Overview of Finite Element Method

BASIC CONCEPT

The basic idea in the finite element method is to find the solution of a complicated problem by replacing it by a simpler one. Since the actual problem is replaced by a simpler one in finding the solution, we will be able to find only an approximate solution rather than the exact solution. The existing mathematical tools will not be sufficient to find the exact solution (and sometimes, even an approximate solution) of most of the practical problems. Thus in the absence of any convenient method to fine even the approximate solution of a given problem, we have to prefer the finite element method. Moreover, in the finite element method, it will often be possible to improve or refine the approximate solution by spending more computational effort.

In the finite element method, the solution region is considered as built up of small, interconnected subregions called finite elements. As an example of how a finite element model might be used to represent a complex geometrical shape, consider the milling machine structure. Since it is very difficult to find the exact response (like stresses and displacements) of the machine under any specified cutting (loading) condition, this structure is approximated as composed of several pieces in the finite element method. In each piece or element, a convenient approximate solution is assumed and the conditions of overall equilibrium of the structure are derived. The satisfaction of these conditions will yield an approximate solution for the displacements and stresses.

ENGINEERING APPLICATIONS OF THE FINITE ELEMENT METHOD

The finite element method was developed originally for the analysis of aircraft structures. However, the general nature of its theory makes it applicable to a wide variety of boundary value problems in engineering. A boundary value problem is one in which a solution is sought in the domain (or region) of a body subject to the satisfaction of prescribed boundary (edge) conditions on the dependent variables or their derivatives. The table given below gives specific applications of the finite element method in the three major categories of boundary value problems, namely, (i) Equilibrium or steady state or time independent problems. (ii) Eigenvalue problems, and (iii) Propagation or transient problems.

In an equilibrium problem, we need to find the steady state displacement or stress distribution if it is a solid mechanics problem, temperature or heat flux distribution if it is a heat transfer problem, and pressure or velocity distribution if it is a fluid mechanics problem.

In eigenvale problems also, time will not appear explicitly. They may be considered as extensions of equilibrium problems in which critical values of certain parameters are to be determined in addition to the corresponding steady state configurations. In these problems, we need to find the natural frequencies or buckling loads and mode shapes if it is a solid mechanics or structures problem, stability of laminar flows if it is fluid mechanics problem, and response characteristics if it is an electrical circuit problem.

The propagation or transient problems are time dependent problems. This type of problem arises, for example, whenever we are interested in finding the response of a body under time varying force in the area of solid mechanics and under sudden heating or cooling in the field of heat transfer.

What is the finite element method

The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought. A continuous physical problem is transformed into a discretized finite element problem with unknown nodal values. For a linear problem a system of linear algebraic equations should be solved. Values inside finite elements can be recovered using nodal values.

Two features of the FEM are worth to be mentioned:
1) Piece-wise approximation of physical fields on finite elements provides good precision even with simple approximating functions (increasing the number of elements we can achieve any precision).
2) Locality of approximation leads to sparse equation systems for a discretized problem. This helps to solve problems with very large number of nodal unknowns.

How the FEM works

To summarize in general terms how the finite element method works we list main steps of the finite element solution procedure below.
1. Discretize the continuum. The first step is to divide a solution region into finite elements. The finite element mesh is typically generated by a preprocessor program. The description of mesh consists of several arrays main of which are nodal coordinates and element connectivities.
2. Select interpolation functions. Interpolation functions are used to interpolate the field variables over the element. Often, polynomials are selected as interpolation functions. The degree of the polynomial depends on the number of nodes assigned to the element.
3. Find the element properties. The matrix equation for the finite element should be established which relates the nodal values of the unknown function to other parameters. For this task different approaches can be used; the most convenient are: the variational approach and the Galerkin method.
4. Assemble the element equations. To find the global equation system for the whole solution region we must assemble all the element equations. In other words we must combine local element equations for all elements used for discretization. Element connectivities are used for the assembly process. Before solution, boundary conditions (which are not accounted in element equations) should be imposed.

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. The next few sections of this chapter cover general steps that are common to most analyses.

A typical ANSYS analysis has three distinct steps:

- Build the model.
- Apply loads and obtain the solution.
- Review the results.

Building a Model

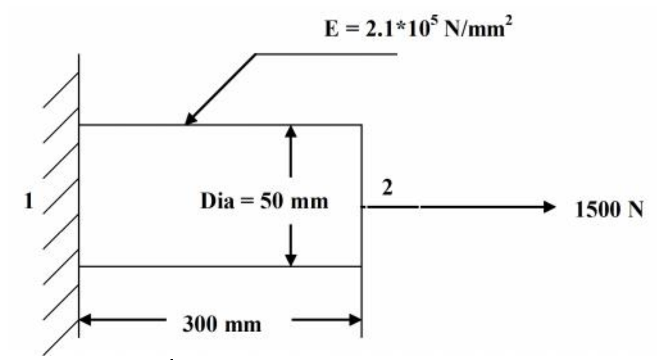
Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a jobname and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.

ANALYSIS OF BARS

Modeling & Stress analysis of Bars of constant cross section area, tapered cross section area and stepped bar

Exercise 1

Consider the bar shown in figure below. Determine 1) The nodal displacement 2) stress in each element 3) reaction forces



Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -xxx -ok

File -Change title -Enter new title -yyy -ok

Step 2 -Ansys Main Menu –Preferences

Select -STRUCTURAL -ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- Link -2D spar 1- ok -close

Real constants -Add/Edit/Delete -Add -ok -real constant set no -Area : 22/7*50**2/4- ok

Material properties - Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2.1 E5 – (If required enter PXY) - ok- enter-Close.

Step 4 -Preprocessor

Modeling -Create -Nodes -in active CS -apply (first node is created)- x,y,z location in CS -300 (x value w.r.t first node)-ok Create -elements - ele attributes -Auto numbered-thru nodes -pick 1 & 2- ok (elements are created through nodes).

Step 5 -Preprocessor

Loads - Define Loads - Apply -Structural Displacement -on nodes -pick node 1 -apply- DOF to be constrained -All DOF-ok

Loads - Define Loads - Apply - Force/Moment -on nodes -pick node 2 -apply- direction of Force/Moment- FX- Force/Moment value- 1500 (value) -ok

Step 6 -Ansys Main Menu -Solution

Solve -solve current LS -ok (If everything is ok -solution is done is displayed) -close

Step 7 -Ansys Main Menu -General Post Processor

Element table -define table -add-results data item- By sequence num- LS- LS, 1-ok

Step 8 -General Post Processor

Plot results - Contour Plots - Line elem results-element table item at node I -LSI -element table item at node J -LS 1- ok (line stress diagram will be displayed)

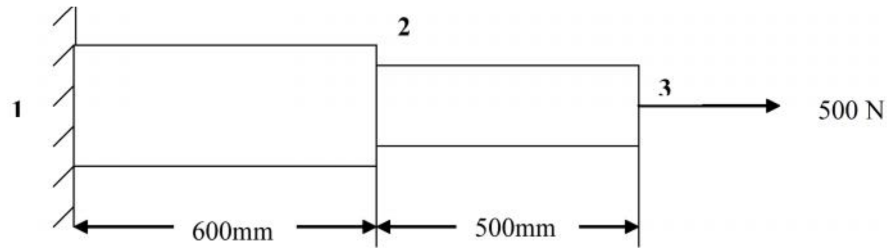
List results

Reaction solution -items to be listed -all items -ok (reaction forces will be displayed with the node numbers)

Nodal loads -loads to be listed -all items -ok (Nodal loads will be displayed):

Nodal solution -translation in UX direction -ok (for displacement in X- direction):

Exercise 2 Consider the bar shown in figure below (Stepped bar)



$E_1 = 2.0*10^5$ N/mm^2	$A_1 = 900$mm^2
$E_2 = 0.7*10^5$ N/mm^2	$A_2 = 600$ mm^2

Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -xxx –ok

File -Change title -Enter new title -yyy -ok

Step 2 -Ansys Main Menu -Preferences

Select -STRUCTURAL -ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- Link -2D spar 1- ok -close

Real constants -Add/Edit/Delete -Add -ok -real constant set no - real constant c/s area -900-apply set no -2- c/s area -600- ok-close

Material properties – Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2E5 – (If required enter PXY) - ok- enter

Material Model – Material Model Number 2 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 0.7E5 – (If required enter PXY) - ok- enter-Close.

Step 4 -Preprocessor

Modeling -Create -Nodes -in active CS -apply -x,y,z location in CS - 600- apply -1100- ok

Create -elements -ele attributes- Element type no- 1 Link1- Material no-1- Real const set no-1-ok-Auto numbered-thru nodes -pick 1 & 2-apply -ele attributes- Element type no- 1 Link1- Material no-2-Real const set no-2-ok- Auto numbered-thru nodes -pick 2 & 3 -ok (elements are created through nodes).

Step 5 -Preprocessor

Loads - Define Loads - Apply -Structural Displacement -on nodes -pick node 1 -apply- DOF to be constrained -All DOF-ok

Loads - Define Loads - Apply - Force/Moment -on nodes -pick node 3 -apply- direction of Force/Moment- FX- Force/Moment value- 500 (value) –ok

Step 6 -Ansys Main Menu -Solution

Solve -solve current LS -ok (If everything is ok displayed) solution is done is-Close

Step 7 -Ansys Main Menu -General Post Processor

Element table -define table -add-results data item- By sequence num- LS- LS ,1-ok

Step 8 -General Post Processor

Plot results- Contour Plots - Line elem results-element table item at node I -LS1 -element table item at node J -LS 1- ok (stress diagram will be displayed)

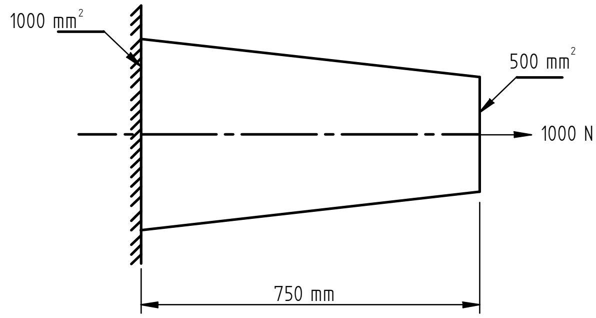
List results

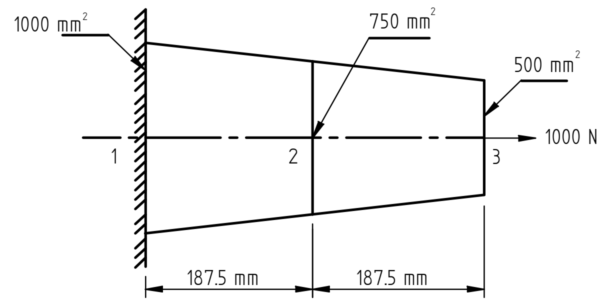
Reaction solution -items to be listed -all items -ok (reaction forces will be displayed with the node numbers)

Nodal loads -loads to be listed -all items -ok (Nodal loads will be displayed)

Nodal solution -translation in UX direction -ok (for displacement in X- direction)

Exercise 3 Consider the bar shown in figure below (Tapered bar)





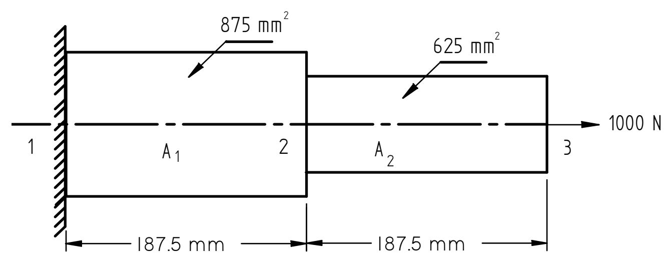
$E = 2.0*10^5$ N/mm^2 $A_1 = 1000$ mm^2 $A_2 = 500$ mm^2

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

(1000+500)/2 = 750 mm^2

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

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



Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -xxx –ok

File -Change title -Enter new title -yyy -ok

Step 2 -Ansys Main Menu -Preferences

Select -STRUCTURAL -ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- Link -2D spar 1- ok -close

Real constants -Add/Edit/Delete -Add -ok -real constant set no - real constant c/s area -875-apply set no -2- c/s area -625- ok-close

Material properties – Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2E5 – (If required enter PXY) - ok- enter

Step 4 -Preprocessor

Modeling -Create -Nodes -in active CS -apply -x,y,z location in CS - 1875- apply -375- ok

Create -elements -ele attributes- Element type no- 1 Link1- Material no-1- Real const set no-1-ok-Auto numbered-thru nodes -pick 1 & 2-apply - pick 2 & 3 -ok (elements are created through nodes).

Step 5 -Preprocessor

Loads - Define Loads - Apply -Structural Displacement -on nodes -pick node 1 -apply- DOF to be constrained -All DOF-ok

Loads - Define Loads - Apply - Force/Moment -on nodes -pick node 3 -apply- direction of Force/Moment- FX- Force/Moment value- 1000 (value) –ok

Step 6 -Ansys Main Menu -Solution

Solve -solve current LS -ok (If everything is ok displayed) solution is done is-Close

Step 7 -Ansys Main Menu -General Post Processor

Element table -define table -add-results data item- By sequence num- LS- LS 1-ok

Step 8 -General Post Processor

Plot results- Contour Plots - Line elem results-element table item at node I -LS1 -element table item at node J -LS 1- ok (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)

Nodal loads -loads to be listed -all items -ok (Nodal loads will be displayed)

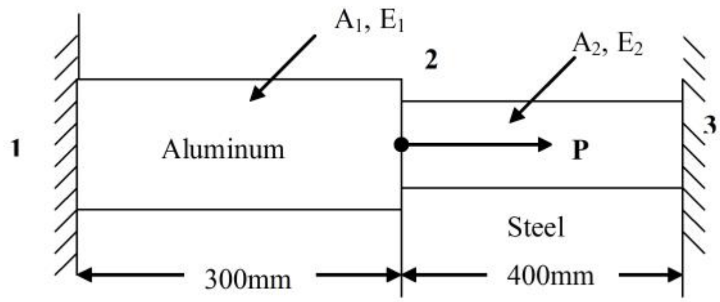
Nodal solution -translation in UX direction -ok (for displacement in X- direction)

EXERCISE PROBLEMS

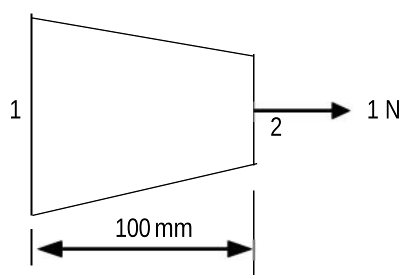
Exercise 4: Determine the nodal displacements, stress in each material, and the reaction forces

A_1 = 2400 mm^2, A_2 = 600 mm^2, E_1= 70 x 10^9 N/m^2, E_2= 200 x 10^9 N/m^2. An axial load

P = 200 x 10^3 N is applied as shown.



Exercise 5: Tapered bar, use two elements



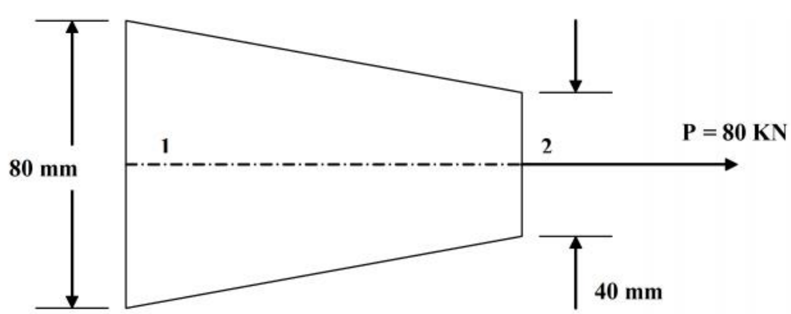
E = 2*10^5 N/mm^2

A_1 = 200 mm^2 A_2 = 100 mm^2

(Hint: Take element Beam 2D tapered 54)

Exercise 6: The steel bar shown in figure has length of 500 mm, under an axial load of 80 KN. Find its extension. Take E = 2 *10^5 N/mm^2.

.



ANALYSIS OF TRUSS

Exercise 1

AIM: To carry out analysis of TRUSS shown in FIG. and to determine the reactions at supports and forces on links by using FEM package.

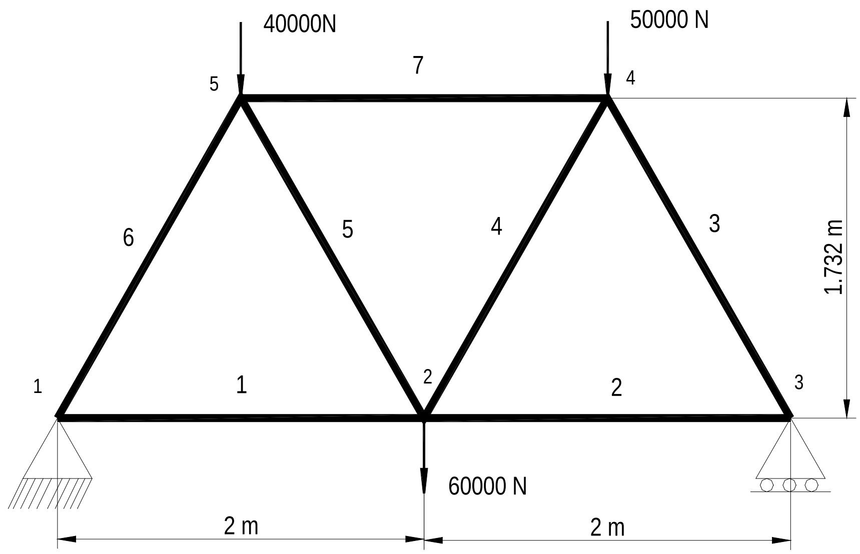
Material Properties:

Young's Modulus : 2 x 10^{11} N/m^2 or 2E11 N/m^2

Poisson's Ratio : 0.3

Type of Element : LINK1 (LINK - 2D SPAR)

Real Constants : Area = 5E-5 m^2



PROCEDURE

1. Select Preferences - STRUCTURAL
2. Select PREPROCESSOR - ELEMENT TYPE- ADD - LINK - 2D SPAR .
3. Select PREPROCESSOR - REAL CONSTANTS- Enter Area
4. Select PREPROCESSOR - MATERIAL PROPERTIES- ISOTROPIC-Enter Young's Modulus & Poissons Ratio

Modelling:

5. Select PREPROCESSOR- CREATE- KEYPOINTS- IN ACTIVE CS - Enter 5 Keypoints with coordi nates (0,0,0), (2,0,0), (4,0,0), (3,1.732,0) and (1,1.732,0).
6. Select PREPROCESSOR- CREATE- NODES-ON KEYPOINTS - Select the key points 1 to 5.
7. Select PREPROCESSOR- CREATE- ELEMENTS- Auto Numbered- THROUGH NODES- Select Nodes 1-2, 2-3, 3-4, 2-4, 2,5, 1-5 and 4-5.

Boundary Conditions:

8. Select SOLUTION - APPLY-DISPLACEMENTS- ON NODES - Select Nodes at 1 - Fix All DOF.
   Select SOLUTION - APPLY-DISPLACEMENTS- ON NODES - Select Nodes at 3 - Fix U_y.
9. Select SOLUTION - APPLY-FORCE - ON NODES - Select Node at 2 - Force -60000 N (downward)
   Select SOLUTION - APPLY-FORCE - ON NODES - Select Node at 4 - Force -50000 N (downward)
   Select SOLUTION - APPLY-FORCE - ON NODES - Select Node at 5 - Force -40000 N (downward)

Solving

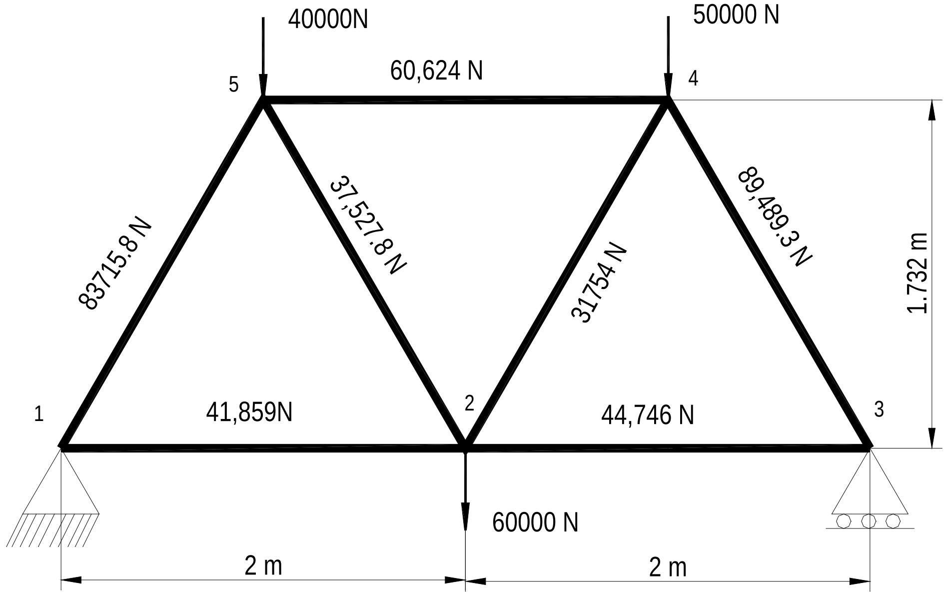
10. Select SOLUTION - SOLVE - CURRENT LS

Results

11. Select GENERAL POST PROCESSOR - LIST RESULTS- ELEMENT SOLUTION -Nodal Force Data- All Forces FORC - OK

Select GENERAL POST PROCESSOR - LIST RESULTS- REACTION SOLUTION -All Items - OK

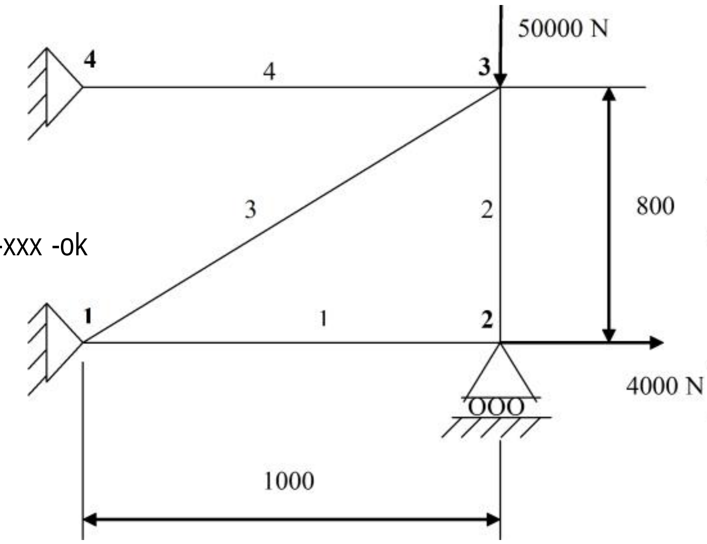
12. Compare the results with theoretical Calculation (Shown in sketch) .



Element	Nodes	F_x	F_y	F FEM	F Theoretical

Show Theoretical Calculations with detailed Force Analysis, Tabulate and compare with FEM Results.

Exercise 2: Consider the four bar truss shown in figure. For the given data, find 1) stress in each element 2) reaction forces 3) Nodal displacement. E = $2x10^5$ KN/ mm^2 A = 10 mm^2



Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -xxx -ok

File -Change title -Enter new title -yyy -ok

Step 2 -Ansys Main Menu -references

Select -STRUCTURAL -ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- Link -2D spar 1- ok –close

Real constants -Add -ok -real constant set no -1- c/s area -1- ok

Material properties - Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2E5 – (If required enter PXY) - ok- enter-Close.

Step 4 Preprocessor

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) -1000,800 (x & y value w.r.t first node)- apply (third node is created) -0, 800(x & y value w.r.t first node)- ok (fourth node is created)

Create -elements -ele attributes- Element type no- 1 Link1- Material no-1- Real const set no-1-ok-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).

Step 5 -Preprocessor

Loads - Define Loads - Apply -Structural Displacement -on nodes -pick nodes 1 & 4 - apply- DOF to be constrained -All DOF-ok -on nodes -pick nodes 2 - apply- DOF to be constrained -UY -ok .

Loads - Define Loads - Apply - Force/Moment -on nodes -pick node 2 -apply- direction of Force/Moment- FX- Force/Moment value- 4000(value) -apply- pick node -3- apply direction of Force/Moment- FY- Force/ Moment value- -50000(value) -ok

Step 6 -Ansys Main Menu -Solution

Solve -solve current LS -ok (If everything is ok -solution is done is displayed) -close

Step 7 -Ansys Main Menu -General Post Processor

Element table -define table -add-results data item- By sequence num- LS- LS1-ok

Step 8 -General Post Processor

Plot results - Contour Plots - Line elem results-element table item at node I -LS1 -element table item at node J -LS 1- ok (Stress diagram will be displayed)

-deformed shape- pick def+undeformed -ok (deformed + undeformed shape of truss will be displayed)

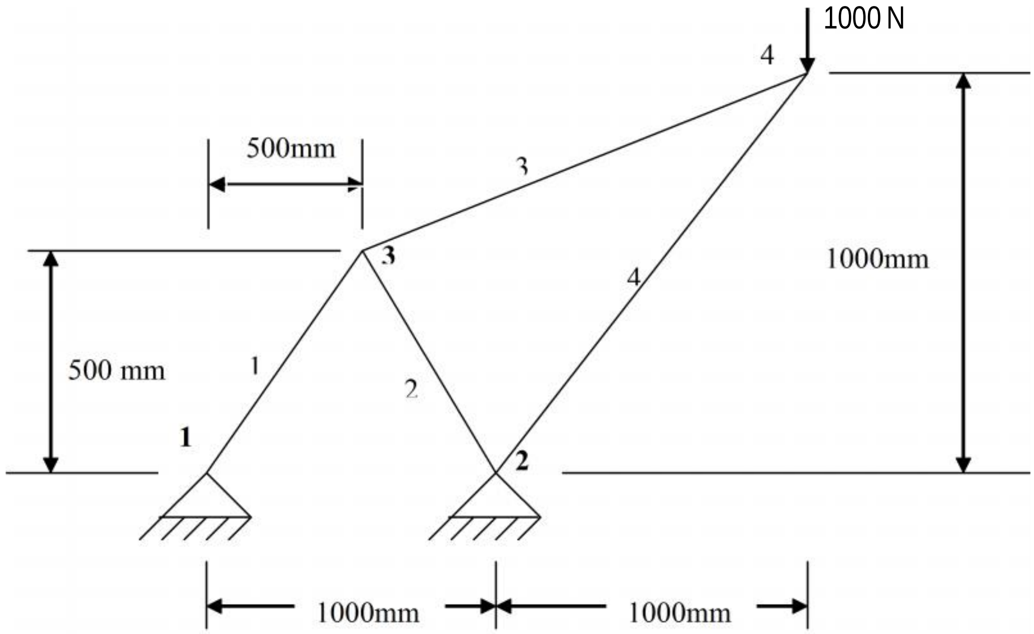
Plot ctrls -animate-deformed shape-def + undeformed-ok

List results

Reaction solution -items to be listed -all items -ok (reaction forces will be displayed with the node numbers)

Nodal solution-DOF solution-All DOF- ok (Nodal displacements will be displayed)

Exercise 3 : Find the nodal displacement and the internal stresses developed in the planar truss shown in figure when a vertically downward load of 1000 N is applied as shown. The Pertaining data given below.



Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -xxx -ok

File -Change title -Enter new title -yyy –ok

Step 2 -Ansys Main Menu -Preferences

Select -STRUCTURAL –ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- Link -2D spar 1- ok –close

Real constants -Add -ok -real constant set no -1- c/s area -200- apply- real constant set no -2- c/s area -100- ok

Material properties - Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2E5 – (If required enter PXY) - ok- enter-Close.

Step 4 -Preprocessor

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) -500,500 (x & y value w.r.t first node)- apply (third node is created) -1000, 1000(x & y value w.r.t first node)- ok (fourth node is created), Create -elements -ele attributes- Element type no- 1 Link1- Material no-1- ,"Real const set no-1-ok-Auto numbered-thru nodes -pick 1 & 3-apply -pick2 & 3 -apply-element attributes- Element type no- 1 Link1- Material no- I- Real const set no-2-ok-Auto numbered-thru nodes -pick 3 & 4-apply -pick 2 & 4 -ok (elements are created through nodes).

Step 5 -Preprocessor

Loads - Define Loads - Apply -Structural Displacement -on nodes -pick nodes 1 & 2 -apply- DOF to be constrained -All DOF-ok .

Loads - Define Loads - Apply –Force/Moment -on nodes -pick node 4 -apply- direction of Force/Moment-FY -Force/Momentum value- -I0000(value) -ok

Step 6 -Ansys Main Menu –Solution

Solve -solve current LS -ok (If everything is ok -solution is done is displayed) –close

Step 7 -Ansys Main Menu -General Post Processor

Element table -define table -add-results data item- By sequence num- LS-LS 1-ok

Step 8 -General Post Processor

Plot results - Contour Plots - Line elem results-element table item at node I -LSI –element table item at node J -LS 1- ok (Stress diagram will be displayed)-deformed shape- pick def+undeformed -ok (deformed + undeformed shape of truss will be displayed)

Plot ctrls -animate-deformed shape-def+undeformed-ok

List results

reaction solution -items to be listed -all items -ok (reaction forces will be displayed with the node numbers)

Nodal solution-DOF solution-All DOF- ok (Nodal displacements will be displayed)

ANALYSIS OF BEAMS

Exercise 1

AIM: To carry out analysis of beams with different end conditions and loads by using FEM package and comparing with theoretical results.

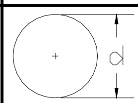
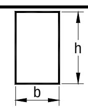
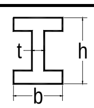
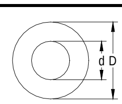
Material Properties:

Young's Modulus : 2×10^{11} N/m^2 or 2e11 N/m^2

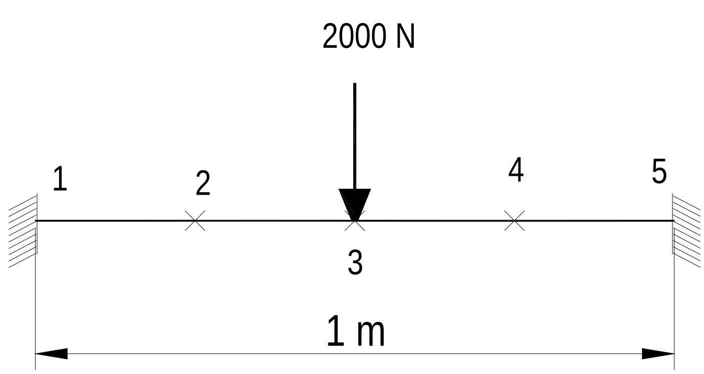
Poisson's Ratio : 0.3

Type of Element : Beam 2d Elastic

Beam Sections and Constants :

Cross Section	Figure	Dimension m	Area m^2		Moment of Inertia m^4		Height m
A		d=30 mm =0.03 m	$\frac{\text{®}\, d^2}{4}$	7.069e-4	$\frac{\text{®}\, d^4}{64}$	3.98e-8	0.03 m
B		b=0.02 m h=0.04 m	$b \times h$	8.0e-4	$\frac{b \times h^3}{12}$	1.0667e-7	0.04 m
C		b=0.02 m h=0.04 m t=0.005 m		3.5e-2		7.292e-8	0.04 m
D		D=0.05 m d=0.02 m	$\frac{\text{®}\left(D^2 - d^2\right)}{4}$	1.65e-3	$\frac{\text{®}\left(D^4 - d^4\right)}{64}$	2.989e-9	0.05 m
E							

CASE 1: Beam Fixed at Both ends and centre Load



Max. Deflection is at centre (3) given by $y_{max} = \frac{FL^3}{192EI}$

PROCEDURE

1. Select Preferences - STRUCTURAL
2. Select PREPROCESSOR - ELEMENT TYPE- ADD - BEAM -2D ELASTIC.
3. Select PREPROCESSOR - REAL CONSTANTS- Enter Area, Area Moment of Inertia, and Height of the beam corresponding to Cross Section A.
4. Select PREPROCESSOR - MATERIAL PROPERTIES- Material Model ISOTROPIC-Enter Young's Modulus & Poissons Ratio

Modelling:

5. Select PREPROCESSOR- CREATE- KEYPOINTS- IN ACTIVE CS - Enter 5 Keypoints with coordi nates (0,0,0), (0.25,0,0), (0.5,0,0), (0.75,0,0) and (1.0,0,0).
6. Select PREPROCESSOR- CREATE- NODES-ON KEYPOINTS - Select the key points 1 to 5.
7. Select PREPROCESSOR- CREATE- ELEMENTS- Auto Numbered- THROUGH NODES- Select Nodes 1-2, 2-3, 3-4 and 4-5.

Boundary Conditions:

8. Select SOLUTION - APPLY-DISPLACEMENTS- ON NODES - Select Nodes at 1 and 5 - Fix All DOF.
9. Select SOLUTION - APPLY-FORCE - ON NODES - Select Node at 3 - Force -2000 N (downward)

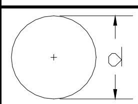
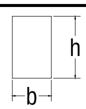
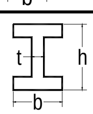
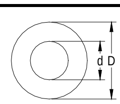
Solving

10. Select SOLUTION - SOLVE - CURRENT LS

Results

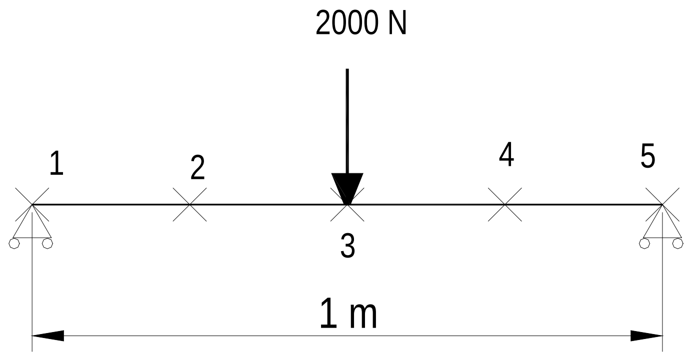
11. Select GENERAL POST PROCESSOR - PLOT RESULTS- NODAL SOLUTION -DOF solution - Trans lation U_y.
12. Compare the results with theoretical Calculation.

TABULAR COLUMN

Cross Section	Figure	Max.Deformation (Theoretical) m	Max.Deformation (FEM) m
A			
B			
C			
D			
E			

NOTE: To solve for other sections, Change the REAL CONSTANTs, SOLVE current LS (keeping the boundary conditions same) and look for the results.

CASE 2: Beam Freely Supported at Both ends and centre Load



Max. Deflection is at centre (3) given by $y_{\max} = \dfrac{FL^3}{48EI}$

PROCEDURE:

1. Follow Steps 1 through 7 specified in previous section
2. Select - SOLUTION- APPLY - DISPLACEMENTS - ON NODES. Select Nodes 1 and 5 - Constrain U_y (i.e. motion in y direction)
3. Select SOLUTION - APPLY-FORCE - ON NODES - Select Node at 3 - Force -2000 N (downward)

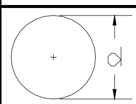
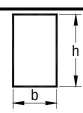
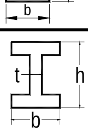
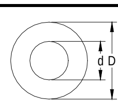
Solving

4. Select SOLUTION - SOLVE - CURRENT LS

Results

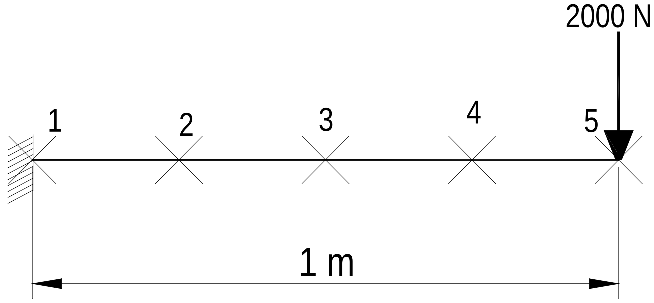
5.. Select GENERAL POST PROCESSOR - PLOT RESULTS- NODAL SOLUTION -DOF solution - Translation $U_{y.}$
6. Compare the results with theoretical Calculation.

TABULAR COLUMN

Cross Section	Figure	Max.Deformation (Theoretical) m	Max.Deformation (FEM) m
A			
B			
C			
D			
E			

NOTE: To solve for other sections, Change the REAL CONSTANTs, SOLVE current LS (keeping the boundary conditions same) and look for the results.

CASE 3: Beam Supported at one end and Load at the Other End



Max. Deflection is at centre (5) given by $y_{max} = \dfrac{FL^3}{3EI}$

PROCEDURE:

1. Follow Steps 1 through 7 specified in CASE 1.
2. Select - SOLUTION- APPLY - DISPLACEMENTS - ON NODE. Select Node 1 - Constrain All DOF's (i.e. motion in all directions)
3. Select SOLUTION - APPLY-FORCE - ON NODES - Select Node at 5- Force -2000 N (downward)

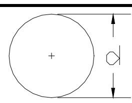
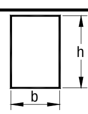
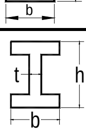
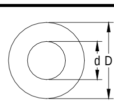
Solving

4. Select SOLUTION - SOLVE - CURRENT LS

Results

5.. Select GENERAL POST PROCESSOR - PLOT RESULTS- NODAL SOLUTION -DOF solution - Translation $U_{y.}$
6. Compare the results with theoretical Calculation.

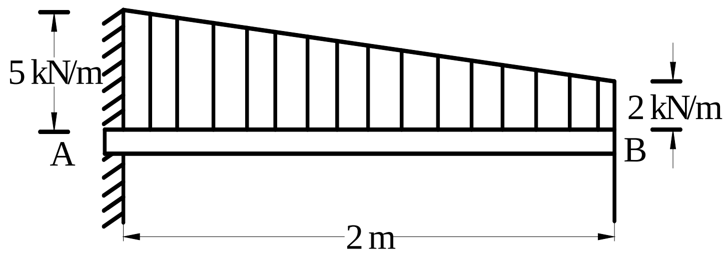
TABULAR COLUMN

Cross Section	Figure	Max.Deformation (Theoretical) m	Max.Deformation (FEM) m
A			
B			
C			
D			
E			

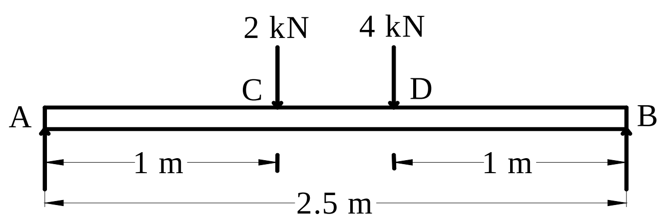
NOTE: To solve for other sections, Change the REAL CONSTANTs, SOLVE current LS (keeping the boundary conditions same) and look for the results.

Exercises on Beams:

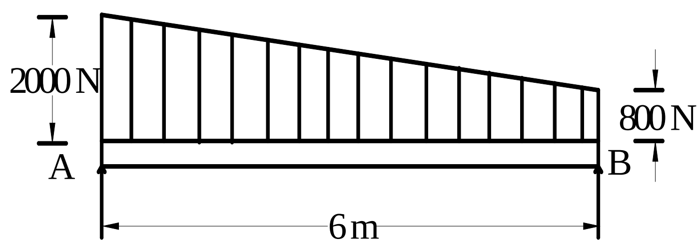
Draw the Shear Force (SFD) and Bending Moment (BMD) Diagrams for the following problems on beams. Also, determine the maximum deflection of beams



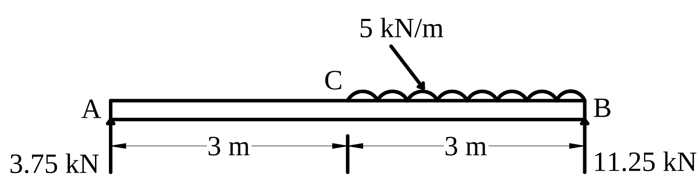
Problem No.1.



Problem No2.



Problem No. 3



Problem No. 4

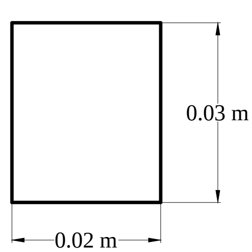
Take : Poisson's Ratio = 0.3

E= 200 GPa (i.e. 2×10^{11} N/m^2) = 2E11 N/m^2 .... when dimensions are in metre

E = 200×10^3 MPa = 200×10^3 N/mm^2 = 2E5 N/mm^2 ...... when dimensions are in mm.

Poissons Ratio = 0.3

Take the following cross section:



NOTE:

1. <u>To apply UDL on the Beam</u>

Select SOLUTION - APPLY-PRESSURE - ON BEAMS - Select ELEMENTS - pressure value at NODE I - <u>2000 N/m</u>

2. <u>To apply UNIFORMLY VARYING LOAD on the Beam</u>

Select SOLUTION - APPLY-PRESSURE - ON BEAMS - Select ELEMENTS - pressure value at NODE I - <u>XX</u> NODE J - <u>YY</u>.

3. <u>To apply Couple at a node on the Beam</u>

Select SOLUTION - APPLY-FORCE/MOMENT - ON NODES - Select NODE - OK
-Lab Direction of force /mom - select MZ
- Value Force/ Moment Value = 2000

(NOTE: Moment 2000 N-m positive sign indicates CCW Moment, Negative moment indicates CW moment)

4. <u>To get Shear Force Diagram (SFD) and Bending Moment Diagram (BMD)</u>

General Post Processor -Element Table - Define Table -
Add - By Sequence Number - SMISC, 2 - Apply
- By Sequence Number - SMISC, 8 - Apply
- By Sequence Number - SMISC, 6 - Apply
- By Sequence Number - SMISC, 12 - OK.

To get SFD (Shear Force Diagram)
General Post Processor -Plot Results - Line Element Results -
Lab1 Elem table item at Node I - SMIS2
Lab1 Elem table item at Node I - SMIS8 - OK

To get BMD (Bending Moment Diagram)
General Post Processor -Plot Results - Line Element Results -
Lab1 Elem table item at Node I - SMIS6
Lab1 Elem table item at Node I - SMIS12 - OK

ANALYSIS OF PLATE

2D ANALYSIS OF PLATE WITH CIRCULAR HOLE

Exercise 1

AIM: To carry out analysis of PLATE shown in FIG. and to determine the DISPLACEMENTS and STRESS distributions by using FEM package.

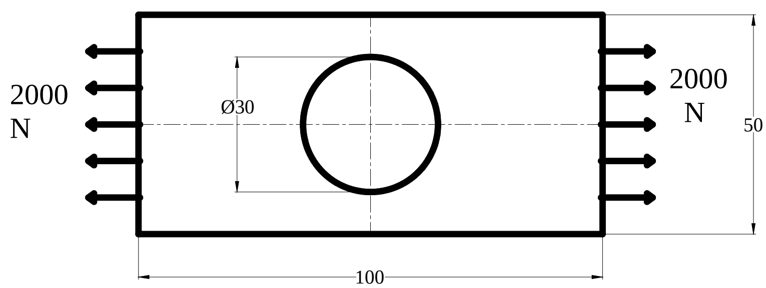
Material Properties:

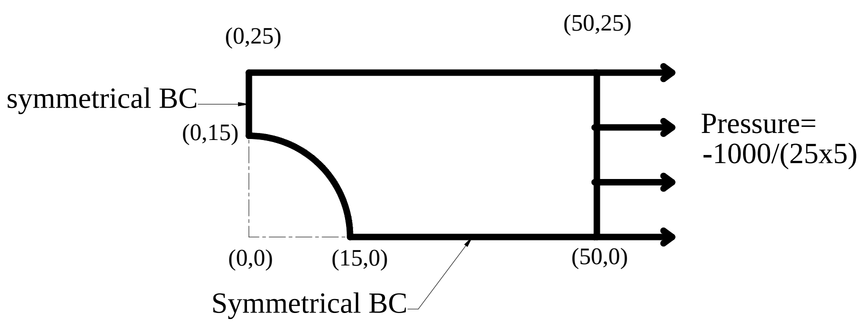
Young's Modulus : 2 x 10^5 N/mm^2 or 2E5 N/mm^2

Poisson's Ratio : 0.3

Type of Element : SOLID (PLANE 42) 4 NODED QUADRILATERAL
(OPTION - Plane Stress with Thickness)

Real Constants : Thickness of the Plate (t = 5 mm)





PROCEDURE

1. Select Preferences - STRUCTURAL
2. Select PREPROCESSOR - ELEMENT TYPE- ADD - LINK - SOLID (PLANE 42) 4 NODED QUADRILATERAL. Click OPTIONS in element menu and choose element behaviour as plane stress with thickness.
3. Select PREPROCESSOR - REAL CONSTANTS- Enter thickness (0.01)
4. Select PREPROCESSOR - MATERIAL PROPERTIES- ISOTROPIC-Enter Young's Modulus(2e+11) & Poissons Ratio (0.3)

Modelling:

5. Select PREPROCESSOR- CREATE- AREA - RECTANGLE - By two corners - width (50) and height(25).
6. Select PREPROCESSOR- CREATE- AREA - CIRCLE - SOLID CIRCLE - Center (0,0) - Radius 15.
7. Select PREPROCESSOR- OPERATE - BOOLEANS - SUBTRACT - AREAS - Subtract circular area from rectangle.

8. Select PREPROCESSOR - MESH TOOL - MESH - Select the area.

Boundary Conditions:

9. Select SOLUTION - APPLY-DISPLACEMENTS- Symmetry B.C. on LINES - Select Lines at the left and bottom as shown in Figure..

10. Select SOLUTION - APPLY-PRESSURE - ON LINES - Select Lines on the left and right and enter the pressure value (-1000/(25x5)). Negative value indicates pressure outward.

Solving

11. Select SOLUTION - SOLVE - CURRENT LS

Results

12. Choose GENERAL POST PROCESSOR - PLOT RESULTS - NODAL SOLUTIONS. Choose displacement or stress results to be displayed on the screen.

NOTE: You can also model the plate by first creating keypoints, lines through keypoints and arc by specifying start and end keypoints, keypoint for center of arc and radius of arc. . Then to create the area - Preprocessor - create- area - arbitrary- by lines - select the lines to create the area.

RESULTS:

MAX. DISPLACEMENTS :

MAX. STRESS at change of section :

NOMINAL STRESS at change in section : $\frac{2000}{(50-30)\times 5}$ = 20 N/mm^2

STRESS CONCENTRATION FACTOR: $K_t = \frac{\pm_{Max}}{\pm_{Nominal}}$

ANALYSIS OF PLATE

2D ANALYSIS OF PLATE WITH FILLET

Exercise 2

AIM: To carry out analysis of PLATE shown in FIG. and to determine the DISPLACEMENTS and STRESS distributions by using FEM package.

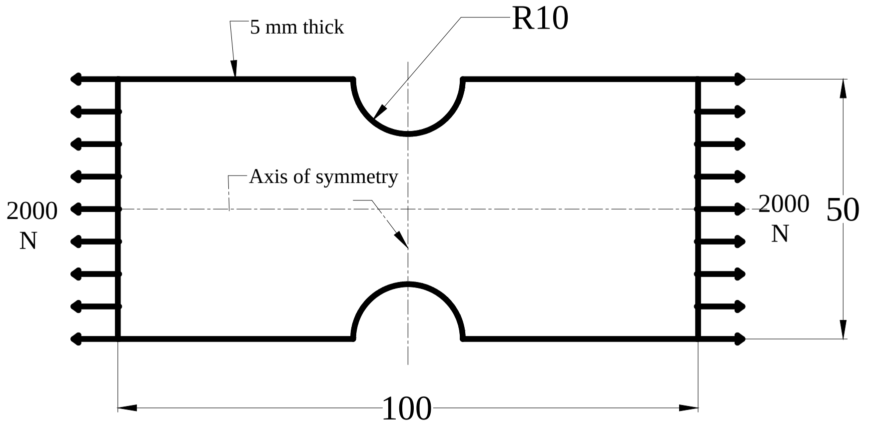
Material Properties:

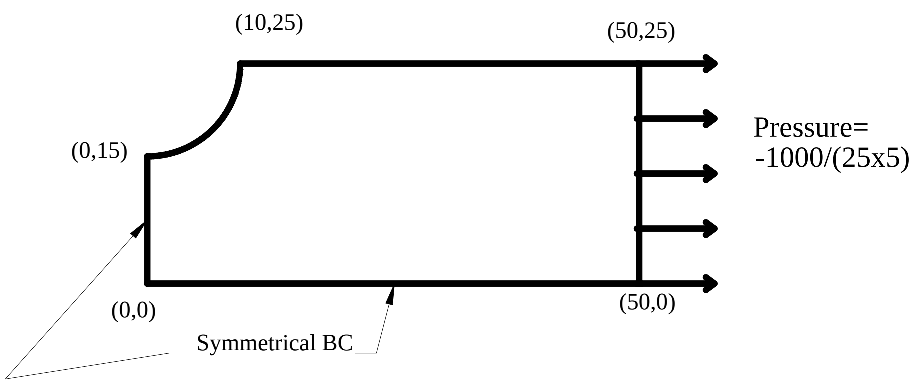
Young's Modulus : 2×10^5 N/mm^2 or 2E5 N/mm^2

Poisson's Ratio : 0.3

Type of Element : SOLID (PLANE 42) 4 NODED QUADRILATERAL
(OPTION - Plane Stress with Thickness)

Real Constants : Thickness of the Plate (t = 5 mm)





RESULTS:

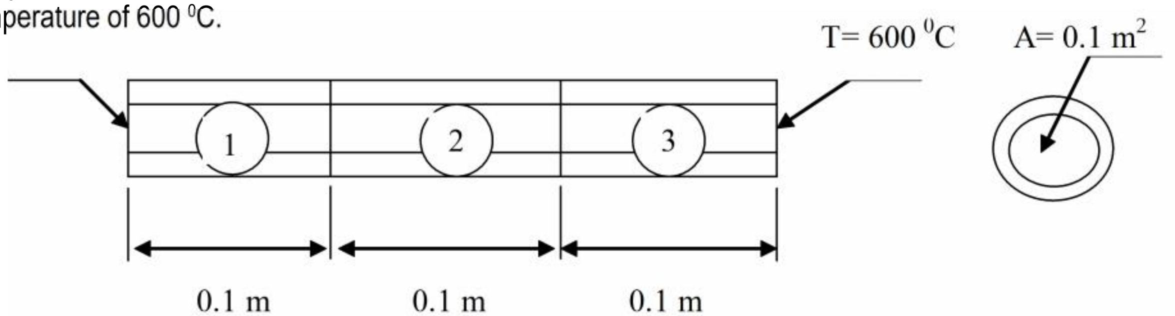
MAX. DISPLACEMENT :.............................. MAX. STRESS :..........................

NOMINAL STRESS: $\frac{2000}{(50-20)\times 5}$ = 13.33 N/mm^2

STRESS CONCENTRATION FACTOR: $K_t = \frac{\pm_{Max}}{\pm_{Nominal}}$

THERMAL ANALYSIS

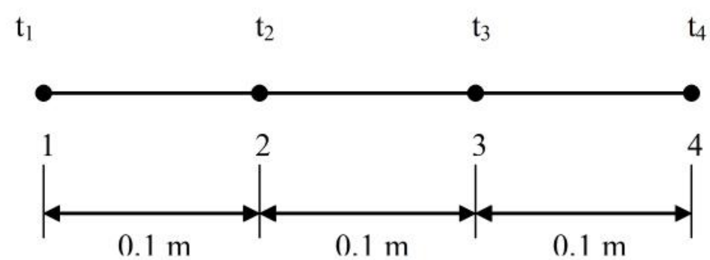
Exercise 1. For the composite wall idealized by the 1-D model shown in figure below, determine the interface temperatures. For element 1, let K_1 = 5 W / m ^{0}C, for element 2, K_2 = 10 W / m ^{0}C and for element 3, K_3 = 15 W / m ^{0}C. The left end has a constant temperature of 200 ^{0}C and the right end has a constant temperature of 600 ^{0}C.



Solution:
Given:

t_1 = 200 ^{0}C	**t_4 = 600 ^{0}C**	**A = 0.1 m^2**
K_1= 5 W / m ^{0}C	**K_2= 10 W / m ^{0}C**	**K_3= 15 W / m ^{0}C**

Finite Element model:



Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -probl –ok

File -Change title -Enter new title -yyy –ok

Step 2 -Ansys Main Menu -Preferences

Select -THERMAL -ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- LINK -2 D CONDUCTION -32- ok -close

Real constants - Add/Edit/Delete-Add-type 1-link 32- real constant set no 1 – cross sectional area –enter -0.1-ok-close

Material properties – Material models – material model number 1-thermal – conductivity –isotropic -conductivity for material - thermal conductivity (K_{xx}) –5 -ok- material –new model – define material ID -2-ok-thermal – conductivity –isotropic - conductivity for material - thermal conductivity (K_{xx}) –10 –ok- material –new model – define material ID -3-ok- thermal – conductivity –isotropic - conductivity for material - thermal conductivity (K_{xx}) –15 –ok-close

Preprocessor-Modeling -Create -Nodes -in active CS -x,y,z location in CS-0,0,0 (x,y value w.r.t first node)-apply (first node is created)- 0.1,0,0 -apply (second node is created) -0.2,0,0 –apply (third node is created)- 0.3,0,0 -apply (fourth node is created) -ok

Create –elements-element attributes-select material number-1-real constant set number -1-Auto numbered – thru nodes – pick 1, 2 (element 1 is created)-ok - element attributes-select material number-2-real constant set number -1-Auto numbered – thru nodes – pick 2, 3 (element 2 is created)-ok- element attributes-select material number-3-real constant set number -1-Auto numbered – thru nodes – pick 3, 4 (element 3 is created)-ok

Loads - Define Loads - Apply –thermal –select temperature –on nodes-pick node 1-apply-select –TEMP-value -200-apply- pick node 4-apply-select –TEMP- value -600-ok

Step 4 -Ansys Main Menu –Solution

Solve -solve current LS -ok (;If everything is ok, -solution is done is displayed) –close

Step 5 -Ansys Main Menu -General. Post Processor.

Plot results- Contour Plots - Nodal solution –DOF solution.-temp -ok (Temp distribution plot)

List results -nodal solution -Dof solution -temp -ok (Temperature at all the nodes will be displayed)

Plotctrls- Animate – Deformed results – Dof solution – Temperature – ok (for animatiom)

Exercise 2:

2D with convection and conduction boundary conditions

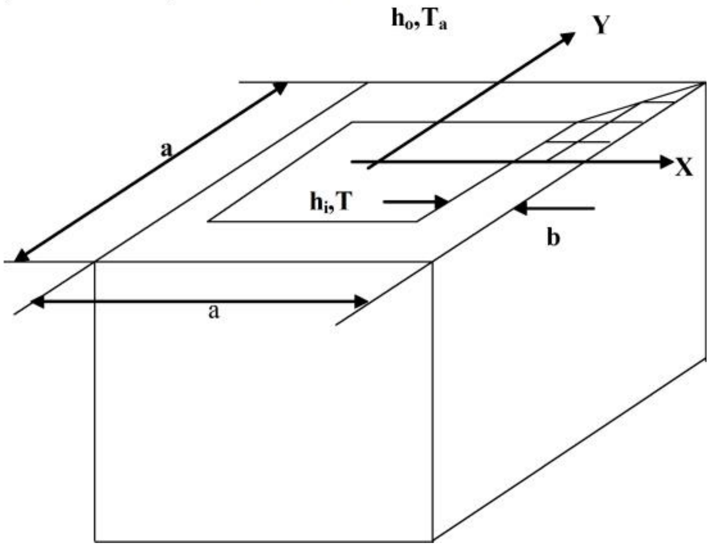
Determine the temperature distribution and the rate of heat flow" q " per metre of the height for a tall chimney whose cross section is shown below. Assume that the inside gas temp is T_g = 311 K ,the inside convection coefficient is h_i., the surrounding air temp is T_a = 255 K and the outside convection coefficient is h_o.

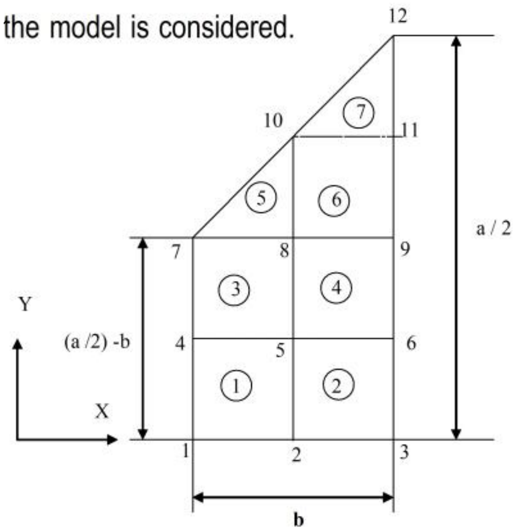
Element type -Thermal solid element -PLANE 55

K = 1.7307 W / m K , h_i = 68.14 w / m^2 K, h_o = 17.04 w / m^2 K

Geometric properties-For solving the problem only 1/8 th of the model is considered.

(see 3-D view) a = 4 m and b = 1 m





Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -probl –ok

File -Change title -Enter new title -yyy –ok

Step 2 -Ansys Main Menu -Preferences

Select -THERMAL -ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- SOLID -QUAD 4 NODE -55- ok -close

Real constants -No real constants

Material properties – Material models – thermal – conductivity –isotropic - conductivity for material - thermal conductivity (K_{xx}) –1.7307 -ok- close

Modeling -Create -Nodes -in active CS -x,y,z location in CS-1,0 (x,y value w.r.t first node)- apply (first node is created)- 1.5,0 -apply (second node is created) -2,0 -apply (third node is created) -1,0.5 -apply (fourth node is created)-1.5,0.5- apply (fifth node is created)- 2, 0.5 -apply(sixth node is created)- 1,1 -apply (seventh node is created) -1.5,1 - apply (eighth node is created) -2 , 1 -apply (ninth node is created)-1.5 ,1.5- apply (tenth node is created)- 2 , 1.5- apply (eleventh node is created)-2 , 2 -apply (twelfth node is created) –ok

Create –elements-Auto numbered – thru nodes – pick 1, 2,5 & 4 (anticlockwise , element 1)-apply- pick 2, 3, 6 & 5 (element 2) -apply- pick 4 , 5 , 8 & 7 (element 3)- apply-pick 5, 6 , 9 & 8 (element 4) -apply -pick 7, 8, 10 & 10 (element 5)- apply-pick 8, 9, 11 &10 (element 6::.)- apply -pick 10, 11, 12 & 12 (element 7)-ok (total seven elements ate created through nodes).

Step 4 –Preprocessor

Loads - Define Loads - Apply –thermal -Convection -: on nodes -pick inner surface by box option - apply- Film coefficient (inner) -68.14 - temperature (at inner surface)- 311 (value)-ok

Convection -on nodes -pick outer surface by box option -apply- Film coefficient (outer) -17.04- Temperature (at inner surface) –(255) (value) ~Ok

Step 5 -Ansys Main Menu –Solution

Solve -solve current LS -ok (;If everything is ok, -solution is done is displayed) –close

Step 6 -Ansys Main Menu -General. Post Processor.

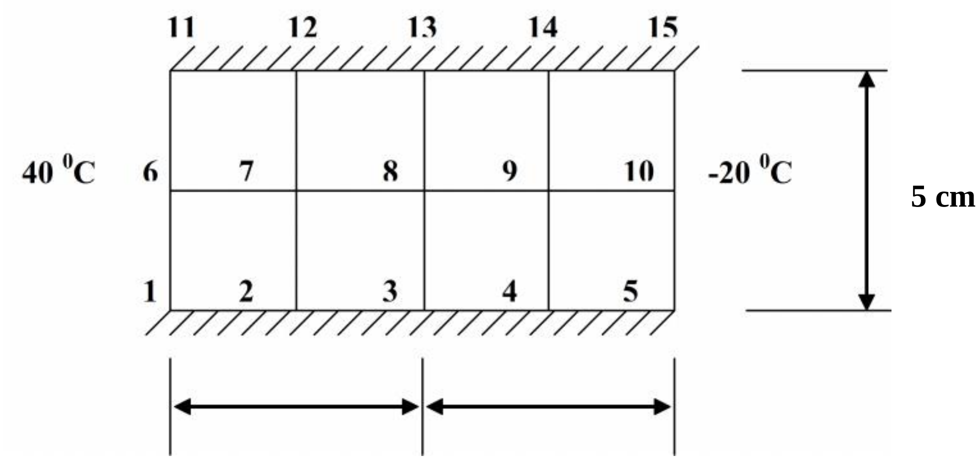
Plot results- Contour Plots - Nodal solution –DOF solution.-temp -ok (Temp distribution plot)

List results -nodal solution -Dof solution -temp -ok (Temperature at all the nodes will be displayed)

Plotctrls- Animate – Deformed results – Dof solution – Temperature – ok (for animatiom)

Exercise 3:

For the body shown in figure, determine the temperature distribution. The body is insulated along the top and bottom edges, $K_{xx} = K_{yy} = 1.7307$ W/m ^{0}C. No internal heat generation is present.



Step 1- Ansys Utility Menu

File -Clear and start new -Do not read file -ok

File -Change job name -Enter new job name -probl –ok

File -Change title -Enter new title -yyy –ok

Step 2 -Ansys Main Menu -Preferences

Select -THERMAL -ok

Step 3 -Preprocessor

Element Type -Add/Edit/Delete -Add- SOLID -QUAD 4 NODE -55- ok -close

Real constants -No real constants

Material properties – Material models – thermal – conductivity –isotropic - conductivity for material - thermal conductivity (K_{xx}) –1.7307 -ok- close

Modeling -Create -Nodes -in active CS -x,y,z location in CS : 0,0 (x,y value w.r.t first node)- apply (first node is created) : 2,0 -apply (second node is created) : 4,0 -apply (third node is created) : 6,0 -apply (fourth node is created) : 8,0- apply (fifth node is created) : 0, 2.5 -apply(sixth node is created) : 2, 2.5 -apply (seventh node is created) : 4, 2.5 - apply (eighth node is created) : 6, 2.5 -apply (ninth node is created): 8, 2.5 - apply (tenth node is created): 0, 5- apply (eleventh node is created): 2, 5-apply (twelfth node is created) : 4, 5-apply (thirteenth node is created) : 6, 5-apply (fourteenth node is created) : 8, 5-apply (fifteenth node is created) –ok

Create –elements-Auto numbered – thru nodes – pick 1, 2,7 & 6 (anticlockwise , element 1)-apply- pick 2, 3, 8 & 7 (element 2) -apply- pick 3 ,4 , 9 & 8 (element 3)- apply-pick 4 ,5 , 10 & 9 (element 4) -apply -pick 6,7, 12 & 11 (element 5)- apply-pick 7, 8, 13 &12 (element 6:)- apply -pick 8, 9, 14 & 13 (element 7) pick 9, 10, 15 & 14 (element 8)-ok (total eight elements are created through nodes).

Step 4 –Preprocessor

Loads - Define Loads - Apply –thermal -temperature -: on nodes -pick inner surface by box option - apply- - temperature (at inner surface)-40(value)-apply

temperature -on nodes -pick outer surface by box option -apply- Temperature (at inner surface) (-20) (value) ~Ok

Loads - Define Loads - Apply –thermal-heat flux- on nodes- pick top surface by box option - apply- - heat flux – 0 (value)-apply- heat flux- on nodes- pick bottom surface by box option - apply- - heat flux – 0 (value)-apply

Step 5 -Ansys Main Menu –Solution

Solve -solve current LS -ok (;If everything is ok, -solution is done is displayed) –close

Step 6 -Ansys Main Menu -General. Post Processor.

Plot results- Contour Plots - Nodal solution –DOF solution.-temp -ok (Temp distribution plot)

List results -nodal solution -Dof solution -temp -ok (Temperature at all the nodes will be displayed)

Plotctrls- Animate – Deformed results – Dof solution – Temperature – ok (for animation)

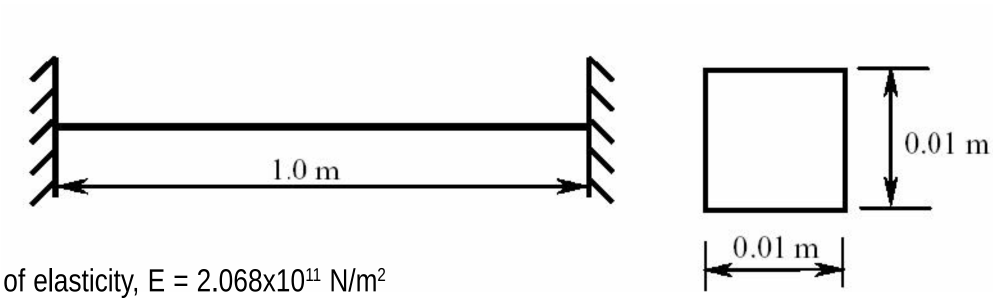
DYNAMIC ANALYSIS

1) Fixed – fixed beam for natural frequency determination Modal Analysis of a Fixed – Fixed Beam

Exercise 1

Aim: To determine the natural frequencies (Eigen values) and corresponding mode shapes (Eigen vectors) of a given fixed – fixed beam

To do a simple modal analysis of fixed – fixed beam shown below. Five lowest natural frequencies and their corresponding mode shapes of the beam are obtained.



Modulus of elasticity, E = 2.068×10^{11} N/m^2

Poisson's ratio, μ= 0.3

Density, r =7830 kg/m^3

Creation of any beam can be obtained through GUI (Graphic User Interface) of ANSYS using the following steps:

PREPROCESSING

1. Define Analysis Type:

Solution > Analysis type > New analysis > Modal

ANTYPE, 2

2.Creation of Fixed – Fixed Beam used in Modal Analysis:

Creation of any beam can be obtained through GUI (Graphic User Interface) of ANSYS using the following steps:

1 Open preprocessor menu

2 Give example a Title

Utility Menu > File > Change Title - - -

Enter Fixed-Fixed Beam for the title

3. Give example a Jobname

Utility Menu > File > Change Jobname - - -

Enter Dynamic for the jobname

4. Define Element Types

Preprocessor > Element Type > Add/Edit/Delete ….

For this problem, the BEAM3 (Beam 2D elastic) element is used. This element has 3 d.o.f (i.e. translation along X and Y axes and rotation about Z axis). With only 3 d.o.f, the BEAM3 element can only be used in 2D analysis.

5. Define Real Constants

Preprocessor > Real Constants … > Add

In the 'Real Constants for BEAM3' window, enter the following geometric properties:

i.) Cross sectional area 'AREA': 0.01 x 0.01 = 0.0001 m^2

ii.) Area Moment of Inertia 'I_{zz}': $\frac{bd^3}{12} = 8.33e\text{-}10\ m^4$

iii. Total beam height 'HEIGHT': 0.01 m^4

6. Define Element Material Properties

Preprocessor >Material Props >Material Models >Structural > Linear >Elastic >Isotropic

In the window that appears, enter the following material properties of the beam (here it is steel)

i. Young's Modulus, EX: $2.068e^{11}$

ii. Poisson's Ratio, PRXY: 0.3

To enter density of the material, double click on 'Linear' followed by 'Density' in the Define Material Model Behavior' window and enter a density of 7830.

Note: for dynamic analysis, both the stiffness and the material density have to be specified.

7. Create Keypoints

Preprocessor > Modeling > Create > Keypoints > In Active CS

Define 2 keypoints (the beam vertices) for this structure along with the coordinates as given in the following table:

Keypoint	Coordinates (x, y, z)
1	(0, 0, 0)
2	(1, 0, 0)

8. Define Lines

Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Create a line between Keypoint 1 and keypoint 2.

9. Define Mesh Size

Preprocessor > Meshing > Size Cntrls > Manual Size > Lines > All Lines . . .

For this example 10 element divisions are specified along the line.

10. Mesh the Frame

Preprocessor > Meshing > Mesh > Lines > Click 'Pick All' in the small window appears

SOLUTION

1. Set Options for Analysis Type

Solution > Analysis Type > Analysis Options . . .

The following window will appear

- As shown, select the Subspace method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand modes shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'.

Note that the default mode extraction method chosen is the Reduced Method. This method as it reduces the system matrices to only consider the master d.o.f (see steps for reduced method next to the Subspace method.). The Subspace Method extracts modes for all d.o.f's. It is therefore more exact but it also takes longer time to compute (especially when the complex geometries).

- For this problem, we will use default options. So click on 'OK'.

2. Apply Constraints

Solution > Define Loads > apply > structural > Displacement > On Keypoints

- Fix all DOFs constraints on Keypoint 1 and click 'Apply' in a small window that appears. Once you click apply in a small window another window will appear, select 'All DOF' and click apply.
- Fix keypoint 2 also for fixed-fixed beam and click 'OK' in small window.

3. Solve the System

Solution > Solve > Current LS

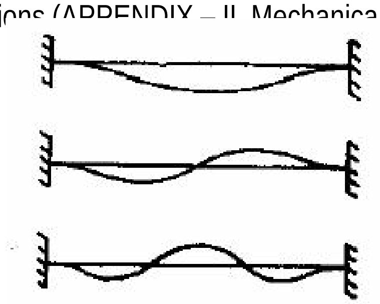
SOLVE

POSTPROCESSING: Viewing the Results

1. Verify Extracted Modes against Theoretical Predictions

General Postproc > Results Summary …

The above theoretical values for the first 3 natural frequencies are obtained by the following theoretical expressions (APPENDIX – II, Mechanical Vibrations by G. K. Grover)



$$\omega_1 = \frac{22.0\,a}{l^2}$$

$$\omega_2 = \frac{61.7\,a}{l^2}$$

$$\omega_3 = \frac{121.0\,a}{l^2}$$

Where, $a = \sqrt{\dfrac{EI}{\rho A}}$ and $f_i = \dfrac{1}{2\pi}\omega_i$ Hz $\quad (i = 1,2,3)$

2. View Mode Shapes

- General Postproc > Read Results > First Set

  This selects the results for the first mode shape.
- General Postproc > Plot Results > Deformed Shapes > Select Def + Undef edge

  The first mode shape is now appear in the graphic window.
- To view the mode shapes, select

General Postproc > Read Results > Next set

General Postproc > Plot Results > Deformed Shapes > Select Def + Undef edge.

Repeat the above steps for the remaining mode shapes.

3. Animate Mode Shapes

- Select Utility Menu (Menu at the top) > Plot Ctrls >Animate >Mode Shapes
- Keep the default setting and click 'OK', the animated mode shapes will appear in the window.

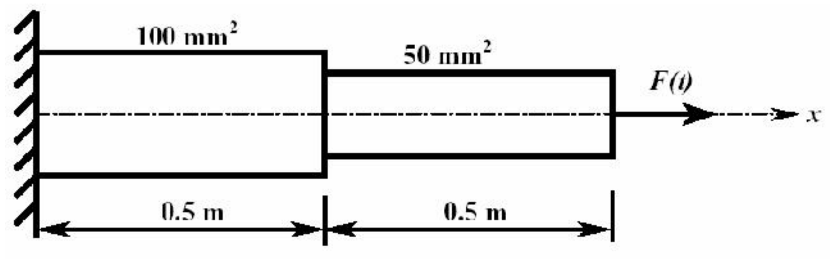
DYNAMIC ANALYSIS OF BARS SUBJECTED TO FORCING FUNCTION

Dynamic Analysis of Bar subjected to forcing function

Harmonic Analysis of a Axial Bar

Exercise 2 : Aim: To conduct harmonic analysis of a given axial stepped bar.

To outline the steps required to do the simple harmonic analysis of a stepped bar subjected to a cyclic load as shown below.



Modulus of elasticity, E = 2.068×10^{11} N/m^2

Poisson's ratio, μ= 0.3

Density, r =7830 kg/m^3

Load:

Cyclic Load

Magnitude = 100N

Frequency Range: 0 – 300 Hz

Note: ANSYS provides 3 methods such as Full, Reduced and Modal Superposition methods for conducting a harmonic analysis. This example demonstrates the Full method because it is simple and ease to use as compared to other methods. However, this method makes use of full stiffness and mass matrices and thus is the slower and costlier option.

PREPROCESSING

1. Define Analysis Type:

Solution > Analysis type > New analysis > Harmonic

ANTYPE, 3

2. Creation of Stepped Bar used in Harmonic Analysis:

Creation of any beam can be obtained through GUI (Graphic User Interface) of ANSYS using the following steps:

1. Open preprocessor menu

2.Give example a Title

Utility Menu > File > Change Title - - -

Enter Stepped Bar for the title

3. Give example a Jobname

Utility Menu > File > Change Jobname - - -

Enter Dynamic for the jobname

4. Define Element Types

Preprocessor > Element Type > Add/Edit/Delete ….

For this problem, the Link (2D spar 1) element is used. This element has 3 d.o.f (i.e. translation along X and Y axes and rotation about Z axis).

5. Define Real Constants

Preprocessor > Real Constants … > Add …

Enter the following 2 sets of geometric properties for element – 1 and element - 2:

i. Cross sectional area 'AREA': For type 1: 0.0001 m^2 and for type 2: 0.00005m^2.

Note: For dynamic analysis the units of area, Young's modulus and density should be in m^2, N/m^2 and kg/m^3 respectively.

6. Define Element Material Properties

Preprocessor >Material Props >Material Models >Structural > Linear >Elastic >Isotropic

In the window that appears, enter the following material properties of the bar (here it is steel)

i. Young's Modulus, EX: 2.068e11 kg/m2

ii. Poisson's Ratio, PRXY: 0.3

To enter density of the material, double click on 'Linear' followed by 'Density' in the Define Material Model Behavior' window and enter a density of 7830 kg/m^3.

Note: If the material properties of two elements are different, create another new model in main menu of the window and enter the material properties of the second element.

7. Create Nodes

Since the real constants such as areas and/or lengths of a stepped bar are different and number of elements to be used are less, the creation of model through node points is simple as compared to the creation of model through keypoints. Hence the model, in this example, is created through nodes.

Preprocessor > Modeling > Create > Nodes > In Active CS

Define 3 node points for this structure along with the coordinates as given in the following table:

Nodepoint	Coordinates (x, y, z)
1	(0, 0, 0)
2	(0.5, 0, 0)
3	(1, 0, 0)

8. Create Element

Preprocessor > Modeling > Create > Elements > Element Attributes.

Check the material No. and real constant set No. for element 1 in the window that appears (for element 1, both should be 1)

Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru Nodes

- Click on node 1 and 2 and click 'OK' in small window.

Preprocessor > Modeling > Create > Elements > Element Attributes.

Change the material No. and real constant set No. for element 2 in the window that appears (for element 2, Material No.= 1 and Real Const. Set No. = 2, in this case)

Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru Nodes

- Click on node 2 and 3 and click 'OK' in small window.

SOLUTION

1. Set Options for Analysis Type

Solution > Analysis Type > Analysis Options . . .

- select the Full solution method, the Real + imaginary DOF printout format

and do not use lumped mass approximation.

- Click ‘OK’.

2. Apply Constraints

Solution > Define Loads > apply > structural > Displacement > On Nodes

- Fix all DOFs constraints on node at x=0 and click ‘Apply’ in a small window that appears. Once you click apply in a small window another window will appear, select ‘All DOF’ and click apply.

3. Apply Loads

Solution > Define Loads > apply > structural > Force/Moment > On Nodes

- Select the node at free-end of bar (i.e. at x = 1.0)
- The following window will appear. Fill it as shown to apply a load with real value of 100 and an imaginary value of 0 in the positive ‘x’ direction.

Note: By specifying a real and imaginary value of the load, we are providing information on magnitude and phase of the load. In this case the magnitude of the load is 100 N and its phase is 0^{o}. Phase information is most important when we have two or more cyclic loads being applied to the structure as these loads could be in or out of phase. For harmonic analysis, all loads must have the same frequency.

4. Set the Frequency Range

Solution > Load Step Opts > Time/Frequency > Freq and Substps . . .

- specify a Harmonic frequency range of 0 – 4500 Hz, 100 substeps and stepped b.c . .

5. Solve the System

Solution > Solve > Current LS

SOLVE

POSTPROCESSING: Viewing the Results

We want to observe the response at x = 0.5 m (where the load is applied) as a function of frequency. We cannot do this with General Postprocessing (POST1), rather we must use TimeHist PostProcessing (POST26). Post26 is used to observe certain variables as a function of either time or frequency.

1. Open the TimeHist Processing (POST26) Menu

Select TimeHIst Postproc from the ANSYS Main Menu

2. Define Variables

Note: Here we have to define variables that we want to see plotted. By default, Variable 1 is assigned either Time or Frequency. In this case it is assigned frequency. We want to see the displacement UY at the node at x = 0.5. (To get a list of nodes and their attributes, select Utility Menu > List > Nodes).

TimeHist PostPro > Variable Viewer . . .

- Select Add (the green ‘+’ sign in the upper left corner) from this window and the following window will appear.
- We are interested in the Nodal solution > DOF Solution > X – Component. Click OK.

  Nodal solution > DOF Solution > Y – Component. Click OK.
- Graphically select node 2 when prompted and click OK. The ‘Time History Variables’ window should now look as follows.

3. List Stored Variables

- In the ‘Time History Variable’ window, click the ‘List’ button (3 buttons to the left of ‘Add’ button).

4. Plot UX vs Frequency

- In the ‘Time Variable’ window click the ‘Plot’ button (2 buttons to the left of ‘Add’

button).

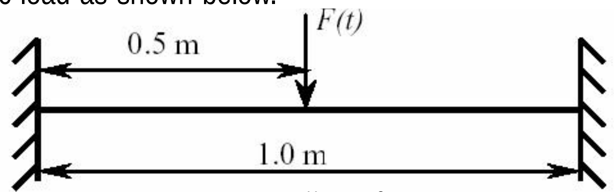
Note that we get peaks at frequencies of approximately 1650 and 4250 Hz. This corresponds with the predicted natural frequencies of 1627.7 and 4220.5 Hz respectively during modal analysis of the same stepped bar.

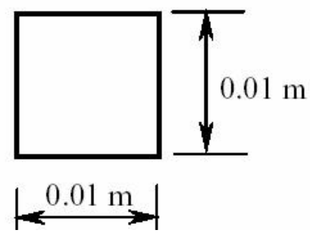
Obtain the response with log scale of UX similar to the previous case. Then, we will now see the response at free-end of bar for the cyclic load with frequency range 0 – 5000 Hz

Harmonic Analysis of a Fixed – Fixed Beam

Exercise 3 : Aim: To conduct harmonic analysis of a given fixed – fixed beam.

To outline the steps required to do the simple harmonic analysis of a fixed – fixed beam subjected to a cyclic load as shown below.





Modulus of elasticity, E = 2.068×10^{11} N/m^2, Poisson's ratio, μ= 0.3,

Density, r =7830 kg/m^3, Load:Cyclic Load, Magnitude = 100N,

Frequency Range: 0 – 300 Hz

Note: ANSYS provides 3 methods such as Full, Reduced and Modal Superposition methods for conducting a harmonic analysis. This example demonstrates the Full method because it is simple and easy to use as compared to other methods. However, this method makes use of full stiffness and mass matrices and thus is the slower and costlier option.

PREPROCESSING

1. Define Analysis Type:

Solution > Analysis type > New analysis > Harmonic

ANTYPE, 3

2. Creation of Fixed – Fixed Beam used in Harmonic Analysis:

Create the fixed – fixed beam using same steps used in modal analysis case.

SOLUTION

1. Set Options for Analysis Type

Solution > Analysis Type > Analysis Options . . .

- As shown, select the Full solution method, the Real + imaginary DOF printout format and do not use lumped mass approximation.
- Click 'OK'.

2. Apply Constraints

Solution > Define Loads > apply > structural > Displacement > On Nodes

- Fix all DOFs constraints on node at x=0 and click 'Apply' in a small window that appears. Once you click apply in a small window another window will appear, select 'All DOF' and click apply.
- Fix end node at x = 1 also for fixed-fixed beam and click 'OK' in small window.

3. Apply Loads

Solution > Define Loads > apply > structural > Force/Moment > On Nodes

- Select the node at mid-point of the beam (i.e. at x=0.5)
- The following window will appear. Fill it as shown to apply a load with real value of 100 and an imaginary value of 0 in the positive 'y' direction.

Note: By specifying a real and imaginary value of the load, we are providing information on magnitude and phase of the load. In this case the magnitude of the load is 100 N and its phase is 0^o. Phase information is most important when we have two or more cyclic loads being applied to the structure as these loads could be in or out of phase. For harmonic analysis, all loads must have the same frequency.

4. Set the Frequency Range

Solution > Load Steo Opts > Time/Frequency > Freq and Substps . . .

- As shown in the window below, specify a frequency range of 0 – 300 Hz, 300 substeps and stepped b.c . . .

By doing this we will be subjecting the beam to loads at 1 Hz, 2 Hz, 3 Hz, . . . , 300 Hz. We will specify a stepped boundary condition (KBC) as this will ensure that the same amplitude (100 N) will be applied for each of the frequencies. The ramped option, on the other hand, would ramp up the amplitude where at 1 Hz the amplitude would be 1 N and at 100 Hz, the amplitude would be100 N.

5. Solve the System

Solution > Solve > Current LS

SOLVE

POSTPROCESSING: Viewing the Results

We want to observe the response at x = 0.5 m (where the load is applied) as a function of frequency. We cannot do this with General Postprocessing (POST1), rather we must use TimeHIst PostProcessing (POST26). Post26 is used to observe certain variables as a function of either time or frequency.

1. Open the TimeHist Processing (POST26) Menu

Select TimeHist Postproc from the ANSYS Main Menu

2. Define Variables

Note: Here we have to define variables that we want to see plotted. By default, Variable 1 is assigned either Time or Frequency. In this case it is assigned frequency. We want to see the displacement UY at the node at x = 0.5. (To get a list of nodes and their attributes, select Utility Menu > List > Nodes).

TimeHist PostPro > Variable Viewer. .

- Select Add (the green '+' sign in the upper left corner) from this window and the following window will appear.
- We are interested in the Nodal solution > DOF Solution > Y – Component. Click OK.
- Graphically select node 2 when prompted and click OK. The 'Time History Variables'

3. List Stored Variables

- In the 'Time History Variable' window, click the 'List' button (3 buttons to the left of 'Add' button).

4. Plot UY vs Frequency

- In the 'Time Variable' window click the 'Plot' button (2 buttons to the left of 'Add' button).

Note that we get peaks at frequencies of approximately 53 and 286 Hz. This corresponds with the predicted first and third natural frequencies of 52.815 and 145.6 Hz respectively during modal analysis of the same beam. It can also be noted that at second natural frequency 145.6 Hz, the deflection of beam at its center was zero during modal analysis and hence there is no peak occurs at second natural frequency.

To get a better view of the response, view the log scale of UY.

- Select Utility Menu > PlotCtrl > Style > Graphs > Modify Axis
- As in the above window, change the Y – axis scale (LOGY) to 'Logarithmic'.
- Select Utility Menu > Plot > Replot

We will now see the response at mid-point for the cyclic load for frequency range 0 – 300 Hz