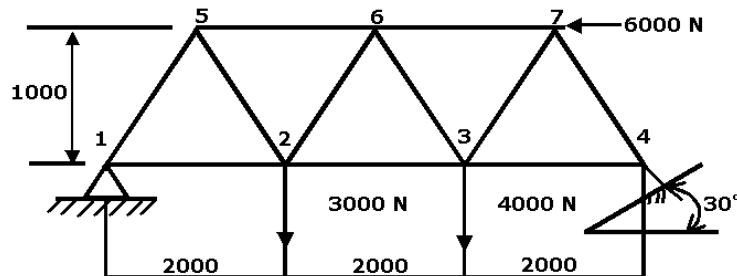
| Element No | Stress (Theory) | Stress (ANSYS) |
|------------|-----------------|----------------|
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |

3. Reaction Solution

| Node No | Fx (Theory) | Fx (ANSYS) | Fy (Theory) | Fy (ANSYS) |
|---------|-------------|------------|-------------|------------|
| 1 | | | | |
| 2 | | | | |
| 3 | | | | |
| 4 | | | | |

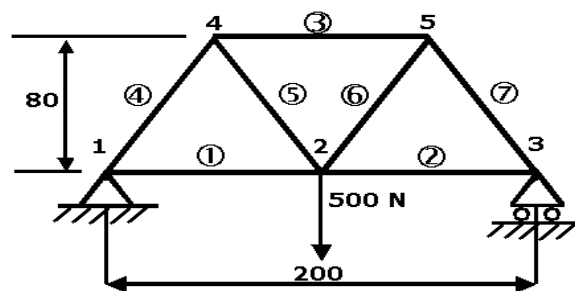
Problem 24: (For Practice)

The below figure shows a truss with a roller support at node 4. The area of cross section of the elements of the truss is 120 Sq.mm. The force acting on the nodes are shown in the figure. It is supported at node 1 by a hinged joint, Node 4 is supported on a roller arrangement and the support allows a freedom of movement at an angle of 30° . Determine the nodal displacement, reactions and stresses in the truss member shown in figure.



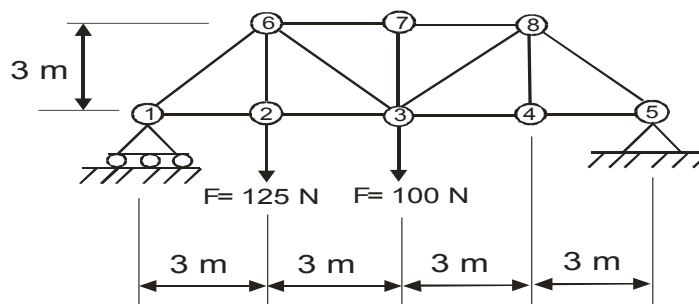
Problem 25: (For Practice)

The figure shows a truss element loaded with 500N at node 2. All horizontal members have an area of 100 Sq.mm and inclined members an area of 80 sq.mm. Young's modulus (E) of element 3= 80Gpa and other element is 210 Gpa. Determine the nodal displacement, reactions and stresses in the truss member shown in figure.



Problem 26: (For Practice)

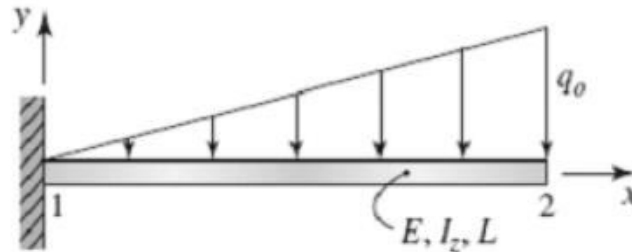
Consider a Truss as shown in the below figure. And solve the problem for displacement stress and Reaction Forces Cross-sectional area of truss members = $3.0E-4 \text{ m}^2$; Modulus of Elasticity = $2.07E11 \text{ N/m}^2$. Circled numbers shown are node numbers



PART A. 5: STRESS ANALYSIS OF BEAMS

Problem 27

Consider a cantilever beam of length 1000mm, subjected to uniformly varying load 0 to 500N/mm. Young's modulus $2e5$ N/mm². The cross section is rectangular with Breadth and Height of 10mm. Obtain the shear force and bending moment diagrams. Obtain the values of deflection, bending moment, maximum bending stress.



Procedure:

1. **Utility Menu > File > Change Job name**
2. **Utility Menu > File > Change Title > Cantilever Beam**
3. **Preprocessor > Element type > Add/Edit/Delete > Add > Beam3**
4. **Preprocessor > Real constant > Add/Edit/Delete > Add > OK > Area = 100 > Area**
moment of inertia = 833.33 > Height = 10.
5. **Preprocessor > Material Property > Material models > Young's Modulus >**
Enter $E_x = 2e5$.
6. **Preprocessor > Modeling > Create > Nodes > In Active CS > Create three nodes**
Node number 1 = (0, 0, 0)
Node number 2 = (1000, 0, 0)
7. **Preprocessor > modeling > create > element > auto numbered > thru node >**
(Pick nodes (1, 2) and create one element).
8. **Preprocessor > Load > Define load > Apply > Structural > Displacement > On Nodes >**
Select node 1 > Apply 'ALL DOF'.
9. **Preprocessor > Load > Define load > Apply > Structural > pressure > on beam > Select**
the First element > OK > give the pressure value '0' in VALI and '500' in VALJ.
10. **Solution > Solve > Current LS.**
12. **General postprocessor > plot Result > Deform shape**
13. **Postprocessor > Contour plot > Nodal solution > Displacement vector sum.**
14. To find Bending moment
Postprocessor > element table > Define table > add > by sequence number > SMISC 6 > ok
Postprocessor > element table > Define table > add > by sequence number > SMISC 12 > ok
14. To plot the Bending moment results
General Postprocessor > Contour plot > Line elem Res
15. Similarly for shear force, max and min stress
Postprocessor > element table > create table > add > by sequence number > SMISC 2 > ok
Postprocessor > element table > create table > add > by sequence number > SMISC 8 > ok
Postprocessor > element table > create table > add > by sequence number > NMISC 1 > ok

**Postprocessor>element table>create table> add>by sequence number>NMISC
3>ok**

**Postprocessor>element table>create table> add>by sequence number>NMISC
2>ok**

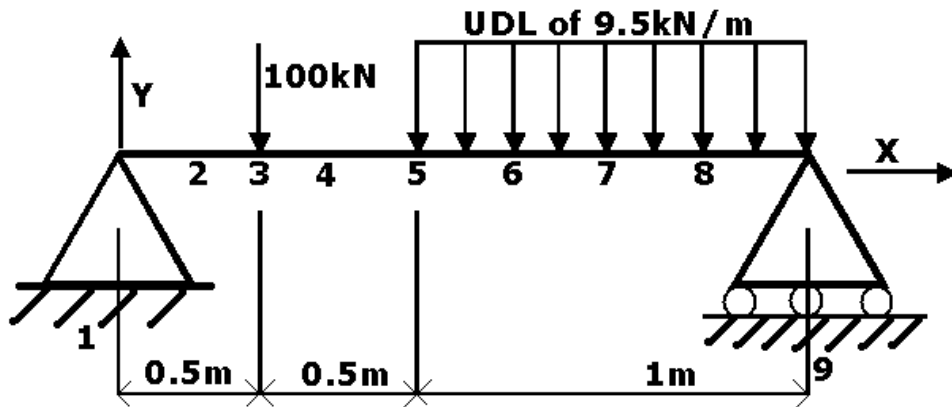
**Postprocessor>element table>create table> add>by sequence
number>NMISC4>ok**

Result:

| Loading | Deflection (mm) | | Bending Moment (N/mm) | | Max.Bending Stress (N/mm ²) | |
|---------|-----------------|--------|-----------------------|--------|---|--------|
| | ANSYS | Theory | ANSYS | Theory | ANSYS | Theory |
| 1 | | | | | | |
| 2 | | | | | | |
| 3 | | | | | | |

Problem 28:

The figure shows a loaded beam subjected to various types of loading. $E = 200\text{Gpa}$ & $I = 4 \times 10^6 \text{mm}^4$.



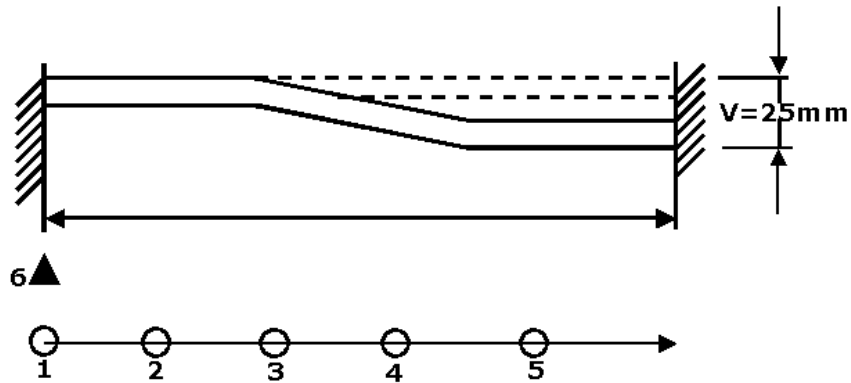
Results:

| Result | Node No | Deflection (mm) | Node No | Bending Moment (N-mm) | Element No | Max. Bending Stress (N/mm ²) |
|--------|---------|-----------------|---------|-----------------------|------------|--|
| ANSYS | 1 | | 1 | | 1- 2 | |
| | 2 | | 2 | | 2-3 | |
| | 3 | | 3 | | 3-4 | |
| | 4 | | 4 | | 4-5 | |
| | 5 | | 5 | | 5-6 | |
| | 6 | | 6 | | 6-7 | |
| | 7 | | 7 | | 7-8 | |
| | 8 | | 8 | | 8-9 | |
| | 9 | | 9 | | | |
| Theory | 1 | | 1 | | 1- 2 | |
| | 2 | | 2 | | 2-3 | |
| | 3 | | 3 | | 3-4 | |
| | 4 | | 4 | | 4-5 | |
| | 5 | | 5 | | 5-6 | |
| | 6 | | 6 | | 6-7 | |
| | 7 | | 7 | | 7-8 | |
| | 8 | | 8 | | 8- 9 | |
| | 9 | | 9 | | | |

Problem 29:

Determine the end forces of a clamped beam due to a 25mm settlement at the right end.

$A = 50 \times 50 \text{ Sq.}$ $E = 200 \text{ Gpa}$ $I = 2\text{mm}^4$

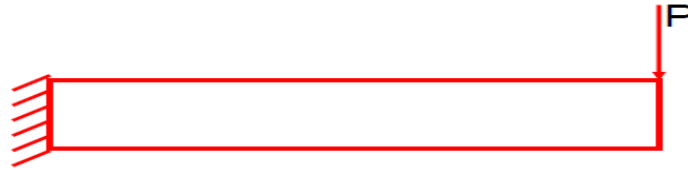


Results:

| Result | Node No | Deflection (mm) | Node No | Bending Moment (N-mm) | Element No | Max. Bending Stress (N/mm ²) |
|--------|---------|-----------------|---------|-----------------------|------------|--|
| ANSYS | 1 | | 1 | | 1- 2 | |
| | 2 | | 2 | | 2-3 | |
| | 3 | | 3 | | 3-4 | |
| | 4 | | 4 | | 4-5 | |
| | 5 | | 5 | | | |
| Theory | 1 | | 1 | | 1- 2 | |
| | 2 | | 2 | | 2-3 | |
| | 3 | | 3 | | 3-4 | |
| | 4 | | 4 | | 4-5 | |
| | 5 | | 5 | | | |

Problem 30:

A 2m Long Cantilever with rectangular section (60mmx90mm) is subjected to concentrated load of 4KN at the free end of the beam. Determine the Deflection, maximum stress due to bending, Bending moment and Shear Force. Take $E = 200\text{Gpa}$. $P = 4 \times 10^3 \text{ N}$, Length $L = 2\text{m} = 2000 \text{ mm}$

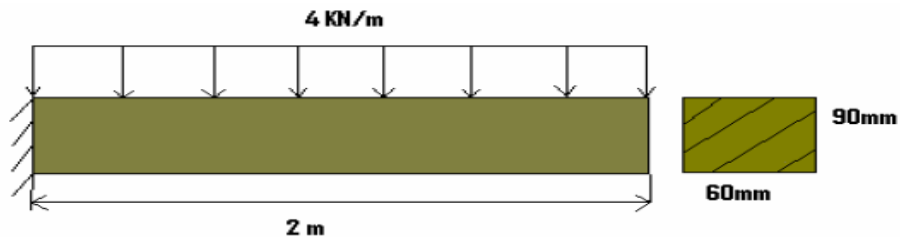


Results:

| Loading | Deflection (mm) | | Bending Moment (N/mm) | | Max.Bending Stress (N/mm ²) | |
|---------|-----------------|--------|-----------------------|--------|---|--------|
| | ANSYS | Theory | ANSYS | Theory | ANSYS | Theory |
| 1 | | | | | | |

Problem 31:

A 2m Long Cantilever with rectangular section (60mmx90mm) is subjected to Uniformly Distributed load of 4 KN/m through out its Length. Determine the Deflection, maximum stress due to bending, Bending moment and Shear Force. Take $E = 200\text{Gpa}$. Length $L = 2\text{m} = 2000 \text{ mm}$

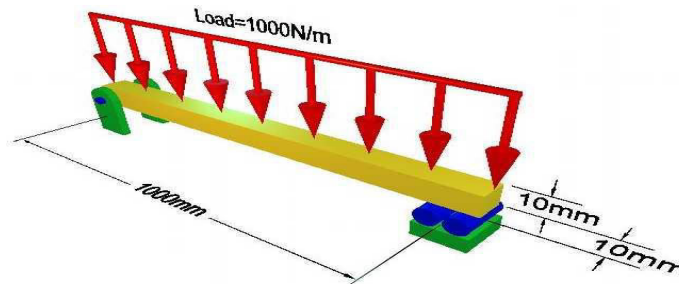


Results:

| Loading | Deflection (mm) | | Bending Moment (N/mm) | | Max.Bending Stress (N/mm ²) | |
|---------|-----------------|--------|-----------------------|--------|---|--------|
| | ANSYS | Theory | ANSYS | Theory | ANSYS | Theory |
| 1 | | | | | | |

Problem 32:

A distributed load of 1000 N/m (1 N/mm) will be applied to a solid steel beam with a rectangular cross section as shown in the figure below. The cross-section of the beam is (10mm x 10mm) while the modulus of elasticity of the steel is 200GPa. Determine the Deflection, maximum stress due to bending,

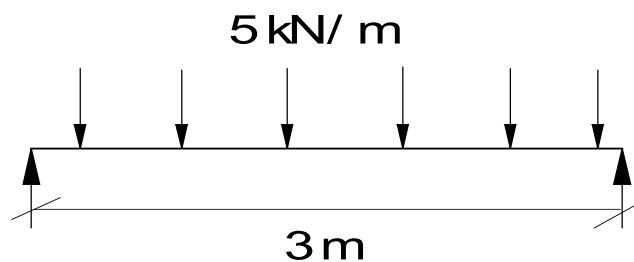


Results:

| Loading | Deflection (mm) | | Max.Bending Stress (N/mm ²) | |
|---------|-----------------|--------|---|--------|
| | ANSYS | Theory | ANSYS | Theory |
| 1 | | | | |

Problem 33:

A simply supported Beam of 3m Span as shown in the figure is subjected to a load of 5 kN/m. determine the max deflection and Stress $E=200 \text{ Gpa}$, $I= 10^8 \text{ mm}^4$

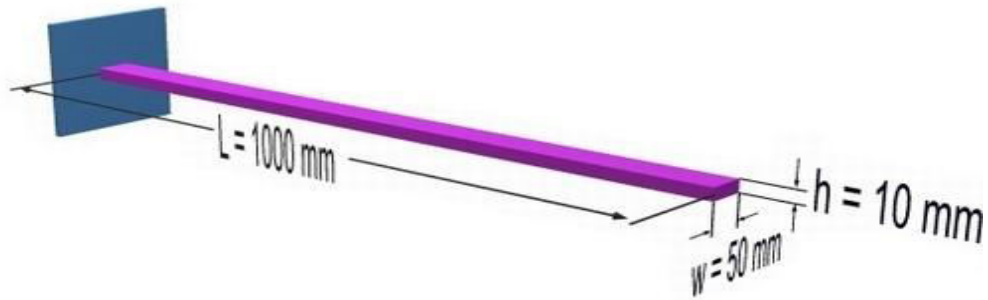


Results:

| Loading | Deflection (mm) | | Max.Bending Stress (N/mm ²) | |
|---------|-----------------|--------|---|--------|
| | ANSYS | Theory | ANSYS | Theory |
| 1 | | | | |

Problem 34:

Loads will not be applied to the beam shown below the beam is to be made of steel with a modulus of elasticity of 200 GPa. Find the max deflection and Stress due to the Self Weight of the Beam.



Results:

| Loading | Deflection (mm) | | Max.Bending Stress (N/mm ²) | |
|---------|-----------------|--------|---|--------|
| | ANSYS | Theory | ANSYS | Theory |
| 1 | | | | |

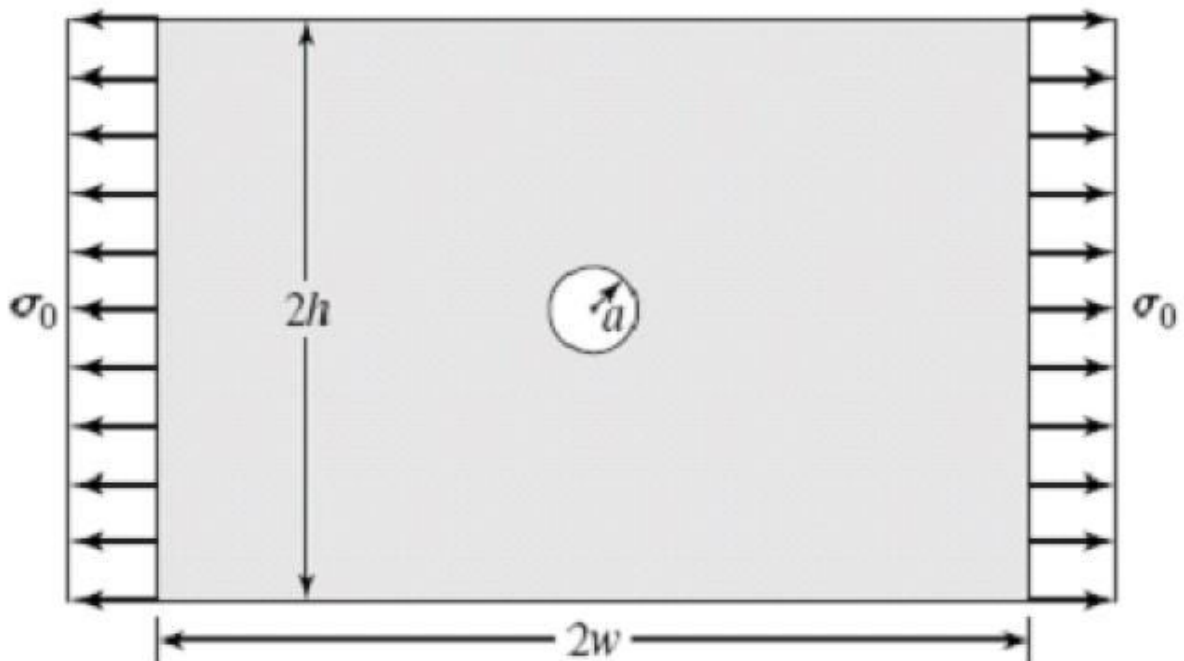
Chapter 4

ANSYS EXERCISE (PART – B)

PART B.1: STRESS ANALYSIS OF RECTANGULAR PLATE WITH A CIRCULAR HOLE

Problem 35

Stress analysis of a rectangular Plate with Hole. Let $2w = 100\text{mm}$, $2h = 50\text{mm}$, $a = 10\text{mm}$, $\sigma_0 = 100\text{ Mpa}$.



Procedure -

1. **Utility Menu > File > Change Job name**
2. **Utility Menu > File > Change Title > Stress analysis of a Plate**
3. **Preprocessor > Element type > Add/Edit/Delete > Add > Solid > Quad 4 node 42**
4. **Preprocessor>Material Property> Young's Modulus > Enter $E_x = 2e5$ and Poisson's ratio = 0.3.**
5. **The model is symmetric we are going to create $\frac{1}{4}$ of the model. Preprocessor>modeling >create> Area > Rectangle> By Dimensions> 0, 100 (X_1, X_2) (0, 50) for (Y_1, Y_2).**
6. **Preprocessor>Modeling > Create> Area >Circle > Solid circle > (define dimensions). Enter WP X = 50, WP Y = 25, Radius = 10.**
6. **Preprocessor>modeling>operate>Booleans>subtract>areas> (select the bigger Area first click apply and select the smaller one and click OK).**
8. **Preprocessor >Meshing>Mesh tool>Smart size ON>Free> Quad >Mesh > Pick all.**
9. **Preprocessor>Load>apply>Structural>Displacement>symmetry B.C>on line (Select the line on X coordinates Zero and Y coordinates Zero and apply Symmetric BCs).**
10. **Preprocessor>Load>apply>Structural> pressure> on line > Enter VALI = 100.**
11. **Solution > Solve > Current LS**

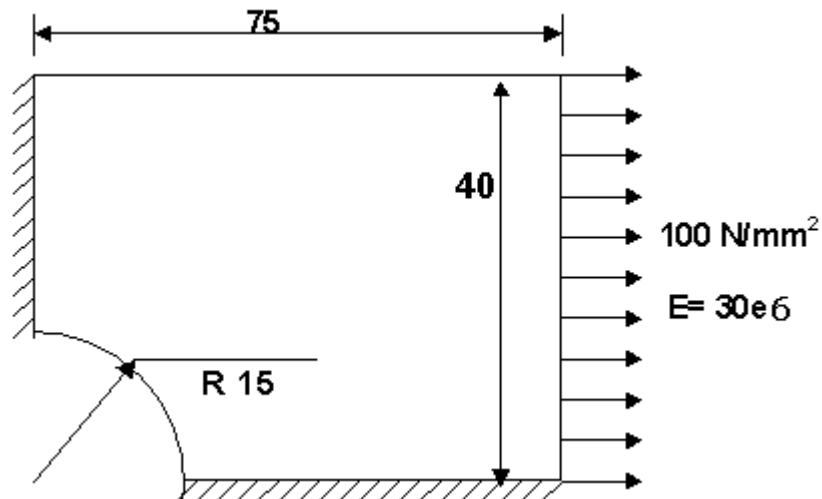
-
-
12. **General postprocessor > Plot Result > Deform shape**
 13. **General postprocessor> Plot Result> Contour Plot> Nodal Solution>Displacement Vector Sum.**
 14. **General postprocessor > Plot Result > Contour Plot > Nodal Solution >Stress >von Misses stress**
 15. **To Expand the Model.
Plot control > Style > Symmetry Expansion > Periodic /Cyclic Symmetry>1/4 Expansion > OK.**

Results:

| | ANSYS | Theory |
|-------------|-------|--------|
| Deformation | | |
| Stress | | |

Problem 36

Determine the location and magnitude of the maximum stress in the plate as shown in figure.

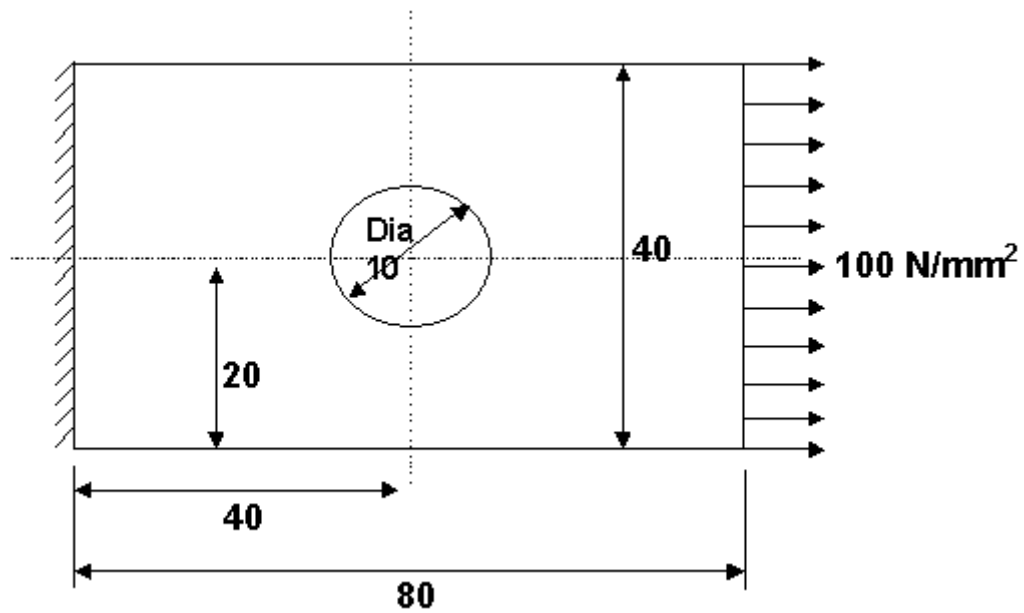


Results:

| | ANSYS | Theory |
|-------------|-------|--------|
| Deformation | | |
| Stress | | |

Problem 37

Determine the location and magnitude of the maximum stress in the plate as shown in figure.

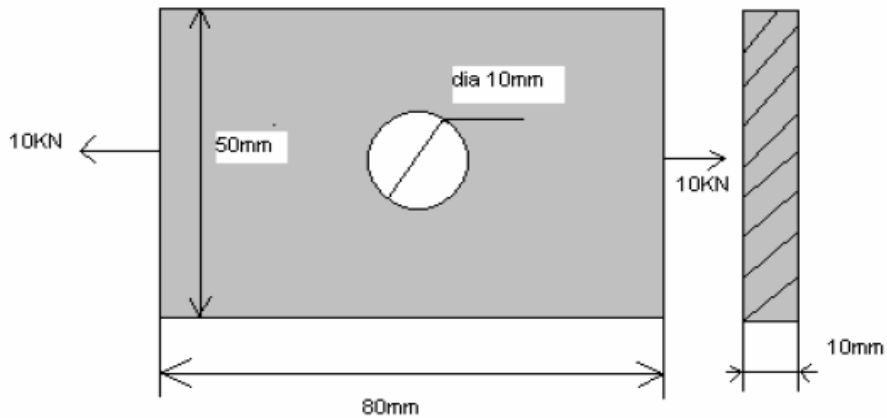


Results:

| | ANSYS | Theory |
|-------------|-------|--------|
| Deformation | | |
| Stress | | |

Problem 38:

Determine the maximum stress for a rectangular plate of 50mm x 80 mm with hole of 10 mm diameter in the center is loaded in axial tension. Thickness of the Plate is 10mm. Take $E = 200 \text{ GPa}$

**Analytical Solution**

$$\sigma_{\text{nom}} = F / (w - a) t = 10000 / (50 - 10) 10 = 25 \text{ N/mm}^2 \quad \text{Results:}$$

$$\text{Stress concentration factor } K_{\sigma} = \sigma_{\text{max}} / \sigma_{\text{nom}}$$

$$2.5 = \sigma_{\text{max}} / 25$$

$$\sigma_{\text{max}} = 2.5 \times 25 = 62.5 \text{ N/mm}^2$$

| | ANSYS | Theory |
|-------------|-------|--------|
| Deformation | | |
| Stress | | |

Problem 39: (For Practice)

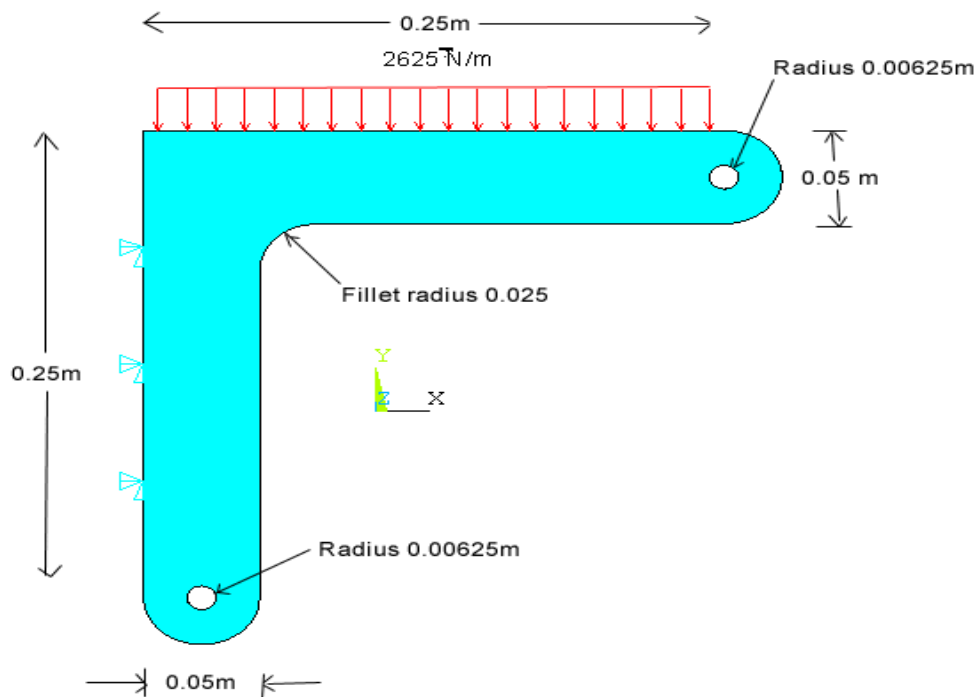
We will model the bracket as a solid 8 node plane stress element. By a plane stress element we are assuming that there are no stresses in the thickness direction of the bracket.

Geometry: The thickness of the bracket is 3.125 mm

Material: Assume the structure is made of steel with modulus of elasticity $E=200$ GPa.

Boundary conditions: The bracket is fixed at its left edge.

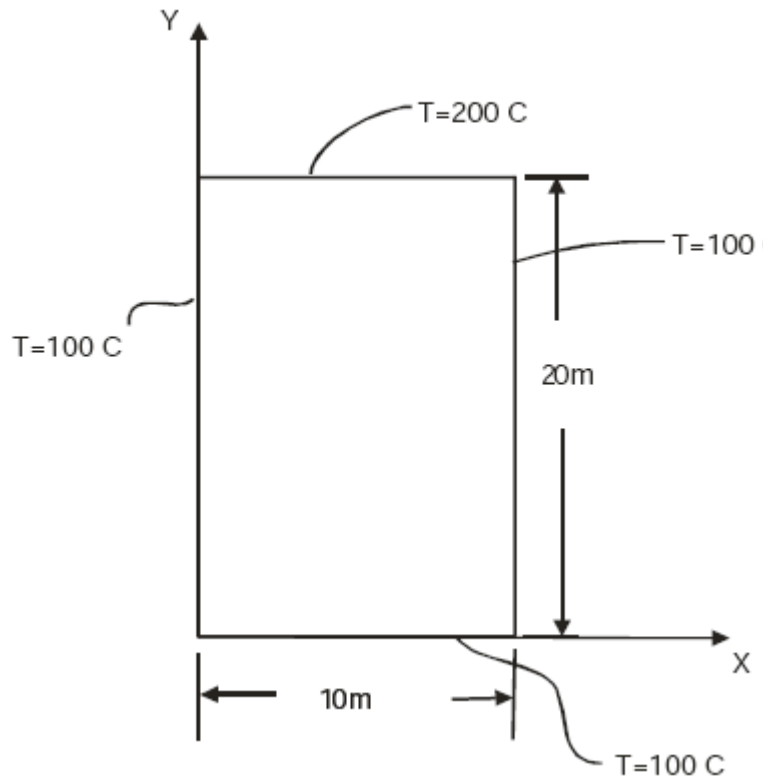
Loading: The bracket is loaded uniformly along its top surface. The load is 2625 N/meter.



PART B.2: THERMAL ANALYSIS

Problem 40

For the rectangular slab shown in the above figure find out the temperature distributions for the boundary conditions indicated. Assume material to be made of Copper.



1. Utility Menu > File >

1. **Change Job name**
2. **Utility Menu > File > Change Title > Thermal Analysis.**
3. **Preprocessor > Element type > Add/Edit/Delete > Add > Thermal > solid > plane55**
4. **Preprocessor > Material Property > Thermal > Conductivity > Isotropic > (Give conductivity value in Kxx) > Enter Kxx = 386 W/mK**
5. **Preprocessor > Modeling > create > Area > Rectangle > By Dimension > (give dimension) X2=10 and Y2=20.**
6. **Preprocessor > Meshing > Mesh tool > smart size ON > Free > Quad > Mesh > pick all**
7. **Preprocessor > Load > Apply > Thermal > Temperature > on line > Give 100 for 3 lines and for the top line give 200.**
8. **Solution > Solve > Current LS.**
9. **General Postprocessor > Plot Result > contour Plot > Nodal Solution > DOF Solution > Nodal Temperature**

ii) **In this example you will learn to model a 2D block subjected to varying boundary conditions. Using ANSYS will allow you to output the temperature distribution in an extremely simple and accurate way. By using any combination of these boundary conditions one can model almost any 2D heat transfer situation. We assume that our block is a rectangle (1m x 2m) made entirely of steel.**

Boundary Conditions:

- 1) The top is insulated.
- 2) The right side has a constant temperature of 100K.
- 3) The left side has constant heat flux into the block of 50 W/m^2 .
- 4) The bottom side is exposed to a convective boundary layer.

Material Properties: (Steel)

Convection film coeff $h = 50 \text{ W/ (m}^2\text{*K)}$

Thermal conductivity $k = 20 \text{ W/m K}$.

Objective: To determine the nodal temperature distribution and create contour plot.

1. **Utility Menu > File > Change Job name.**
2. **Utility Menu > File > Change Title.**
3. **Preprocessor> Element type>Add/Edit/Delete>Add>Thermal>Solid>Plane55**
4. **Preprocessor>Material Property>Thermal> Conductivity>Isotropic> (Give conductivity value in $K_{xx} = 20 \text{ W/M K}$)**
5. **Preprocessor>Modeling>create>Area>Rectangle>By Dimension> (give dimension) $X2=1\text{m}$ and $Y2=2\text{m}$**
6. **Preprocessor>Meshing>Mesh tool>smart size ON>Free>Quad>Mesh>pick all**
7. **Preprocessor>Load>Apply>Thermal>Temperature>on line> Give 100 to right side Line.**
8. **Preprocessor>Load>Apply>Thermal>Convection>on line> Select the bottom line and Enter values for film coefficient = 50 and bulk temperature = 200K.**
9. **Preprocessor>Load>Apply>Thermal>Heat Flux>On line> Enter the value = 50.**
10. **Solution>Solve>Current LS.**
11. **General Postprocessor>Plot Result>contour Plot>Nodal Solution>DOF Solution >Nodal Temperature.**

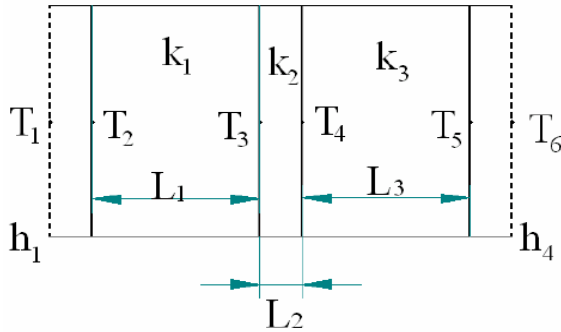
Results:

| Quantity | ANSYS | Theory |
|----------|-------|--------|
| T_2 | | |
| T_3 | | |
| T_4 | | |
| T_5 | | |

Problem 41:

A furnace wall is made of inside Silica brick ($K = 1.5 \text{ W/mk}$) and outside magnesia brick ($K = 4.9 \text{ W/mK}$), each 10 cm thick. The inner and outer surfaces are exposed to fluids at temperature of $820 \text{ }^\circ\text{C}$ and $110 \text{ }^\circ\text{C}$ respectively. The contact resistance for inner and outside surfaces is equal to $35 \text{ w/m}^2\text{k}$. Find the heat flow through the wall per unit area per unit time and temperature distribution across the wall

COMPOSITE WALL1

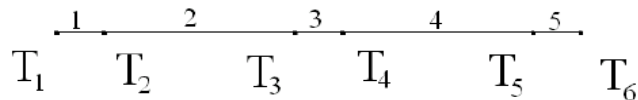


- $k_1 = 1.5 \text{ W/mK}$
- $k_3 = 4.9 \text{ W/mK}$
- $L_1 = L_2 = 100\text{mm}$
- $T_1 = 820^\circ\text{C}$
- $T_6 = 110^\circ\text{C}$
- $R_c = 0.001\text{m}^2\text{K/W}$.
- $h_1 = h_4 = 35 \text{ W/m}^2\text{K}$.

$$R_c = \frac{L_2}{K_2 A}$$

- Take $L_2 = 1\text{mm}$,
- $k_2 = 1 \text{ W/mK}$
- $A = 1\text{m}^2$

COMPOSITE WALL1- ANSYS analysis



For FE model,

| Element no. | Element Type |
|-------------|--------------|
| 2,3,4 | LINK 32 |
| 1,5 | LINK 34 |

COMPOSITE WALL1 – Theoretical solution:

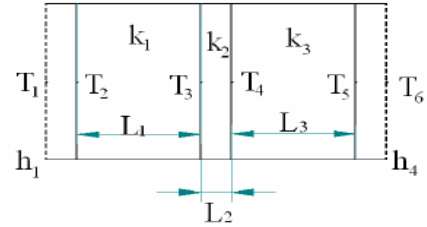
$$Q = \frac{(T_1 - T_6)}{\frac{1}{A} \left(\frac{1}{h_1} + \frac{L_1}{K_1} + \frac{L_2}{K_2} + \frac{L_3}{K_3} + \frac{1}{h_4} \right)} = 4889.21W / m^2$$

$$Q = h_1 A (T_1 - T_2) \Rightarrow T_2 = 680.31^{\circ}C$$

$$Q = \frac{k_1 A (T_2 - T_3)}{L_1} \Rightarrow T_3 = 354.36^{\circ}C$$

$$Q = \frac{k_2 A (T_3 - T_4)}{L_2} \Rightarrow T_4 = 349.47^{\circ}C$$

$$Q = \frac{k_3 A (T_4 - T_5)}{L_3} \Rightarrow T_5 = 249.69^{\circ}C$$



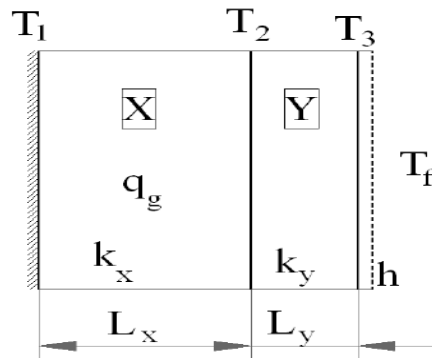
Results:

| Quantity | ANSYS | Theory |
|----------------|-------|--------|
| T ₂ | | |
| T ₃ | | |
| T ₄ | | |
| T ₅ | | |
| Heat flow | | |

Problem 42:

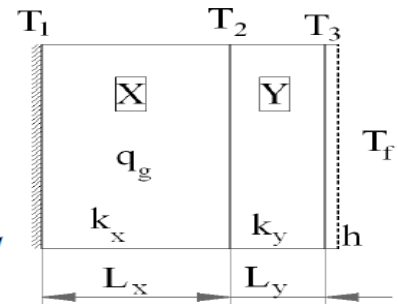
A plane wall 'X' ($K = 75 \text{ W/mk}$) is 60mm thick and has volumetric heat generation of $1.5 \times 10^6 \text{ w/m}^3$. It is insulated on one side while the other side is in contact with surface of another wall 'Y' ($k = 150 \text{ W/mk}$) which is 30 mm thick and has no heat generation. The free surface of wall 'Y' is exposed to a cooling fluid at 20°C with a convection coefficient of $950 \text{ W/m}^2\text{k}$. find steady state temperatures at salient points across the composite wall

COMPOSITE WALL with heat generation –



- $k_x = 75 \text{ W/mK}$
- $k_y = 150 \text{ W/mK}$
- $L_x = 60 \text{ mm}$
- $L_y = 30 \text{ mm}$
- $T_f = 20^\circ\text{C}$
- $q_g = 1.5 \times 10^6 \text{ W/m}^3$
- $h = 950 \text{ W/m}^2\text{K}$
- $A = 1 \text{ m}^2$

COMPOSITE WALL with heat generation – Theoretical solution:



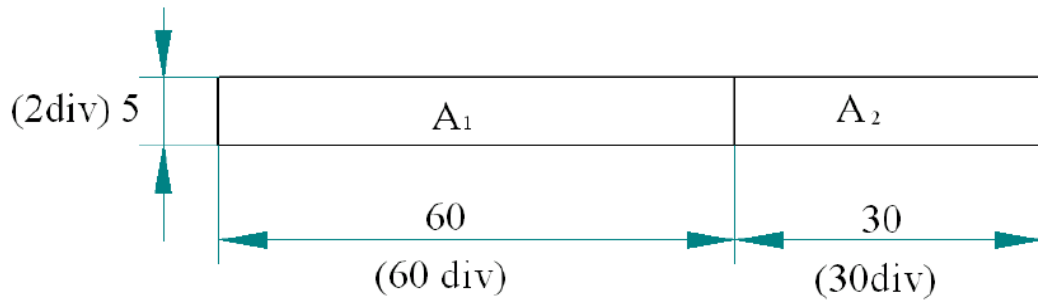
$$Q_g = q_g A = 900000 \text{ W} / \text{m}^2 (= Q)$$

$$Q = hA(T_3 - T_f) \Rightarrow T_3 = 114.7^\circ\text{C}$$

$$Q = \frac{k_y A (T_2 - T_3)}{L_y} \Rightarrow T_2 = 132.7^\circ\text{C}$$

$$T_1 = \frac{q_g}{2k_x} L_x^2 + T_2 \Rightarrow T_1 = 168.7^\circ\text{C}$$

COMPOSITE WALL with heat generation – ANSYS analysis



For FE mode

(for areas A₁, A₂)

Element Type : PLANE 55 (quad 4node)

Option: Plane Thickness

Results:

| Quantity | ANSYS | Theory |
|----------------|-------|--------|
| T ₁ | | |
| T ₂ | | |
| T ₃ | | |

PART B.3: FLUID FLOW ANALYSIS

Problem 43:

Fluid flow analysis in 2D bodies

Air travels between two infinite plates. For the sake of modeling, these plates will be assumed to be 1.5 m long and 5 cm apart. All units are S.I.

Boundary Conditions:

Air enters between the plates at a uniform velocity of 0.1 m/s.

Material Properties:

Air: Density = 1.23 Kg/m³

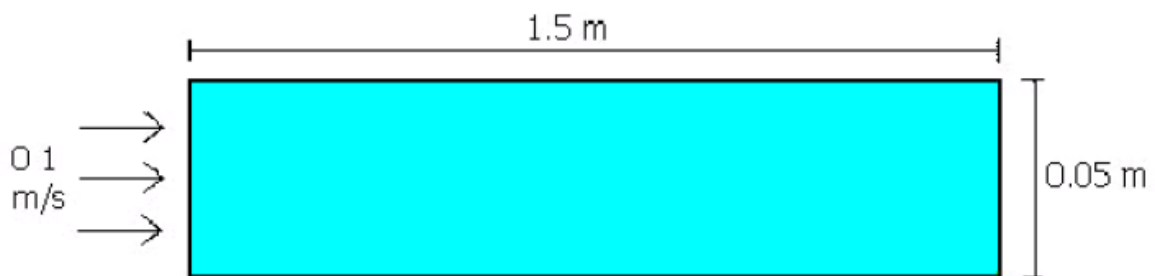
Viscosity = 1.79 e -5

Dimensions

Length = 1.5 m

Width = 0.05 m

Objective: Find the nodal velocity distribution within the air gap between the two plates.



Preprocessing:

1. Start ANSYS

Go to Start > All Programs > ANSYS 11.0 > ANSYS Product launcher
Specify the working directory and job name and click 'Run'.

2. Create areas.

After opening ANSYS,

Click **Preprocessor>-Modeling->** and create the block as shown below.

Hint: Use key points and then create the outer area and the plate within it....OR...

Create the large area using the **Create>Areas>Rectangle** (1.5m x 0.05m) command, then create the small plate and subtract the small plate from the large area.

3. Define the material properties.

Go to **Preprocessor>Flotran Set Up>Fluid Properties.**

On the box, shown below, change the first two input fields to **AIR-SI**, and then click on **OK**. Another box will appear. Accept all defaults on that box by clicking on **OK**.

Go to the ANSYS Main Menu.

Click **Preprocessor > Material Props > Material Models**. The following window will appear as shown below, in this menu

Choose **CFD>Density** and enter the value 1.23 and choose **CFD>Viscosity** and enter 1.79e-5.

4. Define fluid element type.

(2D Flotran 141 element, which is a 2-D element for fluid analysis)

Click **Preprocessor>Element Type>Add/Edit/Delete...** In the 'Element Types' window that opens click on Add... The following window opens.

Type **1** in the **Element type reference number**.

Click on **Flotran CFD** and select **2D Flotran 141**. Click **OK**. Close the **Element types** window. So now we have selected Element type **1** to be solved using **Flotran**, the

computational fluid dynamics portion of ANSYS. This finishes the selection of element type.

5. Specify meshing controls / Mesh the areas to create nodes and elements.

Under Ansys Main menu Choose > Meshing > Mesh tool > Set the smart size value to '1'.

Solution:

6. Specify boundary conditions.

Go to Preprocessor>**Loads>Define Loads>Apply>Fluid CFD>Velocity>on lines.**

Pick

the left edge of the outer block and Click **OK**. The following window comes up. Enter 0.1 in the VX value field and click OK. The 0.1 corresponds to the velocity of 0.1 meter per second of air flowing from the left side. Repeat the above and **set the Velocity to ZERO** for the air along all of the edges of the flat plate. ($VX=VY=0$ for all sides).

Now Atmospheric Pressure must be set for the top, bottom, and right side of the outer area. To do this, select **Preprocessor>Loads>Define Loads>Apply>Fluid CFD>Pressure DOF>on lines.** Click the right most line.

Enter **0** for the constant pressure value for these faces and click **OK**. This sets the pressure to Atmospheric.

Once all the Boundary Conditions have been applied, the boundary conditions on the plates should look like this:

7. Specify number of solution iterations.

Go to ANSYS **Main Menu>Solution>Flotran Set Up>Execution Ctrl.** The following window appears. Change the first input field value to **20**, as shown. No other changes are needed. Click **OK**. (The reason behind setting the iterations so high is that when you run the Flotran Analysis, it stops only when the solution converges, or the solution reaches the number of iterations).

8. Solve.

Go to **Solution>Run FLOTRAN.**

9. Postprocessing:

Plot the contour plot of the velocity distribution.

Go to **General Postproc >Read Results>Last Set.**

Then go to **General Postproc>Plot Results>Contour Plot>Nodal Solution.**

The following window appears –

10. Exit:

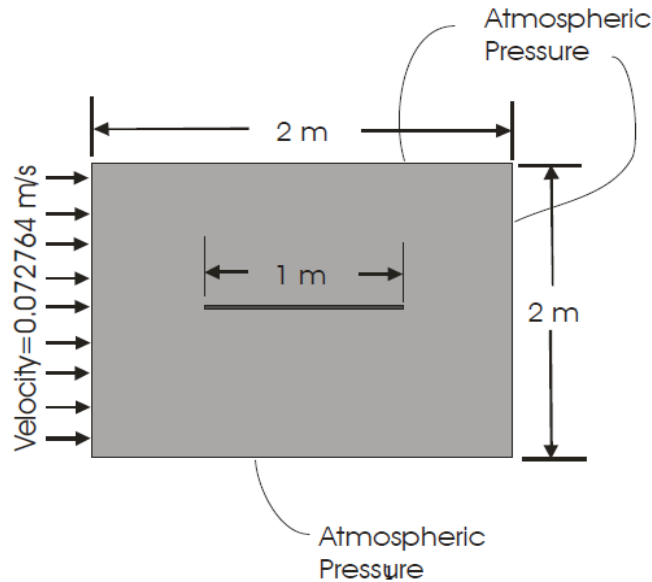
Toolbar > QUIT

Utility Menu > File > Exit

Use the /EXIT command in the Input Window

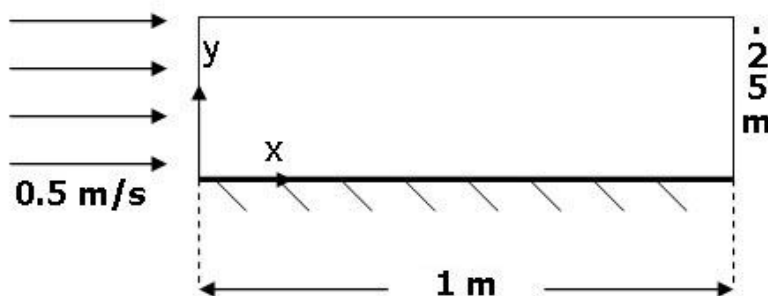
Problem 44:

The fluid is air, with density, $\rho=1.23 \text{ kg/m}^3$, and dynamic viscosity, $\mu =1.79\text{E-}5 \text{ N-s/m}^2$.The plate is 1 m long, as shown, and is very thin. It will be modeled with a thickness of 0.001m.The plate is modeled in a square field, with edge lengths of 2 m. The 2 m edge length dimensions are arbitrary. These lengths are chosen large enough such that the effects of the flatplate on the flow are captured completely within the square field. Also, the flow velocity in all directions is zero along the sides of the flat plate.

**Problem 45:**

Compute and plot the Velocity and Temperature distribution of a flow of air over a flat plate. Dimensions:

- The plate is **1 m** long.
- The flow area is **1 m by .25 m**. This arbitrary size serves to set up the boundary conditions of air traveling over the plate.
- The **velocity of the air** at infinite distance from the plate is **0.5 m/s**.
- **Atmospheric pressure** is assumed on all faces except the face where velocity is input into the system.



Problem 46:

Air at 100 degrees Celsius travels between two infinite plates. For the sake of modeling, these plates will be assumed to be 1.5 m long and 5 cm apart. All units are S.I.

Boundary Conditions:

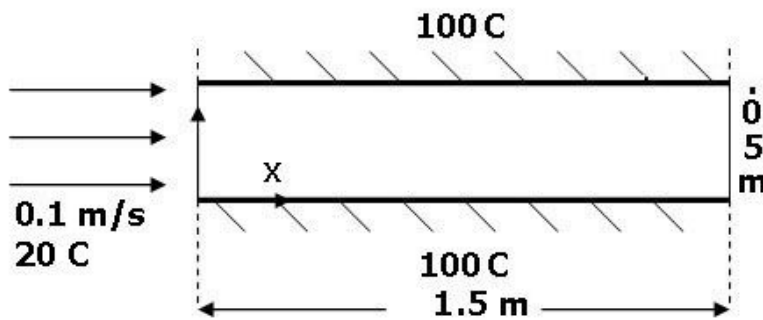
- 1) Air enters between the plates at a uniform velocity of 0.1 m/s and 20 degrees Celsius.
- 2) The wall temperature is 100 degrees Celsius

Material Properties: AIR-SI

Dimensions

Length = 1.5 m

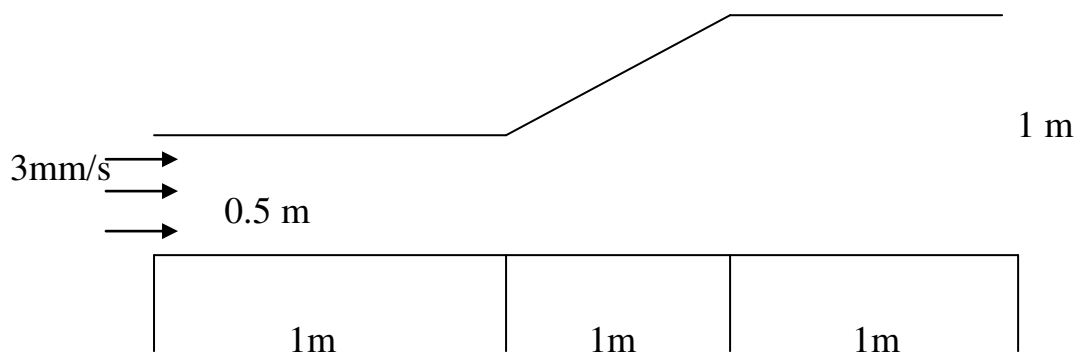
Width = 0.05 m

**Problem 47:**

Compute and plot the velocity distribution of a flow

Dimensions:

- The duct has an entrance region of 1m by 0.5m and an exit region of 1m by 1m. These two regions are separated by a horizontal distance of 1m.
- The **velocity of the air** at infinite distance from the plate is 3 mm/s (0.003m/s). This velocity best demonstrates the recirculation zone while keeping the flow inside the duct laminar.

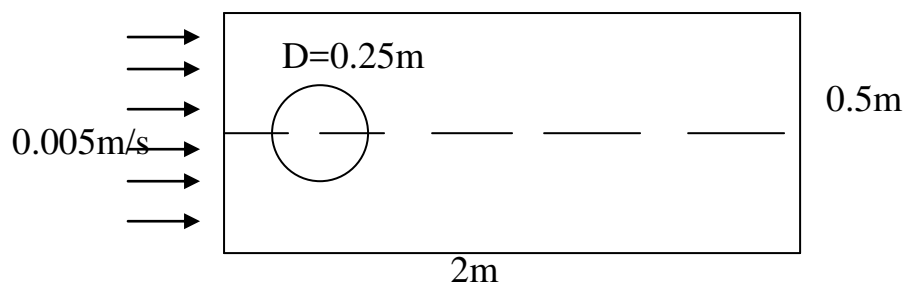


Problem 48:

Compute and plot the velocity distribution of a flow of air over an infinite cylinder.

Dimensions:

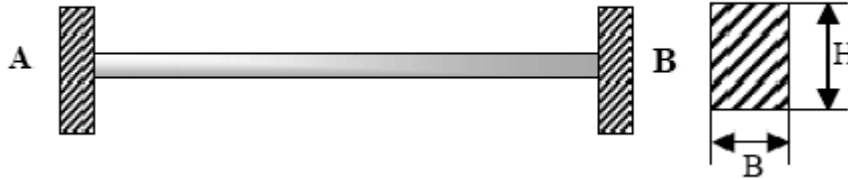
- The cylinder is **0.25 m** in diameter
- Because of symmetry, the problem is simplified so that only half of the cylinder is computed. The results are assumed to be the same on the other side of the x axis (axis of symmetry)
- The flow area is **2 m by 0.5 m**. This arbitrary size serves to set up the boundary conditions of flow over a cylinder.
- The **velocity of the air** at infinite distance from the plate is **5 mm/s (0.005m/s)**.



PART B.4: DYNAMIC ANALYSIS

Problem 49:

Obtain the first five natural frequencies for the beam show below. Take material Properties young's modulus $2e5\text{N/mm}^2$, Density $7800\text{e-}12\text{ Tonns/mm}^3$.length of the Beam 1000mm cross section $10\text{X}10\text{mm}$.



1. *Utility Menu > File > Change Job name > Give Job name 'Simply supp'*
2. *Utility Menu > File > Change Title*
3. *Preprocessor > Element type > Add/Edit/Delete > Add > Beam3*
4. *Preprocessor > Real constant > Add/edit/delete > Add > Ok > Area (100) > Area Moment of Inertia (833.33) > Height (10)*
5. *Preprocessor > Material Property > structural > Linear > Elastic > Isotropic > give value for Young's Modules, Poisson ratio and density*
6. *Preprocessor > Modeling > Create > Key points > In Active CS > Enter the co-ordinates Here.*
7. *Preprocessor > Modeling > Create > Lines > Lines > Straight Line > Create a line between Key point 1 and Key point 2 and so on.*
8. *Preprocessor > Meshing > Mesh tool > Line set > No of division 10 > Preprocessor > Meshing > Mesh tool > Mesh > pick all*
9. *Preprocessor > Load > Define load > Apply > Structural > Displacement > select key points on either side and constrain 'ALL DOF'.*
10. *Solution > Analysis Type > New Analysis > Set to Modal*
11. *Solution > Analysis Type > Analysis Option > No. of .modes to extract (10) > No of modes to expand 10 > ok > ok*
12. *Solution > Solve > Current LS.*
13. *General Postprocessor > Read Results > By Pick > (you can see the frequency values)*
14. *Postprocessor > Contour plot > Nodal solution > Displacement vector sum.*

Results:

| SI No | Frequency |
|-------|-----------|
| 1 | |
| 2 | |
| 3 | |
| 4 | |
| 5 | |

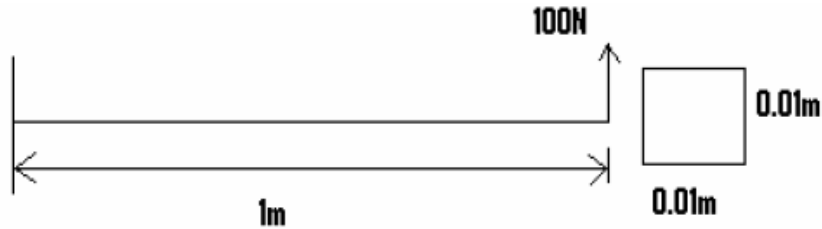
Problem 50:

Simple Harmonic Analysis of a Cantilever beam as shown below

$$E = 206800e^6 \text{ N/m}^2$$

$$I = 8.33e^{-10} \text{ mm}^4$$

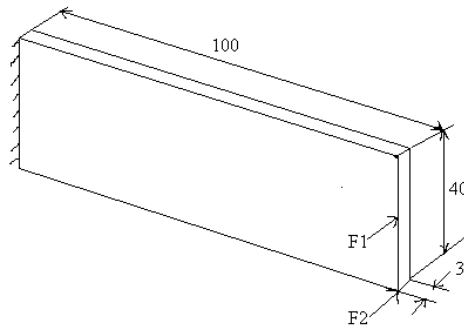
$$\rho = 7830 \text{ Kg/m}^3$$



Results : Value (VS) Frequency Graph

Problem 51:

A Rectangular plate of dimensions 100x40x3 is fixed at one its ends. It is subjected to 2 point loads at its free end. It is required to determine the natural frequencies and the frequency response of the plate. Also transient analysis is carried out.

**Material Properties**

$$E = 10000 \text{ kg/mm}^2$$

$$\nu = 0.27$$

$$\text{Density} = 7.920e-10 \text{ kg/mm}^3$$

Loading(in Z-direction)

$$F1 = 0.5 \text{ kg}$$

$$F2 = 0.5 \text{ Kg}$$

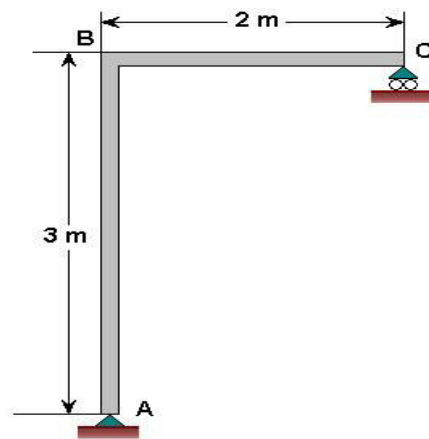
Analysis Type(s): Mode-frequency Analysis

Mode Superposition Harmonic Response Analysis

Mode Superposition Transient analysis

Problem 52:

The problem considered here is the vibration analysis of the right-angle frame.



$$E = 200 \times 10^9 \text{ Pa}$$

$$\nu = 0.29$$

$$\rho = 7860 \text{ kg/m}^3$$

$$I = \frac{1 \times 10^{-4}}{12}$$

Problem 53:

In this example we will find the first five natural modes and frequencies of a square plate with simply supported edges and a circular hole at its center, shown in Figure 6.1.

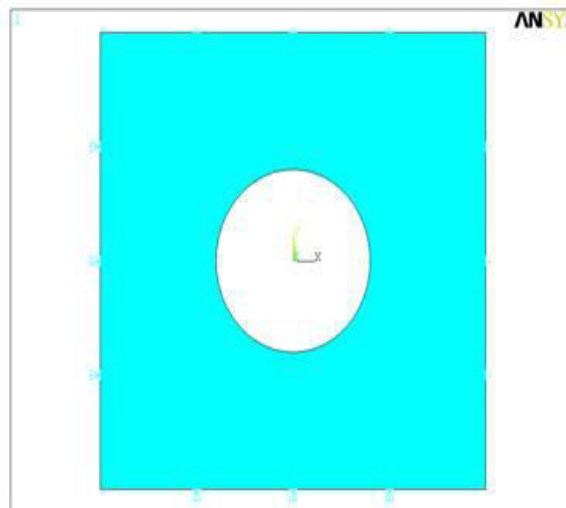
$E = 70 \times 10^9$ (70 GPa) and

$\nu = 0.3$.

$\rho = 2.7 \times 10^3$ (2700 kg/cubic meter).

Dimension = 1 m x 1 m

Circle = 0.2 m

**Results:**

| MODE | Frequency |
|------|-----------|
| 1 | |
| 2 | |
| 3 | |
| 4 | |
| 5 | |

Problem 54:

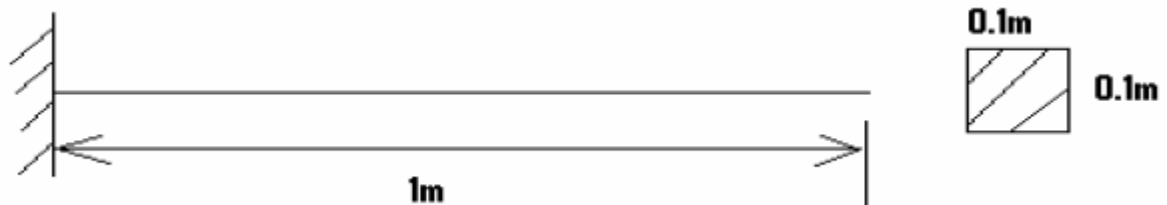
Simple Modal Analysis of a Cantilever Beam as shown below , Objective is to get the Frequency and Different modes of Vibration

$$E = 2e11 \text{ N/m}^2$$

$$I = 8.33e-06 \text{ m}^4$$

$$\text{Area } A = 0.01 \text{ m}^2$$

$$\text{Density} = 7830 \text{ Kg/m}^3$$



To Determine the Natural Frequencies of a Cantilever Beam

Given Data

| | | | |
|----------------------|------------|-------------|-------------------------------------|
| Length | L = | 1 | m |
| Breath | B = | 0.1 | m |
| Depth | D = | 0.1 | m |
| Young's Modulus | E = | 2.00E+11 | N/m ² |
| Density | $\rho =$ | 7.830E+03 | Kg/m ³ |
| Aceln due to gravity | g = | 9.810 | m/Sec ² |
| PIE | $\pi =$ | 3.143 | |
| Area | A = | 0.01 | m ² |
| M.I | I = | 8.33333E-06 | m ⁴ |
| Density/L | $\rho_1 =$ | 7.830E+01 | Kg/m |
| S-Densitv | $\rho =$ | 7.982E+02 | Kg-Sec ² /m ⁴ |

Theoretical Results:**Beam Constant**

| | | |
|-----------------|-----------------|-----------------|
| $(\beta_1 l)^2$ | $(\beta_2 l)^2$ | $(\beta_3 l)^2$ |
| 3.52 | 22.00 | 61.70 |

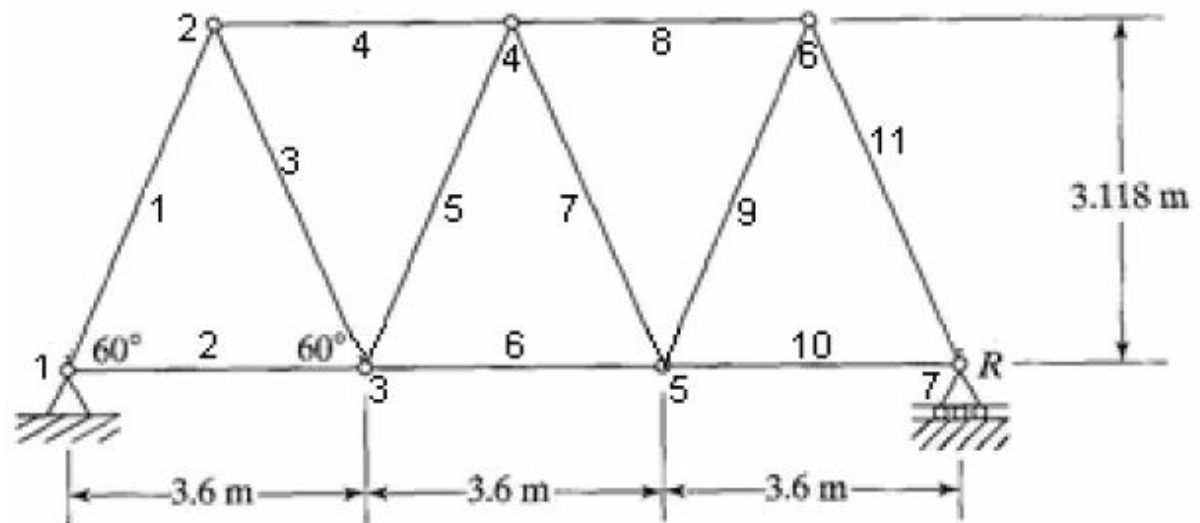
Circular Frequency

| | | |
|-----------------|--|-----------|
| $\omega_{n1} =$ | $(\beta_1 l)^2 \sqrt{(EI)/(\rho I^4)}$ | 1.608E+03 |
| $\omega_{n2} =$ | $(\beta_2 l)^2 \sqrt{(EI)/(\rho I^4)}$ | 1.005E+04 |
| $\omega_{n3} =$ | $(\beta_3 l)^2 \sqrt{(EI)/(\rho I^4)}$ | 2.819E+04 |

| | | |
|----------------------------|-------------------------------|-----------|
| Natural Frequencies | $f_{n1} = \omega_{n1}/2\pi =$ | 255.89749 |
| | $f_{n2} = \omega_{n2}/2\pi =$ | 1599.3593 |
| | $f_{n3} = \omega_{n3}/2\pi =$ | 4485.4759 |

Results:

| Mode | ANSYS (Frequency) | Theory(Frequency) |
|------|--------------------|-------------------|
| 1 | | |
| 2 | | |
| 3 | | |

Problem 55:**Determine the Modal Deflections for the Truss System below** $E = 200\text{GPa}$, $A = 3250\text{ mm}^2$ 

Results:

| Mode | Frequency |
|------|-----------|
| 1 | |
| 2 | |
| 3 | |
| 4 | |

COMPUTER AIDED MODELING AND ANALYSIS LAB
VIVA QUESTIONS

1. What is the total degree of freedom of Ansys commercial package.
2. What are the different menus in ANSYS.
3. What is Work space and Swap space.
4. What is default value of workspace and Swap space?
5. What file format ANSYS can support
6. What is the current ANSYS version
7. What is optimization in ANSYS.
8. What is sub structuring and adaptive meshing.
9. Which package is better – ANSYS or NASTRAN.
10. What is Resume in ANSYS.
11. What is P-method and H-method
12. What is scalar parameters in ANSYS.
13. What are primary nodes and Secondary nodes.
14. What is Mirror (Reflection).
15. What is subroutines.

VIVA QUESTIONS ON FEM

Q.1) What are the different approximate solution methods.

Different approximate solution methods are

1. Functional approximation
2. Finite Difference method
3. Finite Element method

Q.2) What do you mean by continuum.

Structure which is considered for analysis is called continuum.

Q.3) Define term node

The element which is connected with another element at junction is called node.

Q.4) Define term element

Discretised structure is called an element .

Q.5) What is convergence

Process of achieving value to actual solution.

Q.6) What are the types convergence

p-convergence

h-convergence

Q.7) What is p- convergence

Convergence by increasing the elements

Q.8) What is h convergence

Convergence by increasing nodes

Q.9) What is higher order elements

The element which contain more no. of nodes are called higher order element .

Q.10) Give example for higher order elements

Higher order elements are CST, Quadrilateral element.

Q.11) What do you mean by compatible elements.

Elements, which are compatible with adjacent element, like no discontinuity, overlap or sudden slope.

Q.12) What is geometric invariance.

The property in which the shape of the element will not change with change in local coordinates is called geometric invariance.

Q.13) Why do we use Pascal's triangle in FEA

If the displacement equation doesn't contain all the required terms then balancing is done by the Pascal's triangle.

Q.14) What are the steps involved in FEA

Steps involved in FEA:

1. Modelling
2. Discretization of structure
3. Derivation of elemental stiffness matrix.
4. Assembly of elemental equation
5. Applying boundary conditions
6. Computation of stress and strain
7. Interpretation of results

Q.15) what is stiffness matrix

The matrix when contain parameters like E, A, displacement and applied force is stiffness matrix

Q.16) How to obtain stiffness matrix

Stiffness matrix can be obtained by applying condition of minimum potential energy to potential energy equation.

Q.17) What is displacement function

: Displacement function is the assumed polynomial equation, which satisfies boundary conditions.

Q.18) How to identify order of elements

Order of elements depends on the no. of nodes.

Q.19) Mention different types of elements

Different types of elements are bar elements, beam element, truss element, shell element, axis symmetric.

Q.20) Mention some application of FEA

Mechanical, Aerospace, Civil, structure analysis, biomedical, geo -mechanic, electromagnetic.

Q.21) What is connectivity.

Relation between the connected elements is connectivity.

Q.22) What are the methods to improve problem solution

Problem solution can be improved by increasing no. of elements or no. of nodes .

Q.23) Define symmetry in matrix

It is the square matrix in which the element of the row are same as that of element of column.

Q.24) What is plane stress

Stress acting on 2-D element.

Q.25) What is plane strain

Strain occurring in 2-D element.

Q.26) Compare FEA with solid mechanics

complicated irregular structures are difficult in solid mechanics but FEA it's easier with greater accuracy.

Q.27) What are the packages available for FEA

packages available for FEA: ANSYS, I-DEAS, NASTRAN, ABAQUS, COSMOS, ALGOR, PATRAN.

Q.28) Define potential energy

Energy possessed by the body due to it's position

Q.29) Define minimum potential energy

for conservative system of all kinematically admissible displacement field those corresponding to equilibrium extremise the total potential energy if extreme condition is a minimum the equilibrium state is stable

Q.30) Write potential energy equation for cantilever beam

$$\Pi = EI/2 \int (d^2y/dx^2) dx - \text{pie}$$

Q.31) Mention 2 different methods to approach the model of physical system .

Discrete system , continuum

Q.32) Difference between global co ordinate and local co ordinate

| <u>Global co-ordinate</u> | <u>Local co ordinate</u> |
|---------------------------------|------------------------------|
| 1. Contain 2 degrees of freedom | Contain 1 degrees of freedom |
| 2. Represented without prime | Represented with prime |
| 3. Systematic schemes are used | No schemes are used |

Q.33) What is local coordinate

local coordinate contains 1-D.O.F. at each node.

Q.34) What is global co ordinate

Global coordinate contains 2-D.O.F. at each node.

Q.35) General assumption made in stress

Assumptions:

- 1.Truss elements are connected by fracture less pin.
- 2.Load is applied on the load.
- 3.Only two forces compressive and tensile are considered.

Q.36)What is shape function

It is mathematical polynomial, which gives displacement within the element.

Q.37)What are two general natural coordinate

Two general co-ordinates are η (eta) and ζ .

Q.38) Mention the range of natural co-ordinate

Range of natural coordinates is -0 to 1 and -1 to 1 .

Q.39) Number of shape function in CST

Number of shape function in CST are three.

Q.40) Number of shape function in quadrilateral

Number of shape function in quadrilateral are 4.

Q.41) Why we are using natural integration

we are using natural integration to simplify the problem.

Q.42) Explain one point formula

1-point shape function contains one term $\int f(x)dx=w.f(x)$

Q.43) Explain two point formula

2-point shape function contains two term $\int f(x).dx=w_1f(x_1) w_2f(x_2)$

Q.44) Why we are using polynomial equation in FEA

Polynomial equation gives continuous solution & it is simple to solve problem.

Q.45) What are the two important characteristics in stiffness matrix

Characteristics of stiffness matrix is symmetric and bonded

Q.46) Mention two schemes to represent band width

2-Schemes are

Horizontal numbering

Vertical numbering

Q.47)What are forces involved in work potential

Forces involved are Body force, Traction force and Point force

Q.48)What are the different methods to apply boundary condition

Elimination, multiconstraint, penalty method

Q.49) What is isoparametric elements

These are those the S.F. used to define variables of displacement equal to S.F. used to represent geometry.

Q.50) What is orthotropic elements

Material which has three orthogonal planes of symmetry said to be orthotropic elements. Only nine constants are required to describe constituent equation

Q.51) What is anisotropic elements

The material which doesn't contain any plane of symmetry

Q.52) What is isotropic elements

isotropic material is one in which every plane is plane of symmetry only two constants are enough to describe constituent equation

Q.53) What is super parametric elements

GSF > DSF (geometric shape function, displacement shape function)

Q.54) What is sub parametric elements

GSF < DSF

Q.55) Different coordinates involved in chain rule.

Different coordinates involved in chain rule are normal , local and displacement coordinates .

Q.56) What are the 2 different approaches to study elasticity ?

Strength of material , Theory of elasticity

Q.57) Mention any two methods to solve continuum problems.

1. Raleighritz method
2. Galerkin method

Q.58) List the properties of shape functions.

SF for 1D bar element $N_1=0, N_2= 0$ at node 1

$N_1=0 N_2= 1$ at node 2 . Diff. Of S.F. S.F are constant

Q.59) Define truss.

Structural member which is subjected to either tensile or compression

Q.60)What is weighted residual methods.

It's a method in FEA for accurate solution avoiding error
(residue = error)

Q.61)Different methods to solve weighed residual problem.

Point allocation, Sub domain , Galerkein,least square.

Q.62)Explain the principle of virtual work.

If the force and displacement are unrelated by cause effect relation then the work is said to be virtual work.

Q.63) Explain the principle of virtual displacement.

Actual displacement is considered without bothering amount of force is called virtual displacement .

Q.64)What are different types of shape functions.

Diff types are Lagrangian and Hermite Shape function

Q.65) Differentiate two types of shape functions.

Lagrangian shape function only for variable

Hermite shape function is for both variable and its derivative

Q.66) Mention some advantages of FEA over solid mechanics.

1. Applied for complicated structure

2. Analysis is simple

3. More accurate solution

Q.67) Define Young's Modulus and Poisson's Ratio.

E - it is the ratio of stress and strain

μ - It is the ratio of lateral strain and longitudinal strain

Q.68) Mention different types of elastic constants.

Young's modulus, shear modulus, bulk modulus

Q.69)Specify the terms required to solve FEA problem.

Meshing , properties of material, boundary condition and initial condition.

Q.70) What are the assumptions made in linear static problems.

all displacement is small, material is isotropic, linear, elastic solid with E and μ

Q.71) Which is the most accepted form of numerical integration in FEM?

Gaussian Quadrature

Q.72)List the different approaches to derive integral equation

Direct method , variation method, weighted residual method, energy method

Q.73)What are the advantages of symmetrical matrix.

symmetrical matrix simplifies the calculation

Q.74) What are the different types of errors in FEA?

Modeling error , Discretized and Numerical error

Q.75) What are the advantages of isoparametric elements

Useful in modeling structure with curved edges

They are versatile & they are used in 2-D and 3-D elasticity problems

Q.76). Define frontal method for finite element matrices.

In 3- dimensional problems, the size of the stiffness matrix increases rapidly even with the banded method of modeling. An alternative direct method that results in considerable saving in the use of computer memory is called frontal method.

Q.77) What is the another name of the 3-dimensional frames
Space frames.

Q.78). Define beam elements.
Beam elements are slender members that are used for supporting transverses loading.

Q.79) Explain preprocessor steps.
Determining the Nodal coordinates, connectivity, boundary condition, material information

Q.80) Explain processing steps.
Stiffness generation, modification, solutions to the equation resulting in evolution.

Q.81) Explain post processing steps.
Deformation confirmation, mode shapes, temperature and stress distribution, interpretation

Q.82) What are the difference b/w beams and plane frames.
It is similar to beams expect that axial loads & axial deformations are present. The elements also have different orientation.

Q.83) Mention some common material properties.
Isotropic, orthotropic, ductility, brittleness

Q.84) What are the different types of analysis.
Thermal, structural (load), fluid, electromagnetic analysis

Q.85) Define steady state analysis.
The analysis carried out at constant temperature.

Q.86) What is the advantage of subjecting solids to axisymmetric loading.
Axisymmetric loading reduces the 3-D problem into 2-D problems because of total symmetry about the z-axis.

Q.87) Define CST elements.
The constant strain triangle is that where the displacement inside an element is represented by 3-nodal displacement (3-shape functions).

Q.88) How to generate the data files for larger problems in FEA.
Using MESHGEN program generates data files.

Q.89) Define mesh plotting
It is the convenient way of reviewing the coordinate and connectivity data is by plotting it using computer.

Q.90) Explain lumped mass matrices.
It is the total element mass in each direction is distributed equally to the nodes of the element, and the masses are associated with translational degrees of freedom.

Q.91) Briefly explain steps involved in Lagrangian method

-
- Formulation of potential energy function.
assuming displacement function
 - Checking displacement function considering boundary condition
 - Substitute differential function in potential energy equation
 - Potential energy function is minimized
 - Unknown parameters are determined and substituted in assumed equation

Q.92) Explain steps involved in Galerkins method.

Formulate differential equation of the equilibrium

Assume trial function, which satisfies boundary conditions

Substitute displacement function in differential equation then assume the difference due to approx. function be 'R' (residue)

Use gallerkin formula

Determine unknown terms and then substitute in differential function

Q.93) Define jacobian matrix.

The matrix which is defined explicitly in terms of the local coordinate is known as jacobian(J).

Q.94) Mention six components of stress.

3 linear stress along x, y, z direction

3 lateral stress along x, y, z direction

Q.95) Mention six components of strain.

3 linear strain along x, y, z direction

3 lateral strain along x, y, z direction

Q.96) Define Winkler foundations.

Large beams are supported on soil form a class of applications known as winkler foundations.

Q.97) Define variational principle

The problem which specifies a scalar quantity potential energy is defined in an integral form.

Q.98) How to solve the prismatic problems.

The coefficient of the ordinary differential equation are independent of one of the coordinate and the solution of the system can frequently carried out efficiently by standard analytical methods.

Q.99) Define stress & strain.

Stress is the ratio of applied load to its area.

Strain is the ratio of change in length to its original length.

Q.100) Mention the two distinct procedures available for obtaining the approximation in the integral forms.

Method weighed residuals.

Method of variation functional.