JAIPUR INSTITUTE OF TECHNOLOGY-
GROUP OF INSTITUTIONS
JAIPUR

FINITE ELEMENT METHODS LAB MANUAL

DEPARTMENT OF MECHANICAL ENGINEERING

INDEX

7ME8A: FINITE ELEMENT LAB.

B.Tech. (Mechanical) 7th Semester — **Max. Marks: 100**

0L+0T+3P — **Exam Hours: 3**

SN	LABORATORY WORK/NAME OF EXPERIMENT	CONTACT HOURS
1	Laboratory work for the solution of solid mechanics problems, heat transfer problems, and free vibration problems **A: by using FE packages such as NASTRAN/ ANSYS/ SIMULIA/ ABAQUS**	
2	Introduction of GUI of the software in the above mentioned areas realistic problems.	
3	Analysis of beams and frames (bending and torsion problems)	
4	Plane stress and plane strain analysis problems	
5	Problems leading to analysis of axisymmetric solids	
6	Problems leading to analysis of three dimensional solids (a) Heat transfer problems (b) Modal analysis problem	
	B: by writing own code for finite element analysis using MATLAB for:	
7	Plane stress and plane strain analysis problems	
8	Modal Analysis problem	

Introduction to ANSYS

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

- ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering:
  - Structural
  - Thermal
  - Fluid (CFD, Acoustics, and other fluid analysis)
  - Low – and High – frequency electromagnetics
- A partial list of industries in which ANSYS is used:
  - Aerospace – electronics & Appliances
  - Automotive – Heavy equipment & Machinery
  - Biomedical – MEMS – Micro electromechanical systems
  - Bridges & Buildings
- 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
  - ANSYS Professional – linear structural and thermal analysis, a subset of ANSYS Mechanical capabilities in the workbench environment

- **Structural Analysis:** is used to determine deformations, strains, stresses, and reaction forces.
- **Static Analysis:**
  - Used for static loading conditions.
  - Nonlinear behavior such as large deformations, large strain, contact, plasticity, hyper elasticity, and creep can be simulated.
- **Dynamic Analysis:**
  - Includes mass and damping effects.
  - *Modal analysis* calculated natural frequencies and mode shapes.
  - *Harmonic analysis* determines a structure's response to sinusoidal loads of known amplitude and frequency.
  - *Transient dynamic analysis* determines a structure's response to time-varying loads and can include nonlinear behavior.
- **Other structural capabilities**
  - Spectrum analysis
  - Random vibrations
  - Eigen value buckling
  - Sub-structuring, sub-modeling

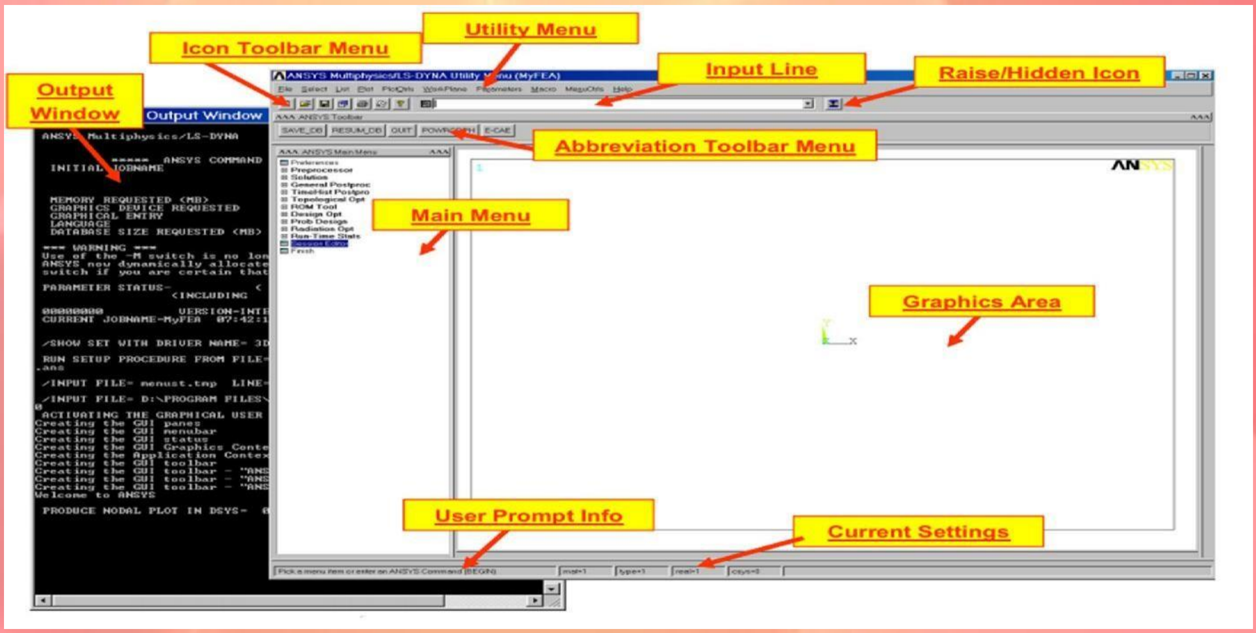
- **Explicit Dynamics with ANSYS/LS – DYNA:**
  - Intended for very large deformation simulations where inertia forces are dominant.
  - Used to simulate impact, crushing, rapid forming, etc.
- **Thermal analysis:** is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux. All three primary heat transfer modes can be simulated: Conduction, Convection, and 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 electromagnetic:
  - To simulated 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.
  - Application: aerospace, electronic packaging, automotive design.
  - Typical quantities of interest are velocities, pressure, temperature, and film coefficients.

ANSYS

Opening ANSYS session:

ANSYS can be opened in windows operating system through

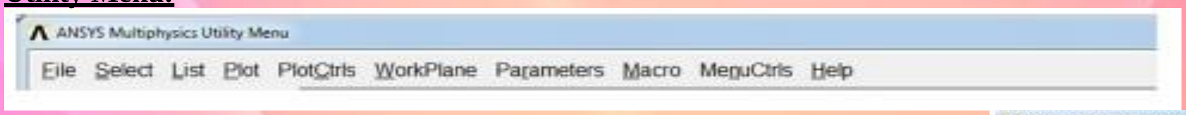
- **Start > programs > ANSYS 14.0 > Mechanical APDL (ANSYS) 14.0 ANSYS Menu:**



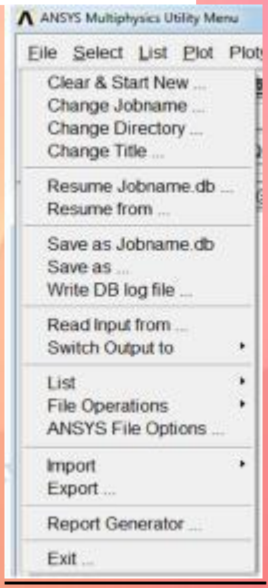
By default ANSYS opens 6 Menus. They are:

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

Utility Menu:



a) File:



The file contains

- Clear & Start: To clear the database & Start a new job
- Resume from: To resume the previously stored job
- Save as: Save the database as filename.db
- Read Input from: If input is taken from outside programmed file
- Switch Output: To external file or by default files in *.iges format is supported without any additional software. By CATIA, UG, PRO-E. You can import the geometry
- Export: To export to use in other software's.
- Exit: To close the ANSYS session.

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

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

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

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

e) Plot Controls: This option is very important and contains

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

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

g) Parameters: These are the scalar parameters represented with values.

h) Macros: These are grouping of ANSYS commands to fulfill particular work. These can be taken equivalent to C, C++ & Java functions.

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

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

Main Menu: This menu contains

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

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

Tool Bar: This contains options like saving the file, resuming the file database, quitting the ANSYS session and graphics type.

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

<u>Output Window:</u> This shows the status of the work being carried out.

<u>WORKPLANE:</u>

Although the cursor appears as a point on the screen, it actually represents a line through space, normal to the screen. In order to define an imaginary plane that, when interested by the normal line of the cursor, will yield a unique point in space. This imaginary plane is called a working plane.

Working plane is an infinite plane with an origin, a 2D coordinate system, a snap increment and a display grid. You can define only one working plane at a time. (Creating a new working plane eliminates the existing working plane). The working plane is separate from the coordinate systems; for example; the working plane can have a different plane of origin and rotation than the active coordinate system. Work plane can be positioned wherever required and model can be created.

<u>COORDINATE PLANE:</u> The ANSYS program has several types of coordinate systems, each used for a different reason:

- Global and local coordinates systems are used to locate geometry items (Nodes, Keypoints, etc.) in space.
- The display coordinate system determines the system in which geometry items are listed or displayed.
- The nodal coordinate system defines the degree of freedom directions at each node and the orientation of nodal results data.
- The element coordinate system determines the orientation of material properties and element results data.
- The results coordinate system is used to transform nodal or element results data to a particular coordinate system for listings, displays, or general post processing operations (POST1).

<u>SCALAR PARAMETERS:</u> These are useful to change the model dimensions at any time. These are useful when macros or batch programs are coded. For example in b = 10, b is considered as scalar parameter. For optimization the model should be represented in scalar parameters. Here is another way to set parameters is *b = 10 and can be changed any time.

MACROS: These are grouping of commands for particular purpose. These are equivalent to functions in C and sub – routines in FORTRAN. They are very powerful and are based on APDL (ANSYS Parametric Design Language). To get expertise with ANSYS, one should be thorough with usage of macros.

MODELING: This is the important step of creating the physical object in the system. They are two types of modeling in ANSYS.

    - Direct Modeling
    - Solid Modeling

- **DIRECT MODELING:** In this approach the physical structure is represented by nodes and elements directly. The problem is solved once after the boundary conditions are applied. This approach is simple and straightforward. Takes very little time computation. But this can be applied only for simple problems. When problem becomes complex, this method becomes tedious to apply.
- **SOLID MODELING:** Models are directly created either using ANSYS preprocessor or imported from popular CAD softwares like Mechanical desktop, ProE, CATIA, SOLID WORKS, etc. Once the structural model created, by using mesh tool, the model can be meshed and problem can be solved by applying the boundary conditions. In ANSYS solid modeling is carried out using two methods:
    - *Bottom up Approach:* To create model, Entities are required. Keypoints, lines, Areas, Volumes are the entities in ANSYS. If model is constructed through keypoints to lines, from lines to areas, and from areas to volumes the approach of modeling is called bottom up approach. This approach is useful when models are complex.
    - *Top Down Approach:* A 3D model can be created directly using the volumes. Once volumes are created, all the entities below the volumes (areas, lines, Keypoints) are automatically created. This approach is easy but can be applied to simple problems.

ELEMENTS: Elements are FE representation of physical structures or discretized parts of the continuum. These elements are like functions designed for a specific purpose. For example bar element can take only axial compressive or tensile loads. And a truss element can take only horizontal and vertical loads in the global directions. So, a truss element cannot take any transverse loading across the element or a moment. So, proper element should be selected based on the problem and loading. Usually the number of elements of its library measures capacity of a software. ANSYS contains more than 200 elements designed for specific purposes.

GRAPHICS DISPLAY: There are two methods available for graphic display.

- Full mode display: This option can be used with all elements.
- Power Graphics: Power graphics method is the default when ANSYS GUI is on. This method is valid for all the element types except for circuit elements. Power graphics method offers significantly faster performance than the full mode method.

ELEMENT TABLE: The primary data results are directly available for all elements in post processor. The secondary data or derived data (Stresses, strains, Von misses stress, principal stress, etc.) is available only for solid elements. The problems where solid model is created and meshed directly through nodal solution results in the post processor, but not available to line elements like (beam, link, etc.). To get the secondary data for line elements, we need to define the element table for the particular element to get the required data. For example to get axial stress for the link element, you must go to ANSYS help, type link1 and see the Link1 definitions and sequence no. for link1. Through the post processor you have to create element table > define > by sequence no. – LS1 and Plot > element table > LS1 gives the axial stress for the problem.

PICKING & PLOTTING

In this course you will be using geometry entities such as volumes, areas, lines and Keypoints as well as FEA entities such as nodes and elements. This topic introduces the following techniques used to display and manipulated those entities within GUI

- Plotting
- Picking
- Select Logic
- Components and assemblies

PLOTTING:

It is often advantages to plot only certain entities in the model. Within the Utility Menu > Plot, you will see that geometric, finite element and other entities can be plotted.

With Multi-plots, a combination of entities can be plotted.

The **plotCtrls** menu is used to control how the plot is displayed:

- Plot orientation
- Zoom
- Colors
- Symbols
- Annotation
- Animation
- Etc.

Among these, changing the plot orientation (/view) and zooming are the most commonly used function.

Use Dynamic mode – a way to orientation dynamically using the control key and mouse buttons.

-Ctrl + Left mouse button pans the model.

-Ctrl + Middle mouse button: Zoom the model.
-Ctrl + Right mouse button rotates the model:

- About screen X
- About screen Y

Note, the shift Right button on a two – button mouse is equivalent to the middle mouse button on a three – button mouse.

Use the model control toolbar icons to change view. The model control toolbar also includes a dynamic rotate option.

PICKING:

- Picking allows you to identify model entities or locations by clicking in the graphics window.
- A picking operation typically involves the use of the mouse and a picker menu. It is indicated by a + sign on the menu.
- For example, you can create keypoints by picking locations in the graphics window and then pressing OK in the picker.

Two types of picking:

- **Retrieval Picking**
  - o Picking existing entities for a subsequent operation.
  - o Allows you to enter entity number in the picker window.
  - o Use the pick *All* button to indicate all entities.
- **Locational picking**:
  - o Locating coordinates of a point, such as a keypoint or node.
  - o Allows you to enter coordinates in the picker window.
- Note, you must hit the <Enter> key, after entering information in the picker window, then hit [OK] or [Apply].

Mouse button assignments for Picking:

- **Left** mouse button pick (or unpicks) the entity or location closet to the mouse pointer. Pressing and dragging allows you to ―preview‖ the item being picked (or Unpicked).
- **Middle** mouse button does apply. Saves the time required to move the mouse over to the picker and press apply button. Use shift – right button on a two – button mouse.
- **Right** mouse button toggles between pick and unpick mode.
  Note, the shift – Right button on a two – button mouse is equivalent to the middle mouse button on a three – button mouse.

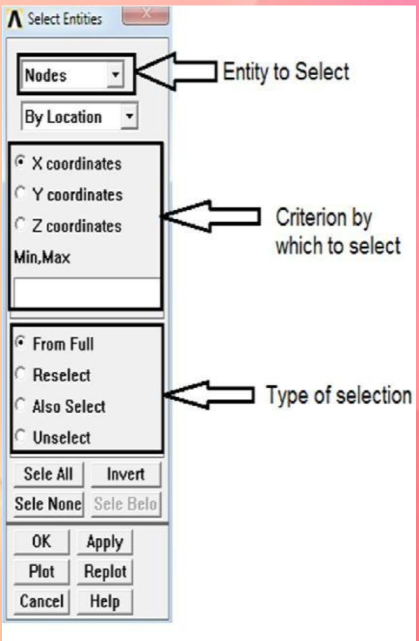
Hotspot locations for picking:

- **Areas and volumes** have one hotspot near the centroid of the solid model entity.
- **Lines** have three hotspots – one in the middle and one near each end.
- **Why this is important:** when you are required to ―pick‖ an entity, you must pick on the hotspot.
- **Note:**
  - o Show locational picking by creating a few keypoints. Also show the use of middle and right mouse buttons.
  - o Show retrieval picking by creating a few lines.
  - o Show ―loop‖ by creating an All area
  - o Show ― pick all‖ by deleting area only
  - o Do KPLOT, LPLOT, etc. with and without numbering. Type in a few of these commands.
  - o Show the use of Pan – zoom – Rotate

- **Suppose you wanted to do the following:**
  - o Plot all areas located in the second quadrant
  - o Delete all arcs of radius 0.2 to 0.3 units
  - o Apply a convection load on all exterior lines o Write out all nodes at Z = 3.5 to a file
  - o View results only in elements made of steel

  The common –theme‖ in these tasks is that they all operate on a subset of the model.

- **Select Logic** allows you to select a subset of entities and operate only on those entities.
- **Three steps:**
  - o Select a subset
  - o Perform operations on the subset
  - o Reactivate the full set



__Selecting Subsets__

- Most selecting tools are available in the select entities dialog box:
  Utility Menu > Select > Entities….
- Or you can use the xSEL family of commands: KSEL, LSEL, ASEL, VSEL, NSEL, ESEL

- **Criterion by which to select:**

  o By **Num/Pick:** to select based on entity numbers or by picking

  o **Attached to:** to select based on attached entities. For example, select all lines attached to the current subset of areas.

  o **By location:** to select based on X,Y,Z location. For example, select all nodes at X = 2.5. X,Y,Z are interpreted in the active coordinate system.

  o **By Attributes:** to select based on material number, real constant set number, etc. Different attributes are available for different entities. o **Exterior:** to select entities lying on the exterior.

  o **By results:** to select entities by results data, e.g., nodal displacements.
- **Type of selection:**
  - o **From full:** select a subset from the full set of entities.
  - o **Reselect:** selects (again) a subset from the current subset.
  - o **Also select:** adds another subset to the current subset.
  - o **Unselect:** deactivates a portion of the current subsets.
  - o **Invert:** toggles the active and inactive subsets.
  - o **Select None:** deactivates the full set of entities.
  - o **Select all:** reactivates the full set of entities.

Operations of the subset

- Typical operations are applying loads, listing results for the subset, or simply plotting the selected entities.
    - The advantages of having a subset selected are that you can use the [Pick all] button when the picker prompts you pick desired entities. Or you can use the ALL label when using commands.
    - Note that most operations in ANSYS, including the SOLVE command, act on the currently selected subset.
- Another ―operation‖ is to assign a name to the selected subset by creating a component.

Reactivating the full set

- After all desired operations are done on the selected subset, you should reactivate the full set of entities.
    - If all nodes and all elements are not active for solution, the solver will issue a warning to that effect.
- The easiest way to reactivate the full set to select ―everything‖:
    - Utility Menu > Select > Everything – or issue the command ALLSEL

You can also use the [Sele All] button in the select entities dialog box to reactive each entity set separately. [Or issue KSEL, ALL; LSEL, ALL; etc.]

COMPONENTS

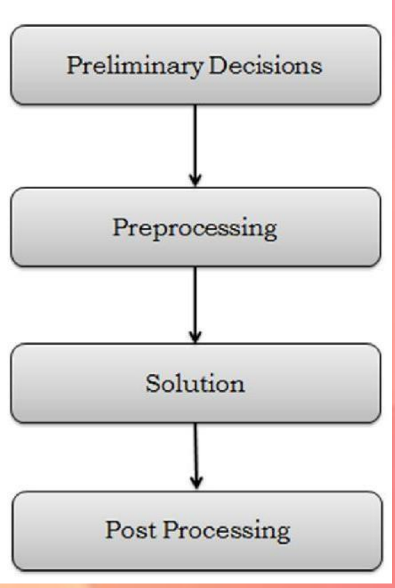
- **Components** are user – named subsets of entities. The name can then be used in dialog boxes or commands in place of entity numbers or the label ALL.
- A group of nodes, or elements, or Keypoints, or lines, or areas, or volumes can be defined as a component. Only one entity type is associated with a component.
- Component can be selected or unselected. When you select a component, you are actually selecting all of the entities in that component.
- **Component Manager** is used to create, Display, List and Select components and Assemblies.
    - **Utility Menu > Select > Component Manager….**
- **Creating a component**
    - **Utility Menu > Select > Component Manager – Click on the Create Component Icon**
- All of the currently selected entities will be included in the component, or you can select [pick] the desired entities at this step.
- **Enter a name**
    - Up to 32 characters – letters, numbers, and _ [underscore] – are allowed.
    - Beginning a component with _ [underscore] will make it a ―hidden component‖ and it cannot be picked from the list.
      This is not recommended.
    - Suggestion: Use the first letter of the name to indicate the entity type. For example, use N_HOLES for a node component.

- **Creating Assembly**
  - Highlight the components for the assembly
  - Click on the create assembly Icon and enter a name
  - Checking the box next to a component under the assembly number will also put a component in an assembly.

GENERAL PROCEDURE IN FEM

The objective of this chapter is outline of general analysis procedure to be used to solve a simulation. Regardless of the physics of the problem, the same general procedure can be followed.

Every analysis involves four main steps:



- **Preliminary Decisions**
  - Which analysis type?
  - What to model?
  - Which element type?
- **Preprocessing**
  - Define Material
  - Create or import the model geometry
  - Mesh the geometry
- **Solution**
  - Apply loads
  - Solve
- **Postprocessing**
  - Review results
  - Check the validity of the solution

Which analysis type?

The analysis type usually belongs to one of the following disciplines:

Structural : Motion of solid bodies, pressure on solid bodies, or contact of solid bodies.

Thermal : Applied heat, high temperature, or changes in temperature

Electromagnetic : Devices subjected to electric currents (AC or DC), Electromagnetic waves and voltage or charge excitation.

Fluid : Motion of gases/fluids, or contained gases/fluids

Coupled – field : Combination of any.

PERFORMING A TYPICAL ANSYS ANALYSIS

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

Build the model.

Apply loads and obtain the solution.

Review the results.

Building a Model

Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a job name and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.

Specifying a Job name and Analysis Title

This task is not required for an analysis, but is recommended.

Defining the Job name

The job name is a name that identifies the ANSYS job. When you define a job name for an analysis, the job name becomes the first part of the name of all files the analysis creates. (The extension or suffix for these files' names is a file identifier such as .DB.) By using a job name for each analysis, you insure that no files are overwritten. If you do not specify a job name, all files receive the name *FILE* or *file*, depending on the operating system.

Command(s): **/FILNAME**

GUI: **Utility Menu>File>Change Job name**

Defining Element Types

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID96, etc. The following element categories are available

The element type determines, among other things:

The degree-of-freedom set (which in turn implies the discipline-structural, thermal, magnetic, electric, quadrilateral, brick, etc.)

Whether the element lies in two-dimensional or three-dimensional space.

For example, BEAM4, has six structural degrees of freedom (UX, UY, UZ, ROTX, ROTY, ROTZ), is a line element, and can be modeled in 3-D space. PLANE77 has a thermal degree of freedom (TEMP), is an eight-node quadrilateral element, and can be modeled only in 2-D space.

Defining Element Real Constants

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. For example, real constants for BEAM3, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS). Not all element types require real constants, and different elements of the same type may have different real constant values.

As with element types, each set of real constants has a reference number, and the table of reference number versus real constant set is called the *real constant table*. While defining the elements, you point to the appropriate real constant reference number using the **REAL** command

(**Main Menu> Preprocessor>Create>Elements>Elem Attributes**).

Defining Material Properties

Most element types require material properties. Depending on the application, material properties may be:

Linear or nonlinear

Isotropic, orthotropic, or anisotropic

Constant temperature or temperature-dependent.

As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the *material table*. Within one analysis, you may have multiple material property sets (to correspond with multiple materials used in the model). ANSYS identifies each set with a unique reference number.

Main Menu > Preprocessor> Material Props > Material Models.

Creating the Model Geometry

Once you have defined material properties, the next step in an analysis is generating a finite element model-nodes and elements-that adequately describes the model geometry.

There are two methods to create the finite element model: solid modeling and direct generation.

With solid modeling, you describe the geometric shape of your model, and then instruct the ANSYS program to automatically mesh the geometry with nodes and elements. You can control the size and shape of the elements that the program creates. With *direct generation*, you "manually" define the location of each node and the connectivity of each element. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc. are available.

Apply Loads and Obtain the Solution

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7 preprocessor.

Applying Loads

The word loads as used in this manual includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads. Loads in the ANSYS program are divided into six categories:

DOF Constraints

Forces

Surface Loads

Body Loads

Inertia Loads

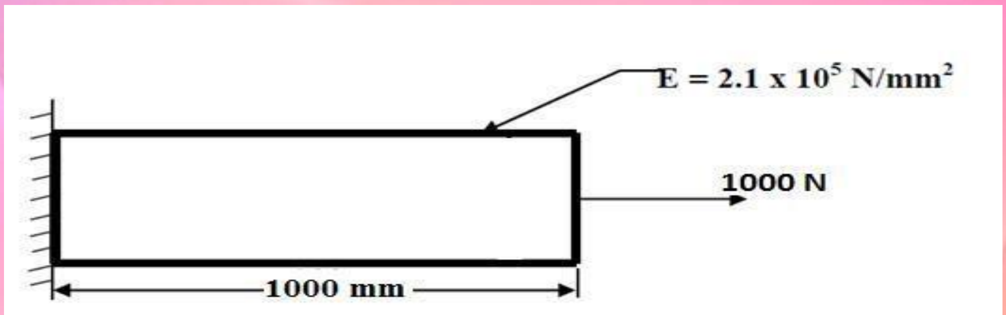
Coupled-field Loads

You can apply most of these loads either on the solid model (key points, lines, and areas) or the finite element model (nodes and elements).

Two important load-related terms you need to know are load step and sub step. A *load step* is simply a configuration of loads for which you obtain a solution. In a structural analysis, for example, you may apply wind loads in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

Problem 1.1: Bar of Constant Cross-section Area

Consider the bar shown in figure below. Young's modulus is $2.1\times10^5 N/mm^2$ and Area is $500mm^2$. Determine the Nodal Displacement, Stress in each element, Reaction forces.



1. Ansys Main Menu – Preferences-Select – STRUCTURAL – h method- ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finite stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no 1 – c/s area – 500 – ok.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5 – PRXY – 0.27 – ok – close.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 1000 (x value w.r.t first node) – ok (second node is created).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –DOFs to be constrained – All DOF – ok. Loads –
8. Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 1000 (+ve value) – ok.
9. Solve – current LS – ok (Solution is done is displayed) – close.
10. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
11. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
12. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
13. Plot results- nodal solution-ok-DOF solution- x component of displacement-ok.
14. Animation: Plot Ctrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:
Analytical approach:

Calculation:

Displacement:______________________

Stress:____________________________

Reaction force: ____________________

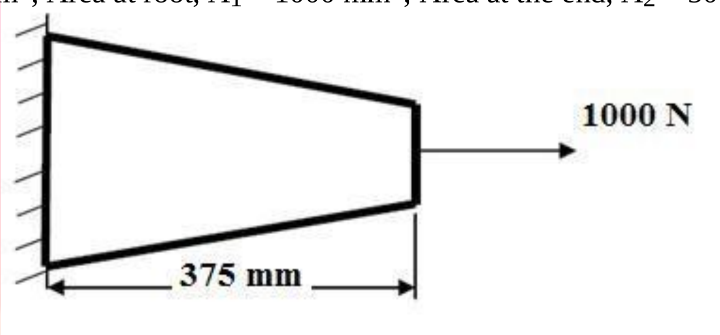
Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

Problem 1.2: Bars of Tapered Cross section Area

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

$E = 2 \times 10^5$ N/mm^2, Area at root, $A_1 = 1000$ mm^2, Area at the end, $A_2 = 500$ mm^2.



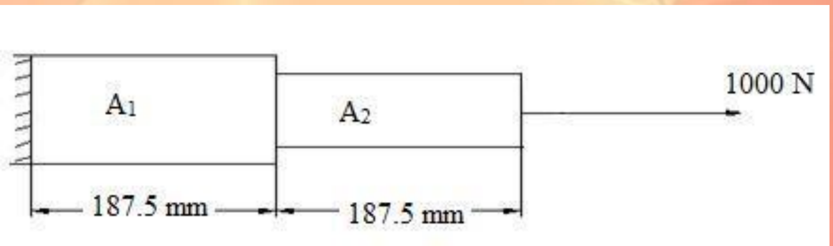
Solution: The tapered bar is modified into 2 elements as shown below with modified area of cross section.

$(A_1 + A_2)/2= (1000+500)/2=750$ mm^2

$A_1 = (1000+750)/2= 875$ mm^2

$A_2= (500+750)/2=625$ mm^2

$L_1 = 187.5$ mm & $L_2 = 187.5$ mm



1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method– ok
2. Element type – Add/Edit/Delete – Add – link, 3D Finit stn 180 – ok- close.
3. Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA1 – 875 – apply-ok
4. Add – ok – real constant set no – 2 – cross-sectional AREA 2 – 625-ok
5. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 –PRXY – 0.3 – ok – close.
6. Modeling – Create – keypoints– In Active CS, =0, Y=0 – Apply (first key point is created) – location in active CS, X= 187.5, Y=0, apply (second key point is created) - location in active CS X=375, Y=0(third key point is created) -ok.
7. Modeling-Create – lines-straight lines-pick key points 1 & 2-ok- pick key points 2 & 3-ok
8. Meshing-mesh attributes-picked lines (pick the lines)-ok-material no= 1, real constants set no = 1, element type no =1, link 1, element section= none defined-pick the other line-ok-material number 2-define material id 2- real constants set no = 2, element type no =2-element section= none defined-ok.
9. Meshing-size controls-manual size-lines-all lines- no of element divisions=10(yes)-ok

10. Meshing-mesh tool-mesh-pick the lines-ok (the color changes to light blue)
11. Loads – Define loads – apply – Structural – Displacement – on key points- pick key point 1 – apply –DOFs to be constrained – ALL DOF, displacement value=0 – ok.
12. Loads – Define loads – apply – Structural – Force/Moment – on key points- pick last key point – apply – direction of For/Mom – FX – Force/Moment value – 1000 (+ve value) – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
15. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
16. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. Plot results- nodal solution-ok-DOF solution- x component of displacement-ok.
18. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

Analytical approach:

Calculation:

Displacement:______________________

Stress:____________________________

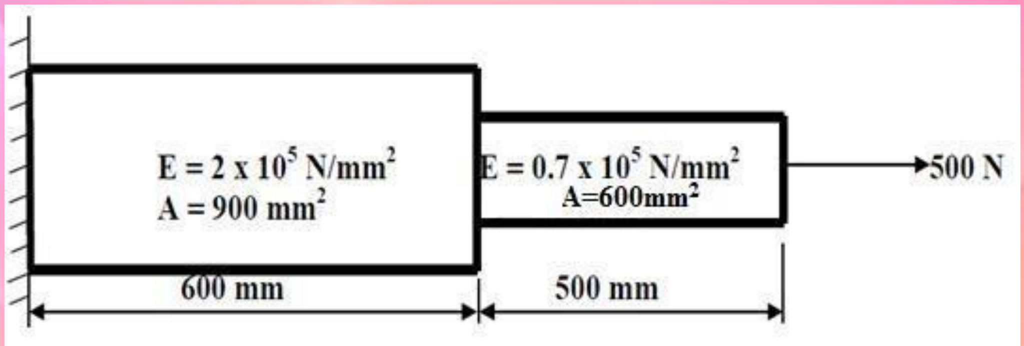
Reaction force: _____________________

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

Problem 1.3: Stepped Bar

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



1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – link, 3D Finit stn 180 – ok- close.
3. Real constants – Add – ok – real constant set no – 1 – cross-sectional AREA 1 – 900 – apply-ok
4. Add – ok – real constant set no – 2 – cross-sectional AREA 2 – 600-ok
5. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 –PRXY – 0.3- material- new material-define material id=2- Structural – Linear – Elastic – Isotropic – EX – 0.7e5 –PRXY – 0.3– ok – close.
6. Modeling – Create – key points– In Active CS, =0, Y=0 – Apply (first key point is created) – location in active CS, X= 600, Y=0, apply (second key point is created) - location in active CS X=1100, Y=0(third key point is created) -ok.
7. Modeling-Create – lines-straight lines-pick key points 1 & 2-ok- pick key points 2 & 3-ok
8. Meshing-mesh attributes-picked lines (pick the lines)-ok-material no= 1, real constants set no = 1, element type no =1, link 1, element section= none defined-pick the other line-ok-material number 2-define material id 2- real constants set no = 2, element type no =2-element section= none defined-ok.
9. Meshing-size controls-manual size-lines-all lines- no of element divisions=10(yes)-ok
10. Meshing-mesh tool-mesh-pick the lines-ok (the color changes to light blue)
11. Loads – Define loads – apply – Structural – Displacement – on key points- pick key point 1 – apply –DOFs to be constrained – ALL DOF, displacement value=0 – ok.
12. Loads – Define loads – apply – Structural – Force/Moment – on key points- pick last key point – apply – direction of For/Mom – FX – Force/Moment value – 500 (+ve value) – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
15. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
16. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. Plot results- nodal solution-ok-DOF solution- x component of displacement-ok.
18. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

Analytical approach:

Calculation:

Displacement:________________________

Stress in each element:________________

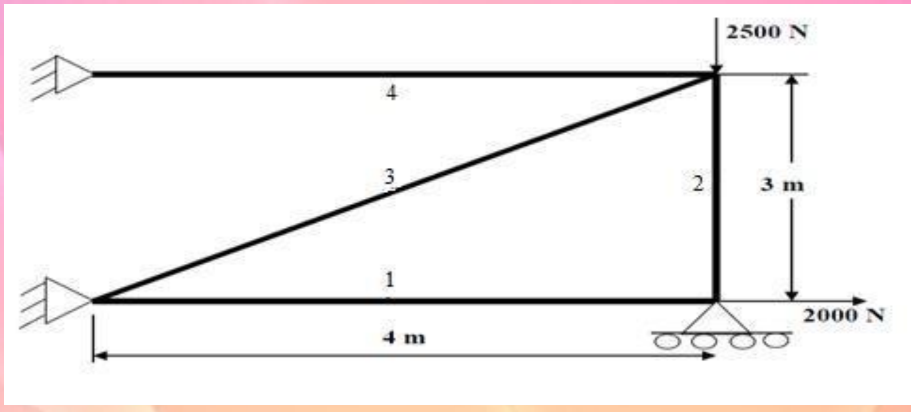
Reaction force: ______________________

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

2. TRUSSES

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



1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– Ok – close.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 4 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 4, 3 (x, y value w.r.t first node) – apply (third node is created) – 0, 3 (x, y value w.r.t first node) – ok (forth node is created).
6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number – 1 – ok
7. Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 1 – apply pick 3 & 4 – ok (elements are created through nodes).
8. Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 4 – apply – DOFs to be constrained – All DOF – ok – on Nodes – pick node 2 – apply – DOFs to be constrained – UY – ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok – Structural –
10. Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.
11. Solve – current LS – ok (Solution is done is displayed) – close.
12. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.
13. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.
14. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
15. Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.
16. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

Analytical approach:

Calculation:

Displacement:________________________

Stress:______________________________

Reaction force: ______________________

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

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

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

RESULT:

Analytical approach:

Calculation:

Displacement:_______________________

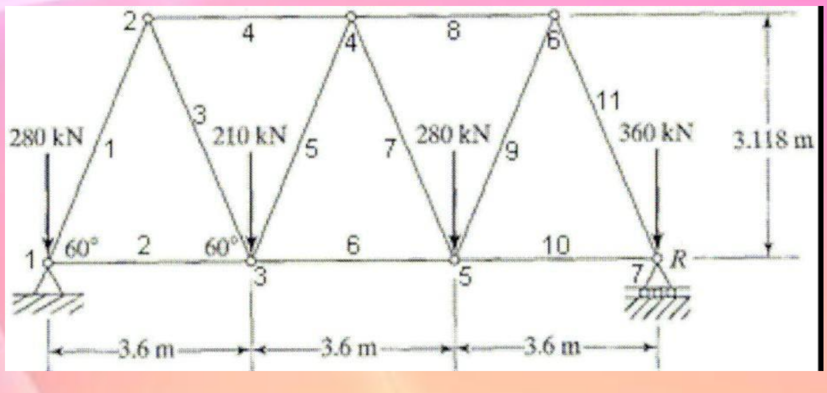
Stress:______________________________

Reaction force: ______________________

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

Problem 2.3: Determine the nodal deflections, reaction forces, and stress for the truss system shown below(E = 200GPa, A = 3250mm2).



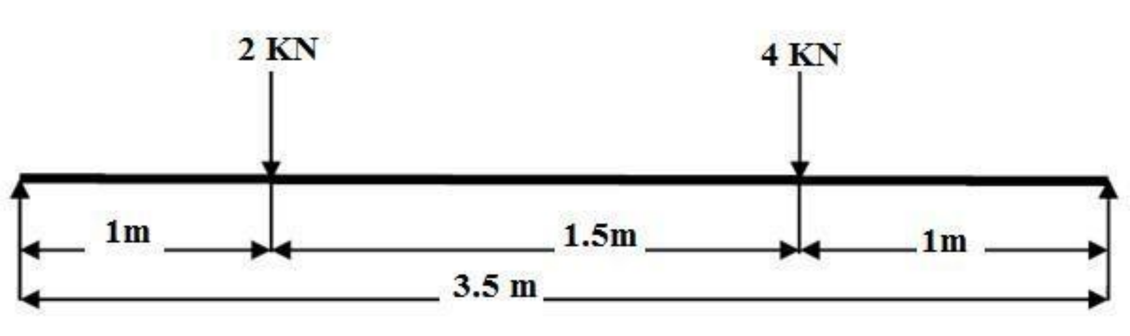
Ansys results:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

3. BEAMS

Problem 3.1: Simply Supported Beam

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



1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node BEAM 188– ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic –EX – 2.10e5– PRXY – 0.27 – ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) -enter b=100, h=100- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 1000 (x value w.r.t first node) – apply (second node is created) – 2500 (x value w.r.t first node) – apply(third node is created)- x,y,z location in CS- 3500 (x value w.r.t first node)-ok.
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 apply – pick 2 & 3 apply – pick 3 & 4 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply –DOFs to be constrained – all DOF – ok.
8. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply –direction of For/Mom – FY – Force/Moment value – -2000(-ve value) – ok- Force/Moment – on Nodes- pick node 3 – apply –direction of For/Mom – FY – Force/Moment value – -4000(-ve value) – ok.
9. Solve – current LS – ok (Solution is done is displayed) – close.
10. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
11. Stress: Plot Results – Contour plot – Nodal solution – stress – vonmises stress – ok.
12. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

13. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).
15. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
    NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19, for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.
16. Animation: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

RESULT:

Analytical approach:

Calculation:

Displacement: ________________________

Shear force: ___________________________

Bending moment: _____________________

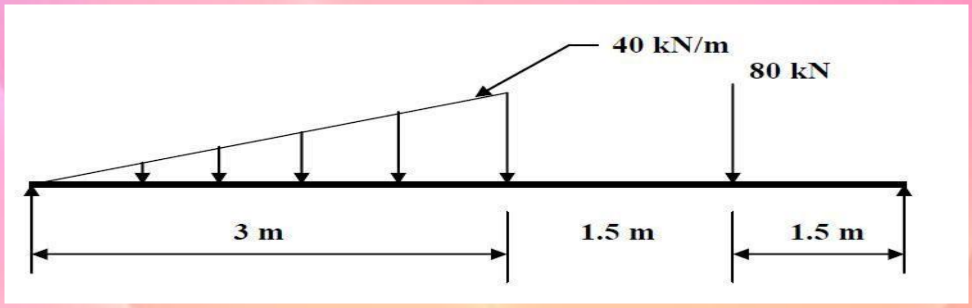
Stress:_______________________

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

Problem 3.2: Simply Supported Beam with uniformly varying load.

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100mm * 100m m, Young's modulus of 2.1×10^5 N/mm^2, Poisson's ratio= 0.27.



1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method- ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 nodes Beam 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=100, h=100- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 3000 (x value w.r.t first node) – apply (second node is created) – 4500 (x value w.r.t first node) –apply (third node is created) – 6000 (x value w.r.t first node) – ok (forth node is created).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply –pick 3 & 4 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply –DOFs to be constrained – all DOF – ok.
8. Loads – Define loads – apply – Structural – Pressure – on Beams – pick element between nodes 1 & 2–apply–pressure value at node I– 0 (value)– pressure value at node J – 40000–ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – (-80000) (-ve value) – ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).
16. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. Animation: PlotCtrls – Animate – Deformed results – DOF solution – deformed + undeformed – ok.

RESULT:

Analytical approach:

Calculation:

Deflection:______________________

Shear force: ________________________

Bending moment:___________________

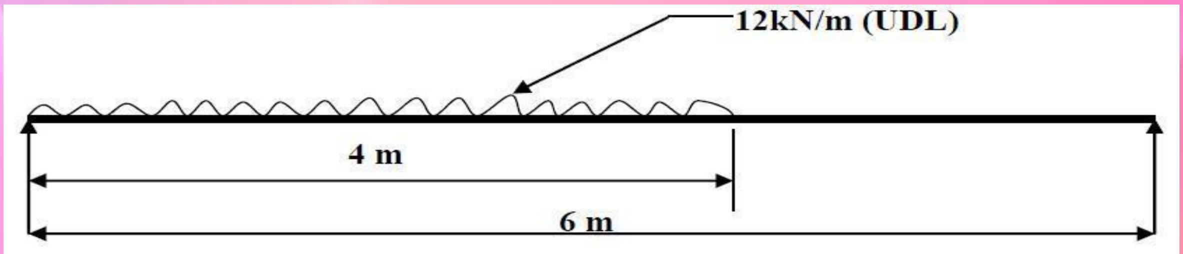
Stress:_____________________________

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

Problem 3.3: Simply Supported Beam with Uniformally distributed load.

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.1 m * 0.1 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



1. Ansys Main Menu – Preferences-select – STRUCTURAL – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188 – ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=100, h=100- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 4 (x value w.r.t first node) – apply (second node is created) – 6 (x value w.r.t first node) – ok (third node is created).
6. Create – Nodes – Fill between Nds – pick 1 & 2 – apply – number of nodes to fill 7 – startingnode no – 4 – ok.
7. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 4 apply– pick 4 & 5 apply– pick 5 & 6 apply– pick 6 & 7 apply– pick 7 & 8 apply– pick 8 & 9 apply – pick 9 & 10 apply– pick 10 & 2 apply – pick 2 & 3 – ok (elements are created through nodes).
8. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok.
9. Loads – Define loads – apply – Structural – Pressure – on Beams – pick all elements betweennodes 1 & 2 – apply – pressure value at node I – 12000 – pressure value at node J – 12000 –ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS 6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS 3 – Elem table item at node J – SMIS 16 – ok (bending moment diagram will be displayed).
16. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
17. PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

RESULT:

Analytical approach:

Calculation:

Deflection:________________________

Shear force: __________________________

Bending moment: ____________________

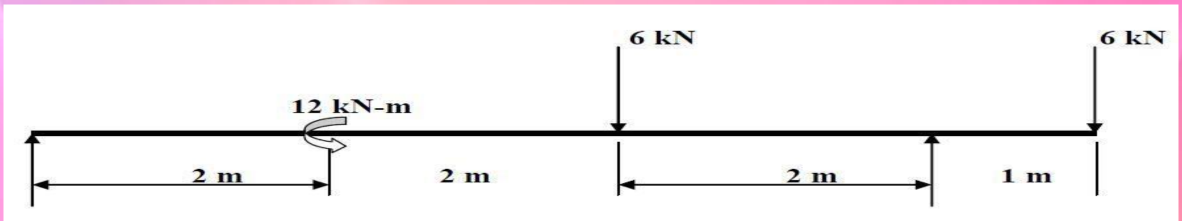
Stress:_______________________________

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

Problem 3.4: Beam with moment and overhung

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



1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method- ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1^{st} element) - enter b=200, h=300- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 2 (x value w.r.t first node) – apply (second node is created) – 4 (x value w.r.t first node) –apply (third node is created) – 6 (x value w.r.t first node) – apply (forth node is created) – 7 (x value w.r.t first node) – ok (fifth node is created).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply –pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 –apply –DOFs to be constrained – UY – ok.
8. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply direction of For/Mom – MZ – Force/Moment value - 12000 (anticlockwise, +ve value) – apply –pick node 3 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) –apply – pick node 5 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) – ok.
9. Solve – current LS – ok (Solution is done is displayed) – close.
10. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
11. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
12. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
13. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

15. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).
16. Animation: PlotCtrls – Animate – Deformed results – DOF solution – deformed + undeformed – ok.

RESULT:

Analytical approach:

Calculation:

Deflection:________________________

Shear force: __________________________

Bending moment:______________________

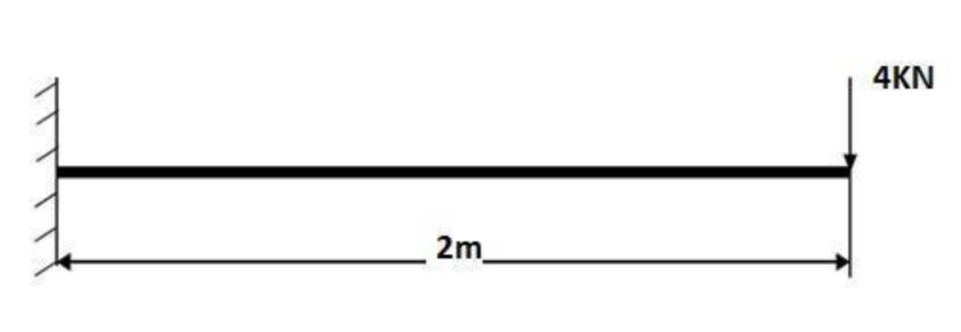
Stress: ______________________

Ansys results:

	Ansys	Theoretical
Deflection		
SFD		
BMD		
Stress		

Problem 3.5: Cantilever Beam

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



1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node Beam 188 – ok- close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=200, h=300- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 2 (x value w.r.t first node) – ok (second node is created).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).
7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –
8. DOFs to be constrained – ALL DOF – ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value –(-40000) (-ve value) – ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).
16. Reaction forces: List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

RESULT:

Analytical approach:

Calculation:

Deflection:_______________________

Shear force: ________________________

Bending moment: ____________________

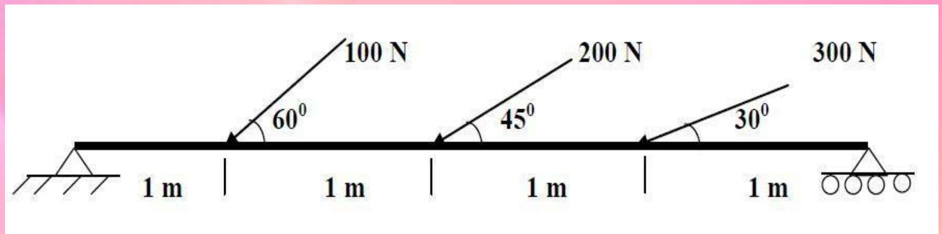
Stress: _____________________

Ansys results:

	Ansys	Theoretical
Deflection		
SFD		
BMD		
Stress		

Problem 3.6: Beam with angular loads

Compute the Shear force and bending moment diagrams for the beam shown in fig such that one end hinged and at the other end is having roller support and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



1. Ansys Main Menu – Preferences select – STRUCTURAL – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2node 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– PRXY – 0.27 –ok – close.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=200, h=300- preview-ok.
5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 1 (x value w.r.t first node) – apply (second node is created) – 2 (x value w.r.t first node) – apply (third node is created) – 3 (x value w.r.t first node) – apply (forth node is created) – 4 (x value w.r.t first node) – ok (fifth node is created).
6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).
7. Create – Nodes – Rotate nodes CS – by angles – pick node 2 – apply – about nodal z- axis – 60 – apply – pick node 3 – apply about nodal z- axis – 45 – apply – pick node 4 – apply – about nodal z –axis – 30 – ok.
8. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –DOFs to be constrained – UX & UY – apply – pick node 5 – apply – DOFs to be constrained –UY – ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply direction of For/Mom – FX – Force/Moment value - -100 (-ve value) – apply – pick node 3 – apply – direction of For/Mom – FX – Force/Moment value - -200 (-ve value) – apply – pick node 4 – apply – direction of For/Mom – FX – Force/Moment value - 300 (-ve value) – ok.
10. Solve – current LS – ok (Solution is done is displayed) – close.
11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.
12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.
13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC –SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num –SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.
14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).
15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

RESULT:

Analytical approach:

Calculation:

Deflection:________________________

Shear force: __________________________

Bending moment: ____________________

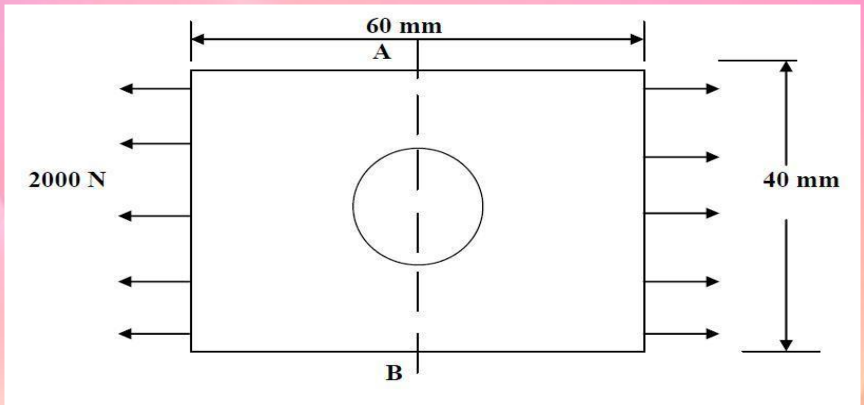
Stress:_______________________________

Ansys results:

	Ansys	Theoretical
Deflection		
Shear force		
Bending moment		
Stress		

Stress analysis of a rectangular plate with circular hole

Problem 4.1: In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use t = 1 mm). E = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 10 mm, Analysis assumption – plane stress with thickness is used.



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

RESULT:

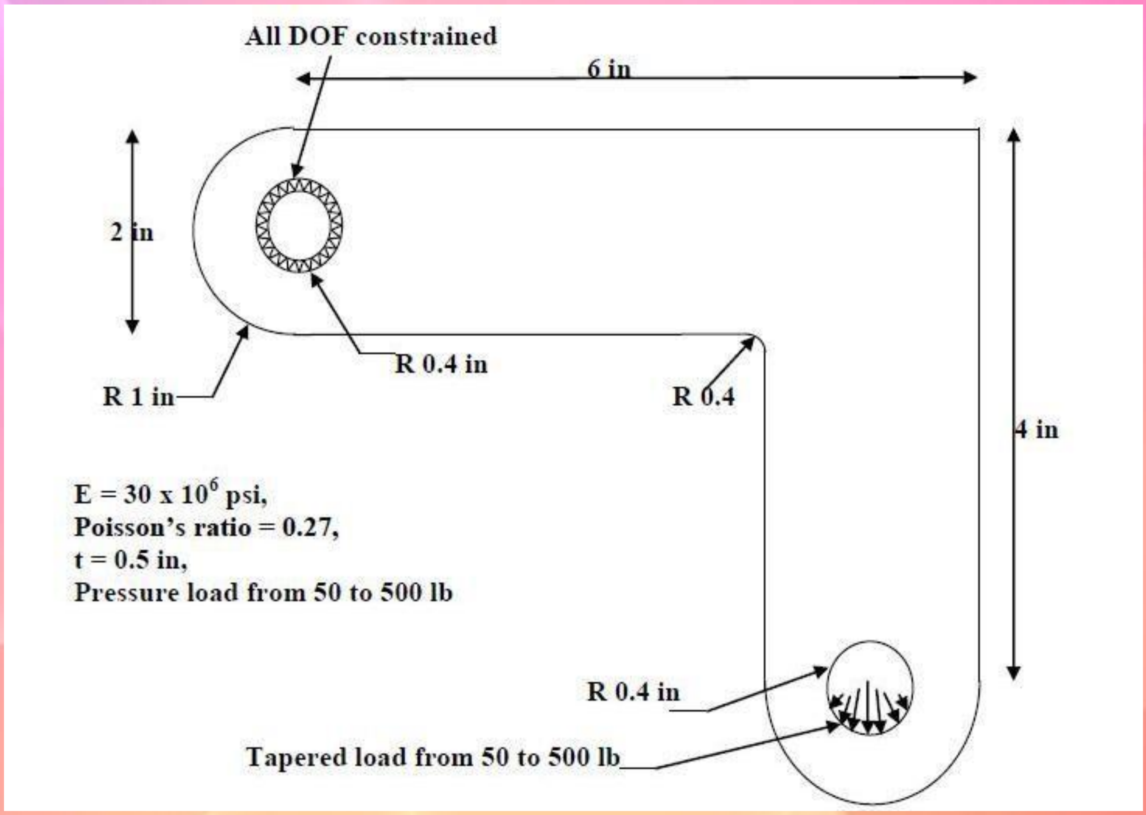
Analytical approach:

Calculation:

Ansys results:

	Ansys	Theoretical
Deformation		
Stress		

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



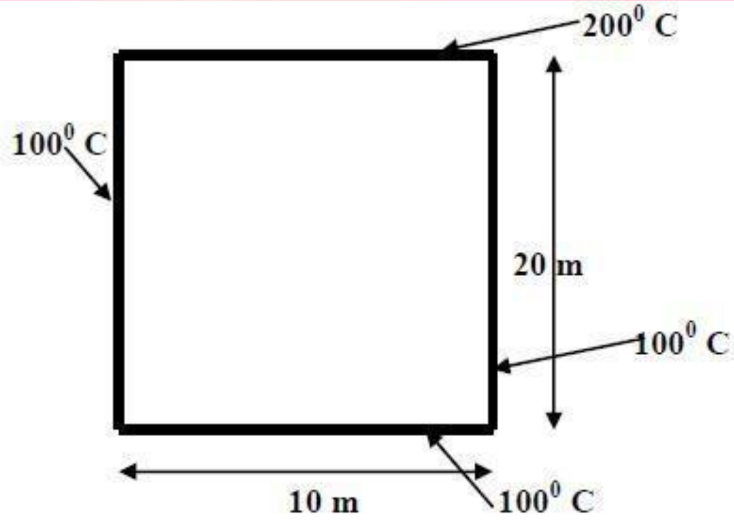
1. Ansys Main Menu – Preferences select – STRUCTURAL – ok
2. Element type – Add/Edit/Delete – Add – Solid – Quad 8 node – 82 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – Thickness – 0.5 – ok.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 30e6 –PRXY – 0.27 – ok – close.
5. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 6, 0, 2 – apply –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 4, 6, -2, 2 – ok. Create – Area –Circle – solid circle – X, Y, radius – 0, 1, 1 – apply – X, Y, radius – 5, -2, 1 – ok.
6. Operate – Booleans – Add – Areas – pick all.
7. Create – Lines – Line fillet – pick the two lines where fillet is required – apply – fillet radius – 0.4 – ok. Create – Areas – Arbitrary – by lines – pick filleted lines – ok. Operate – Booleans –
8. Add – Areas – pick all. Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 0.4 – apply –X, Y, radius – 5, -2, 0.4 – ok.
9. Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (bracket) – apply – pick areas which is to be deleted (pick two circles) – ok.
10. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine– pick all – Level of refinement – 3 – ok.

11. Loads – Define loads – apply – Structural – Displacement – on Lines – select the inner lines of the upper circle – apply – DOFs to be constrained – ALL DOF – ok.
12. Loads – Define loads – apply – Structural – Pressure – on Lines – Pick line defining bottom left part of the circle – apply – load PRES value – 50 – optional PRES value – 500 – ok. Structural – Pressure – on Lines – Pick line defining bottom right part of the circle – apply – load PRES value – 500 – optional PRES value – 50 – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. Plot Results – Deformed Shape – def+undeformed – ok.
15. Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).
16. PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

THERMAL ANALYSIS

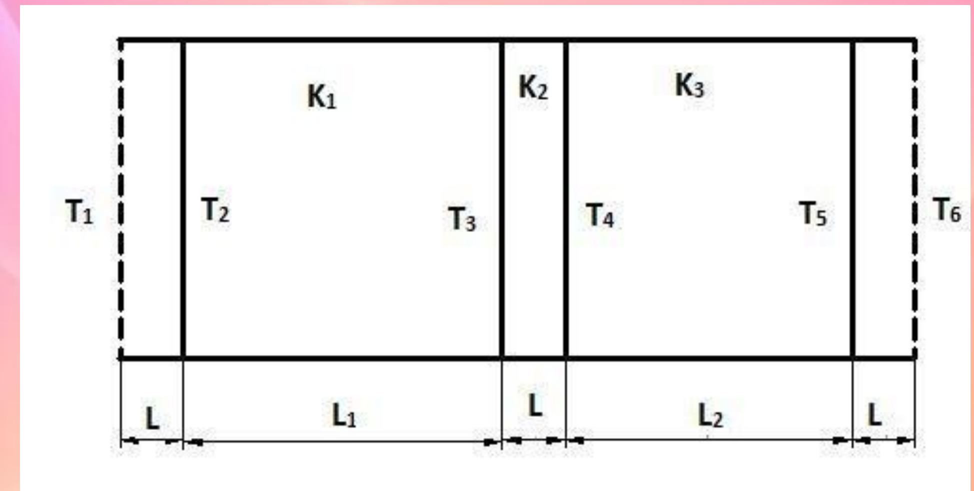
Problem 5.1: Solve the 2-D heat conduction problem for the temperature distribution within the rectangular plate. Thermal conductivity of the plate, KXX=401 W/(m-K).



1. Ansys Main Menu – Preferences-select – THERMAL- h method– ok
2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.
3. Material Properties – material models – Thermal – Conductivity – Isotropic – KXX – 401.
4. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok.
5. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.
6. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 100_0 C lines – apply – DOFs to be constrained – TEMP – Temp value – 100_0 C –ok.
7. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 100_0 C lines –
8. Solve – current LS – ok (Solution is done is displayed) – close.
9. Read results-last set-ok
10. List results-nodal solution-select temperature-ok
11. Observe the nodal solution per node.
12. From the menu bar-plot ctrls-style-size and shape-display of the element-click on real constant multiplier=0.2, don’t change other values-ok.
13. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok
14. Element table-define table-add-enter user label item=HTRANS, select by sequence no SMISC, 1-ok-close.
15. Element table-list table-select HTRANS-ok

RESULT:

Problem 5.2: A furnace wall is made up of silica brick (K=1.5W/m^{o}C) and outside magnesia brick (K= 4.9 W/m^{o}C) each 10cm thick. The inner and outer surfaces are exposed to fluids at temperatures of 820^{o}C and 110^{o}C respectively. The contact resistance is 0.001m^{2o}C/W. The heat transfer coefficient for inner and outer surfaces is equal to 35 W/m^{2}K. Find the heat flow through the wall per unit area per unit time and temperature distribution across the wall. Area= 1m^{2}.



Given: K_1=1.5W/m^{o}C, K_2= 1 W/m^{o}C, K_3= 4.9 W/m^{o}C, h_1=h_4=35 W/m^{2}K, T1=820°C, T6=110 ^{0}C, L1=L2= 10 cm & L= 1mm.

1. Preferences-thermal-h method-ok
2. Preprocessor-Element type-add/edit/delete-add-link, 3d conduction 33,element type reference N0.=1-apply-link, convection 34 element type reference no.2=2-ok-close
3. Real constant- add/edit/delete-add-real constant set no=1-C/S area =1-ok-close.
4. Real constant- add/edit/delete-add-real constant set no=2-C/S area =1-ok-close.
5. Material properties-material model-thermal conductivity-isotropic-KXX=1.5-ok. From the define material model behavior menu bar-material new model
   Enter define material id=2-ok Thermal-conductivity-isotropic-K_{xx}=1-ok
   Define material id=3-ok Thermal-conductivity-isotropic-K_{xx}=4.9-ok
   Define material id=4-ok-convection or film coefficient HF= 35,

close Modeling-create-nodes-in active CS

Enter node no=1,x=0,y=0,z=0-apply

Enter node no=2, X=0.001, Y=0, Z=0-apply

Enter node no=3, X=0.101, Y=0, Z=0-apply

Enter node no=4, X=0.102, Y=0, Z=0-apply

Enter node no=5, X=0.202, Y=0, Z=0-apply

Enter node no=6, X=0.203, Y=0, Z=0-ok.

Modeling-create-element-element attributes

Enter element type no=2 LINK 34 (convection)

Material no=4 (convection or film coefficient)

Real constant set no=2 (convection)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 1 & 2-ok Modeling-create-element-element attributes

Enter element type no=1 LINK 33 (Conduction)

Material no=1 (conduction)

Real constant set no=1 (conduction)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 2 & 3-ok Modeling-create-element-element attributes

Enter element type no=1 LINK 33 (Conduction)

Material no=2 (conduction)

Real constant set no=1 (conduction)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 3 & 4-ok Modeling-create-element-element attributes

Enter element type no=1 LINK 33 (Conduction)

Material no=3 (conduction)

Real constant set no=1 (conduction)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 4 & 5-ok. Modeling-create-element-element attributes

Enter element type no=2 LINK 34 (Convection)

Material no=4 (convection or film coefficient)

Real constant set no=2 (convection)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 5 & 6-ok.

Observe the straight line.
From the menu bar select plot controls-Numbering-Plot numbering control and select element/attributes numbering=element no and don’t change other attributes-ok

6. Solution- Analysis type-new analysis-steady state-ok.

- Solution-define loads-apply-thermal-temperature-on nodes-pick the first nodes-ok-temperature-load-temperature value=820^{0} C-apply.

- Define load-apply-thermal-temperature-on nodes-pick the last node-ok, select temperature-load temperature value=110^{0} C-ok.

- Solution- solve-current LS-ok. Solution is done-close.

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

RESULT:

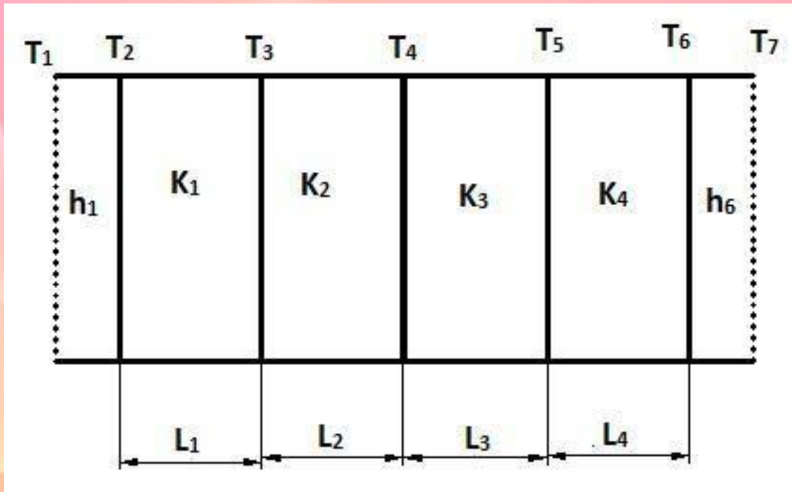
Analytical approach:

Calculation:

Ansys results:

	Ansys	Theoretical
Nodal temperature		
T_1 T_2 T_3 T_4 T_5 T_6		
Heat flux		

Problem 5.3: The exterior wall of a building is constructed of four materials, 12mm thick gypsum board, 75mm thick fibre glass insulation, 20mm thick plywood and 20mm thick hardboard. The inside and outside air temperatures are 20^{o}C and -10^{o}C respectively. The convective heat transfer coefficients on the inner and outer surfaces of the wall are 6W/m^{2} ^{o}C and 10 W/m^{2} ^{o}C respectively. Determine the heat flux and the temperature distribution. Take K for gypsum=0.176W/m^{o}C, K for fibre glass=0.036W/m^{o}C, K for plywood=0.115 W/m^{o}C and K for hardboard=0.215 W/m^{o}C. Area= 1m^{2}.



1. Preferences-thermal-h method-ok
2. Element type-add/edit/delete-add-link, 3d conduction 33,element type reference N0.=1-apply-link, convection 34 element type reference no.2=2-ok-close
3. Real constant- add/edit/delete-add-real constant set no=1-C/S area =1-ok-close.
4. Real constant- add/edit/delete-add-real constant set no=2-C/S area =1-ok-close.
5. Material properties-material model-convection film coefficient-enter HF=6-ok From the define material model behavior menu bar-material new model Enter define material id=2-ok Thermal-conductivity-isotropic-K_{XX}=0.176-ok
   Define material id=3-ok Thermal-conductivity-isotropic-K_{XX}=0.036-ok
   Define material id=4-ok Thermal-conductivity-isotropic-K_{XX}=0.115-ok
   Define material id=5-ok Thermal-conductivity-isotropic-K_{XX}=0.215-ok
   Define material id=6-convection or film coefficient HF= 10, close

Modeling-create-nodes-in active CS

Enter node no=1,x=0,y=0,z=0-apply
Enter node no=2, X=0.001, Y=0, Z=0-apply

Enter node no=3, X=0.013, Y=0, Z=0-apply

Enter node no=4, X=0.088, Y=0, Z=0-apply

Enter node no=5, X=0.108, Y=0, Z=0-apply

Enter node no=6, X=0.128, Y=0, Z=0-apply

Enter node no=7, X=0.129, Y=0, Z=0-o k

Modeling-create-element-element attributes

Enter element type no=2 LINK 34 (convection)

Material no=1 (convection or film coefficient)

Real constant set no=2 (convection)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 1 & 2-ok

Modeling-create-element-element attributes

Enter element type no=1 LINK 33 (Conduction)

Material no=2 (conduction)

Real constant set no=1 (conduction)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 2 & 3-ok

Modeling-create-element-element attributes

Enter element type no=1 LINK 33(Conduction)

Material no=3 (conduction)

Real constant set no=1 (conduction)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 3 & 4-ok

Modeling-create-element-element attributes

Enter element type no=1 LINK 33 (Conduction)

Material no=4 (conduction)

Real constant set no=1 (conduction)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 4 & 5-ok.

Modeling-create-element-element attributes

Enter element type no=1 LINK 33 (Conduction)

Material no=5 (convection or film coefficient)

Real constant set no=1 (conduction)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 5 & 6-ok.

Modeling-create-element-element attributes

Enter element type no=2 LINK 34 (convection)

Material no=6 (convection or film coefficient)

Real constant set no=2 (convection)-ok

Modeling-create-element-auto numbered-through node-pick the nodes 6 & 7-ok

Observe the straight line.

From the menu bar select plot controls-Numbering-Plot numbering control and select element/attributes numbering=element no and don’t change other attributes-ok

6. Solution- Analysis type-new analysis-steady state-ok.
   - Solution-define loads-apply-thermal-temperature-on nodes-pick the first nodes-ok-temperature-load-temperature value=20^0 C-apply.
   - Define load-apply-thermal-temperature-on nodes-pick the last node-ok, select temperature-load temperature value=-10^0 C-ok.
   - Solution- solve-current LS-ok.
7. Read results-last set-ok
8. List results-nodal solution-select temperature-ok
9. Observe the nodal solution per node.
10. From the menu bar-plot ctrls-style-size and shape-display of the element-click on real constant multiplier=0.2, don’t change other values-ok.
11. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok
12. Element table-define table-add-enter user label item=HTRANS, select by sequence no SMISC, 1-ok-close.
13. Element table-list table-select HTRANS-ok

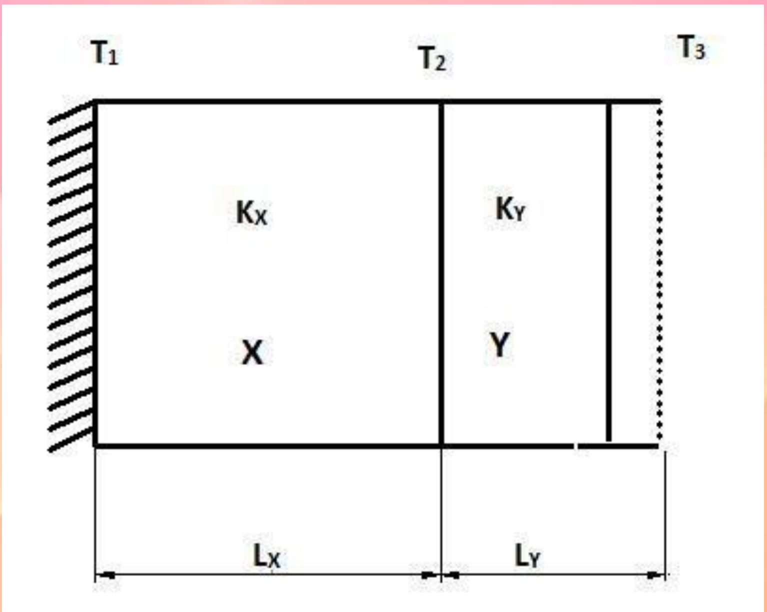
RESULT:

Analytical approach:

Calculation:

	Ansys	Theoretical
Nodal temperature		
T_1		
T_2		
T_3		
T_4		
T_5		
T_6		
T_7		
Heat flux		

Problem 5.4: A plane wall 'X' (K=75W/mK) is 60 mm thick and has volumetric heat generation of 1.5×10^{6} W/m^{3}. It is insulated on one side while the other side is in contact with the surface of another wall 'Y' (K=150W/mK) which is 30mm thick and has no heat generation. The free surface of wall 'Y' is exposed to a cooling fluid at 20^{0}C with a convection coefficient of 950 W/m^{2}K. Find steady state temperatures at salient points across the composite wall. Area= 1m^{2}.



1. Preferences-Thermal-h method-ok
2. Element type-add/edit/delete- add-solid, quad 4node 55-enter reference number=1-select options-element behaviour,K_3= plane thickness-ok
3. Real constants- add/edit/delete-add- enter real constant set no. 1, thickness=2-ok-close
4. Material properties-Material models-material number 1-thermal-conductivity-isotropic-KXX=75-ok.
5. From the menu bar select material-new model-enter material no. ID 2=2-select
6. Material model no.2- thermal-conductivity-isotropic-KXX=150-ok.
7. Modeling-create-areas-rectangles-by dimensions-X_1=0, X_2=0.06, Y_1=0,Y_2=0.03-apply- X_1=0.06, X_2=0.09, Y_1=0,Y_2=0.03-OK Modeling-operate-Boolean-glue-areas-pick the material-ok
8. Meshing-size controls-manual size-picked lines-pick the first vertical line, middle line and the last vertical line-ok-number of element divisions-2-apply
   Meshing-size controls-manual size-picked lines-pick the first rectangle top and bottom lines-number of element divisions=60-apply
   Meshing-size controls-manual size-picked lines-pick the top and bottom lines of second rectangle-number of element divisions=30-ok.
   Meshing-mesh areas- free-pick all-ok.
9. Solution-Analysis type-new analysis-steady state-ok
   define loads-apply-thermal-heat generated on areas-pick the first rectangle-ok

   define loads-apply-thermal-heat generated on areas-apply Hgen on areas as constant value- load Hgen value=1.5e6 define loads-apply-thermal-convection-on lines-pick the back corner line-ok-enter film coefficient=950-bulk temperature=20^{0}C (Don't change other attributes)-ok

define loads-apply-thermal-heat flux-on lines-pick the front corner line-ok-enter heat flux=0-ok

10. Solution-solve-current LS-ok-close
11. Read results-last set-ok
12. List results-nodal solution-select temperature-ok
13. Observe the nodal solution per node.
14. From the menu bar-plot ctrls-style-size and shape-display of the element-click on real constant multiplier=0.2, don't change other values-ok.
15. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok

RESULT:

Analytical approach:

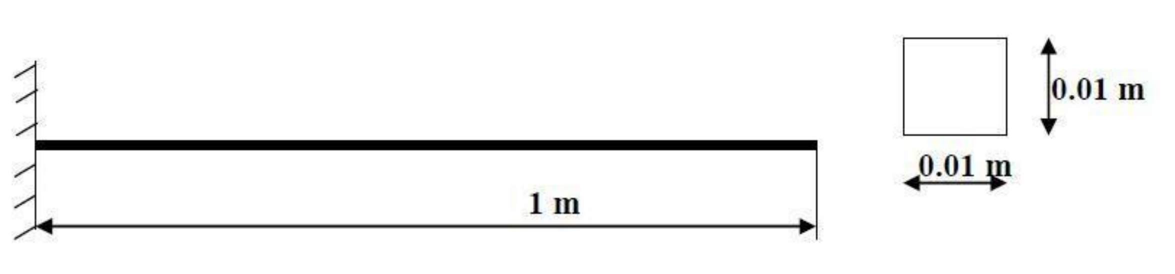
Calculation:

Ansys results:

	Ansys	Theoretical
Nodal temperature		
T_1 T_2 T_3		

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

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



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

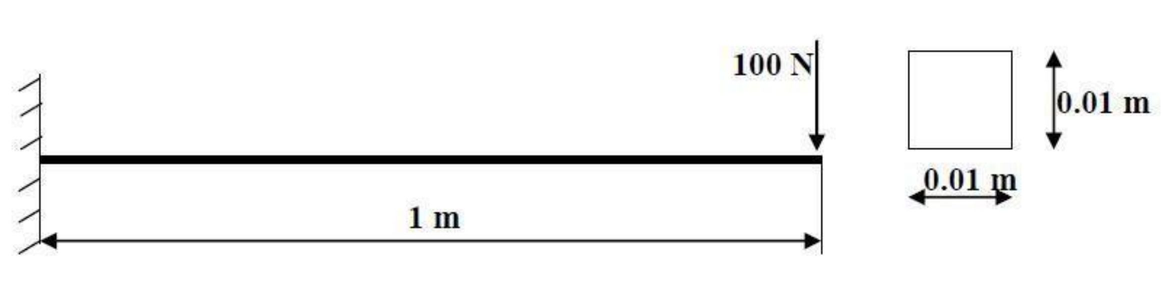
<u>RESULT:</u>

Analytical solution:

Ansys results:

Problem 6.2: Fixed- fixed beam subjected to forcing function

Conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1 - 100 Hz. Modulus of elasticity = 200GPa, Poisson's ratio = 0.3, Density = 7800 Kg/m3.



1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – BEAM – 2 node BEAM 188 – ok – close.
3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 200e9– PRXY – 0.3 – Density – 7800 – ok.
4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=100, h=100- preview-ok.
5. Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations –1,0 – ok (Keypoints created).
6. Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.
7. Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh– Lines – Pick All – ok.
8. Solution – Analysis Type – New Analysis – Harmonic – ok.
9. Solution – Analysis Type – Subspace – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.
10. Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok.
11. Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – Pick second node – apply – direction of force/mom – FY – Real part of force/mom – 100 – imaginary part of force/mom – 0 – ok.
12. Solution – Load Step Opts – Time/Frequency – Freq and Substps... – Harmonic frequency range– 0 – 100 – number of substeps – 100 – B.C – stepped – ok.
13. Solve – current LS – ok (Solution is done is displayed) – close.
14. TimeHistPostpro

    Select 'Add' (the green '+' sign in the upper left corner) from this window – Nodal solution -DOF solution – Y component of Displacement – ok. Graphically select node 2 – ok.

    Select 'List Data' (3 buttons to the left of 'Add') from the window.

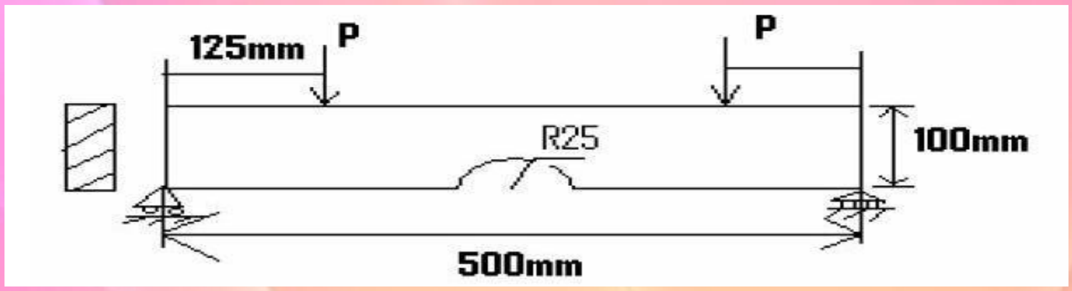
'   Time History Variables' window click the 'Plot' button, (2 buttons to the left of 'Add') Utility Menu – PlotCtrls – Style – Graphs – Modify Axis – Y axis scale – Logarithmic –ok. Utility Menu – Plot – Replot.

    This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.

RESULT:

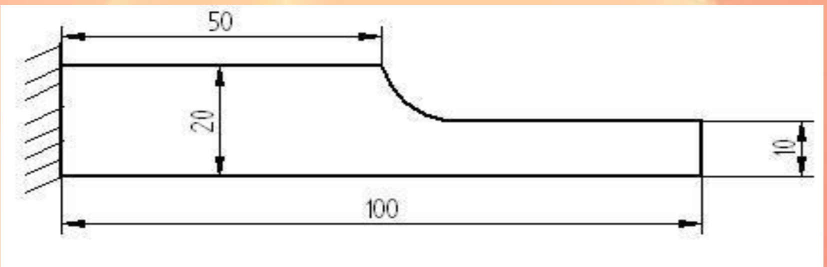
ADDITIONAL PROBLEMS

1. Calculate the stresses and displacement for the plate shown below. Let the load be P = 100N applied at equal distance from both ends and E = 3e7 N/mm2.

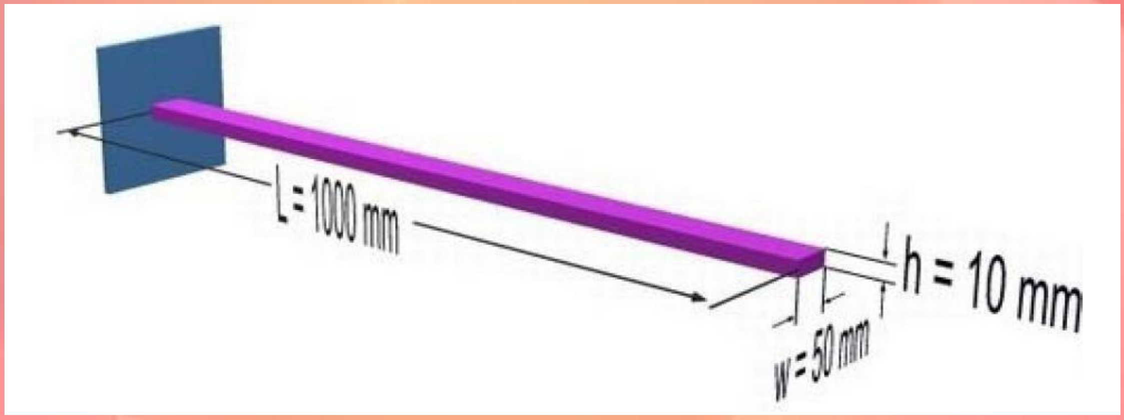


2. Current passes through a stainless steel wire of 2.5 mm diameter (k=200 W/mK) causing volumetric heat generation of 26.14X108 W/m3 .the wire is submerged in a fluid maintained at 500 C and convective heat transfer coefficient at the wire surface is 4000W/m2 K . Find the steady state temperature at the centre and at the surface of the wire.

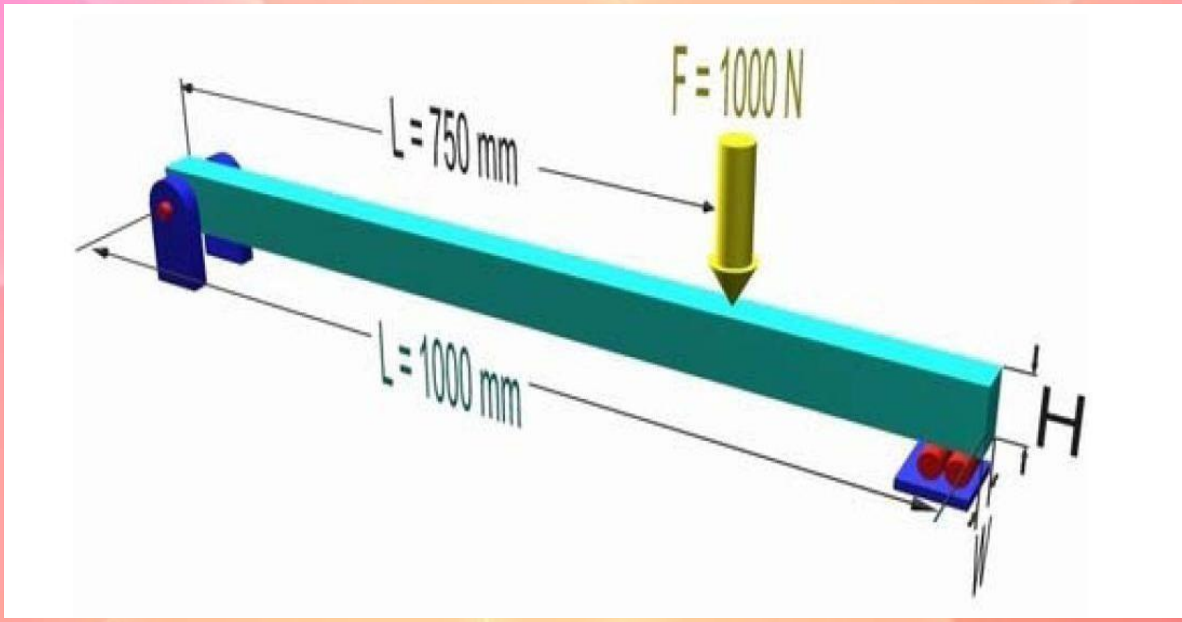
3. Calculate the maximum value of Von-misses stresses in the stepped beam with a rounded plate as shown in the figure. Where Young's modulus, E=210Gpa, Poisson's ratio is 0.3 and the beam thickness is 10mm, the element size is 2mm



4. Loads will not be applied to the beam shown below in order to observe the deflection caused by the weight of the beam itself. The beam is to be made of steel with a Young's modulus of elasticity of 200 Gpa.



5. A beam has a force of 1000N applied as shown below. The purpose of this optimization problem is to minimize the weight of the beam without exceeding the allowable stress. It is necessary to find the cross sectional dimensions of the beam in order to minimize the weight of the beam. However, the width and height of the beam cannot be smaller than 10mm. The maximum stress anywhere in the beam cannot exceed 200 MPa. The beam is to be made of steel with a modulus of elasticity of 200 GPa



VIVA QUESTIONS

1. Theories of failure.

   a. **Maximum Principal Stress Theory**- A material in complex state of stress fails, when the maximum principal stress in it reaches the value of stress at elastic limit in simple tension.

   b. **Maximum Shear Stress Theory**- A material in complex state of stress fails when the maximum shearing stress in it reaches the value of shearing stress at elastic limit in uniaxial tension test.

   c. **Maximum Principal Strain Theory**-Failure in a complex system occurs when the maximum strain in it reaches the value of the strain in uniaxial stress at elastic limit.

   d. **Maximum Strain Energy Theory**- A material in complex state of stress fails when the maximum strain energy per unit volume at a point reaches the value of strain energy per unit volume at elastic limit in simple tension test.

   e. **Maximum Distortion Energy Theory**-This theory is also known as Von-Mises criteria for failure of elastic bodies. According to this theory part of strain energy causes only changes in volume of the material and rest of it causes distortion. At failure the energy causing distortion per unit volume is equal to the distortion energy per unit volume in uniaxial state of stress at elastic limit.

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

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

4. Define: Modulus of rigidity, Bulk modulus
   Modulus of rigidity: It is defined as the ratio of shearing stress to shearing strain within elastic limit.
   Bulk modulus:It is defined as the ratio of identical pressure ‘p’ acting in three mutually perpendicular directions to corresponding volumetric strain.

5. What is proof resilience?

The maximum strain energy which can be stored by a body without undergoing permanent deformation is called proof resilience.

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

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

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

9. State the three phases of finite element method.

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

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

11. What is the difference between static analysis and dynamic analysis?
    Static analysis: The solution of the problem does not vary with time is known as static analysis.E.g.: stress analysis on a beam.
    Dynamic analysis: The solution of the problem varies with time is known as dynamic analysis.E.g.: vibration analysis problem.

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

13. What are natural coordinates?A natural coordinate system is used to define any point inside the element by a set of dimensionless number whose magnitude never exceeds unity. This system is very useful in assembling of stiffness matrices.

14. What is a CST element?

    Three node triangular elements are known as constant strain triangular element. It has 6 unknown degrees of freedom called u1, v1, u2, v2, u3, v3. The element is called CST because it has constant strain throughout it.

15. Define shape function.

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

$\Phi(x,y) = N_1(x,y)\,\Phi 1 + N_2(x,y)\,\Phi 2 + N_3(x,y)\,\Phi 3 + N_4(x,y)\,\Phi 4$ where Φ1, Φ2, Φ3 and Φ4 are the values of the field variables at the nodes and N1, N2, N3 and N4 are interpolation function. N1, N2, N3, N4 are called shape functions because they are used to express the geometry or shape of the element.

16. What are the characteristics of shape function?

The characteristics of the shape functions are as follows:

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

17. Why polynomials are generally used as shape function?
    - Differentiation and integration of polynomials are quite easy.
    - The accuracy of the results can be improved by increasing the order of the polynomial.
    - It is easy to formulate and computerize the finite element equations.

18. State the properties of a stiffness matrix. The properties of the stiffness matrix [K] are:
    - It is a symmetric matrix.
    - The sum of the elements in any column must be equal to zero.
    - It is an unstable element, so the determinant is equal to zero.

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

    The solution of differential equation obtained for physical problems which satisfies some specified conditions known as boundary conditions. If the solution of differential equation is obtained together with initial conditions then it is known as initial value problem. If the solution of differential equation is obtained together with boundary conditions then it is known as boundary value problem.

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

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

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

23. What is a sub parametric element?

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

24. What is a super parametric element?
If the number of nodes used for defining the geometry is more than the number of nodes used for defining the displacements is known as sub parametric element.

25. What is meant by isoparametric element?
If the number of nodes used for defining the geometry is same as number of nodes used for defining the displacements then it is called parametric element.

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

27. What are isotropic and orthotropic materials?

A material is isotropic if its mechanical and thermal properties are the same in all directions. Isotropic materials can have homogeneous or non-homogeneous microscopic structures.
Orthotropic materials: A material is orthotropic if its mechanical or thermal properties are unique and independent in three mutually perpendicular directions.

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

29. Steps in FEM

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

Lab Performance Appraisal Sheet

Date of Allotment	Experiment No & Title of the Experiment	Full Marks (Y)	Marks awarded for				Signature of Lab In charge with Date
			Lab Performance	Viva-voce	Lab Citizenship	Total (X)	
	Total Marks from experiment No------ to------ (Z)						
	Total marks as per syllabus (M)= Z / 10						

Note: Lab citizenship covers Discipline, Punctuality, Lab Meeting Participation, Note book Record Keeping and contribution in up keeping of the lab