Introduction to Computer Aided Manufacturing (CAM)

CAM refers to an automation process that accurately converts product design and drawing of an object into a code format, readable by the machine to manufacture the product. The system allows professionals who have completed Computer Aided Design courses to control machinery in a highly automated environment to manufacture various items and pieces. CAM is also used in all computer-aided operations in a manufacturing plant, such as planning, management, transpiration, and storage.

History

CAM evolved from the technology utilized in the Computer Numerical Control machines that were used in the early 1950s, which involved the use of coded instructions on a punched paper tape and could control simple manufacturing functions. Computer Aided Manufacturing is a logical extension of that idea, as it allows for a whole set of manufacturing functions to be controlled simultaneously.

The system's primary purpose is to speed up the production process of different components with more precise dimensions and material consistency. In some cases, Computer Aided Manufacturing uses only the required amount of raw material, thus minimizing waste and reducing energy consumption.

Practical Applications

The CAM system controls manufacturing operations performed by robotic welding machines and other industrial tools. A professional with Process Piping Training oversees how the system moves the raw material to different machines after the completion of each step. Finished products are also moved within the system to complete other manufacturing tasks such as synthesizing, quality control, packaging and final checks.

Some of the major applications of the system are woodturning, metalworking, glass working, spinning and graphical optimization of the entire manufacturing procedure. Production of surfaces and screw threads, or aspheric optical elements made from glass, crystals, and other metals, can also be done through Computer-Aided Manufacturing.

Since CAM can manufacture three-dimensional solids using ornamental lathes with great intricacy and detail, its practical applications are nearly endless. The system can construct products like: crankshafts, camshafts

Benefits of CAM

Computer Aided Manufacturing makes it easier for professionals with Engineering Training to measure a plant's output and production as whole as well as to optimize the assembly process. CAM systems can maximize utilization of a full range of production equipment, such as high speed, 5-axis turning machines and electrical discharge machines, and can help implementing advanced productivity tools like simulation and optimization to leverage specific professional skills.

In fact, one of the key areas of development is the simulation of performance. Among the most common types of simulation are testing a piece or item for response to stress. In stress tests, model surfaces distort as the part comes under simulated physical or thermal stress. The ease with which a part's specifications can b e changed facilitates the development.

CAM can be applied to the fields of mechanical, electrical, industrial and aerospace engineering. Other applications such as thermodynamics, fluid dynamics, electromagnetism and kinematics are also compatible with Computer-Aided Manufacturing, leveraging the value of skilled professionals that have received excellent training.

USE OF CAM TECHNOLOGY

Machining	**Milling, Turning, Drilling, Grinding, Finishing**
Nontraditional Processes	**Electric Discharge (EDM: Wire & Plunge), Electro Chemical (ECM), Water Jet, PCB & Micro chip Manufacturing**
Rapid Prototyping	**SLA, SLS, FDM, LOM, SGC, Milling**
Robot programming	**Welding, painting, finish, hoisting and conveying, sheet forming**
Measurement & Re-engineering	**CMM (Computerized Measurement Mach.)**
Metal Forming	**Sheet blanking and bending in Punch Press, Plasma Arc Cutting, Sheet bending in CNC Press Brake, Spinning, etc...**
Microelectronic Devices	**Microchips, Printed circuit boards,**

Commercial CAM Software Commercial CAM Software

- I-DEAS
- CATIA
- PRO-Engineer Engineer
- Unigraphics
- Cimatron
- Work-NC
- Power Mill
- Hyper Mill
- CAM Works
- Master CAM
- Surf-CAM
- NC-Gibbs
- Auto-CAD based CAM programs

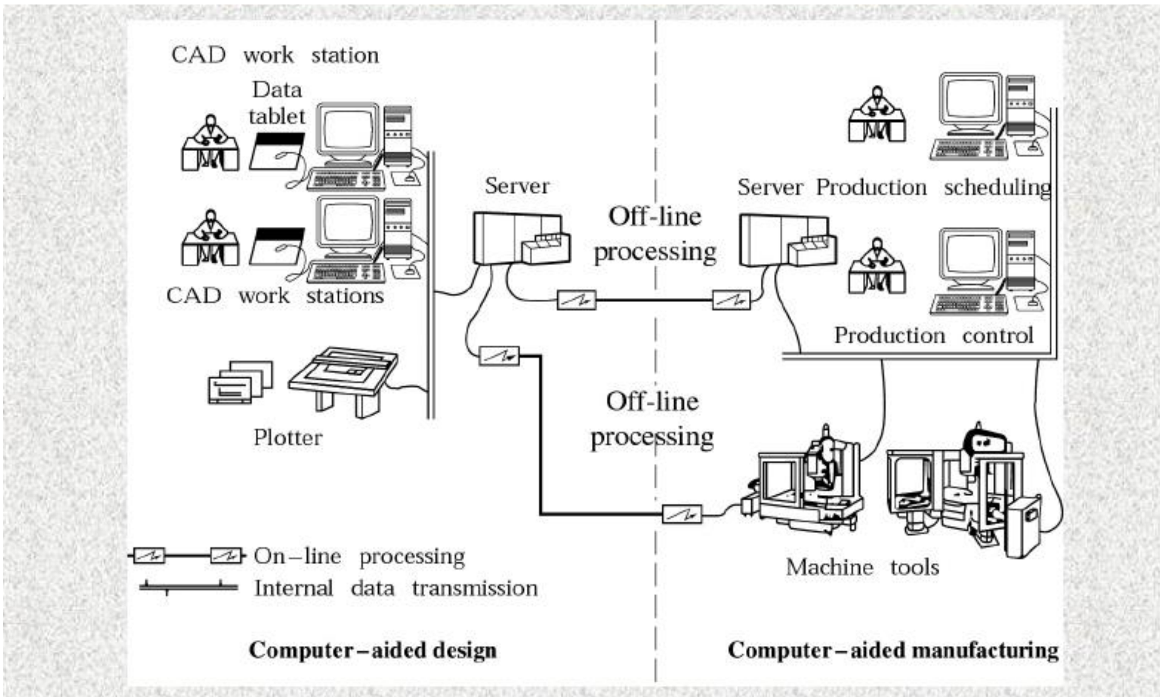
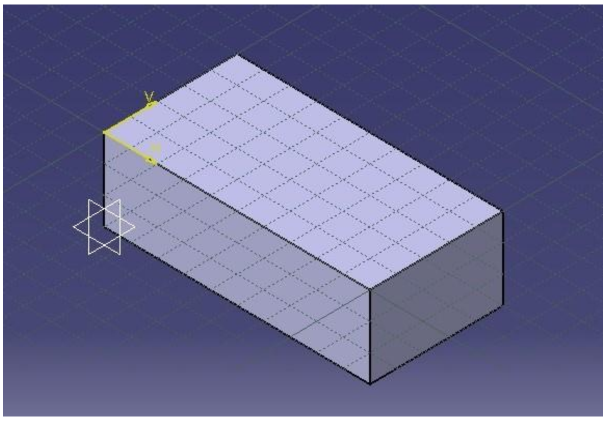


Fig. Information Flow chart in CAD/CAM Application

SURFACE MILLING USING – NX CAM



Create a solid with plane surface such as a rectangular block with respect to given dimensions as shown in U.G. or Solid Edge and save

Step 1 File – open – Part name – ok
Go to start – **select manufacturing** – machining environment (default) – initialize.

Step 2 **Program order view**
select all options one by one
Go to operation navigator (right hand side) – select and delete program, machine tool and geometry, machining methods one by one if any

Step 3 **Format – WCS** – origin – pick on the part – ok – cancel

Step 4 **Create geometry**
Parent group – geometry , Name – work piece – ok
Part – select – select all – ok –material – chose material - ok

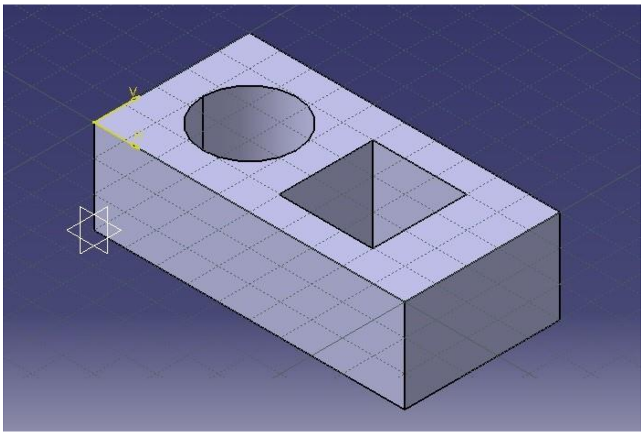
Step 5 **Create tool**
Parent group – generic machine , Name – face mill – ok
Milling tool – 5 parameters – give diameter value – say 10.00 and other dimensions if required – material – chose material – display tool - ok – cancel

Step 6 **Create method**
Parent group – method , Name – mill method,
Cut method – face mill.

Step 7 **Create operation**
Programme – NC programme, Use geometry – work piece, Use tool – face mill
Use method – mill method, Name – face milling, Cut method – zig-zag, Cut area – select – select the surface, Generate – display parameters (all on) – reply – verify – 2D, Dynamic – select animation speed – play – blank required for verification – ok pull – ok, List for APT program

Step 8 File – save as – ok

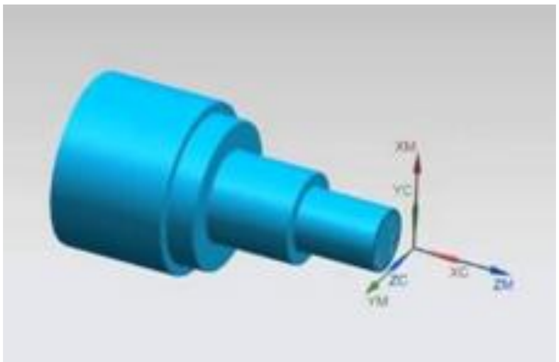
CAVITY MILLING USING – NX CAM



Create rectangular block with cavities with respect to given dimensions as shown in U.G. or Solid Edge and save

Step 1	File – open – Part name – ok Go to start – **select manufacturing** – Cam general – Mill contour – initialize.
Step 2	**Program order view** select all options one by one Go to operation navigator (right hand side) – select MCS spindle – workpiece, Double click on MCS spindle - Double click on workpiece.
Step 3	Select Part – Part body , Select blank – blank body, Give Autoblank (enter some value ex. 5 to add material in all direction except in Z direction.
Step 4	**Create geometry** Parent group – geometry , Name – work piece – ok Part – select – select all – ok –material – chose material - ok
Step 5	**Create tool** Parent group – generic machine , Name – face mill – ok Milling tool – 5 parameters – give diameter value – say 10.00 and other dimensions if required – material – chose material – display tool - ok
Step 6	**Create method** Parent group – method , Name – mill method – apply Cut method – end milling - ok
Step 7	**Create operation** Planar mill – program – NC, Use geometry – work piece, Use tool – mill, Use method – mill method, name – planar mill – apply – planar mill: part select, cut method – zigzag, Generate – display parameters – all on – ok Verify – 2D – dynamic select animation speed – play – blank required for verification – ok – pull the arrow and select the thickness – ok List of APT program
Step 8	File – save as – ok

TURNING USING – NX CAM



Create a solid model with respect to given dimensions as shown in U.G. or Solid Edge and save

Step 1	File – open – Part name – ok Go to start – **select manufacturing** – Cam general - Turning– initialize.
Step 2	**Program order view** Go to operation navigator (right hand side) – MCS spindle – workpiece, Double click on MCS spindle - Double click on workpiece.
Step 3	Select Part – Part body , Select blank – blank body
Step 4	**Create Program** Parent group – geometry , Name – work piece – ok Part – select – select all – ok –material – chose material - ok
Step 5	**Create tool** Select tool – OD_55_L, if required – material – chose material – display tool - ok
Step 6	**Create method** Parent group – method , Name Lathe rough– apply
Step 7	**Create operation** Rough turn OD – program – NC, Use geometry – turning work piece, Use tool – OD_55_L, Use method – Lathe rough, cut method – linear zigzag, Generate – display parameters – all on – ok Verify – 2D – dynamic select animation speed – play – blank required for verification – ok – pull the arrow and select the thickness – ok List of APT program Follow same steps for Finish turn using– Lathe Finish
Step 8	File – save as – ok

COMPUTER AIDED ENGINEERING ANALYSIS

Finite Element Method

FEM has become a powerful tool for the numerical solution of a wide range of engineering problems. Applications range from deformation and stress analysis of automotive, aircraft, building and bridge structures to field analysis of heat flux, fluid flow, magnetic flux, seepage, and other flow problems. With the advances in computer technology and CAD systems, complex problem can be modeled with relative ease. Several alternative configurations can be tested on a computer before the first prototype is built. All of this suggests that we need to keep pace with these developments by understanding the basic theory, modeling techniques, and computational aspects of the finite element method.

How Does Finite Element Method Work?

FEM uses a complex system of points called **nodes,** which make a grid called a **Mesh**. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many **Elements**. An Element is a section of a body obtained from dividing the body up into a finite number of regions.

Why Finite Element Method?

- FEA is the most widely applied computer simulation method in Engineering.
- It is very closely integrated with CAD/CAM applications.
- It is very well proven , tested and validated method for simulating any complex practical scenario in the area of Structural ,Thermal ,Vibration etc..

Application of FEM in Engineering

- Mechanical / Aerospace / Civil Engineering / Automobile Engineering
- Structural Analysis (Static / Dynamic , Linear / Non-Linear)
- Thermal Analysis (Steady State / Transient)
- Electromagnetic Analysis
- Geomechanics
- Biomechanics

Advantages of FEA

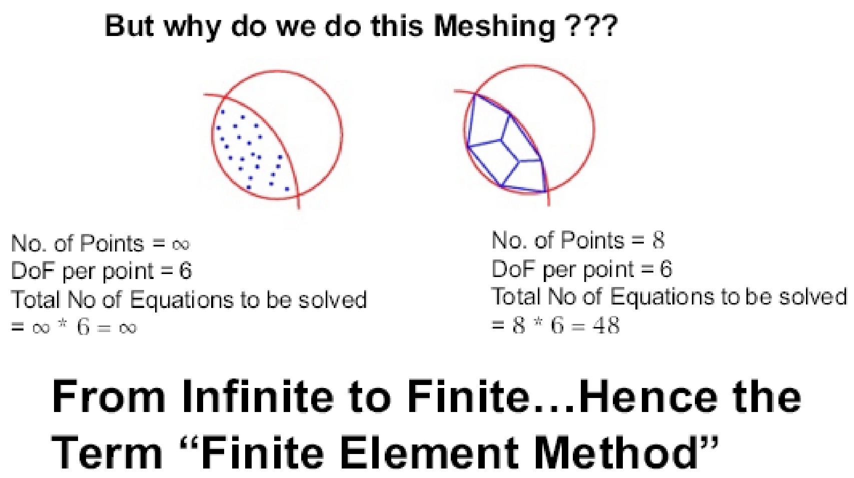
- Cost ↓
- Design Cycle time ↓
- No. of Prototypes ↓
- Testing ↓
- Design Optimization ↑
- Visulalization ↑

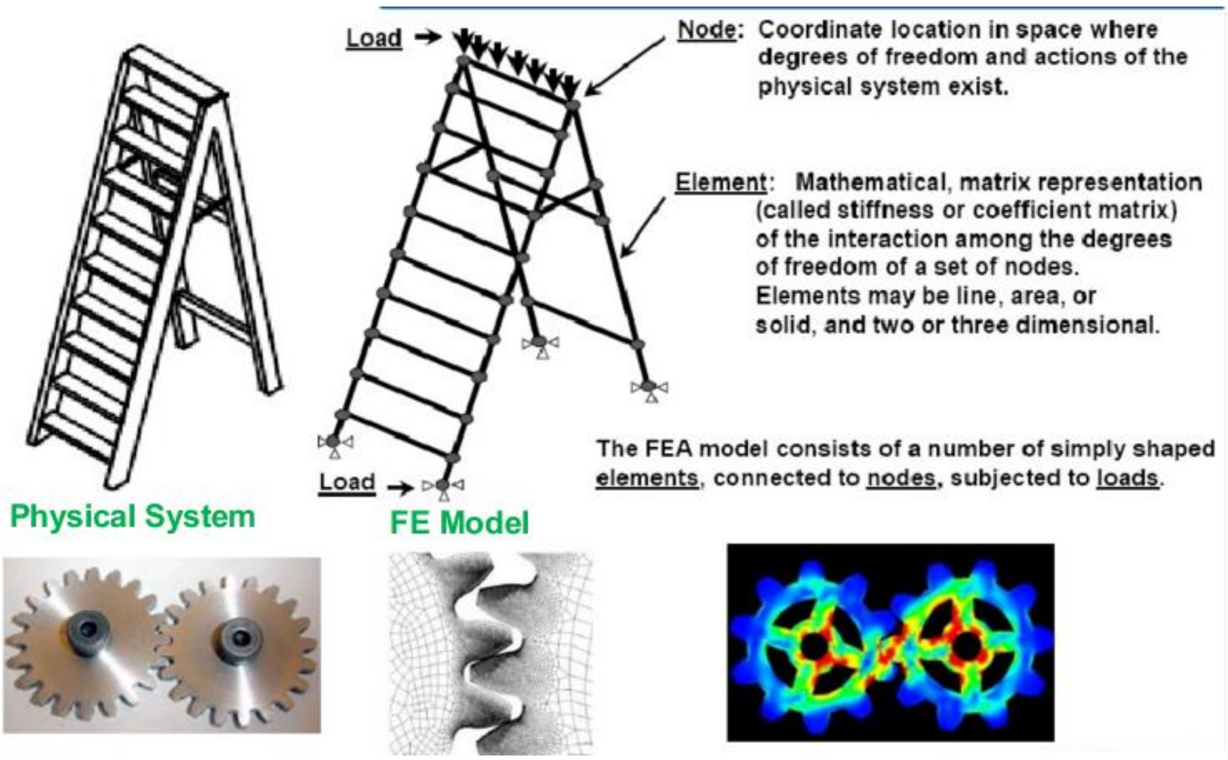
Commercial FEA Tools/Software Packages

- *ANSYS* (General purpose, PC and workstations)
- *SDRC/I-DEAS* (Complete CAD/CAM/CAE package)
- *NASTRAN* (General purpose FEA on mainframes)
- *ABAQUS* (Nonlinear and dynamic analyses)
- *COSMOS* (General purpose FEA)
- *ALGOR* (PC and workstations)
- *PATRAN* (Pre/Post Processor)
- *HyperMesh* (Pre/Post Processor)
- *Dyna-3D* (Crash/impact analysis)

Discretization

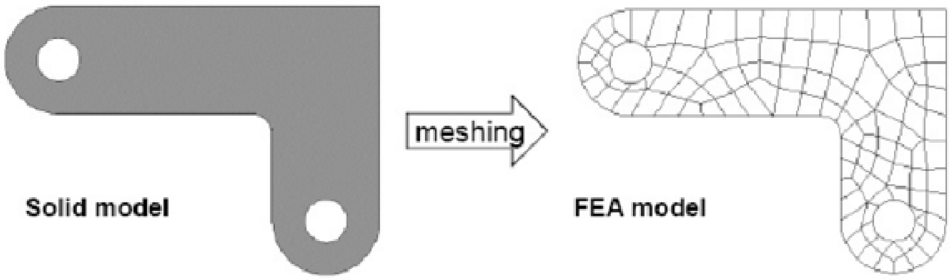
Concept of FEM is all about Discretization (Meshing) i.e. Dividing a big structure / component into small discrete Blocks (Nodes and Element concept)



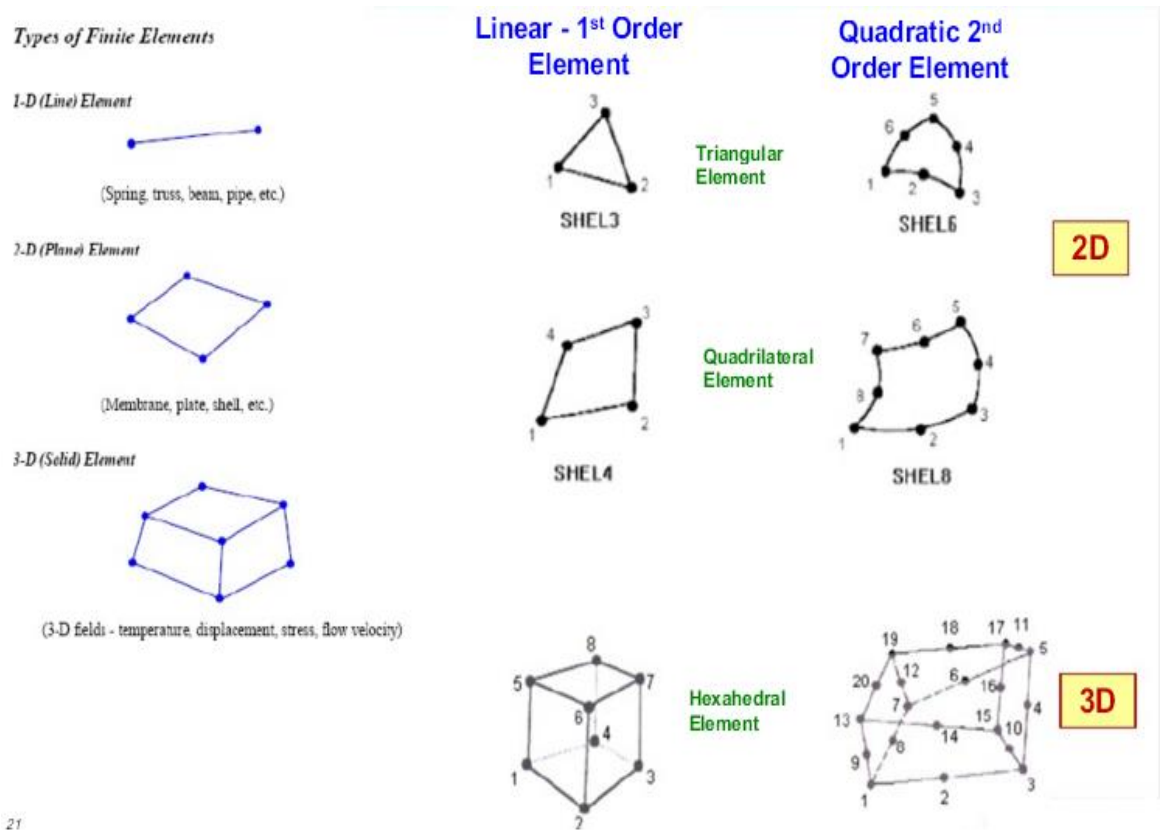


Meshing **is the process used to "fill" the solid model with nodes and elements, i.e, to create the FEA model.**

- **Remember, you need nodes and elements for the finite element solution, not just the solid model. The solid model does NOT participate in the finite element solution.**



Different Type of Elements



About ANSYS

- ANSYS is general-purpose finite element analysis (FEA) software package.
- Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements.
- The software implements equations that govern the behavior of these elements and solves them.
- Results then can be presented in tabulated or graphical forms.
- This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand.
- Systems that may fit into this category are too complex due to their geometry, scale, or governing equation.

ANSYS Interface

There are two methods to use ANSYS.

- The first is by means of the graphical user interface or GUI.
- The second is by means ANSYS Parametric Design Language (APDL) or Macro.

Generic Steps to Solve Problem in ANSYS:

- **Build Geometry**

Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinates system within ANSYS.

- **Define Material Properties**

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

- **Generate Mesh**

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

- **Apply Loads**

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

- **Obtain Solution**

This is actually a step, because ANSYS needs to understand within what state (steady state, transient… etc.) the problem must be solved.

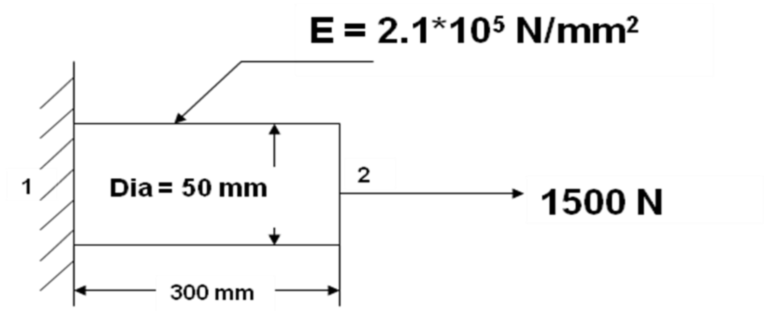
- **Present the Results**

After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots.

Problems on Bars

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 Main Menu -Preferences

Select -STRUCTURAL -ok

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- Link -3D finit stn 180- ok -close

Real constants - Real constants for the link 180 element type are no longer supported (Not required)

Material properties – Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2.1e5 – (If required enter PXY) - ok- enter -close

Step 3 -Preprocessor

Sections- Link - Add -Add link section with ID -1 **,** section name bar 1, link area - 1963.49

Modeling -Create -Nodes -in active CS -apply -x,y,z location in CS - 300- ok. Nodes 1 and 2 created at the location (0,0) and (300,0)

Create -elements -element attributes- Element type no- 1 Link180- Material no-1- section no 1 bar1 -ok- Auto numbered-thru nodes -pick 1 & 2-apply -ok (element is created through nodes).

Step 4 -Preprocessor

Loads - Analysis Type - New Analysis- Static

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- 1500 (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

List Results - Nodal Solutions - DOF solution - pick X,Y, Z components - Displacements at nodal points will be listed

NODE	UX
1	0.0000
2	0.10914E-02

List Results -Element Solutions- Stress- pick X,Y, Z components of stress - stresses are listed

ELEMENT= 1 LINK180

NODE	SX	SY	SZ	SXY	SYZ	SXZ
1	0.76395	0.0000	0.0000	0.0000	0.0000	0.0000
2	0.76395	0.0000	0.0000	0.0000	0.0000	0.0000

Element table -define table -add-results data item- By sequence num- LS- LS 1- By sequence num- LS- LS 2- ok

Step 7 -**General Post Processor**

Plot results- **Contour Plots** - **Line element results**-element table item at node I -LS1 -element table item at node J -LS 2- ok (stress diagram will be displayed)

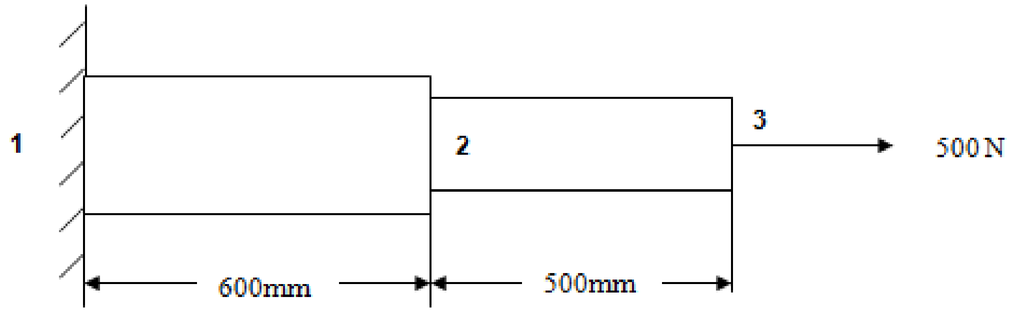
To see the size and shape of the element and for animation of the deformed shape use PLOT and PLOTCtrls.

PLOTCtrls - Style- Size and Shape - Display of element - on/off - on - ok. At the right-hand side of the graphic window choose ISOMETRIC View option. Element will be displayed with its size and shape.

PLOTCtrls- Animate - Deformed Shape - Def+Undeformed - ok. Animation with deformation will be seen.

Exercise 2 - Consider the stepped bar shown in figure below.

Determine 1) The nodal displacement 2) stress in each element 3) reaction forces



$E_1 = 2.0*10^5$ 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 Main Menu -Preferences

Select -STRUCTURAL -ok

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- Link -3D finit stn 180- ok -close

Real constants - Real constants for the link 180 element type are no longer supported (Not required)

Material properties – Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2e5 – (If required enter PXY) - ok-

Material-New Model –Define Material ID -2- Material Model Number 2 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 0.7e5 – (If required enter PXY) - ok- enter -close

Step 3 -Preprocessor

Sections- Link - Add -Add link section with ID -1 **,** section name bar 1, link area - 900 -apply

Add link section with ID -2 **,** section name bar 2, link area - 600 -ok

Modeling -Create -Nodes -in active CS -apply -x,y,z location in CS - x -600- apply -x,y,z location in CS - x- 1100-ok. Nodes 1 ,2 and 3 are created at the location (0,0) , (600,0) and (1100,0)

Create -elements -element attributes- Element type no- 1 Link180- Material no-1- section no 1 bar1 -ok- Auto numbered-thru nodes -pick 1 & 2-apply -ok (element 1 is created through nodes), element attributes- Element type no- 1 Link180- Material no-2- section no 2 bar2 -ok-Auto numbered-thru nodes - pick 2 & 3-apply -ok (element 2 is created through nodes)

Step 4 -Preprocessor

Loads - Analysis Type - New Analysis- Static

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 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

List Results - Nodal Solutions - DOF solution - pick X,Y, Z components - Displacements at nodal points will be listed

NODE	UX
1	0.0000
2	0.16667E-02
3	0.76190E-02

List Results -Element Solutions- Stress- pick X,Y, Z components of stress - stresses are listed

ELEMENT= 1 LINK180

NODE	SX	SY	SZ	SXY	SYZ	SXZ
1	0.55556	0.0000	0.0000	0.0000	0.0000	0.0000
2	0.55556	0.0000	0.0000	0.0000	0.0000	0.0000

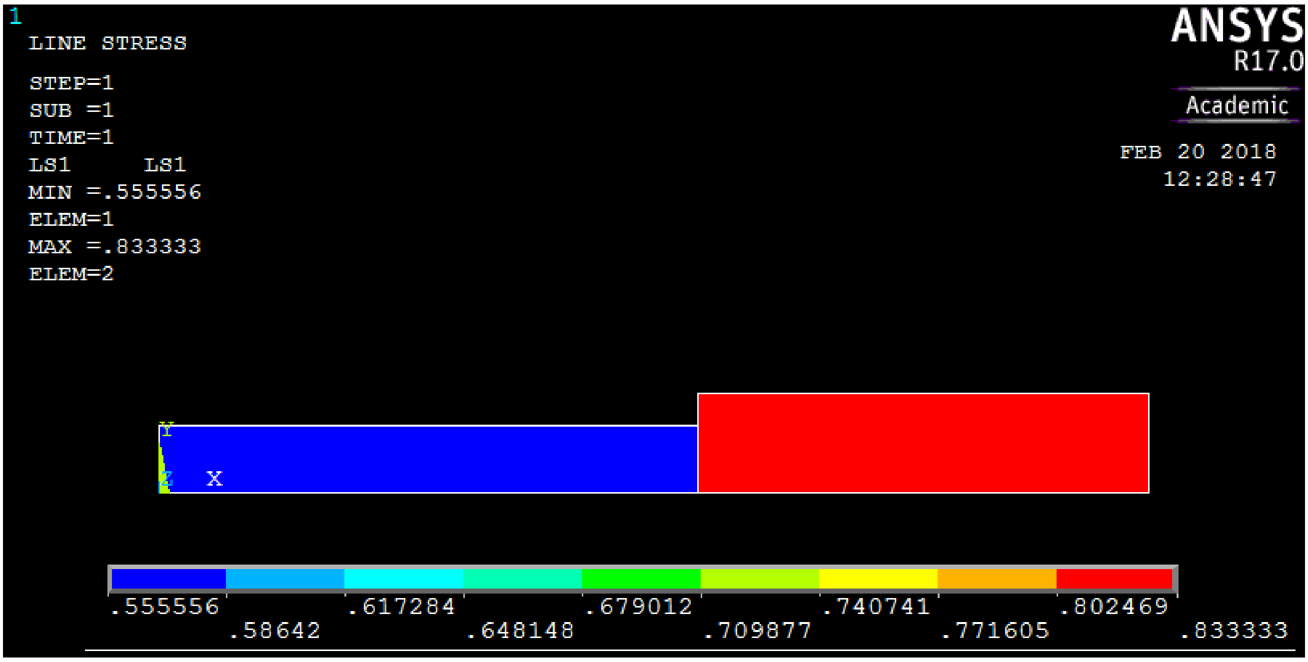
ELEMENT= 2 LINK180

NODE	SX	SY	SZ	SXY	SYZ	SXZ
2	0.83333	0.0000	0.0000	0.0000	0.0000	0.0000
3	0.83333	0.0000	0.0000	0.0000	0.0000	0.0000
2	0.76395	0.0000	0.0000	0.0000	0.0000	0.0000

Element table -define table -add-results data item- By sequence num- LS- LS 1- By sequence num- LS- LS 2- ok

Step 7 -General Post Processor

Plot results- **Contour Plots** - **Line element results**-element table item at node I -LS1 -element table item at node J -LS 2- ok (stress diagram will be displayed)



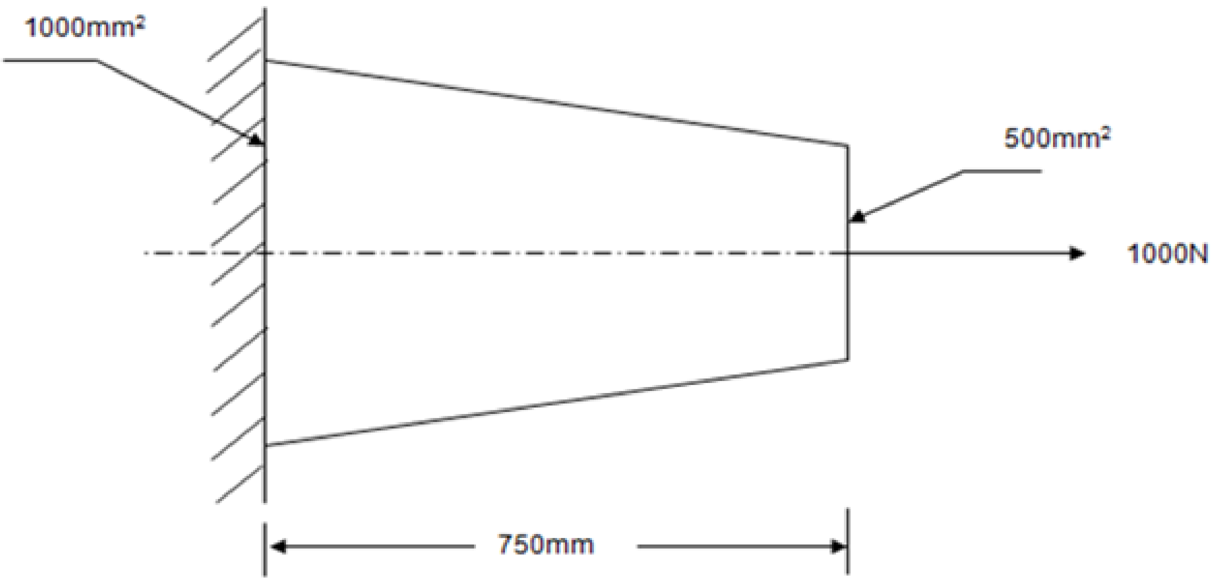
To see the size and shape of the element and for animation of the deformed shape use PLOT and PLOTCtrls.

PLOTCtrls - Style- Size and Shape - Display of element - on/off - on - ok. At the right-hand side of the graphic window choose ISOMETRIC View option. Element will be displayed with its size and shape.

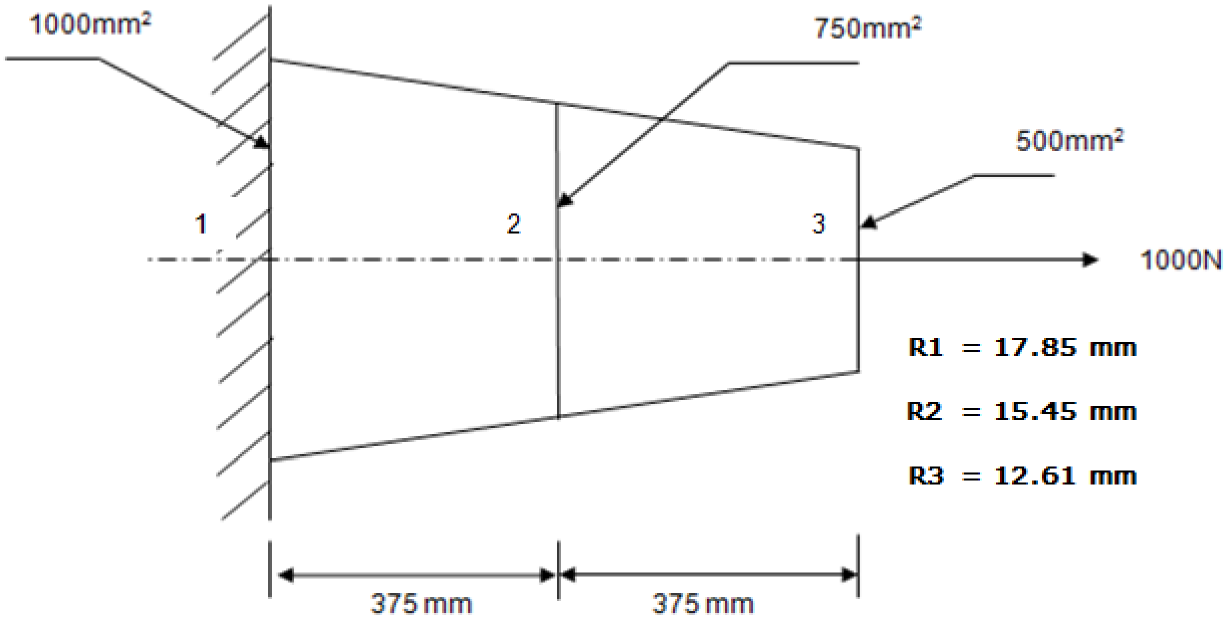
PLOTCtrls- Animate - Deformed Shape - Def+Undeformed - ok. Animation with deformation will be seen.

Exercise 3 – Tapered bar

For the following bar, find the nodal displacements. The cross sectional area (Circular) decreases linearly from 1000 mm² to 500 mm². Use two elements. Take E = 2x 10^5 MPa,



Finite element model:



Note: Since in Ansys No Link element with tapered configuration is available, Beam- 2 Node 188 element with tapered configuration is used to solve this problem.

Step 1 -Ansys Main Menu -Preferences

Select -STRUCTURAL –ok

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- Beam- 2 Node 188- ok -close

Real constants - Beam 188 does not require real constants

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 3 -Preprocessor

Sections- Beam -Common sections - ID -1 Sub type solid circle with offset to -Centroid- Enter R = 17.85, N = 30 and T = 30 -apply (Solid circular section having radius 17.85 mm , number of divisions =30 along radial & angular direction will be created) Repeat the procedure for creating the sections with ID -2 and ID -3 for sections with radii 15.45 mm & 12.61 mm with N = 30 and T = 30 -ok

Sections- Beam -Tapered sections - by XYZ location - New tapered section ID - 4 - Beginning section ID -1, XYZ location of beginning sect - X = 0- Ending section ID -2, XYZ location of ending sect - X = 375 - apply - New tapered section ID - 5 - Beginning section ID -2, XYZ location of beginning sect - X = 375- Ending section ID -3, XYZ location of ending sect - X = 750 - ok

Modeling -Create -Nodes -in active CS -apply -x,y,z location in CS - x -375- apply -x,y,z location in CS - x-750-ok. Nodes 1 ,2 and 3 are created at the location (0,0) , (375,0) and (750,0)

Create -**Elements -element attributes**- Element type no- 1 BEAM188- Material no-1- section no 4 -Target element shape -circle-ok

Auto numbered-thru nodes -pick 1 & 2-apply -ok (element 1 is created through nodes), **Elements -element attributes**- Element type no- 1 BEAM188- Material no-1- section no 5 -Target element shape -circle-ok (element 2 is created through nodes)

Step 4 -Preprocessor

Loads - Analysis Type - New Analysis- Static

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 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**

List Results - Nodal Solutioons - DOF solution - pick X components - Displacements at nodal points will be listed

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UX
1	0.0000
2	0.21529E-02
3	0.51850E-02

List Results -Reaction Solutions- All items-ok

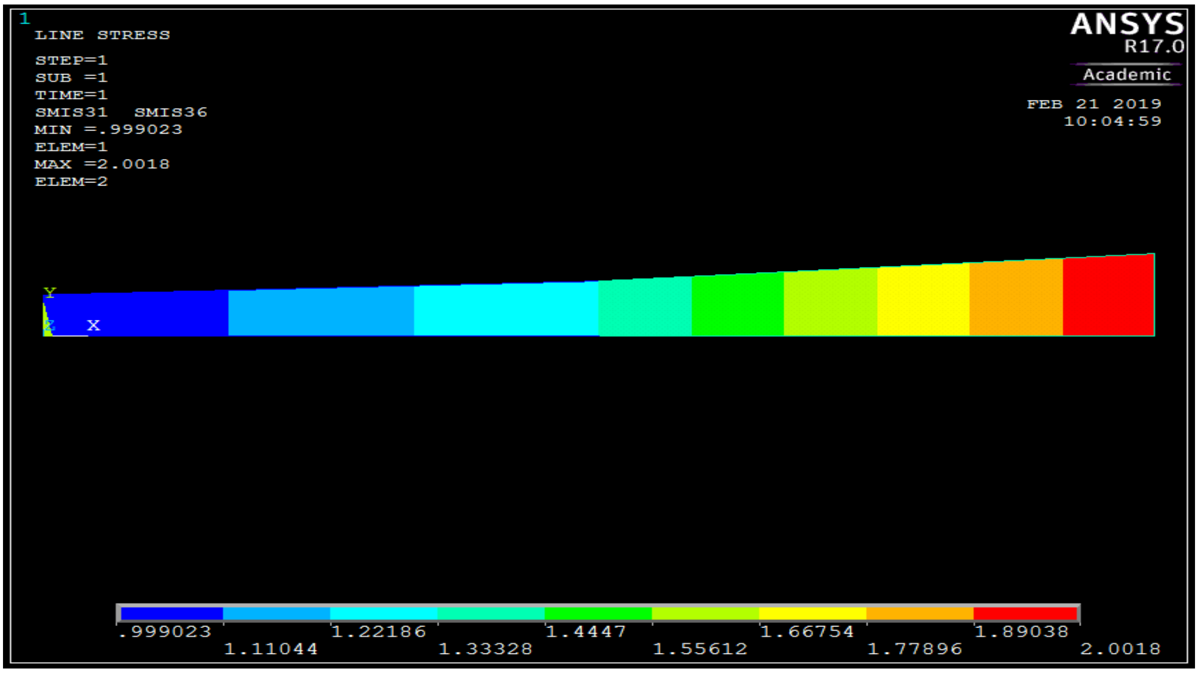
THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	FX	FY	FZ	MX	MY	MZ
1	-1000.0	0.0000	0.0000	0.0000	0.0000	0.0000

Step 7 -**General Post Processor**

Element table -define table -add-results data item- By sequence num- SMISC - SMISC 31 - By sequence num- SMISC - SMISC 36 -ok

Plot results- **Contour Plots** - **Line element results**-element table item at node I -SMISC 31 -element table item at node J - SMISC 36 - ok (stress diagram will be displayed)



Note:

The tapered bar problem can also be solved by using 3D finit stn 180 elements with elements of uniform cross sectional area.

Since Area A_1 = 1000 mm^2, A_2 = 750 mm^2 and A_3 = 500 mm^2 at nodes 1, 2 & 3 respectively, the area for element 1 is calculated taking the average of A_1 & A_2 and area for element 2 is calculated taking the average of A_2 & A_3 and following the general procedure nodal solution, element solution and others can be obtained.

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UX
1	0.0000
2	0.21429E-02
3	0.51429E-02

THE FOLLOWING X,Y,Z VALUES ARE IN GLOBAL COORDINATES

ELEMENT= 1 LINK180

NODE	SX	SY	SZ	SXY	SYZ	SXZ
1	1.1429	0.0000	0.0000	0.0000	0.0000	0.0000
2	1.1429	0.0000	0.0000	0.0000	0.0000	0.0000

ELEMENT= 2 LINK180

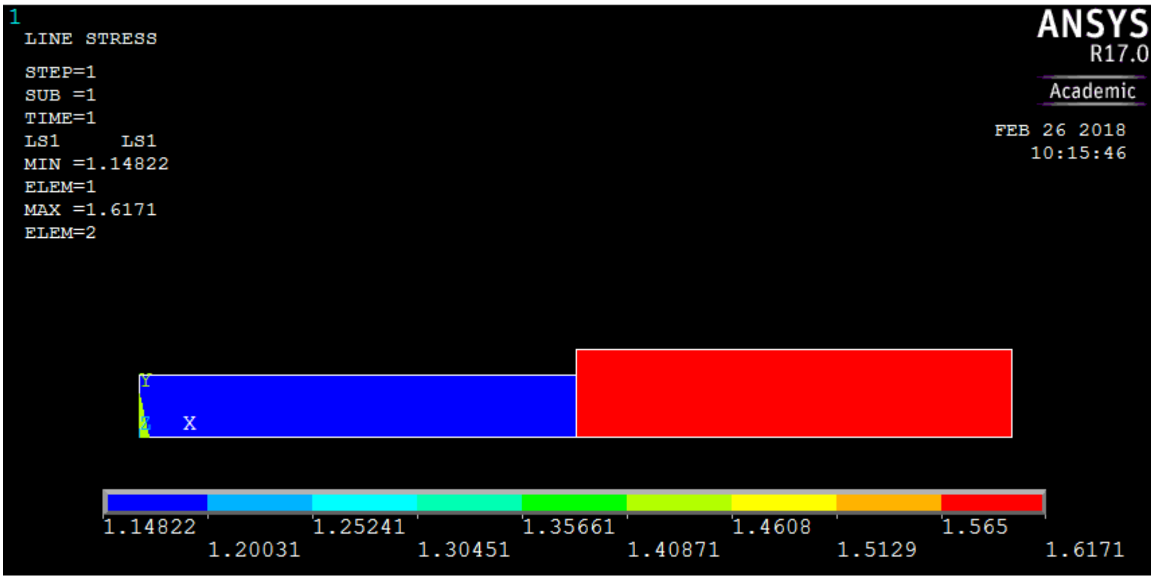
NODE	SX	SY	SZ	SXY	SYZ	SXZ
2	1.6000	0.0000	0.0000	0.0000	0.0000	0.0000
3	1.6000	0.0000	0.0000	0.0000	0.0000	0.0000

THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	FX	FY	FZ
1	-1000.0	0.0000	0.0000

Element table -define table -add-results data item- By sequence num- LS- LS 1- By sequence num- LS- LS 2 - ok

Plot results- **Contour Plots** - **Line element results**-element table item at node I -LS1 -element table item at node J - LS2 - ok (stress diagram will be displayed)

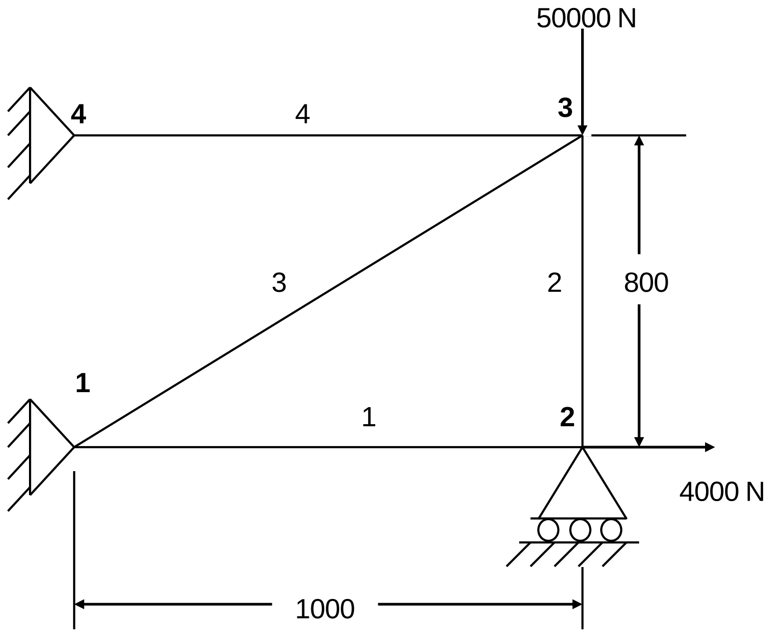


List Results -Element Solutions- Stress- pick X - stresses are listed

SECTION ID = 4, SXX = 1.1482
SECTION ID = 5, SXX = 1.6171

Problems on Trusses

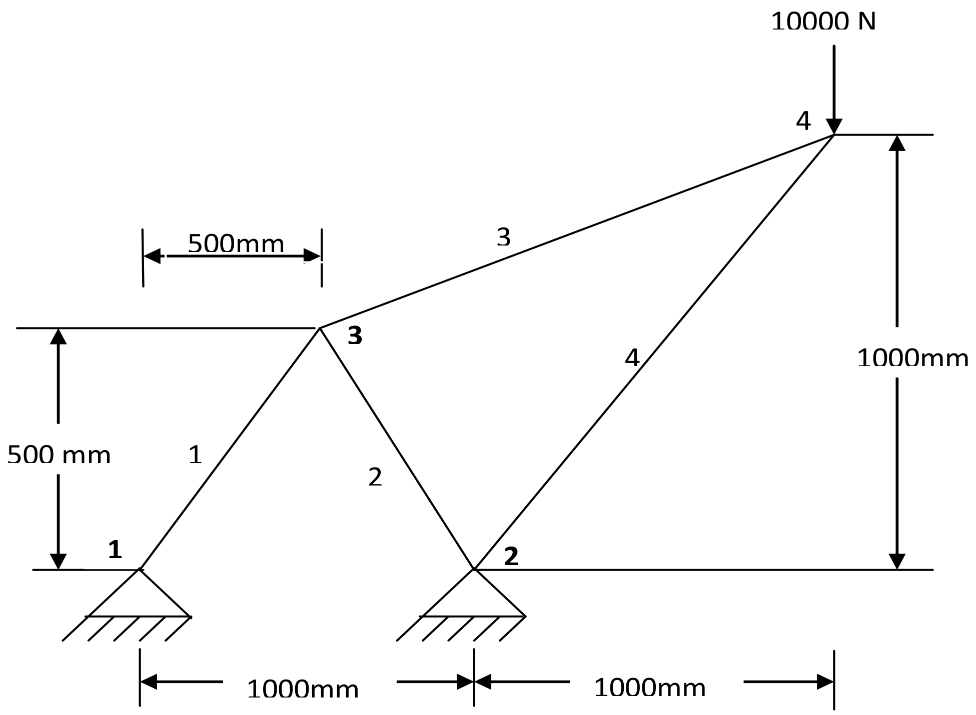
Exersice 1 - Consider the four bar truss shown in figure. For the given data, find 1) stress in each element 2) reaction forces 3) Nodal displacement. E = 2×10^5 N/ mm^2 A = 10 mm^2



Exercise 2 - Find the nodal displacement and the internal stresses developed in the planar truss shown in figure when a vertically downward load of 10000 N is applied as shown.

Take For elements 1 and 2, E = 2×10^5 N/ mm^2 A = 50 mm^2

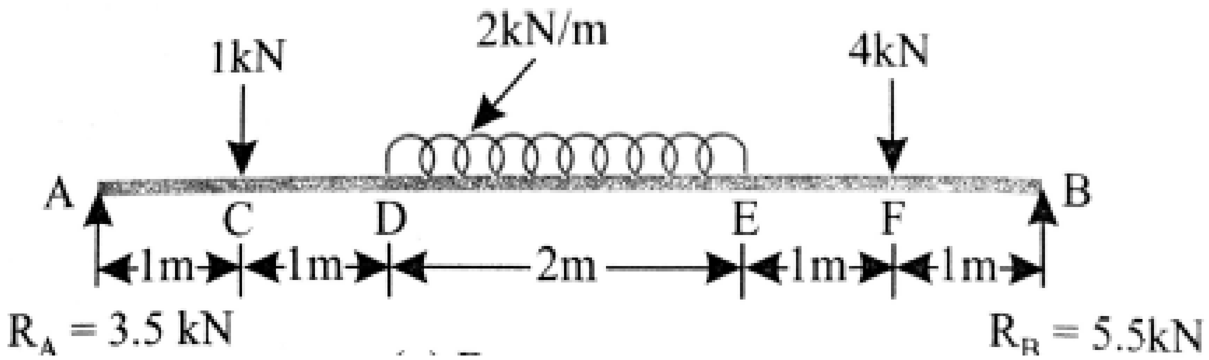
For elements 3 and 4, E = 0.7×10^5 N/ mm^2 A = 50 mm^2



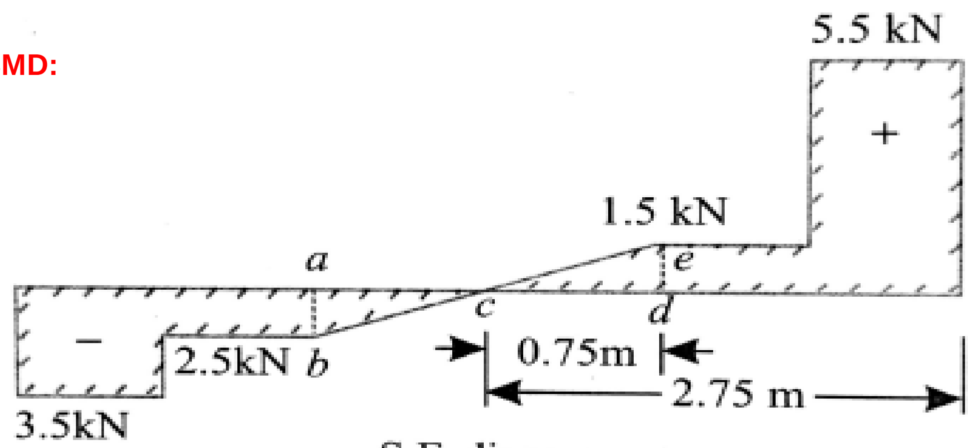
Problems on Beams

Exercise 1: Compute the shear force and bending moment diagrams for the beam shown and find the reactions at the supports.

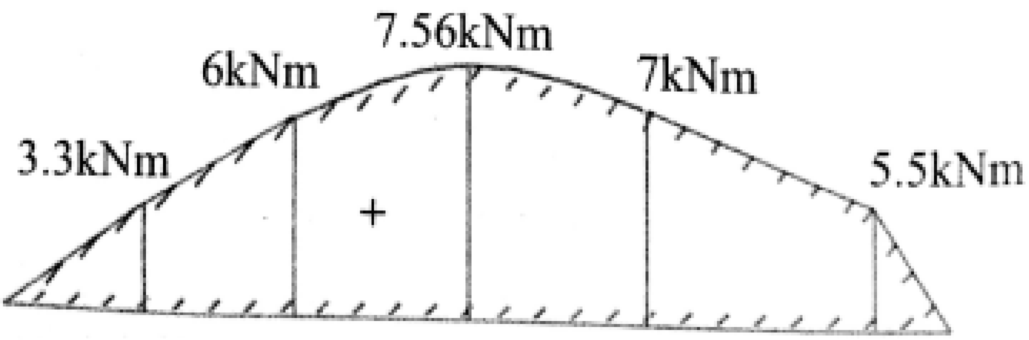
Assume rectangular c/s of 200 mm x 300 mm and Young's Modulus of 2×10^5 N/mm^2 for all problems on beams. Use consistent units for solving the problems.



SFD & BMD:



S.F. diagram



B.M. diagram

Step 1 -Ansys Main Menu -Preferences

Select -STRUCTURAL –ok

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- Beam- 2 Node 188- ok -close

Real constants - Beam 188 does not require real constants

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 3 -**Preprocessor**

Sections- Beam - Common sections- offset to - centroid- B - 200, H - 300-ok

Modeling -Create -key points -in active CS -apply (first key point is created)- x,y,z location in CS -**1000** (x value w.r.t first key point)- apply (second key point is created) - **2000** (x value w.r.t first key point) - apply (third key point is created) - **4000** (x value w.r.t first key point)- apply (fourth key point is created)- **5000** (x value w.r.t first key point)- apply (fifth key point is created)- **6000** (x value w.r.t first key point)- apply (sixth key point is created)- ok

Create - Lines -straight lines through key points - pick 1 & 2-pick 2 & 3 -pick 3 & 4 - pick 4 & 5-pick 5&6 - ok (lines are created through keypoints).

Meshing - Size control-manual size -Line - picked lines-pick lines one by one and mention the no. of divisions on each line viz for lines 1, 2, 4 and 5 give no. of divisions as 10. For the line 3 take more no. of divisions say 20 - ok

Mesh - Line - pick all - meshing is done with elements and nodes

Plot Cntrls- Numbering- Node numbering -on **-** Node numbers will be displayed

Plot- Keypoints -Keypoints - key points will be displayed

Plot Cntrls- Numbering- Key point numbering -on **-** Key point numbers will be displayed

Step 4 -Preprocessor

Loads –Define Loads - Apply -Structural Displacement -on Key point -pick Key point 1 & 6 - apply- DOF to be constrained -Select UX,UY,UZ,ROTX and ROTY -ok

Loads - Define Loads -Apply - Force/Moment -on Key point -pick Key point 2 -apply- direction of Force/Moment- FY -Force/Moment value- **-1000** (-ve value) -apply- pick Key point 5- apply- direction of Force/Moment- FY -Force/Moment value**- - 4000** (-ve value) –ok

To apply UDL:

Loads - Define Loads -Apply - Pressure -on beams (use box option) -pick beams between key points between 3 and 4 -apply- Load key -2 (for applying load in Y direction)- pressure value at node I -2 and pressure value at node J -2 -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

Element table -define table -add-results data item- By sequence num- smisc-smisc 3-apply -By sequence num- smisc-smisc 16-apply -By sequence-num- smisc-smisc 6-apply -By sequence num- smisc-smisc 19-apply

Step 7 -General Post Processor

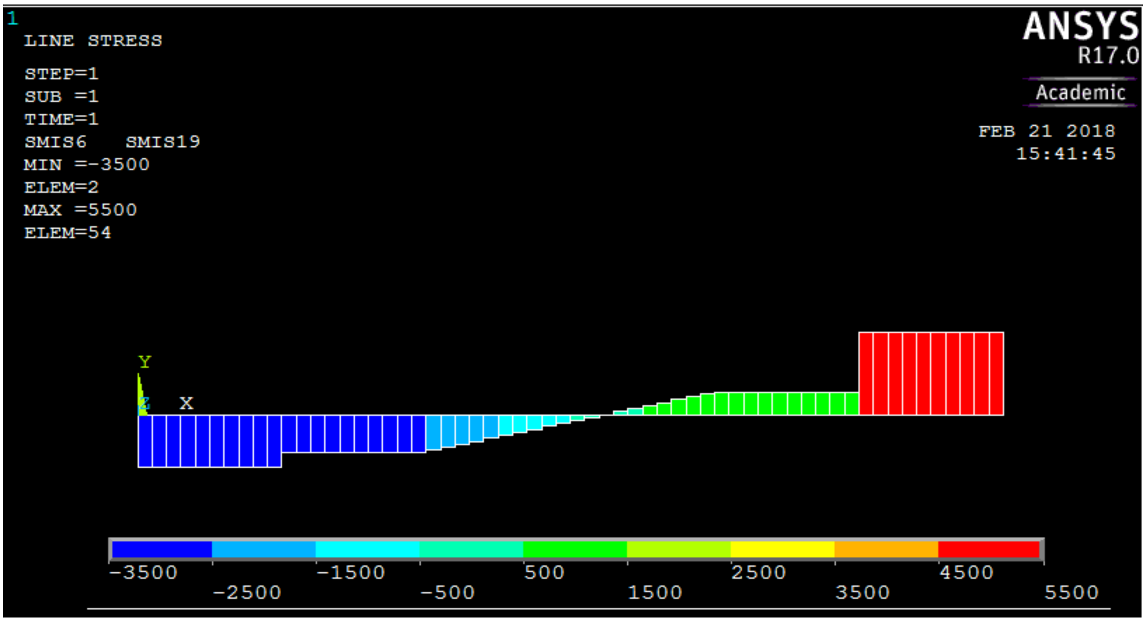
Plot results – Contour Plots - Line element results-element table item at node I -smis 6 -element table item at node J -smis 19- ok **(shear force diagram will be displayed)** Line element - results-element table item at node I -smis 3 -element table item at node J -smis 16- ok (**bending moment diagram will be displayed)**

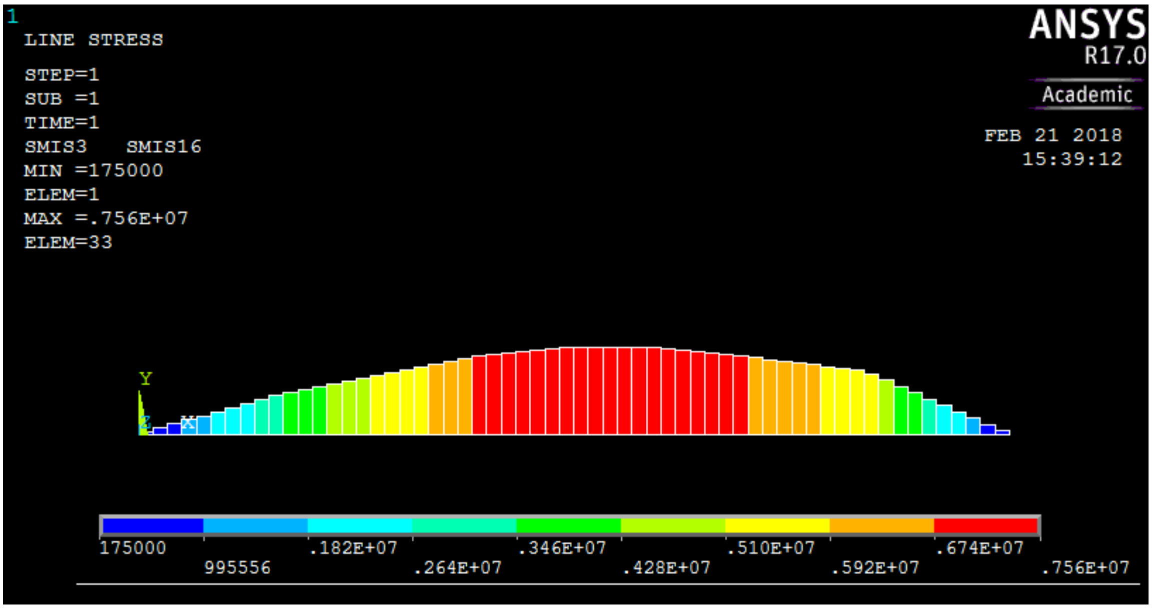
Note:

For Bending Moment diagram use the combination smisc 3 & smisc 16

And for shear force diagram use the combination smisc 6 & smisc 19

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Mz	SMISC	--	3	16
SFy	SMISC	--	6	19



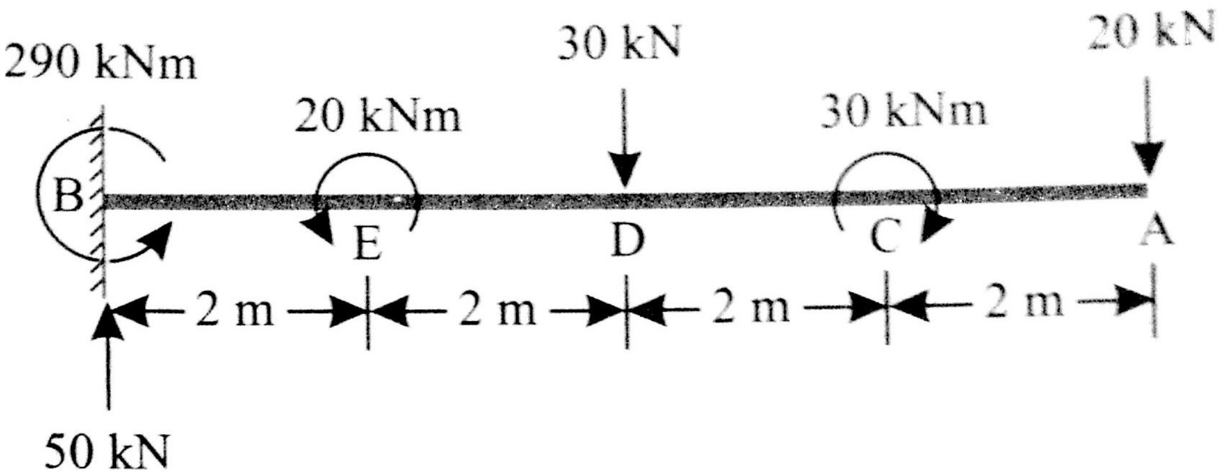


List results

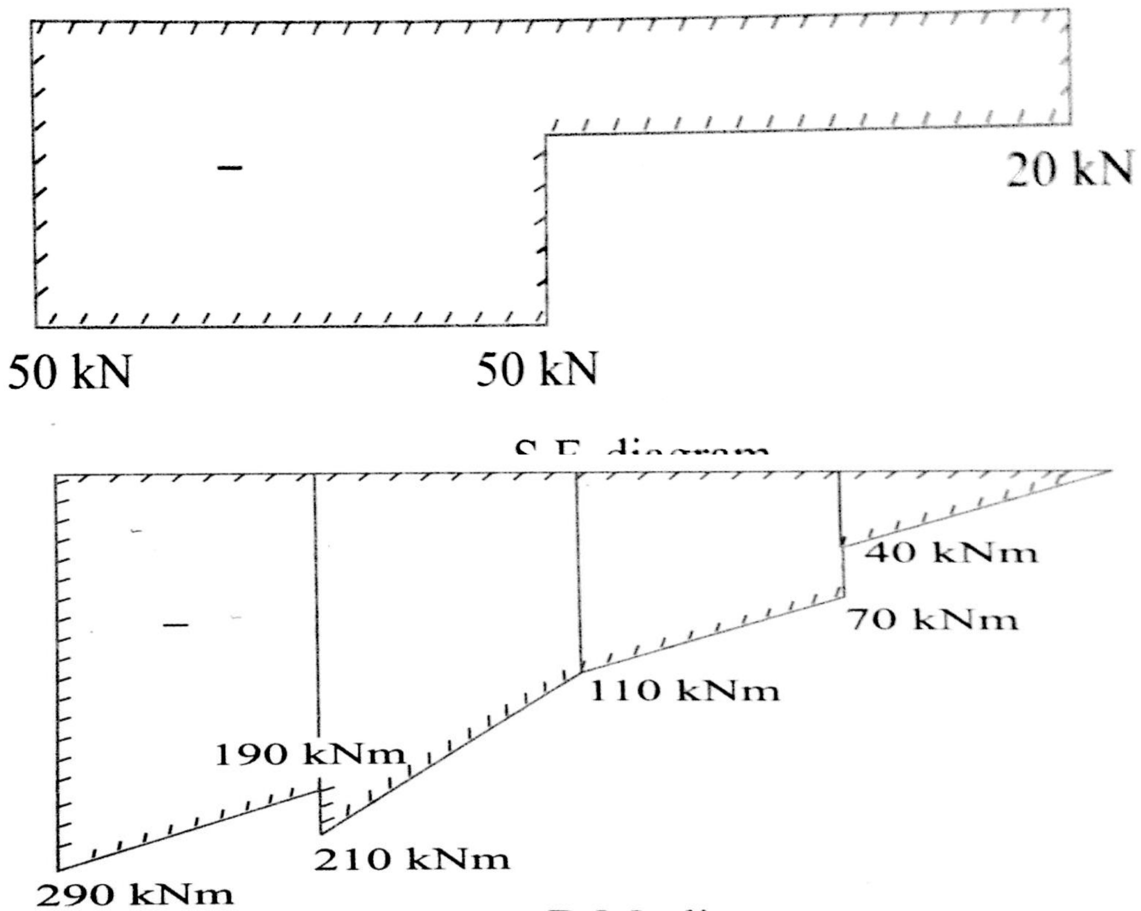
Reaction solution -items to be listed -all items –ok (reaction forces will be displayed with node numbers)

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

Exercise 2:



SFD & BMD:



B.M. diagram

Step 1 -Ansys Main Menu -Preferences

Select -STRUCTURAL –ok

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- Beam- 2 Node 188- ok -close

Real constants - Beam 188 does not require real constants

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 3 -**Preprocessor**

Sections- Beam - Common sections- offset to - centroid- B - 200, H - 300-ok

Modeling -Create -key points -in active CS -apply (first key point is created)- x,y,z location in CS -**2000** (x value w.r.t first key point)- apply (second key point is created) - **4000** (x value w.r.t first key point) - apply (third key point is created) - **6000** (x value w.r.t first key point)- apply (fourth key point is created)- **8000** (x value w.r.t first key point)- apply (fifth key point is created)- ok

Create - Lines -straight lines through key points - pick 1 & 2-pick 2 & 3 -pick 3 & 4 - pick 4 & 5- ok (lines are created through key points).

Meshing - Size control-manual size -Line - picked lines-pick lines one by one and mention the no. of divisions on each line viz for lines 1, 2, 3 and 4 give no. of divisions as 10 - ok

Mesh - Line - pick all - meshing is done with elements and nodes

Plot Cntrls- Numbering- Node numbering -on **-** Node numbers will be displayed

Plot- Keypoints -Keypoints - key points will be displayed

Plot Cntrls- Numbering- Key point numbering -on **-** Key point numbers will be displayed

Step 4 -Preprocessor

Loads –Define Loads - Apply -Structural Displacement -on Key point -pick Key point 1- apply- DOF to be constrained -Select ALL DOF-ok

Loads - Define Loads -Apply - Force/Moment -on Key point -pick Key point 2 -apply- direction of Force/Moment- MZ -Force/Moment value- **-20000 (+value)** -apply- pick Key point 3- apply- direction of Force/Moment- FY -Force/Moment value**- - 30000** (-ve value) –apply- pick Key point 4 -apply- direction of Force/Moment- MZ -Force/Moment value- **-30000** (-ve value) -apply- pick Key point 5- apply- direction of Force/Moment- FY -Force/Moment value**- - 20000** (-ve 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

Element table -define table -add-results data item- By sequence num- smisc-smisc 3-apply -By sequence num- smisc-smisc 16-apply -By sequence-num- smisc-smisc 6-apply -By sequence num- smisc-smisc 19-apply

Step 7 -General Post Processor

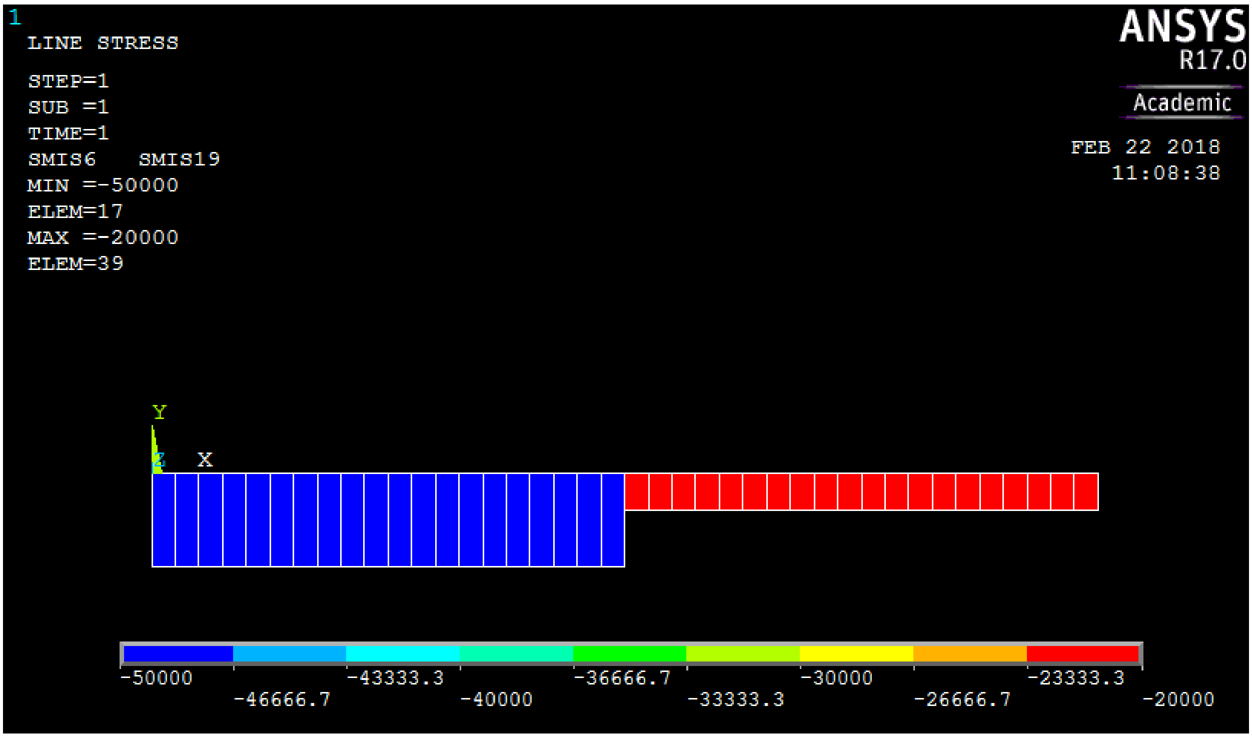
Plot results – Contour Plots - Line element results-element table item at node I -smis 6 -element table item at node J -smis 19- ok **(shear force diagram will be displayed)** Line element - results-element table item at node I -smis 3 -element table item at node J -smis 16- ok (**bending moment diagram will be displayed)**

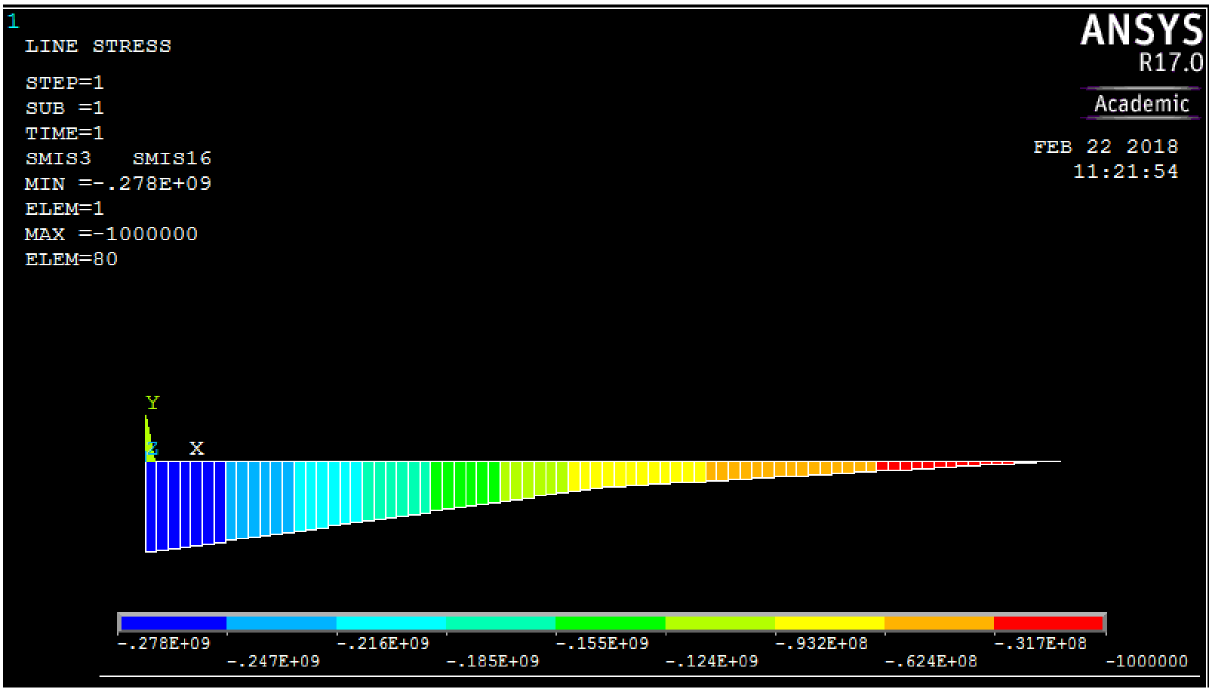
Note:

For Bending Moment diagram use the combination smisc 3 & smisc 16

And for shear force diagram use the combination smisc 6 & smisc 19

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Mz	SMISC	--	3	16
SFy	SMISC	--	6	19



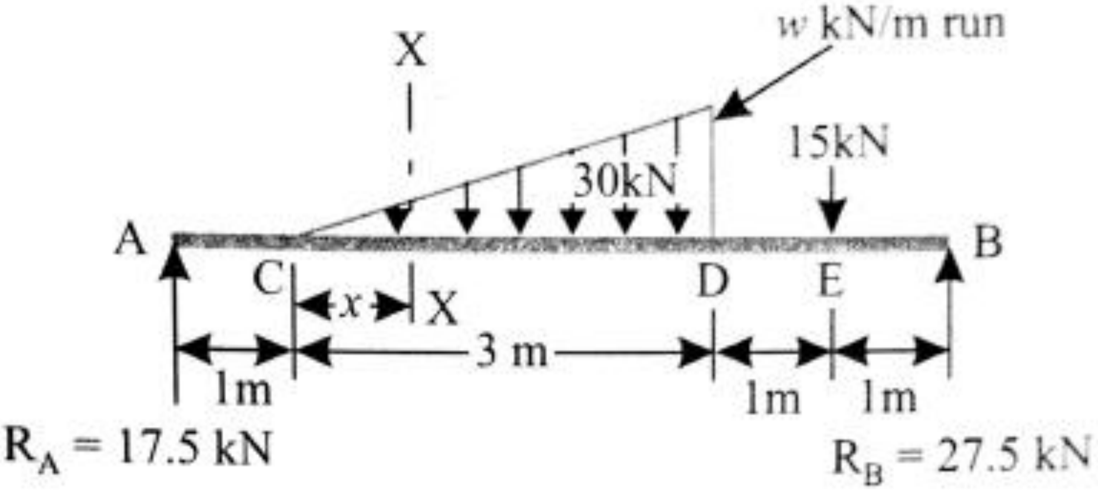


List results

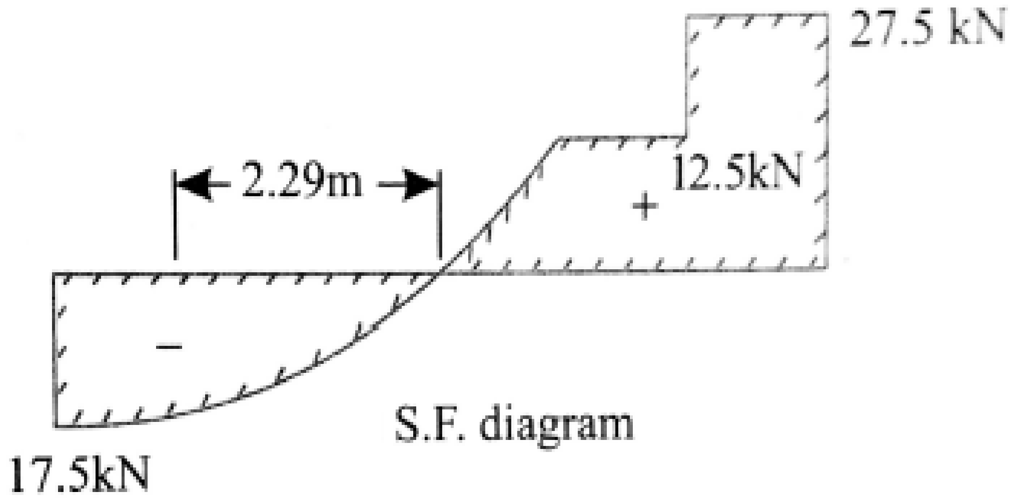
Reaction solution -items to be listed -all items –ok (reaction forces will be displayed with node numbers)

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

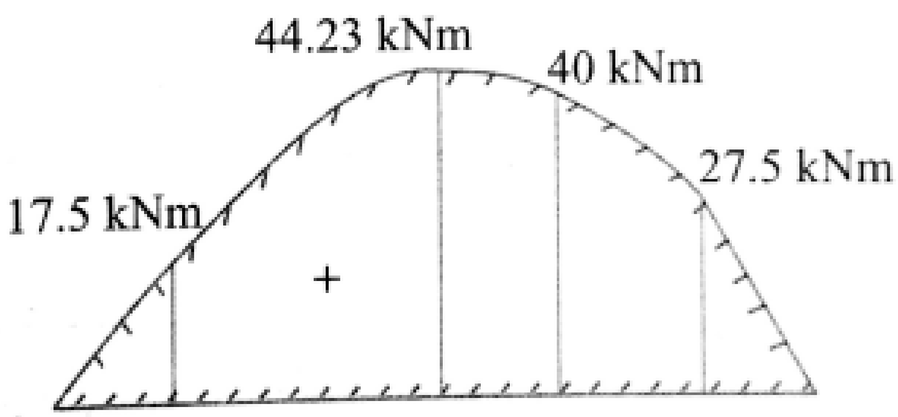
Exercise 3:



SFD & BMD:



S.F. diagram



B.M. diagram

Step 1 -Ansys Main Menu -Preferences

Select -STRUCTURAL –ok

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- Beam- 2 Node 188- ok -close

Real constants - Beam 188 does not require real constants

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 3 -**Preprocessor**

Sections- Beam - Common sections- offset to - centroid- B - 200, H - 300-ok

Modeling -Create -key points -in active CS -apply (first key point is created)- x,y,z location in CS -**1000** (x value w.r.t first key point)- apply (second key point is created) - **4000** (x value w.r.t first key point) - apply (third key point is created) - **5000** (x value w.r.t first key point)- apply (fourth key point is created)- **6000** (x value w.r.t first key point)- apply (fifth key point is created)- ok

Create - Lines -straight lines through key points - pick 1 & 2-pick 2 & 3 -pick 3 & 4 - pick 4 & 5- ok (lines are created through key points).

Meshing - Size control-manual size -Line - picked lines-pick lines one by one and mention the no. of divisions on each line viz for lines 1, 3 and 4 give no. of divisions as 10. For the line 2 take more no. of divisions say 30 - ok

Mesh - Line - pick all - meshing is done with elements and nodes

Plot Cntrls- Numbering- Node numbering -on **-** Node numbers will be displayed

Plot- Keypoints -Keypoints - key points will be displayed

Plot Cntrls- Numbering- Key point numbering -on **-** Key point numbers will be displayed

Step 4 -Preprocessor

Loads –Define Loads - Apply -Structural Displacement -on Key point -pick Key point 1 & 6 - apply- DOF to be constrained -Select UX,UY,UZ,ROTX and ROTY -ok

Loads - Define Loads -Apply - Force/Moment -on Key point -pick Key point 4 -apply- direction of Force/Moment- FY -Force/Moment value- **-15000 (-value)** -ok

For Varying load:

Copy the text written below from a word or text file and paste it in command prompt window. The programme code is to apply pressure varying from 0 kN on element 11 to 20 kN on element 40.

***do, k, 11,40,1**

sfbeam, k, 2, press, (k-11)*20/30, (k-10)*20/30

***enddo**

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

Element table -define table -add-results data item- By sequence num- smisc-smisc 3-apply -By sequence num- smisc-smisc 16-apply -By sequence-num- smisc-smisc 6-apply -By sequence num- smisc-smisc 19-apply

Step 7 -General Post Processor

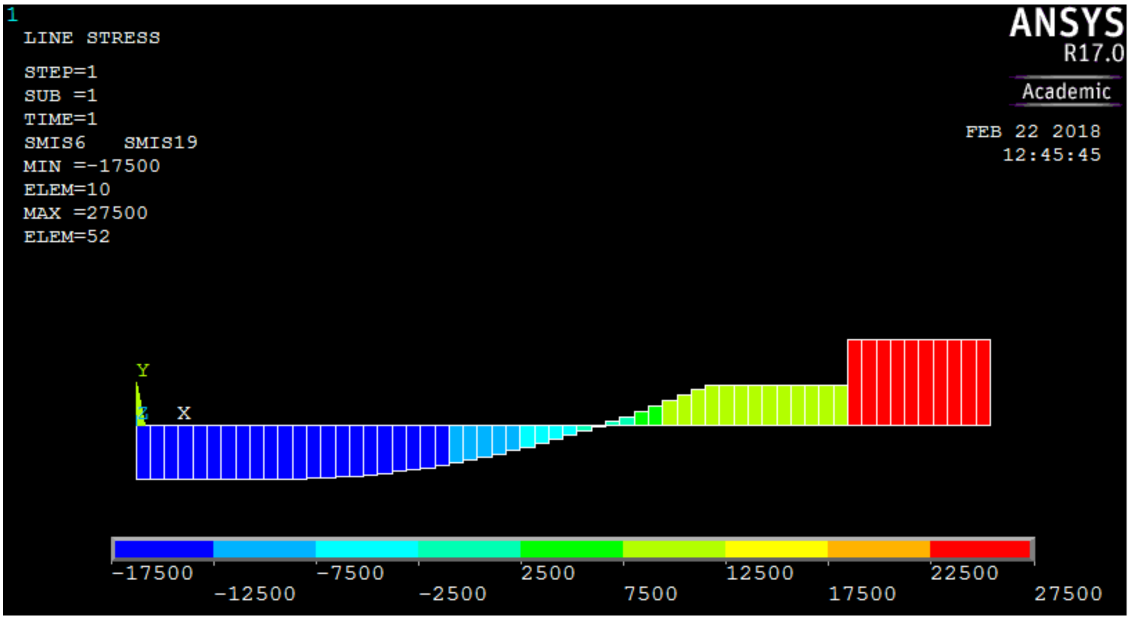
Plot results – Contour Plots - Line element results-element table item at node I -smis 6 -element table item at node J -smis 19- ok **(shear force diagram will be displayed)** Line element - results-element table item at node I -smis 3 -element table item at node J -smis 16- ok (**bending moment diagram will be displayed)**

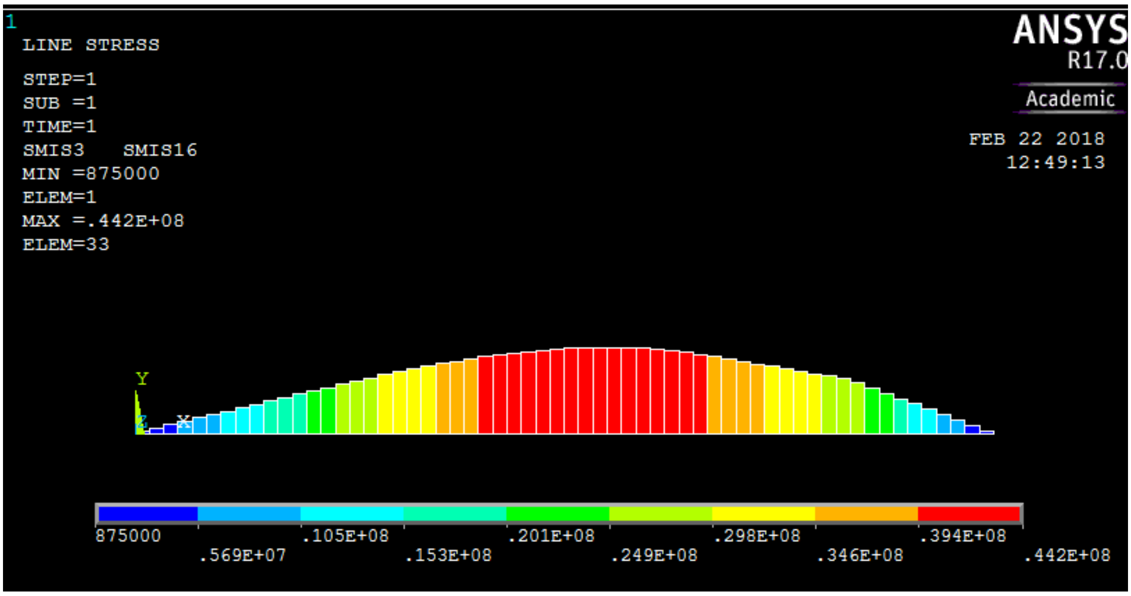
Note:

For Bending Moment diagram use the combination smisc 3 & smisc 16

And for shear force diagram use the combination smisc 6 & smisc 19

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Mz	SMISC	--	3	16
SFy	SMISC	--	6	19



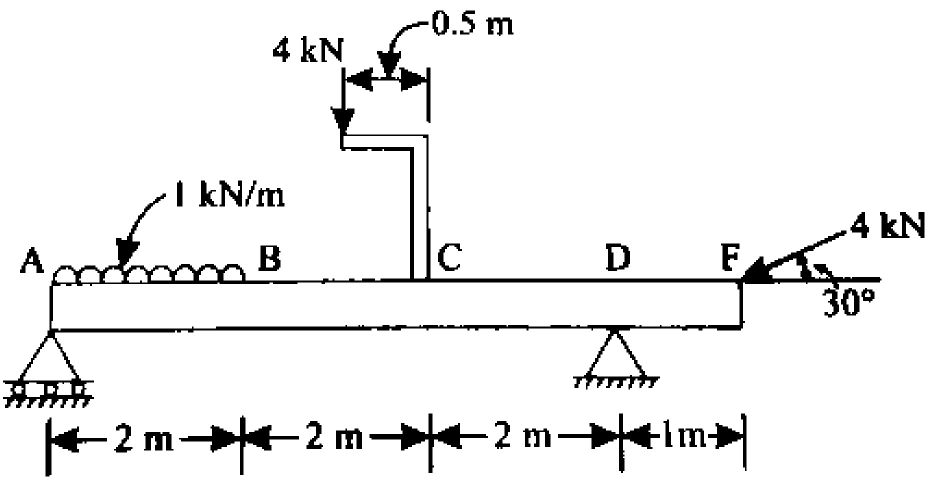


List results

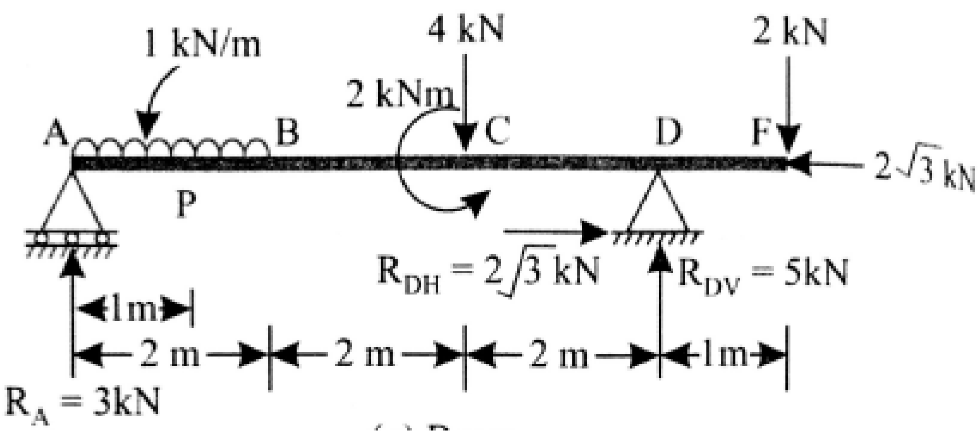
Reaction solution -items to be listed -all items –ok (reaction forces will be displayed with node numbers)

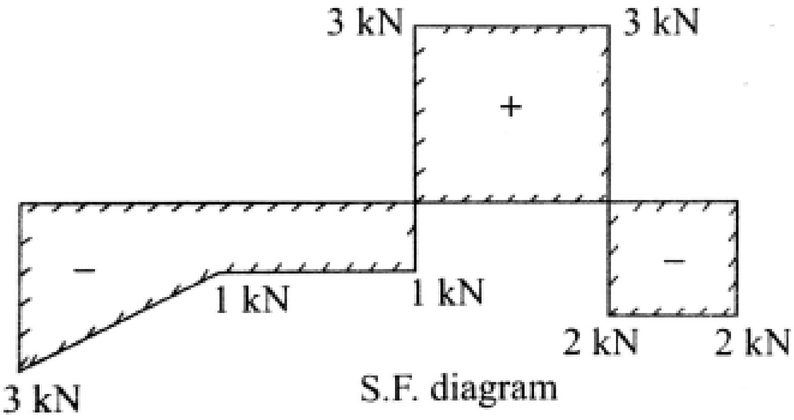
Nodal loads -loads to be listed -all items -ok **(Nodal loads will be displayed)**

Exercise 4:

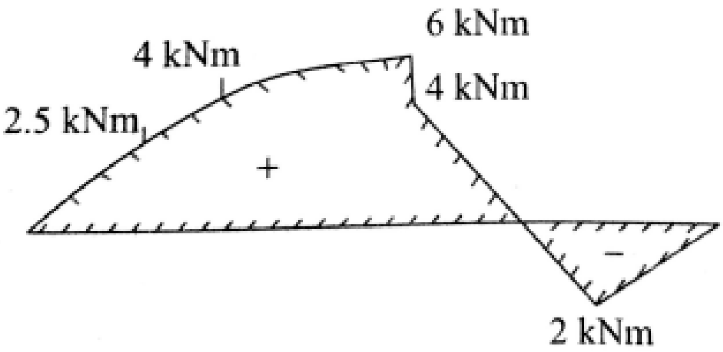


SFD & BMD:

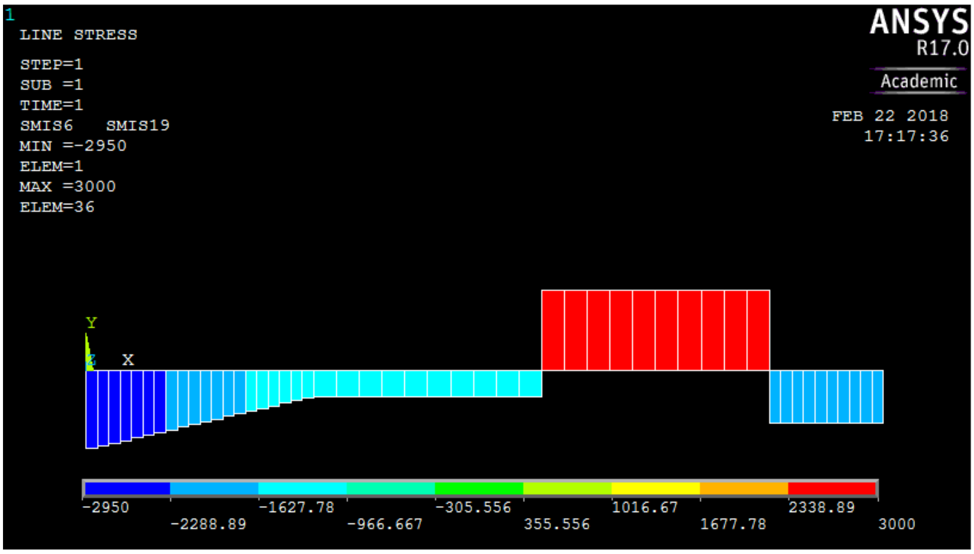


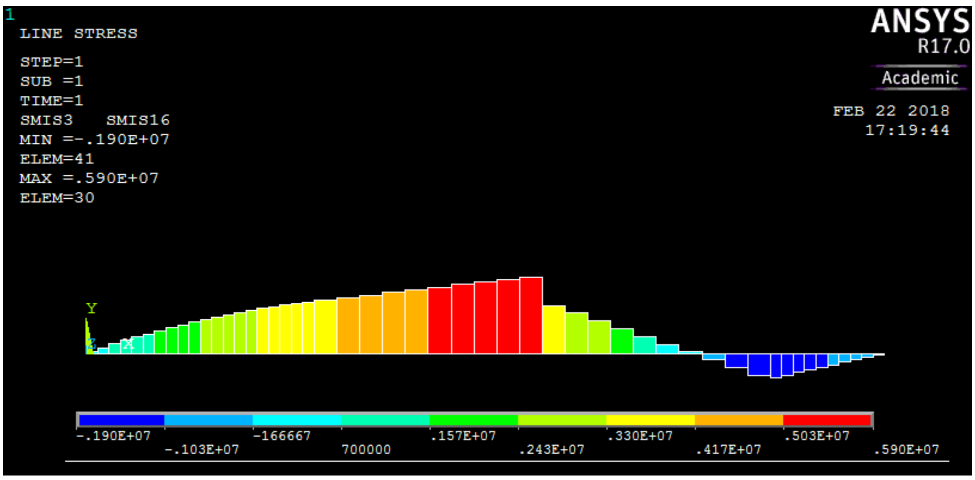


S.F. diagram

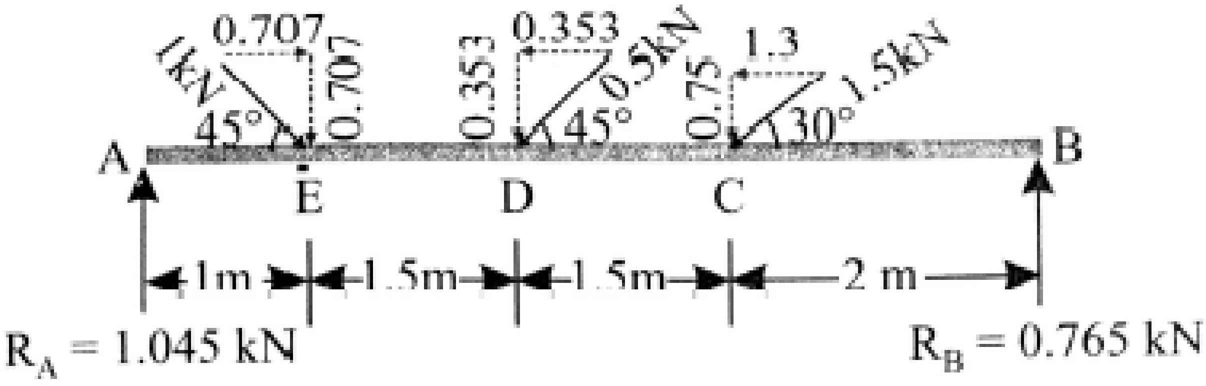


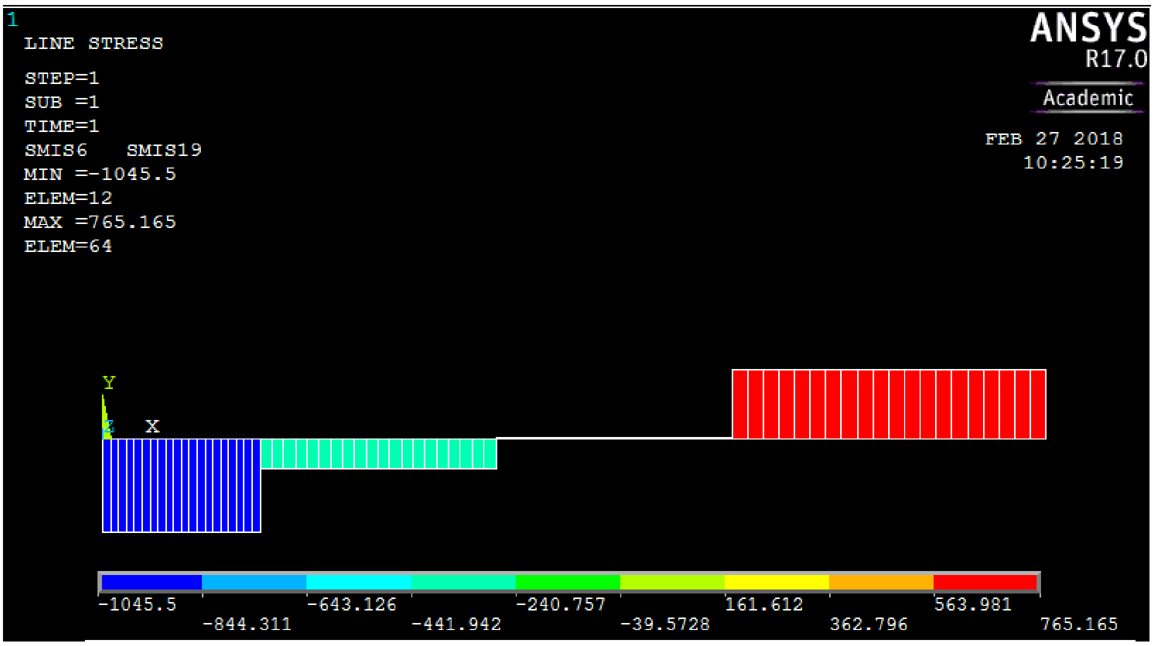
B.M. diagram

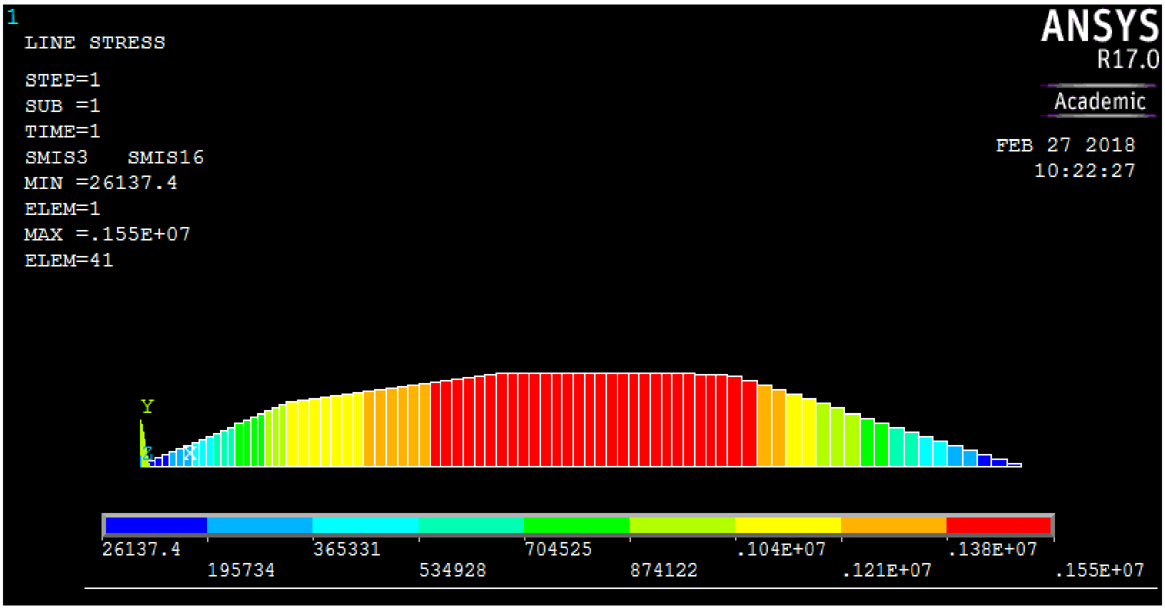




Exercise No. 5:







Step 1 -Ansys Main Menu -Preferences

Select -STRUCTURAL –ok

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- Beam- 2 Node 188- ok -close

Real constants - Beam 188 does not require real constants

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 3 -**Preprocessor**

Sections- Beam - Common sections- offset to - centroid- B - 200, H - 300-ok

Modeling -Create -key points -in active CS -apply (first key point is created)- x,y,z location in CS -**1000** (x value w.r.t first key point)- apply (second key point is created) - **2500** (x value w.r.t first key point) - apply (third key point is created) - **4000** (x value w.r.t first key point)- apply (fourth key point is created)- **6000** (x value w.r.t first key point)- apply (fifth key point is created)- ok

Create - Lines -straight lines through key points - pick 1 & 2-pick 2 & 3 -pick 3 & 4 - pick 4 & 5- ok (lines are created through key points).

Meshing - Size control-manual size -Line - picked lines-pick lines one by one and mention the no. of divisions on each line viz for lines 1, 2, 3 and 4 give no. of divisions as 10 - ok

Mesh - Line - pick all - meshing is done with elements and nodes

Plot Cntrls- Numbering- Node numbering -on **-** Node numbers will be displayed

Plot- Keypoints -Keypoints - key points will be displayed

Plot Cntrls- Numbering- Key point numbering -on **-** Key point numbers will be displayed

Modeling -Create -Nodes-Rotate Node CS- by angles-pick node on key point 2, node to be rotated (default)- about nodal Z-axis- enter 45^0 - apply - pick node on key point 3, node to be rotated (default) - about nodal Z-axis- enter -45^0 - apply- pick node on key point 4, node to be rotated (default)- about nodal Z-axis- enter -60^0 - ok

Step 4 -Preprocessor

Loads –Define Loads - Apply -Structural Displacement -on Key point -pick Key point 1 - apply- DOF to be constrained - Select UX,UY,UZ,ROTX and ROTY-apply- pick Key point 5 - apply- DOF to be constrained - Select UY- ok

Loads - Define Loads -Apply - Force/Moment -on Key point -pick Key point 2 -apply- direction of Force/Moment- FY -Force/Moment value- **-1000 (-value)** -apply- pick Key point 3- apply- direction of Force/Moment- FY -Force/Moment value**- - 500** (-ve value) –apply- pick Key point 4 -apply- direction of Force/Moment- FY -Force/Moment value- **-1500** (-ve 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

Element table -define table -add-results data item- By sequence num- smisc-smisc 3-apply -By sequence num- smisc-smisc 16-apply -By sequence-num- smisc-smisc 6-apply -By sequence num- smisc-smisc 19-apply

Step 7 -General Post Processor

Plot results – Contour Plots - Line element results-element table item at node I -smis 6 -element table item at node J -smis 19- ok **(shear force diagram will be displayed)** Line element - results-element table item at node I -smis 3 -element table item at node J -smis 16- ok (**bending moment diagram will be displayed)**

Note:

For Bending Moment diagram use the combination smisc 3 & smisc 16

And for shear force diagram use the combination smisc 6 & smisc 19

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Mz	SMISC	--	3	16
SFy	SMISC	--	6	19

Finite Element solution of a plane stress problem

In two dimensions, the problems are modeled as plane stress and plane strain.

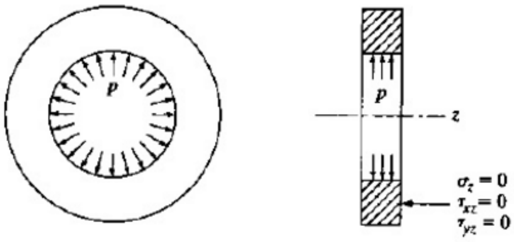
Plane stress: A thin planar body subjected to in-plane loading on its edge surface is said to be in plane stress. In this the Hooke's law relation gives

$\epsilon_x = \frac{\sigma_x}{E} - \vartheta\frac{\sigma_y}{E}$, $\epsilon_y = -\vartheta\frac{\sigma_x}{E} + \frac{\sigma_y}{E}$, $\gamma_{xy} = \frac{2(1+\vartheta)}{E}\tau_{xy}$ and $\epsilon_z = -\frac{\vartheta}{E}(\sigma_x + \sigma_y)$

In matrix form, the inverse relations are given by,

$$\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{Bmatrix} = \frac{E}{(1-\vartheta^2)}\begin{bmatrix}1 & \vartheta & 0\\ \vartheta & 1 & 0\\ 0 & 0 & \frac{1-\vartheta}{2}\end{bmatrix}\begin{Bmatrix}\epsilon_x\\ \epsilon_y\\ \gamma_{xy}\end{Bmatrix} \quad \text{i. e} \quad \sigma = D\,\epsilon$$

and the conditions for plane stress problems are $\sigma_z = 0,\ \tau_{xz} = 0, \tau_{yz} = 0$



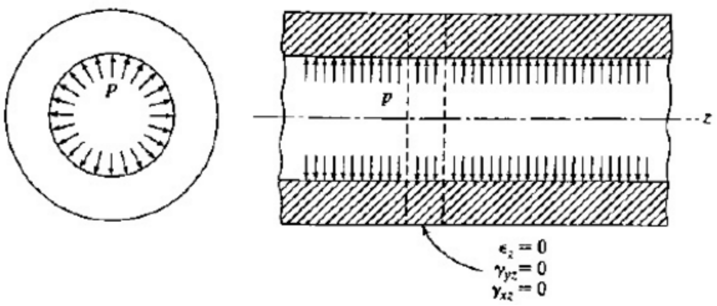
Example: A ring press fitted on a shaft

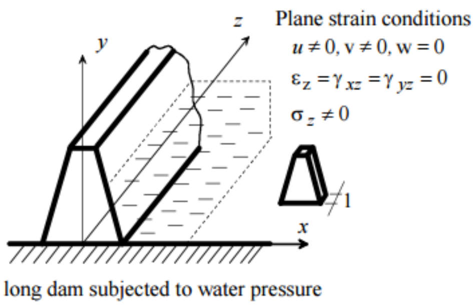
Plane strain: If a long body of uniform c/s is subjected to transverse loading along its length, a small thickness in the loaded area can be treated as subjected to plain strain.

In plane strain problem,

$$\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{Bmatrix} = \frac{E}{(1-\vartheta)(1-2\vartheta)}\begin{bmatrix}1-\vartheta & \vartheta & 0\\ \vartheta & 1-\vartheta & 0\\ 0 & 0 & \frac{1-2\vartheta}{2}\end{bmatrix}\begin{Bmatrix}\epsilon_x\\ \epsilon_y\\ \gamma_{xy}\end{Bmatrix} \quad \text{i. e} \quad \sigma = D\,\epsilon$$

and the conditions for plane strain problems are $\epsilon_z = 0,\ \gamma_{yz} = 0, \gamma_{xz} = 0$ and stress σ_z may not be zero in this case.





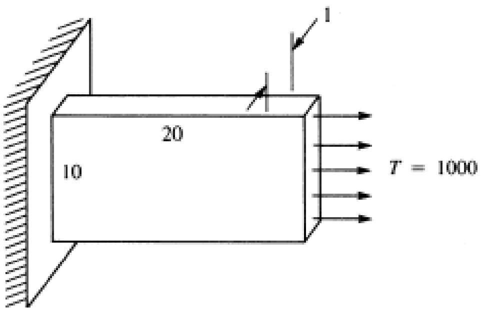
Example: Long dam subjected to water pressure

PROBLEMS ON PLATES

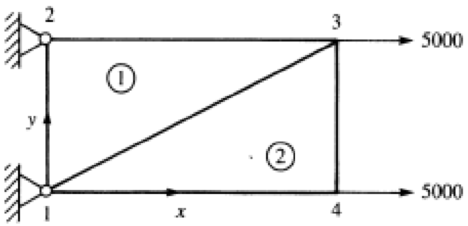
Problem on PLANE STRESS:

Exercise 1:

For a thin plate subjected to the surface traction shown in figure below, determine the nodal displacements and the element stresses. The plate thickness t = 1 unit , E = $30x10^6$ units , and v = 0.30.



Thin plate subjected to tensile stress



Discretized plate

To convert original tensile surface traction to nodal forces, we have,

$$F = \frac{1}{2} T\, A = \frac{1}{2}\, x\, (1000) x (1x10) = 5000\ units$$

The above plate is modeled by using 2-d elements. The plate is discretized into two elements Only. The coarseness of the mesh will not yield as true a predicted behavior of the plate as would a finer mesh particularly near the fixed edge. One can Solve the problem using finer mesh and compare the results

<u>**Solution using ANSYS 17.0**</u>

Preference : Structural

Element type: Select element- **solid -Quad-4 node-182**. In the options of element select **element behavior k_3** and select the option **plane strs w/thk**, since the problem is a plane stress problem.

Real constants: select Add thickness **THK = 1**

Material Properties: Choose Material models -structural- linear-elastic-and enter youngs modulus **EX = 30e+6**, poison's ratio **PRXY = 0.3**

Modelling: **Create Nodes- 4 nodes** using x and y coordinates, for node 1 enter x = 0, y = 0, for node 2 enter x = 0, y = 10, for node 3 enter x = 20 , y = 10 and for node 4 enter x = 20, y = 0.

Create elements through nodes. For element 1, select nodes 1,3 and 2. For element 2, select nodes 1,4 and 3. Even though we have selected 4 noded quad element, elements with 3 nodes are created.

Loads: Apply structural loads

Displacements- on nodes-Selecting the **node numbers 1 and 2 of the first element** apply -choose all dof and then OK.

Force/Moments-on nodes- select node nos. 3 and 4 - apply- **choose FX** and **value = 5000** and OK

Solution:

Analysis type: New Analysis- steady state

Solve-current LS   Solution is obtained.

Genaral post processor: list results-nodal solution to see the displacements at nodes.

list results-element solution to see the element stresses including principal stresses.

Plot results- contour plots- nodal solution

Plot control - Animate-deform results - to see the animation

Answers obtained using FEM:

Displacements:

$$\begin{Bmatrix} d_{3x} \\ d_{3y} \\ d_{4x} \\ d_{4y} \end{Bmatrix} = \begin{Bmatrix} 609.6 \\ 4.2 \\ 663.7 \\ 104.1 \end{Bmatrix} x10^{-6} units$$

Stresses:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} 995 \\ -1.2 \\ -2.4 \end{Bmatrix} units$$

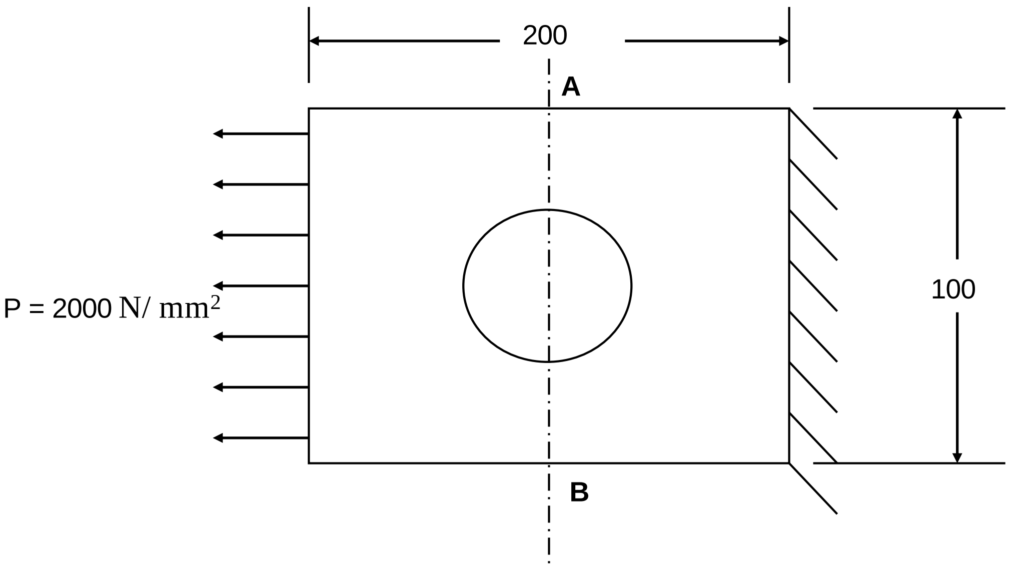
Principal stresses:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \end{Bmatrix} = \begin{Bmatrix} 995 \\ -1.1 \end{Bmatrix} units$$

Note: Solve the problem using four elements and compare the results

Exercise 2:

In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A~B. Take E = 2x10^5 N/ mm^2, t = 10 mm, Poisson's ratio = 0.3, Diameter of circle = 40 mm. Analysis assumption -Plane stress with thickness is used.



***** FOR 2-D and 3-D problems after the geometry has been created meshing is to be done (elements /nodes are created).

Step 1 - ANSYS Main Menu ~Preferences

Select - STRU CTURAL –ok

Step 2 -**Preprocessor**

Element Type -Add/Edit/Delete -Add- Solid-Quad 4node 182- ok option-element behavior k3- plane stress with thickness-ok- close

Real constants -Add -Thickness -10-ok

Material properties – Material Model – Material Model Number 1 –Structural – double click – Linear - double click- Elastic – Isotropic - double click - 2E5 – (If required enter PXY = 0.3) - ok- enter-Close.

Step 3 -Preprocessor

Modeling -Create -Areas- rectangle -by dimensions- X_l, Y_l, X_2, Y_2 = 0, 0, 200,100-ok

-create -areas -circle -solid circle -WP X = 100, WP Y=50 -radius -40- ok

Operate -subtract -areas -pick base area from which to subtract (Rectangle) -ok -pick area to be subtracted (circle)-ok

Mesh Tool- smart size on (select any value, say 4) -mesh -pick area to be meshed -apply- if refining is required at some place -refine -select -box/circle -drag on the area -level-3-ok

Step 4 -Preprocessor

Loads - Define Loads - Apply -Structural Displacement -on nodes -box-drag box right side of area(to pick all nodes)-apply- DOF to be constrained -All DOF-ok.

Loads - Define Loads - Apply -Pressure- on nodes -box-drag box left side of area (to pick all nodes) -apply- Pressure value- -2000(value) –ok-close.

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 - Element solution -stress- Von mises stress -ok (stress distribution diagram will be displayed)

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

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

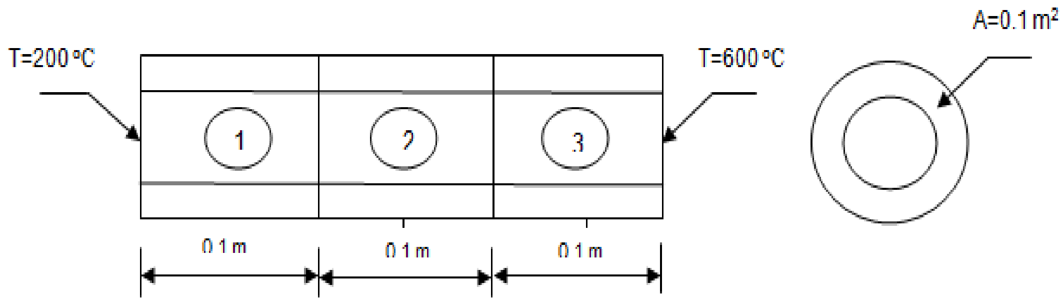
Query results -nodal solution -stress- Von mises stress -ok -select any location on area -value displayed.

List results -nodal solution -DOF solution -All DOF's -ok-values displayed -nodal solution -stress- Von mises stress -ok -values displayed.

Heat Transfer Problems

Exercise 1: One Dimensional Problem with conduction only

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 ^oC, for element 2 K_2=10 W/m ^oC and for element 3 K_3=15 W/m ^oC. The left end has a constant temperature of 200 ^oC and the right end has a constant temperature of 600 ^oC.

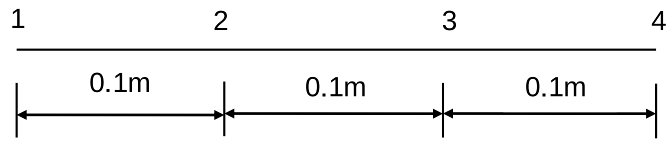


Solution:

Given:

T_1=200 ^oC	T_4=600 ^oC	A= 0.1m^2
K_1= 5 W/ m ^oC	K_2=10 W/ m ^oC	K_3= 15 W/ m ^oC

Finite element model:



Step 1 -Ansys Main Menu -Preferences *Select -THERMAL -ok*

Step 2 -Preprocessor

Element Type -Add/Edit/Delete -Add- LINK – 3D CONDUCTION -33- ok –close

Real Constants - Add/Edit/Delete- Add type 1- link 33 – real constant set no. 1 – cross sectional area – enter – 0.1- ok – close

Material properties – Material models – material model no. 1 - thermal – conductivity –isotropic - conductivity for material - thermal conductivity (K_{xx}) –5 –ok- material – new model – define material 1D – 2 – ok – thermal – conductivity – isotropic – conductivity for material - thermal conductivity (K_{xx}) –10 –ok-material – new model – define material 1D – 3 – ok – thermal – conductivity – isotropic – conductivity for material - thermal conductivity (K_{xx}) –15 –ok-close.

Step 3 -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** - elements attributes- select material no-1- Real const set no-1-ok-Auto numbered-thru nodes -pick 1 & 2-apply -element attributes- select material no-2-Real const set no-1-ok- Auto numbered-thru nodes -pick 2 & 3 –ok - element attributes- select material no-3-Real const set no-1-ok- Auto numbered-thru nodes -pick 3 & 4.

Step 4 -Preprocessor

Loads - Define Loads - Apply – thermal- select temperature -on nodes -pick node 1 -apply- select – TEMP-value – 200- apply – pick node 2 –apply – select – TEMP – value 600 - ok

Step 5 -Ansys Main Menu -Solution

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

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

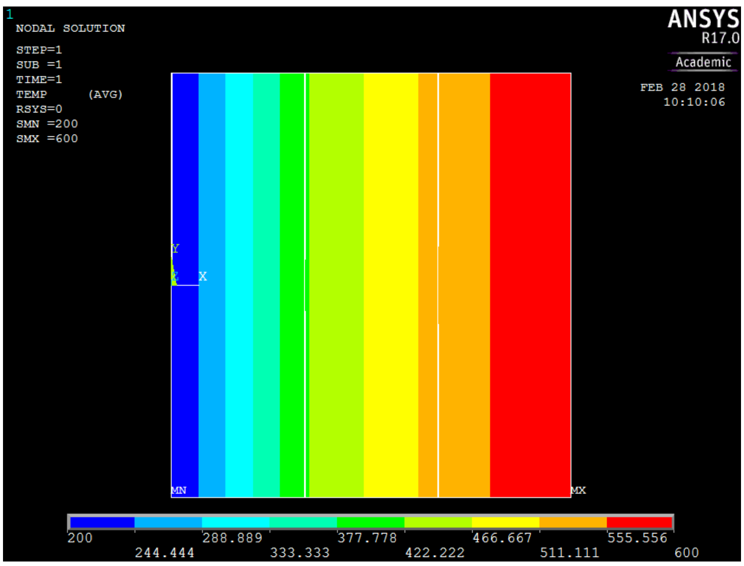
Plot results- Contour Plots – Nodal solution – DOF solution – temp – ok (Temp distribution plot)

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

NODE	TEMP
1	200.00
2	418.18
3	527.27
4	600.00

Plot controls – Style-Size and Shape-display of element-on

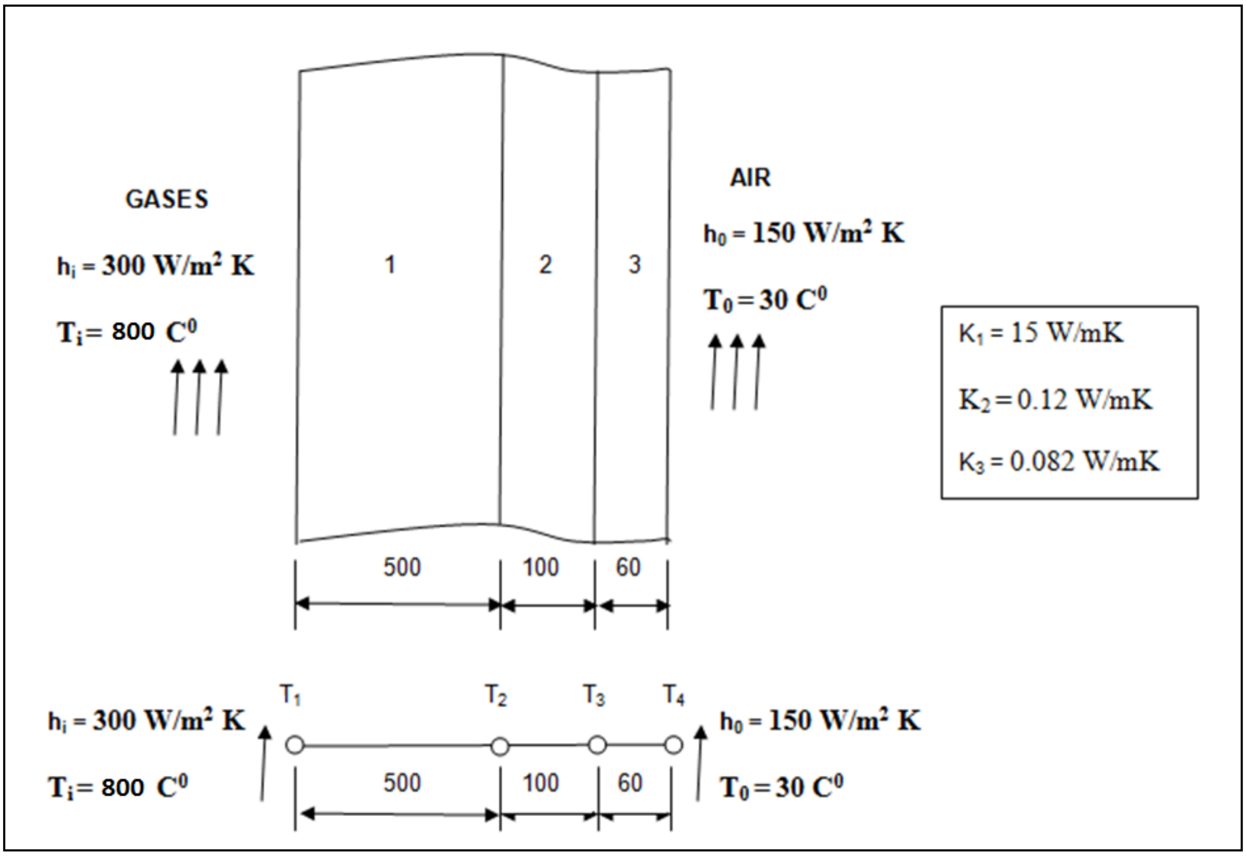
Plot controls -animate – deformed results – DOF solution – Temperature – ok (for animation)



Exercise 2: One Dimensional Problems with conduction and convection

A composite slab consists of one layer of brick 500 mm thick and two layers of insulation. The inner layer of insulation is 100 mm thick and the outer layer is 60mm thick. The thermal conductivities of the brick, inner and the outer layers are 15 W/mK, 0.12W/mK and 0.082 W/mK respectively. The brick side is exposed to gases at 800^{0}C and the outer insulation is exposed to ambient air at 30^{0}C. The brick side and air side heat transfer coefficients are 300 W/m^{2}K and 150 W/m^{2}K respectively. Find the heat transfer through slab and the interface temperatures.

Solution:



FE Model

Step 1 -Ansys Main Menu -Preferences *Select -THERMAL -ok*

Step 2 -Preprocessor

Element type: Select two elements 1.**Link- 3D conduction 33** and 2. **Link -convection 34**

Real constants:

Real constant set No.1, selecting element LInk 33, enter cross sectional area = 1m^2

Real constant set No.2, selecting element LInk 34, enter conventional surface area = 1m^2

Material Properties: Material models :

Material Model No. 1, Thermal, Convection or film coefficient = 300 W/m^2 K

Material Model No. 2, Thermal, Thermal Conductivity = 15 W/m K

Material Model No. 3, Thermal, Thermal Conductivity = 0.12 W/m K

Material Model No. 4, Thermal, Thermal Conductivity = 0.082 W/m K

Material Model No. 5, Thermal, Convection or film coefficient = 150 W/m^2 K

Modelling:

Create 6 nodes taking lengts in m and at locations x=0, x=0.01, x=0.51, x=0.61, x=0.67 and x=0.68

Important: Node Nos. 1 and 6 are created to have convention BCs on both sides of composite slab choosing very small lengths between nodes 1 & 2 , 5 & 6 respectively.

Create elements through nodes choosing attributes like Material Model No. and Rreal Constant set No.

Loads:Apply thermal loads

Selecting the **node No.1 of the first element** apply temperature load = 800 ^{0}C and Selecting the **node No.6 of the last element** apply temperature load = 30 ^{0} C.

Solution:

Analysis type: New Analysis- steady state, Solve-current LS Solution is obtained.

Genaral post processor: list results-nodal solution to see the temperatures at nodes.

Plot results- contour plots- nodal solution

Plot control - Animate-deform results - to see the animation