

MODEL LABORATORY MANUAL

**COMPUTER AIDED ANALYSIS AND SIMULATION
LABORATORY**

ACADEMIC YEAR 2017-18

Introduction to ANSYS

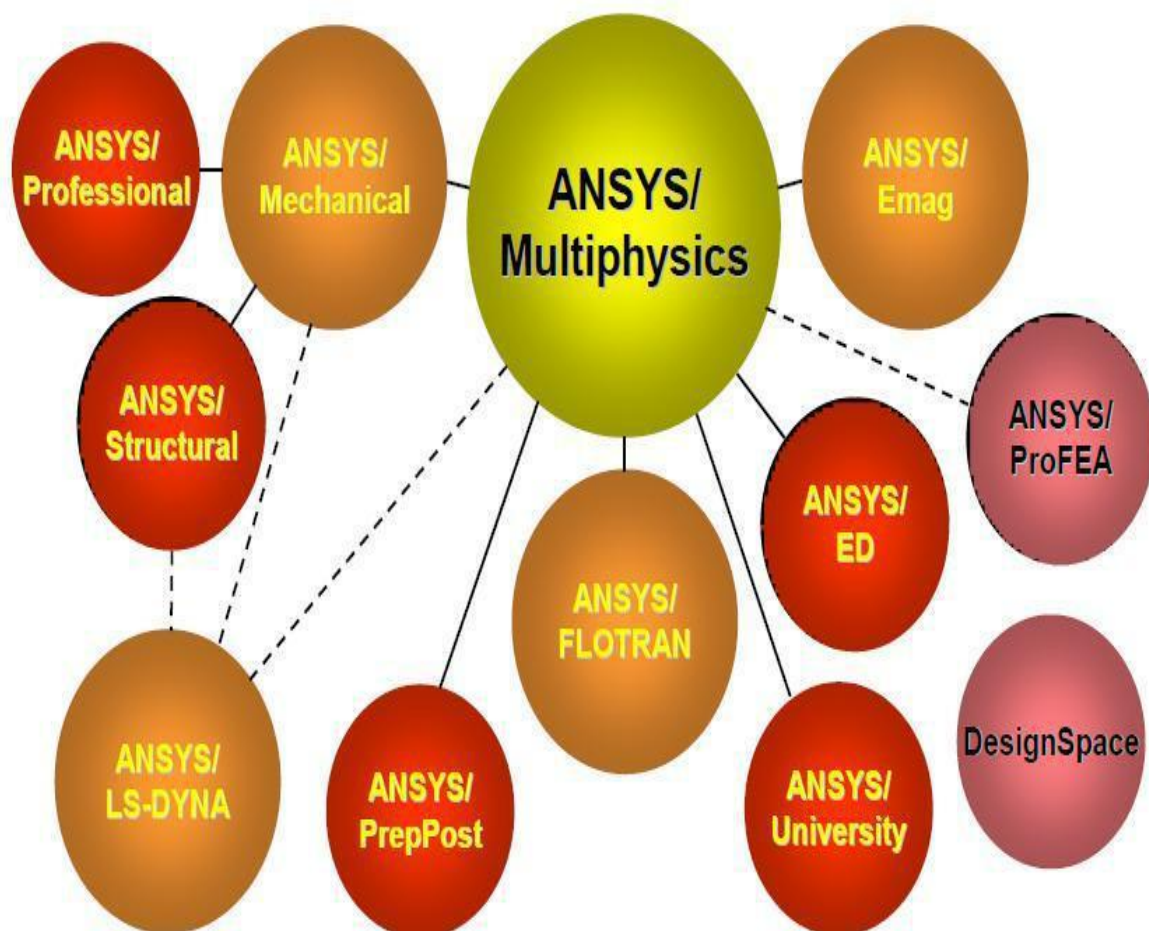
❖ ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering:

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

❖ A partial list of industries in which ANSYS is used:

- Aerospace
- Automotive
- Bio-medical
- Bridges & Buildings

❖ ANSYS/Multiphysics is the flagship ANSYS product which includes all capabilities in all engineering disciplines.



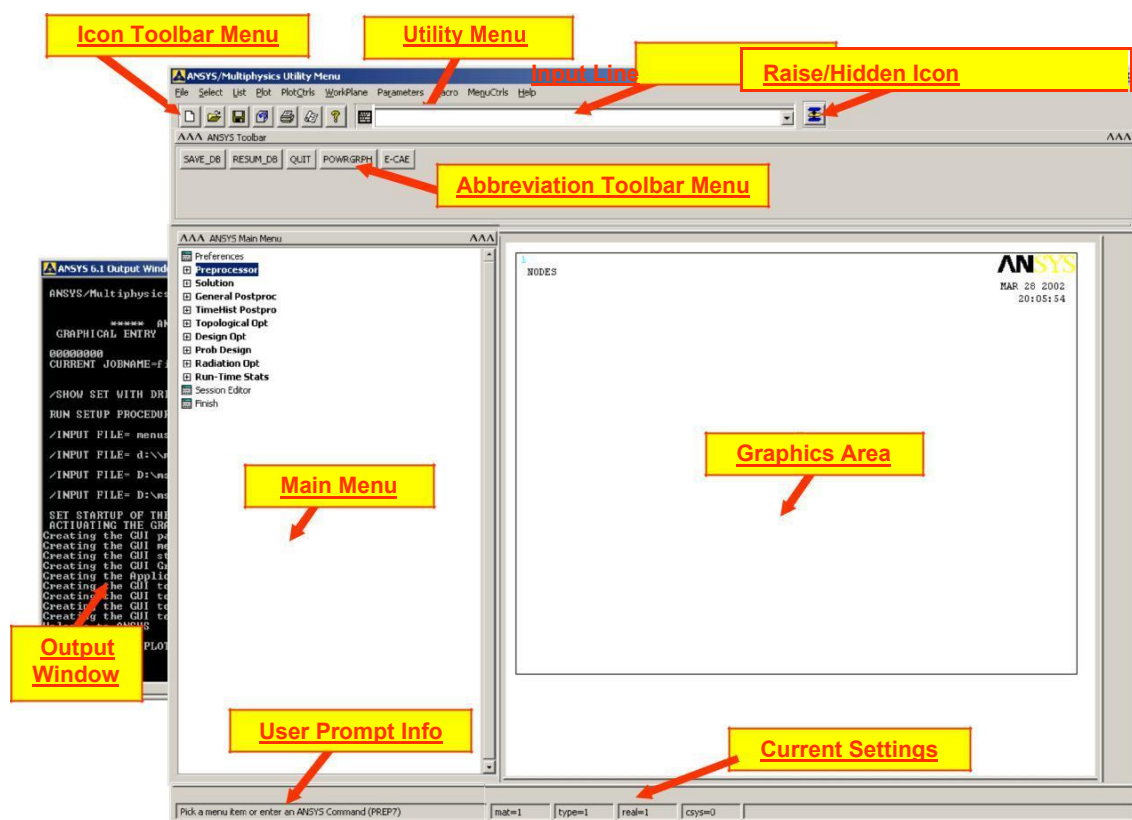
- ❖ There are three main component products derived from ANSYS/Multiphysics:
 - ANSYS/Mechanical - structural & thermal capabilities
 - ANSYS/Emag - electromagnetics
 - ANSYS/FLOTTRAN - CFD capabilities
 - ❖ Other product lines:
 - ANSYS/LS-DYNA - for highly nonlinear structural problems
 - Design Space - an easy-to-use design and analysis tool meant for quick analysis within the CAD environment
 - ANSYS/ProFEA - for ANSYS analysis & design optimization within Pro/ENGINEER
 - ❖ **Structural analysis:** is used to determine deformations, strains, stresses, and reaction forces.
 - Static analysis
 - Used for static loading conditions.
 - Nonlinear behavior such as large deflections, large strain, contact, plasticity, hyper elasticity, and creep can be simulated
 - Dynamic analysis
 - Includes mass and damping effects.
 - Modal analysis calculates natural frequencies and mode shapes.
 - Harmonic analysis determines a structure's response to sinusoidal loads of known amplitude and frequency.
 - Transient Dynamic analysis determines a structure's response to time-varying loads and can include nonlinear behavior.
 - Other structural capabilities
 - Spectrum analysis
 - Random vibrations
 - Eigen value buckling
 - Substructuring, submodeling
 - Explicit Dynamics with ANSYS/LS-DYNA
 - Intended for very large deformation simulations where inertia forces are dominant.
 - Used to simulate impact, crushing, rapid forming, etc.
 - ❖ **Thermal analysis:** is used to determine the temperature distribution in an object. Other quantities of interest include amount of heat lost or gained, thermal gradients, and thermal flux. All three primary heat transfer modes can be simulated: conduction, convection, radiation.
 - Steady-State
 - Time-dependent effects are ignored.
 - Transient
 - To determine temperatures, etc. as a function of time.
 - Allows phase change (melting or freezing) to be simulated.
-

- Electromagnetic analysis is used to calculate magnetic fields in electromagnetic devices.
- Static and low-frequency electro magnetic
 - To simulate devices operating with DC power sources, low-frequency AC, or low- frequency transient signals.

❖ Computational Fluid Dynamics (CFD)

- To determine the flow distributions and temperatures in a fluid.
- ANSYS/FLOTRAN can simulate laminar and turbulent flow, compressible and incompressible flow, and multiple species.
- Applications: aerospace, electronic packaging, automotive design
- Typical quantities of interest are velocities, pressures, temperatures, and film coefficients.

The GUI Layout



Utility Menu

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

Toolbar Menu

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

Graphics Area

Displays graphics created in ANSYS or imported into ANSYS.

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

Main Menu

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

Output

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

Resume:

This is opening a previously saved database. It is important to know that if you simply resume a database, it doesn't change the job name. For example: You start ANSYS with a job name of —file\$. Then you resume my model.db, do some work, then save. That save is done to file.db! Avoid this issue by always resuming using the icon on the toolbar. If you open mymodel.db using this method, it resumes the model and automatically changes the job name to my model.

Plotting:

Contrary to the name, this has nothing to do with sending an image to a plotter or printer. Plotting in ANSYS refers to drawing something in the graphics window. Generally you plot one type of entity (lines, elements, etc.) to the screen at a time. If you want to plot more than one kind of entity use, —Plot → Multiplot\$, which by default will plot everything in your model at once.

Plot Controls:

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

Creating Geometry:

Geometry in ANSYS is created from

—Main Menu → Preprocessor → Modeling → Create\$ and has the following terminology, KEYPOINTS: These are points, locations in 3D space.

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

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

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

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

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

Boolean Operations:

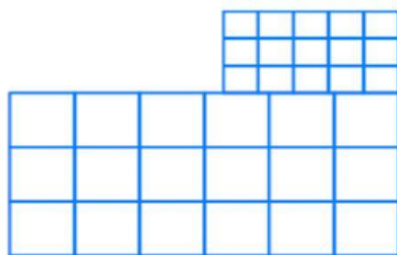
Top Down style modeling can be a very convenient way to work. Instead of first creating key points, then lines from those key points, then areas from the lines and so on (bottom up modeling), start with volumes of basic shapes and use Boolean operations to add them, subtract them, divide them etc. Even if you are creating a shell model, for example a box, you could create the box as a volume (a single command) and then delete the volume keeping the existing areas, lines and key points.

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

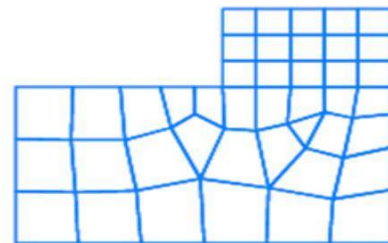
Add: Take two entities that overlap (or are at least touching) and make them one.

Subtract: Subtract one entity from another. To make a hole in a plate, create the plate (area of volume) then create a circular area or cylinder and subtract it from the plate.

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



Meshing without gluing areas.



Meshing after gluing areas.

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

The Working Plane:

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

Working Plane Hints:

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

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

Move the working plane around using

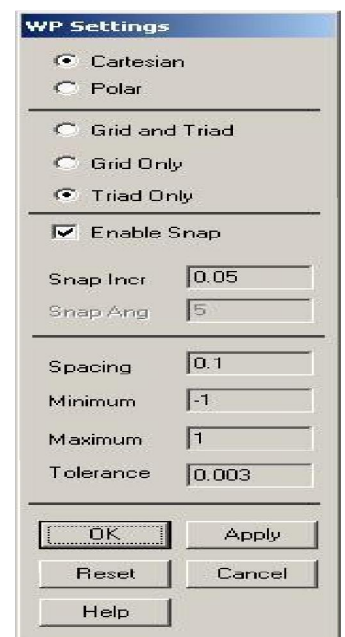
—Utility Menu → Work Plane → Offset WP to...

Align the working plane with various parts of the model using

—Utility Menu → Work Plane → Align WP with...

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

Use the working plane to slice and dice your model. For example to cut an area in pieces use —Main Menu → Modeling → Operate → Booleans → Divide → Area by WrkPlane. Do this for lines and volumes as well.



Select Logic:

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

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

Examples where this would be useful:

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

After working with the selected set,

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

**Select Entities Dialog Box Terminology:**

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

Reselect: Select a subset from the currently selected entities.

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

Unselect: Remove items from the selection set.

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

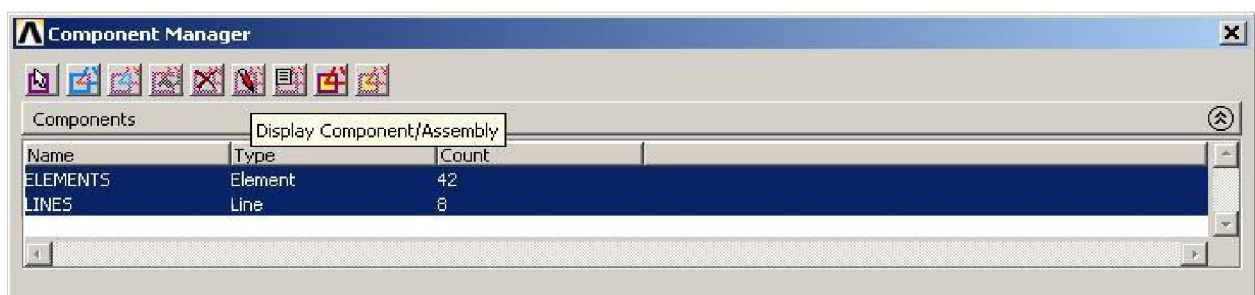
Invert: Reverses the selected and unselected entities (just the entities specified at the top of the dialog).

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

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

Replot: Replots whatever is active in the graphics window.

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

Organizing Your Model Using Components:

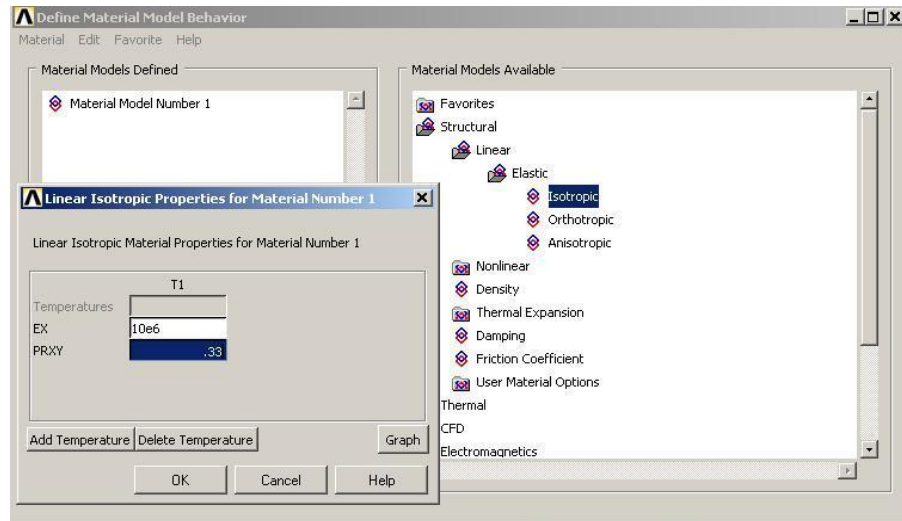
If you select a group of entities and think that you might want to use that selection set again, create a component out of it. Components are groups of entities but hold only one kind of entity at a time. Components can themselves be grouped into Assemblies, so this is how you group different types of entities together. Use —Utility Menu → Select → Comp/Assembly → Create

Component... to create a component. The new Component Manager in Release 8.0 makes it very easy to manage and manipulate groups and select/plot what you want to see to the screen. This is found under —Utility Menu → Select → Component Manager

Creating a Material:

Create the material properties for your model in

—Main Menu → Preprocessor → Material Props → Material Models. This gives you this dialog box where all materials can be created,

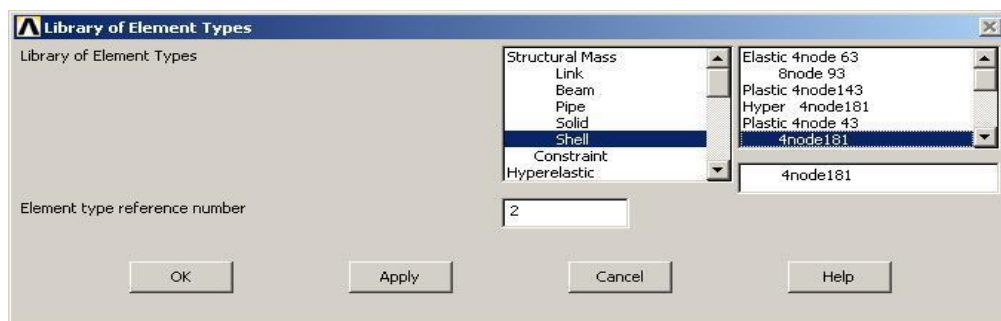


Double click on items in the right hand pane of this window to get to the type of material model you want to create. All properties can be temperature dependant. Click OK to create the material and it will appear in the left hand pane. Create as many different materials as you need for your analysis.

Selecting an Element Type:

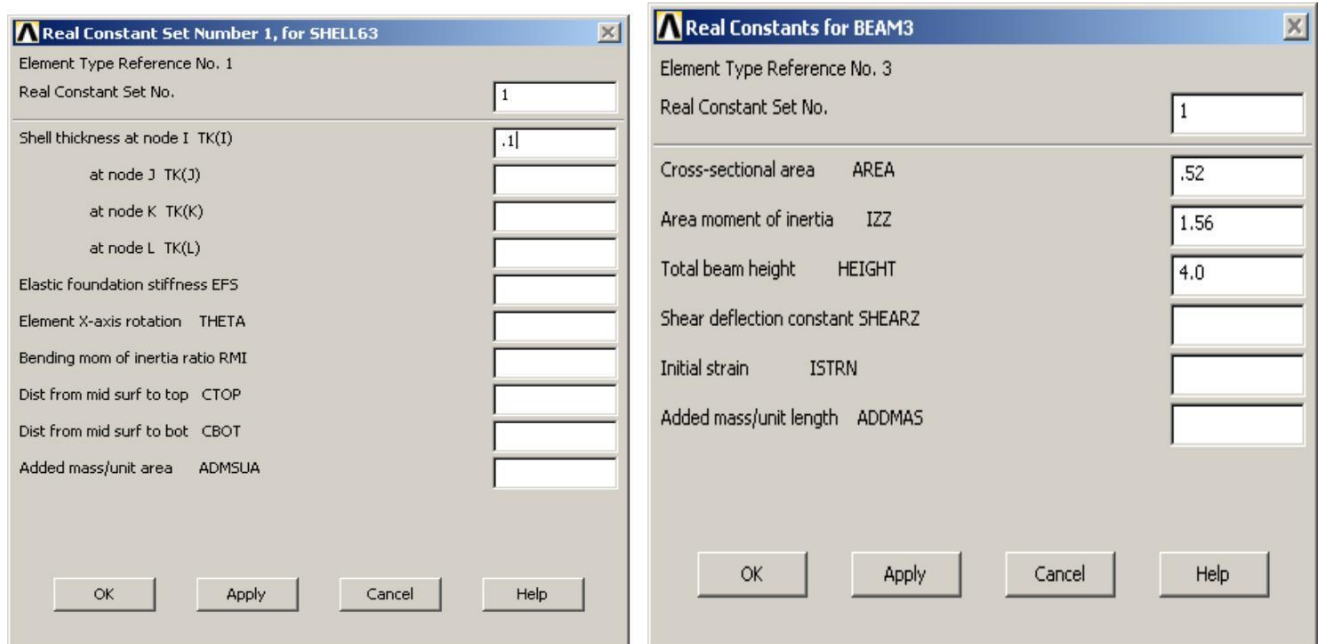
ANSYS has a large library of element types. Why so many? Elements are organized into groups of similar characteristics. These group names make up the first part of the element name (BEAM, SOLID, SHELL, etc). The second part of the element name is a number that is more or less (but not exactly) chronological. As elements have been created over the past 30 years the element numbers have simply been incremented. The earliest and simplest elements have the lowest numbers (LINK1, BEAM3, etc), the more recently developed ones have higher numbers. The —18x1 series of elements (SHELL181, SOLID187, etc) are the newest and most modern in the ANSYS element library.

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



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

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



Creating the Finite Elements Model - Meshing:

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

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

Steps for Creating the Finite Elements:

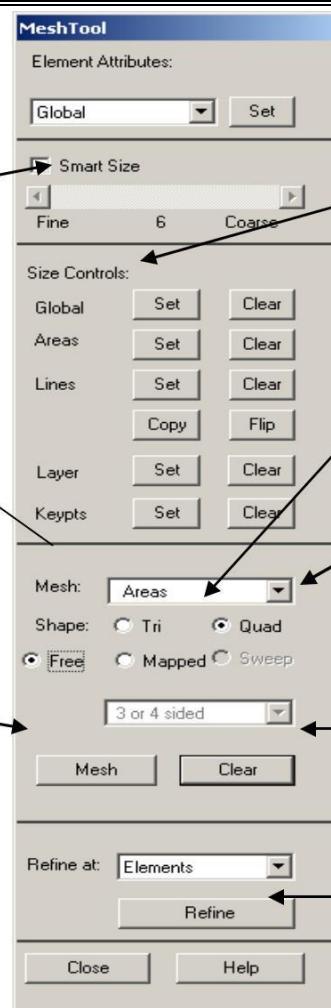
- Assign Attributes to Geometry (materials, real constants, etc).
- Specify Mesh Controls on the Geometry (element sizes you want)
- Mesh.

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

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

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

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



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

Pick what you want to mesh.

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

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

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

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

Applying Loads and Boundary Conditions:

Loads and boundary condition can be applied in both the Preprocessor (—Main Menu → Preprocessor → Loads → Define Loads → Apply), and the Solution processor

(—Main Menu → Solution → Define Loads → Apply).

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

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

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

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

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

Solving:

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

where $[K]$ is the assembled stiffness matrix of the structure, $\{u\}$ is the vector of displacements at each node, and $\{F\}$ is the applied load vector.

This is analogous to a simple spring and is the essence of small deflection theory.

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

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

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

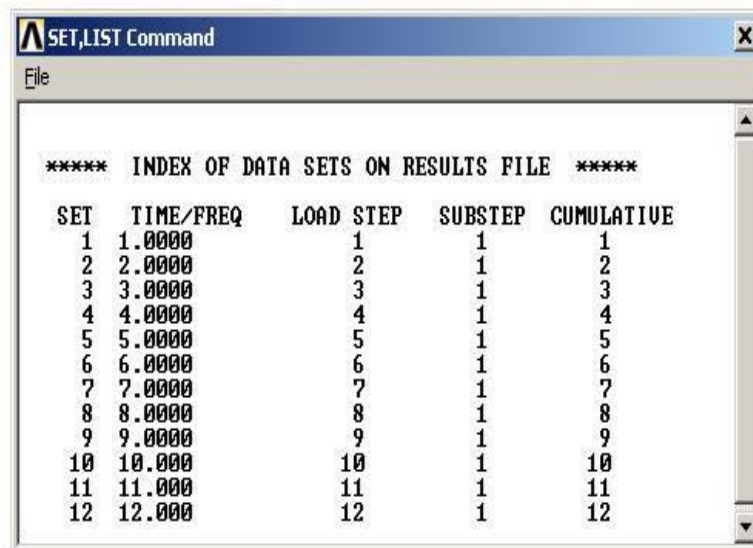
There are several solvers in ANSYS that differ in the way that the system of equations is solved for the unknown displacements. The two main solvers are the sparse solver and the PCG solver.

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

Postprocessing:

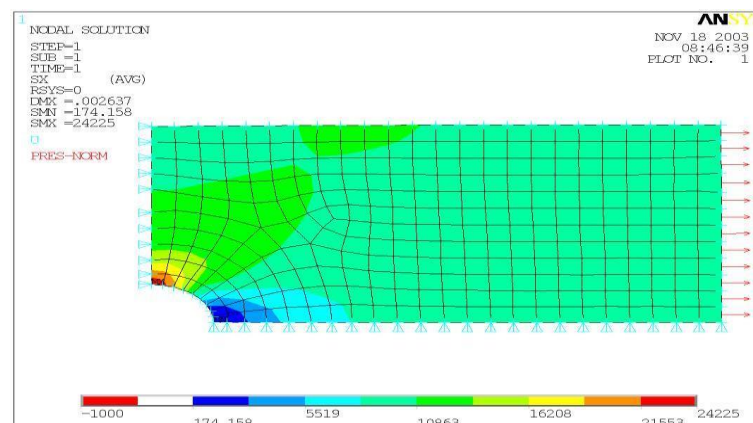
The General Postprocessor is used to look at the results over the whole model at one point in time. This is the final objective of everything we have discussed so far; finding the stresses, deflections, temperature distributions, pressures, etc. These results can then be compared to some criteria to make an objective evaluation of the performance of your design.

The solution results will be stored in the results file as result —sets. For a linear static analysis like we are talking about, the correlation between Load Step numbers and Results Set numbers will be one to one as shown below. Only one set of results can be stored in the database at a time, so when you want to look at a particular set, you have to read it in from the results file. Reading it in clears the previous results set from active memory.



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

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



There are many, many other ways to look at your results data including:

- Listing them to a file.
- Querying with the mouse to find a result at a particular node.
- Graphing results along a path.
- Combining different load cases.
- Summing forces at a point.
- Extracting data and storing it in an APDL array that you can do further operations.

Animate any result on the deformed shape with —Utility Menu → PlotCtrls → Animate \bar{I} . This is very helpful for understanding if your model is behaving in a reasonable way.

Experiment No: 1

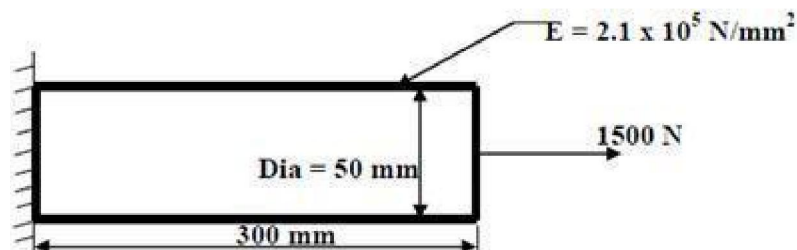
Date:

Stress Analysis of Bars of Constant Cross Section Area**AIM:**

To perform displacement and stress analysis for the given bar using Ansys simulation and analytical expressions.

Problem Description:

A steel rod subjected to tension is modeled by one bar element, as shown in figure. Determine the nodal displacements and the axial stress in each Element and reaction forces. $E = 2.1 \times 10^5 \text{ N/mm}^2$, 0.3 (Poisson's Ratio).



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real constants –Add –ok –Real constant set no –1 –c/s area –22/7*50**2/4 –ok.

Material Properties –Material Models –Structural –Linear –Elastic –Isotropic –EX – 2.1e5 –ok –close.

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z location in CS –300, 0, 0 –ok (second node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –ok (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes- pick node 1 –Apply – DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes - pick node 2 –Apply –

direction of For/Mom –FX –Force/Moment value –1500 (+ve value) –

ok. **Solve** –Current LS –ok (Solution is done is displayed) –close.

Step 5: General Post Processor

Element table –Define table –Add –_Results–By data Sequence item‘num–LS–LS1 –ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1 – Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

List Results –Elem table Data –Items to be listed –LS1 –ok. (Stress will be displayed with the element numbers)

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+ undeformed-ok

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

Comparison between theoretical and Ansys values:

Particulars	Ansys		Theoretical	
	Node 1	Node 2	Node 1	Node2
Deformation				
Stress				
Reaction				

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increasing number of nodes and elements to match the theoretical or analytical results.

Conclusions: Ansys simulation and the software results are near to theoretical or analytical results.

Experiment No: 2

Date:

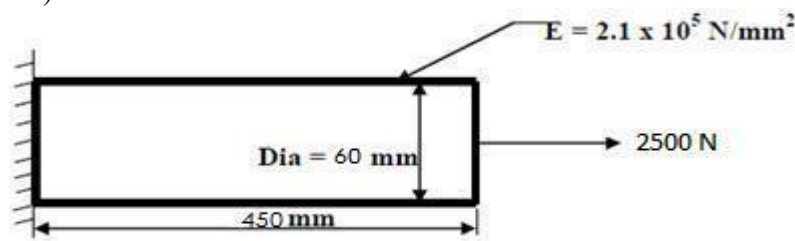
Stress Analysis of Bars of Constant Cross Section Area

AIM:

To perform displacement and stress analysis for the given bar using Ansys simulation and analytical expressions.

Problem Description:

A steel rod subjected to tension is modeled by one bar element, as shown in figure. Determine the nodal displacements and the axial stress in each Element and reaction forces. $E = 2.1 \times 10^5 \text{ N/mm}^2$, 0.3 (Poisson's Ratio).



Software Required: Ansys 14.5.

Procedure:

Step 1: Ansys Utility Menu

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real Constants –Add –ok –Real constant set no –1 –c/s area – $22/7 \times 60^2/4$ –ok.

Material Properties –Material Models –Structural –Linear –Elastic –Isotropic –EX – 2.1×10^5 –PRXY –0.3 –ok –close.

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z location in CS –450, 0, 0 –ok (second node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –ok (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes- pick node 1 –Apply – DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes- pick node 2 –Apply
– direction of For/Mom –FX –Force/Moment value –2500 (+ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1-ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1
– Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

List Results –Elem table Data –Items to be listed –LS1 –ok. (Stress will be displayed with the element numbers)

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok

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

Comparison between theoretical and Ansys values:

Particulars	Ansys		Theoretical	
	Node 1	Node 2	Node 1	Node2
Deformation				
Stress				
Reaction				

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increasing number of nodes and elements to match the theoretical or analytical results.

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

Experiment No: 3

Date:

Stress Analysis of Bars of Constant Cross Section Area

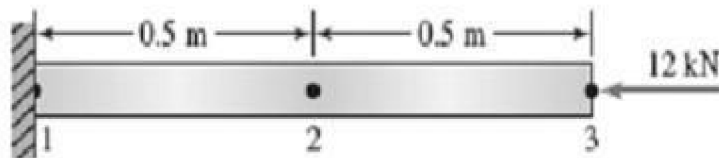
AIM:

To perform displacement and stress analysis for the given bar using Ansys simulation and analytical expressions.

Problem Description:

A steel rod subjected to compression is modeled by two bar elements, as shown in figure. Determine the nodal displacements and the axial stress in each Element. $E=207 \text{ GPa}$,

$$A = 500 \text{ mm}^2$$



Software Required: Ansys 14.5.

Procedure:

Step 1: Ansys Utility Menu

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real constants –Add –ok –Real constant set no –1 –c/s area –500 –ok.

Material Properties –Material models –Structural –Linear –Elastic –Isotropic –EX – 207e3 –ok –close.

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z location in CS –500, 0, 0 –Apply (second node is Created) - x, y, z location in CS – 1000, 0, 0 (third node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –Apply - pick 2 & 3 (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes- pick node 1 –Apply – DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes- pick node 3 –Apply – direction of For/Mom –FX –Force/Moment value → 12000 (-ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1–ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1 – Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.

Comparison between theoretical and Ansys values:

Particulars	Ansys			Theoretical		
	Node 1	Node 2	Node 3	Node 1	Node2	Node 3
Deformation						
Stress						
Reaction						

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

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

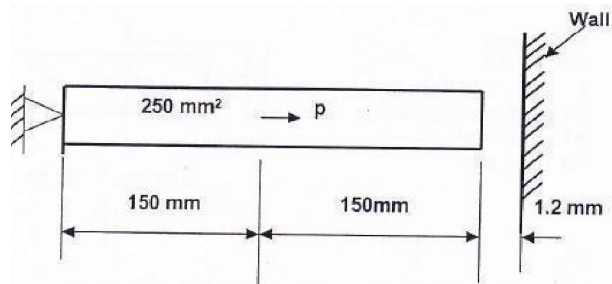
AIM:

To perform displacement and stress analysis for the given bar using Ansys simulation and analytical expressions.

Problem Description:

A load of $P = 60 \times 10^3$ N is applied as shown. Determine the following, a) Nodal Displacement, b) Stress in each member, c) Reaction Forces.

Given Data: $E = 20 \times 10^3$ N/mm², 0.3 (Poisson's Ratio)



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real Constants –Add –ok –Real constant set no –1 –c/s area –250 –ok.

Material Properties –Material Models –Structural –Linear –Elastic –Isotropic –EX –
20e3 –ok –close.

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z
location in CS –150, 0, 0 –Apply (second node is Created) – x, y, z location in CS –300,
0, 0 (third node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –Apply - pick
2 & 3 (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes- pick node 1 –Apply – DOFs
to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes- pick node 3 –Apply
– DOFs to be constrained –UX –VALUE - Displacement Value –1.2 - ok.

Define loads –Apply –Structural –Force/Moment –on Nodes- pick node 2 –Apply
– direction of For/Mom –FX –Force/Moment value –60e3 (+ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1
–ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1
– Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal
solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be
displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.

Comparison between theoretical and Ansys values:

Particulars	Ansys			Theoretical		
	Node1	Node2	Node3	Node1	Node2	Node3
Deformation						
Stress						
Reaction						

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increasing number of nodes and elements to match the theoretical or analytical results.

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

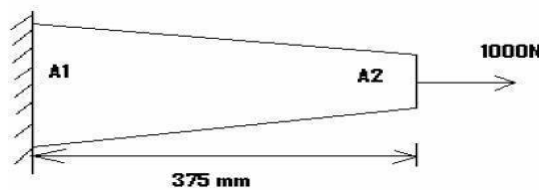
Stress Analysis of Bars of Tapered Cross Section Area

AIM:

To perform displacement and stress analysis for the given Taper bar using Ansys simulation and analytical expressions.

Problem Description:

For the tapered bar shown in the figure determine the displacement, stress and reaction in the bar. Given $A_1 = 1000 \text{ mm}^2$ and $A_2 = 500 \text{ mm}^2$, $E = 2 \times 10^5 \text{ N/mm}^2$

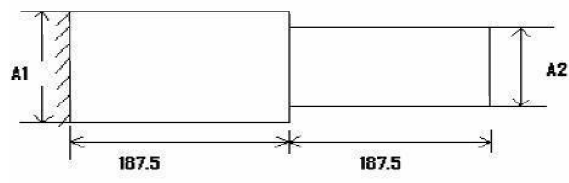


The Tapered bar is modified into 2 elements as shown below with modified area of cross section $(1000+500)/2 = 750 \text{ mm}^2$

Areas of modified two elements:

$$A'_1 = (1000+750)/2 = 875 \text{ mm}^2$$

$$A'_2 = (750+500)/2 = 625 \text{ mm}^2$$



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real constants –Add –ok –Real constant set no –1 –c/s area –875 –ok. Add –ok –
Real constant set no –2 –c/s area 625– ok.

Material Properties –Material models –Structural –Linear –Elastic –Isotropic –EX
– 2×10^5 –ok –close.

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z location in
CS –187.5, 0, 0 –Apply (second node is Created) - x, y, z location in CS – 375, 0, 0

(third node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –Apply - pick 2 & 3 (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes- pick node 1 –Apply – DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes- pick node 2 –Apply – direction of For/Mom –FX –Force/Moment value –1000 (+ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1–ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1 – Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.

Comparison between theoretical and Ansys values:

Particulars	Ansys			Theoretical		
	Node1	Node2	Node3	Node1	Node2	Node3
Deformation						
Stress						
Reaction						

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

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

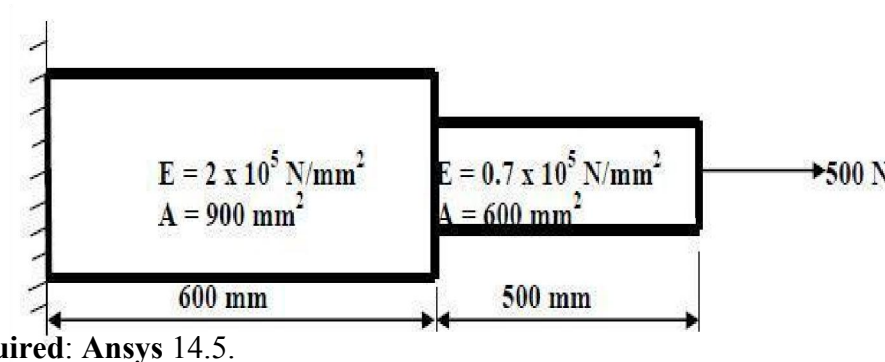
Stress Analysis of Bars Varying In Cross Section or Stepped Bars

AIM:

To perform displacement and stress analysis for the given stepped bar using Ansys simulation and analytical expressions.

Problem Description:

Consider the stepped bar shown in figure below. Determine the nodal displacement stress in each element, reaction forces.



Software Required: Ansys 14.5.

Procedure:

Step 1: Ansys Utility Menu

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok –close.

Real constants –Add –ok –Real constant set no –1 –c/s area –900 –ok. Add –ok –
Real constant set no –2 –c/s area 600 –ok.

Material Properties –Material Models –Structural –Linear –Elastic –Isotropic –EX –
2e5 –ok,

Material –New model –Define material ID –2 –ok –Structural –Linear –Elastic
– Isotropic –EX –0.7e5 –ok –close.

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z
location in CS –600, 0, 0 –Apply (second node is Created) - x, y, z location in CS –
1100, 0, 0 (third node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –Apply -
pick 2 & 3 (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes- pick node 1 –Apply –DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes - pick node 3 –Apply – direction of For/Mom –FX –Force/Moment value –500 (+ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1–ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1 – Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.

Comparison between theoretical and Ansys values:

Particulars	Ansys			Theoretical		
	Node1	Node2	Node3	Node1	Node2	Node3
Deformation						
Stress						
Reaction						

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

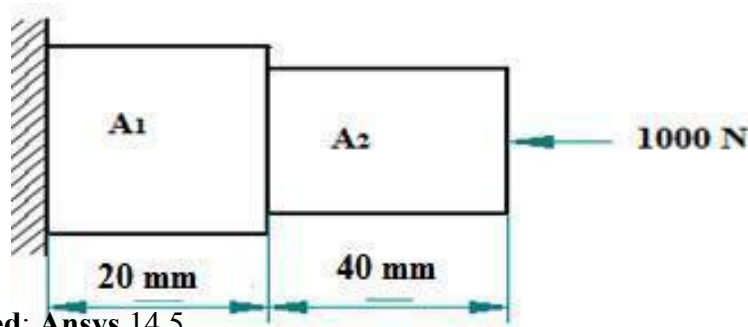
Conclusion: Ansys simulation and the software results are near to theoretical or analytical results.

AIM:

To perform displacement and stress analysis for the given stepped bar using Ansys simulation and analytical expressions.

Problem Description:

Find nodal displacement, stress in the element & reaction forces for the following problem. Given Data: $E = 2 \times 10^5 \text{ N/mm}^2$, $\gamma = 0.3$ (Poisson's Ratio). $A_1 = 40 \text{ mm}^2$, $A_2 = 20 \text{ mm}^2$



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real constants –Add –ok –Real constant set no –1 –c/s area –40 –
ok. Add –ok –Real constant set no –2 –c/s area
– 20 –ok.

Material Properties –Material models –Structural –Linear –Elastic –Isotropic –EX –
2e5 –ok - close

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z
location in CS –20, 0, 0 –Apply (second node is Created) - x, y, z location in CS –
60, 0, 0 (third node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –Apply
- pick 2 & 3 (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes - pick node 1 –Apply –

DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes - pick node 3 –Apply
– direction of For/Mom –FX –Force/Moment value –1000 (-ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1
–ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1
– Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal
solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be
displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.

Particulars	Ansys			Theoretical		
	Node1	Node2	Node3	Node1	Node2	Node3
Deformation						
Stress						
Reaction						

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

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

AIM:

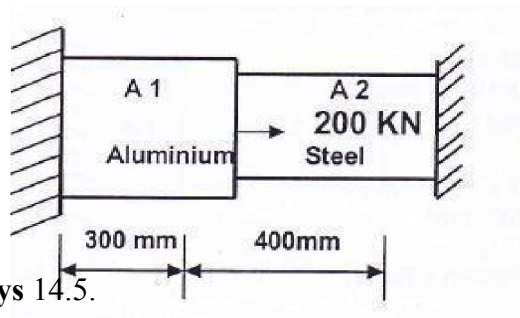
To perform displacement and stress analysis for the given stepped bar using Ansys simulation and analytical expressions.

Problem Description:

Find nodal displacement, stress in each element & reaction forces for the following problem Given

Data: 1) $E_1 = 70 \times 10^3 \text{ N/mm}^2$, $A_1 = 2400 \text{ mm}^2$. 2) $E_2 = 200 \times 10^3 \text{ N/mm}^2$, $A_2 = 600 \text{ mm}^2$

, $\nu = 0.3$ (Poisson's Ratio)



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real constants –Add –ok –Real constant set no –1 –c/s area –
2400 –ok. Add –ok –Real constant set no –2 –
c/s area –600 –ok.

Material Properties –Material Models –Structural –Linear –Elastic –Isotropic
–EX – 70e3 –PRXY –0.3 - ok

Material –New model –Define material ID –2 –ok –Structural –Linear –Elastic
– Isotropic –EX –200e3 –PRXY –0.3 - ok –close.

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x,
y, z location in CS –300, 0, 0 –Apply (second node is Created) - x, y, z
location in CS –700, 0, 0 (third node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –ok
Elements Attributes- Change Real Constant Set No 2 & Material No 2- Ok- Elements –
Auto Numbered - Thru Nodes - Pick 2 & 3 Node-Apply-Ok.

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes- pick node 1 & 3 –Apply –DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes- pick node 2 –Apply – direction of For/Mom –FX –Force/Moment value –200000 (+ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Resultem's–BydataSequence inum –LS –LS1 –ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1 – Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.

Comparison between theoretical and Ansys values:

Particulars	Ansys			Theoretical		
	Node1	Node2	Node3	Node1	Node2	Node3
Deformation						
Stress						
Reaction						

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increasing number of nodes and elements to match the theoretical or analytical results.

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

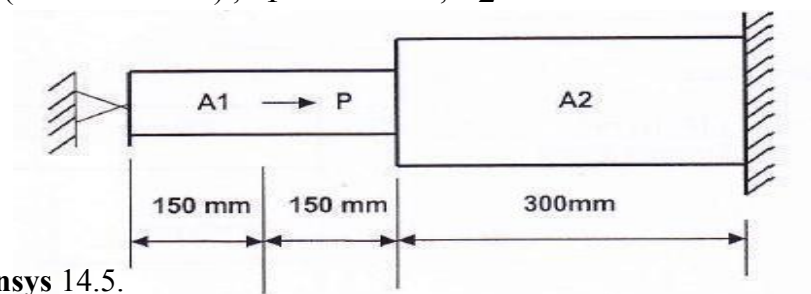
AIM:

To perform displacement and stress analysis for the given stepped bar using Ansys simulation and analytical expressions.

Problem Description:

Consider the bar loaded as shown. Determine the following

1) Nodal displacement, 2) Stress in the element, 3) Reaction forces. Given Data: $P = 300 \text{ KN}$, $E = 200 \times 10^9 \text{ N/m}^2$, $\gamma = 0.3$ (Poisson's Ratio), $A_1 = 250 \text{ mm}^2$, $A_2 = 400 \text{ mm}^2$



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finit stn 180 –ok – close.

Real Constants –Add –ok –Real constant set no –1 –c/s area

–250 –ok Add –ok –Real constant set no –

2 –c/s area –400 –ok

Material Properties –Material Models –Structural –Linear –Elastic –

Isotropic –EX – 200×10^9 –PRXY –0.3 - ok

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –

x, y, z location in CS –150, 0, 0 –Apply (second node is Created) - x, y, z

location in CS –300, 0, 0 –Apply (third node is Created), x, y, z location in

CS –600, 0, 0 –ok (fourth node is Created).

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –

Apply - pick 2 & 3 –ok.

Modeling –Create –Elements –Element Attributes –Change Real Constant Set No 2 –

ok - pick 3 & 4 (elements are Created through nodes).

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes - pick node 1 & 4 –Apply –DOFs to be constrained –All DOF –ok.

Define loads –Apply –Structural –Force/Moment –on Nodes - pick node 2 –Apply – direction of For/Mom –FX –Force/Moment value –300e3 (+ve value) –ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1–ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1 – Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

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

List Results –Nodal solution –DOF solution –Displacement Vector Sum –ok. (Nodal solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.**Comparison between theoretical and Ansys values:**

Particulars	Ansys				Theoretical			
	Node1	Node2	Node3	Node4	Node1	Node2	Node3	Node4
Deformation								
Stress								
Reaction								

Results:

The analysis of the bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increasing number of nodes and elements to match the theoretical or analytical results.

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

AIM:

To perform displacement and stress analysis for the given stepped bar using Ansys simulation and analytical expressions.

Problem Description:

An axial load $P = 300 \times 10^3 \text{ N}$ is applied at 20°C to the rod as shown. The temperature is raised to 60°C . Determine the nodal displacement, stress in the element, reaction forces.

Given Data:

1: Aluminum:

$$E = 70 \times 10^9 \text{ N/m}^2$$

$$\gamma = 0.3 \text{ (Poisson's Ratio)}$$

$$A = 900 \text{ mm}^2$$

$$\alpha = 23 \times 10^{-6} ^\circ\text{C}$$

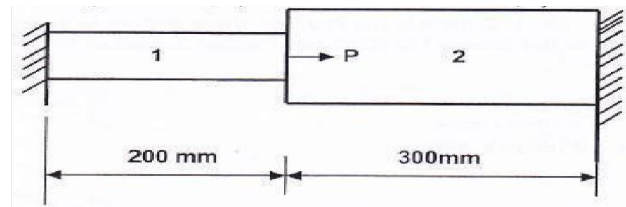
2: Steel:

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

$$\gamma = 0.3 \text{ (Poisson's Ratio)}$$

$$A = 1200 \text{ mm}^2$$

$$\alpha = 11.7 \times 10^{-6} ^\circ\text{C}$$



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Link –3D finite stn 180 –ok – close.

Real constants –Add –ok –Real constant set no –1 –c/s area –900 –ok

Add –ok –Real constant set no –2 –c/s area –1200 –ok –close.

Material Properties –Material models –Structural –Linear –Elastic –Isotropic –EX –
70e3 –PRXY –0.3 –ok

Thermal Expansion - Secant Coefficient - Isotropic - ALPX-23E-6 –ok.

Material - Define material ID –2 –ok —Structural –Linear –Elastic –Isotropic –EX –
– 200e3 –PRXY –0.3 –ok

Thermal Expansion-Secant Coefficient - Isotropic –ALPX -11.7E-6-Ok

Modeling –Create –Nodes –In Active CS –Apply (first node is Created) –x, y, z

location in CS –200, 0, 0 –Apply (second node is Created) - x, y, z location in CS –500,
0, 0 –Apply (third node is Created)

Modeling –Create –Elements –Auto numbered –Thru Nodes –pick 1 & 2 –ok -
Elements Attributes- Change Real Constant Set No 2 & Material No 2- Ok- Elements –
Auto Numbered - Thru Nodes - Pick 2 & 3 Node -Apply- Ok.

Step 4: Solution

Define loads –Settings –Reference Temperature –20 - Ok.

Define Loads - Apply-Structural –Temperature - On Elements - Select Both Elements -
Apply - VAL 1 Temperature at Location N = 60 –ok.

Define Loads - Apply - Structural - Displacement –On Nodes - Pick Node No 1 & 3 -
Apply -All DOF -Apply - OK.

Define Loads –Apply –Structural - Force / Moments –On Nodes - Pick 2nd Node
- Apply - Select FX = 300e3 - Apply - Ok.

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

Step 5: General Post Processor

Element table –Define table –Add –_Results–BydataSequence item‘num–LS–LS1–ok.

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Line Element Results –Elem table item at node I –LS1
– Elem table item at node J –LS1 –ok (Line Stress diagram will be displayed).

Element Table - List Element Table - Select Stress - Ok.

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

List Results –Nodal Solution –DOF solution –Displacement Vector Sum –ok. (Nodal
solution will be displayed with the node numbers).

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be
displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok.

Comparison between theoretical and Ansys values:

Particulars	Ansys				Theoretical			
	Node1	Node2	Node3	Node4	Node1	Node2	Node3	Node4
Deformation								
Stress								
Reaction								

Results:

The analysis of the stepped bar was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increasing number of nodes and elements to match the theoretical or analytical results.

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

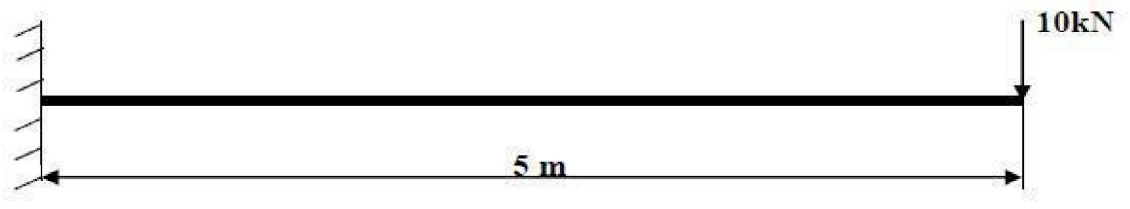
EXPT;11 Stress analysis of Beams

AIM:

Draw the shear force and bending moment diagrams for the given beam due to applied load .

Problem Description:

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



Software Required: Ansys 14.5.

Procedure:

Step 1: Ansys Utility Menu

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –BEAM –2 noded 188 –ok- close.

Material Properties –Material Models –Structural –Linear –Elastic –
Isotropic –EX – 210e3 –PRXY –0.27 –ok –close.

Sections –Beam –Common Section –B = 200, H = 300 –ok.

Modeling –Create –Nodes –In Active CS –Apply (first node is created) –
x, y, z location in CS –5000, 0, 0 –ok (second node is Created).

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

Step 4: Solution

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

Define loads –Apply –Structural –Force/Moment –on Nodes - pick node 2
–Apply – direction of For/Mom –FY –Force/Moment value - -10000 (-ve
value) –ok.

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

Step 5: General Post Processor

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Nodal solu –DOF solution –displacement vector sum – ok.

Element table –Define table –Add –_Results–BydataSequence item‘num–SMISC –SMISC, 6 –Apply, By Sequence num –SMISC –SMISC, 19 –Apply, By Sequence num – SMISC –SMISC, 2 –Apply, By Sequence num –SMISC –SMISC, 15 –ok – close.

NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19, for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.

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).

Plot Results –Contour plot –Line Element Results –Elem table item at node I –SMIS2 –Elem table item at node J –SMIS15 –ok (bending moment diagram will be displayed).

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

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed results –DOF solution –USUM –ok.

Comparison b/w Theoretical & Ansys Results

Particulars	Ansys		Theoretical	
	Maximum	Minimum	Maximum	Minimum
Shear force				
Bending Moment				

Results:

The analysis of the cantilever Beam was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

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

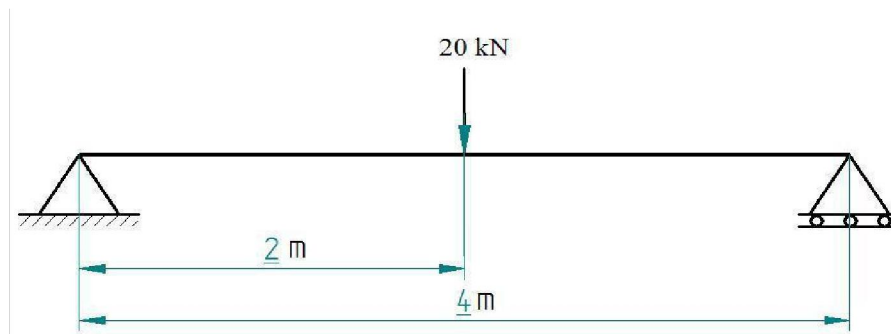
Stress analysis of Beams

AIM:

Draw the shear force and bending moment diagrams for the given beam due to applied load.

Problem Description:

Compute the shear force and bending moment diagrams for the beam shown. Assume rectangular c/s area of $0.2 \text{ m} \times 0.3 \text{ m}$, Young's modulus of 210 GPa , Poisson's ratio 0.27 .



Software Required: Ansys 14.5.

Procedure:

Step 1: Ansys Utility Menu

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL –ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –BEAM –2 noded 188 –ok- close.

Material Properties –Material Models –Structural –Linear –Elastic – Isotropic –EX – 210×10^3 –PRXY –0.27 –ok –close.

Sections –Beam –Common Section –B = 200, H = 300 –ok.

Modeling –Create –Keypoints –In Active CS –Apply (first node is Created) –x, y, z location in CS –2000, 0, 0 - Apply (second node is Created) –4000, 0, 0 –ok - (third node is Created).

Modeling –Create –Lines –Straight lines–In Active Coord –pick keypoints 1 & 2, pick keypoints 2 & 3 –ok.

Meshing –Size cntrls –Manual size –Global –Size –Element edge length –5 –ok.

Meshing –Mesh –Lines –Pick all –ok.

Step 4: Solution

Define loads –Apply –Structural –Displacement –on keypoint - pick keypoint 1 – Apply –DOFs to be constrained –UX, UY, UZ, ROTX, ROTY –Apply –pick keypoint 3 –Apply

–DOFs to be constrained –UY–ok.

Define loads –Apply –Structural –Force/Moment –on keypoint - pick keypoint 2 –
Apply –direction of For/Mom –FY –Force/Moment value –20e3 (-ve value) –ok.

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

Step 5: General Post Processor

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Nodal solu –DOF solution –displacement vector sum -ok.

Element table –Define table –Add –_Results–BydataSequence item‘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.

**NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19,
for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.**

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).

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). **List**

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

Step 6: PlotCtrls –Animate –Deformed results –DOF solution –USUM –ok.

Comparison b/w Theoretical & Ansys Results

Particulars	Ansys		Theoretical	
	Maximum	Minimum	Maximum	Minimum
Shear force				
Bending Moment				

Results:

The analysis of the cantilever Beam was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

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

Experiment No: 13

Date:

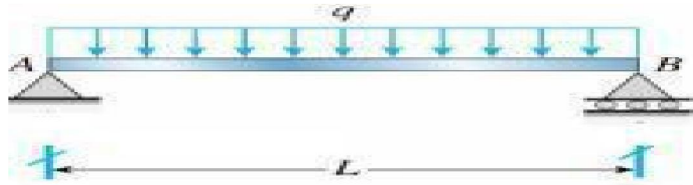
Stress analysis of Beams**AIM:**

Draw the shear force and bending moment diagrams for the given beam due to applied load.

Problem Description:

Draw the shearforce and bending moment diagram for the beam loaded as shown in figure.

Assume rectangular c/s area of 0.2 m * 0.3 m, $E = 200\text{GPa}$, Poisson's ratio = 0.3, Length (L) = 2m, $q = 10\text{kN/m}$.



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –BEAM –2 noded 188 –ok- close.

Material Properties –Material models –Structural –Linear –Elastic –
Isotropic –EX – 200e3 –PRXY –0.3 –ok –close.

Sections –Beam –Common Section –B = 200, H = 300 –ok.

Modeling –Create –Keypoints –In Active CS –Apply (first node is Created)

–x, y, z location in CS –2000, 0, 0 –ok (second node is Created).

Modeling –Create –Lines –Straight lines–in Active Coord –pick keypoints 1 & 2, pick ok.

Meshing –Size Cntrls –Manual size –Global –Size –No of element divisions –100 –ok.

Meshing –Mesh –Lines –Pick all –ok.

Step 4: Solution

Define loads –Apply –Structural –Displacement –on keypoint - pick keypoint

1 – Apply –DOFs to be constrained –UX, UY, UZ, ROTX, ROTY –Apply –

pick keypoint 2 –Apply –DOFs to be constrained –UY–ok.

Define loads –Apply –Structural –on beams –select full line –Load key –2 - Pressure value at node I –10e3 - ok.

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

Step 5: General Post Processor

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Nodal solu –DOF solution –displacement vector sum –ok.

Element table –Define table –Add –_Results–BydataSequence item‘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.

NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19, for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.

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).

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).

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

List Results –Nodal loads –items to be listed –All items –ok (Nodal loads will be displayed with the node numbers).

Step 6: PlotCtrls –Animate –Deformed results –DOF solution –USUM –ok.

Comparison b/w Theoretical & Ansys Results

	Ansys		Theoretical	
	Maximum	Minimum	Maximum	Minimum
Shear force				
Bending Moment				

Results:

The analysis of the Simply Supported Beam was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

Conclusion: Ansys simulation and the software results are near to theoretical or analytical results

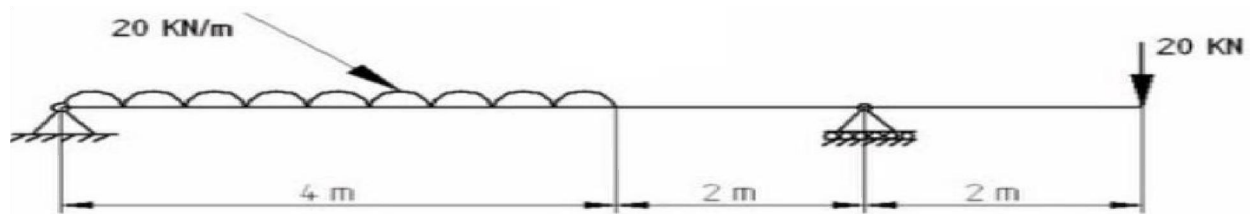
Stress analysis of Beams

AIM:

Draw the shear force and bending moment diagrams for the given beam due to applied load.

Problem Description:

Draw the shear force and bending moment diagram for the beam loaded as shown in figure. Assume rectangular c/s area of $0.2 \text{ m} \times 0.3 \text{ m}$, $E = 200 \text{ GPa}$, Poisson's ratio = 0.3.



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –BEAM –2 noded 188 –ok- close.

Material Properties –Material models –Structural –Linear –Elastic – Isotropic –EX – 200e3 –PRXY –0.3 –ok –close.

Sections –Beam –Common Section –B = 200, H = 300 –ok.

Modeling –Create –Keypoints –In Active CS –Apply (first node is Created) –x, y, z location in CS –4000, 0, 0 - Apply (second node is Created) –6000, 0, 0 –ok - (third node is Created) - 8000, 0, 0 –ok - (fourth node is Created).

Modeling –Create –Lines –Straight lines–in Active Coord –pick keypoints 1 & 2, pick keypoints 2 & 3, pick keypoints 3 & 4 –ok.

Meshing –Size cntrls –Manual size –Global –Size –Element edge length –5 –ok.

Meshing –Mesh –Lines –Pick all –ok.

Step 4: Solution

Define loads –Apply –Structural –Displacement –on keypoint - pick keypoint 1 – Apply –DOFs to be constrained –UX, UY, UZ, ROTX, ROTY –Apply –pick keypoint 3 –Apply –DOFs to be constrained –UY–ok.

Define loads –Apply –Structural –Force/Moment –on keypoint - pick keypoint 4 –
 Apply –direction of For/Mom –FY –Force/Moment value –20e3 (-ve value) –ok.
Define loads –Apply –Structural –on beams –select box option –select from
 1st keypoint to 2nd –Load key –2 - Pressure value at node I –20e3 - ok.
Solve –Current LS –ok (Solution is done is displayed) –close.

Step 5: General Post Processor

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Nodal solu –DOF solution –displacement vector sum –ok.

Element table –Define table –Add –_Results–BydataSequence item ‘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.

**NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19,
 for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.**

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).

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).

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

Step 6: PlotCtrls –Animate –Deformed results –DOF solution –USUM –ok.

Comparison b/w Theoretical & Ansys

	Ansys		Theoretical	
	Maximum	Minimum	Maximum	Minimum
Shear force				
Bending Moment				

Results: The analysis of the Simply Supported Beam was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

Conclusion: Ansys simulation and the software results are near to theoretical or analytical results

Stress analysis of Beams

AIM:

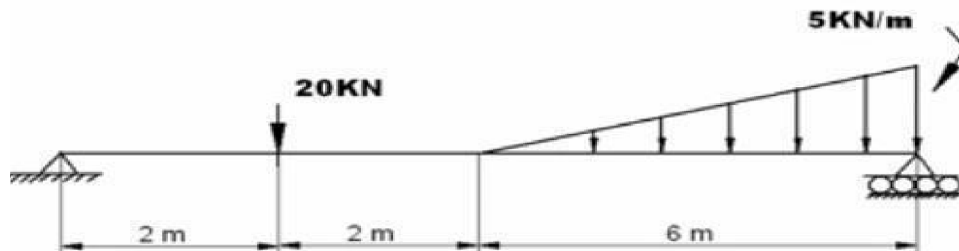
Draw the shear force and bending moment diagrams for the given beam due to applied load.

Problem Description:

Draw the shear force and bending moment diagram for the beam loaded as shown in figure.

Assume rectangular c/s area of $0.2 \text{ m} \times 0.3 \text{ m}$, $E = 200 \text{ GPa}$,

Poisson's ratio = 0.3 .



Software Required: Ansys 14.5.

Procedure:**Step 1: Ansys Utility Menu**

File –clear and start new –do not read file –ok –yes.

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL - ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –BEAM –2 noded 188 –ok- close.

Material Properties –Material models –Structural –Linear –Elastic –Isotropic –EX – 200×10^3 –PRXY – 0.3 –ok –close.

Sections –Beam –Common Section –B = 200, H = 300 –ok.

Modeling –Create –Keypoints –In Active CS –Apply (first node is Created) –x, y, z location in CS –2000, 0, 0 - Apply (second node is Created) –4000, 0, 0 –ok - (third node is Created) - 10000, 0, 0 –ok - (fourth node is Created).

Modeling –Create –Lines –Straight lines–in Active Coord –pick keypoints 1 & 2, pick keypoints 2 & 3, pick keypoints 3 & 4 –ok.

Meshing –Size cntrls –Manual size –Global –Size –No of element divisions –100 –ok.

Meshing –Mesh –Lines –Pick all –ok.

Step 4: Solution

Define loads –Apply –Structural –Displacement –on keypoint - pick keypoint 1 – Apply –DOFs to be constrained –UX, UY, UZ, ROTX, ROTY –Apply –pick keypoint 4 –Apply –DOFs to be constrained –UY –ok.

Define loads –Apply –Structural –Force/Moment –on keypoint - pick keypoint 2 –

Apply –direction of For/Mom –FY –Force/Moment value –20e3 (-ve value) –ok.

Define loads –Apply –Structural –on beams –select box option –select from 1st keypoint to 2nd –Load key –2 - Pressure value at node I –0 - Pressure value at node J –5e3 - ok.

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

Step 5: General Post Processor

Plot Results –Deformed Shape –def+undeformed –ok.

Plot Results –Contour plot –Nodal solu –DOF solution –displacement vector sum –ok.

Element table –Define table –Add –_Results–BydataSequence item ‘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.

NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19, for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.

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).

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).

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

Step 6: PlotCtrls –Animate –Deformed results –DOF solution –USUM –ok.

Comparison b/w Theoretical & Ansys Results

	Ansys		Theoretical	
	Maximum	Minimum	Maximum	Minimum
Shear force				
Bending Moment				

Results:

The analysis of the Simply Supported Beam was carried out using the Ansys simulation and the software results were compared with theoretical or analytical results.

Verification/Validation: Verify the Ansys results by increasing number of nodes and elements to match the theoretical or analytical results.

Conclusion: Ansys simulation and the software results are near to theoretical or analytical results

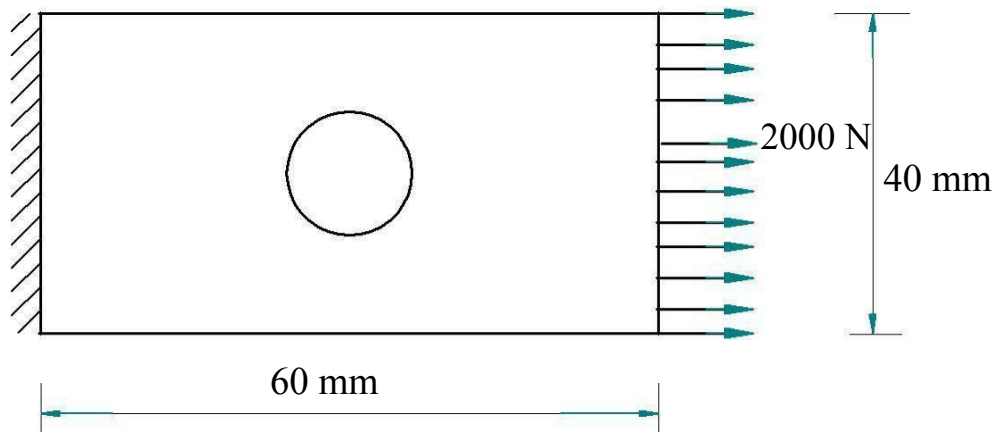
Stress Analysis of a Rectangular Plate with a circular Hole

AIM:

To determine the stress acting on a rectangular plate with a circular hole due to the applied external load.

Problem Description:

In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B. $E = 210\text{GPa}$, $t = 1\text{ mm}$, Poisson's $\nu = 0.3$, Diameter of the circle = 10 mm.



Software Required: Ansys 14.5.

Procedure:

Step 1: Ansys Utility Menu

File - Change Job name

File - Change Title

Step 2: Ansys Main Menu –Preferences

Select –STRUCTURAL –ok

Step 3: Preprocessor

Element type –Add/Edit/Delete –Add –Solid –Quad 4 node 182 –ok –Option – element behavior K3 –Plane stress with thickness –ok –close.

Real constants –Add –ok –Real constant set no –1 –Thickness –1 –ok.

Material Properties –material models –Structural –Linear –Elastic –Isotropic –EX – $210\text{e}3$ –PRXY –0.3 –ok –close.

Modeling –Create –Area –Rectangle –by dimensions –X1, X2, Y1, Y2 –0, 60, 0, 40 –ok.

Modeling –Create –Area –Circle –solid circle –X, Y, radius –30, 20, 5 –ok.

Modeling –Operate –Booleans –Subtract –Areas –pick area which is not to be deleted (rectangle) –Apply –pick area which is to be deleted (circle) –ok.

Meshing –Mesh Tool –Mesh Areas –Quad –Free –Mesh –pick all –ok.

Meshing –Mesh Tool –Refine –pick all –Level of refinement –3 –ok.

Step 4: Solution

Define loads –Apply –Structural –Displacement –on Nodes –select box –drag the left side of the area –Apply –DOFs to be constrained –ALL DOF

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.

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

Step 5: General Post Processor

Plot Results –Deformed Shape –def+undeformed –ok.

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

Step 6: PlotCtrls –Animate –Deformed shape –def+undeformed-ok

Result: Thus the performance of the stress analysis of a Rectangular Plate with a circular hole was analyzed and animated.

Verification/Validation: Verify the Ansys results by increaing number of nodes and elements to match the theoretical or analytical results.

Conclusion: Ansys simulation and the software results are near to theoretical or analytical results