



DEPARTMENT OF MECHANICAL ENGINEERING

LABORATORY MANUAL

ME8711 -SIMULATION AND ANALYSIS LABORATORY

YEAR/SEMESTER : IV/07

STUDY OF

BASICS IN ANSYS

ExNo : 01

Date :

Aim:

To study about the basic procedure to perform the analysis in ANSYS.

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.

Build the model:

1. Defining the Jobname:

The jobname is a name that identifies the ANSYS job. When you define a jobname for an analysis, the jobname becomes the first part of the name of all files the analysis creates. (The extension or suffix for these files' names is a file identifier such as .DB.) By using a jobname for each analysis, you ensure that no files are overwritten.

2. Defining an Analysis Title:

The TITLE command (Utility Menu > File > Change Title), defines a title for the analysis. ANSYS includes the title on all graphics displays and on the solution output. You can issue the /STITLE command to add subtitles; these will appear in the output, but not in graphics displays.

3. Defining Units:

The ANSYS program does not assume a system of units for your analysis. Except in magnetic field analyses, you can use any system of units so long as you make sure that you use that system for all the data you enter. (Units must be consistent for all input data.)

4. Defining Element Types:

The ANSYS element library contains more than 150 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID96, etc. The following element categories are available:

BEAM	MESH
CIRCUIT	Multi-Point Constraint
COMBINATION	PIPE

CONTACT	PLANE
FLUID	PRETS (Pretension)
HF(High Frequency)	SHELL
HYPERELASTIC	SOLID
INFINITE	SOURCE
INTERFACE	SURFACE
LINK	TARGET
MASS	TRANSDUCER
MATRIX	USER
	VISCOELASTIC (or viscoplastic)

The element type determines, among other things:

- ✓ The degree-of-freedom set (which in turn implies the discipline - structural, thermal, magnetic, electric, quadrilateral, brick, etc.)
- ✓ Whether the element lies in 2-D or 3-D space.

5. Defining Element RealConstants:

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

6. Defining MaterialProperties:

Most element types require material properties. Depending on the application, material properties can be linear (see Linear Material Properties) or nonlinear (see Nonlinear Material Properties).

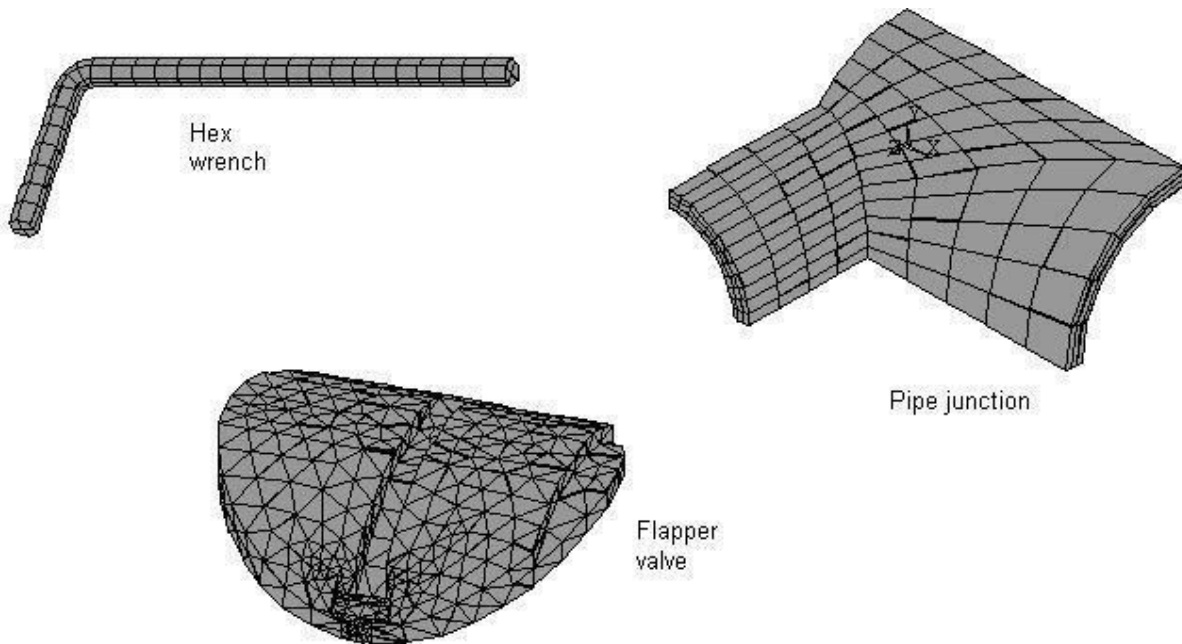
As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the material table. Within one analysis, you may have multiple material property sets (to correspond with multiple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

7. Creating the ModelGeometry:

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

There are two methods to create the finite element model: solid modeling and direct generation. With solid modeling, you describe the geometric shape of your model, then instruct the ANSYS program to automatically mesh the geometry with nodes and elements. You can control the size and shape in the elements that the program creates. With direct generation, you "manually" define the location of each node and the connectivity of each element. Several convenience

operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc. are available.



Sample Finite Element Models

Apply Loads and Obtain the Solution:

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7preprocessor.

1. Defining the Analysis Type and AnalysisOptions:

You choose the analysis type based on the loading conditions and the response you wish to calculate. For example, if natural frequencies and mode shapes are to be calculated, you would choose a modal analysis. You can perform the following analysis types in the ANSYS program: static (or steady-state), transient, harmonic, modal, spectrum, buckling, and substructuring.

Not all analysis types are valid for all disciplines. Modal analysis, for example, is not valid for a thermal model. The analysis guide manuals in the ANSYS documentation set describe the analysis types available for each discipline and the procedures to do those analyses.

Analysis options allow you to customize the analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton-Raphson options.

2. Applying Loads:

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

- ✓ DOF Constraints
- ✓ Forces
- ✓ Surface Loads
- ✓ Body Loads
- ✓ Inertia Loads
- ✓ Coupled-field Loads

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

3. Specifying Load Step Options:

Load step options are options that you can change from load step to load step, such as number of substeps, time at the end of a load step, and output controls. Depending on the type of analysis you are doing, load step options may or may not be required. The analysis procedures in the analysis guide manuals describe the appropriate load step options as necessary.

4. Initiating the Solution:

To initiate solution calculations, use either of the following:

Command(s): SOLVE

GUI: Main Menu> Solution> Solve> Current LS
Main Menu> Solution> solution_method

When you issue this command, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file (Jobname.RST, Jobname.RTH, Jobname.RMG, or Jobname.RFL) and also to the database. The only difference is that only one set of results can reside in the database at one time, while you can write all sets of results (for all substeps) to the resultsfile.

Review the Results:

Once the solution has been calculated, you can use the ANSYS postprocessors to review the results. Two postprocessors are available: POST1 and POST26.

You use POST1, the general postprocessor, to review results at one substep (time step) over the entire model or selected portion of the model. The command to enter POST1 is /POST1 (Main Menu> General Postproc), valid only at the Begin level. You can obtain contour displays, deformed shapes, and tabular listings to review and interpret the results of the analysis. POST1 offers many other capabilities, including error estimation, load case combinations, calculations among results data, and pathoperations.

You use POST26, the time history postprocessor, to review results at specific points in the model over all time steps. The command to enter POST26 is /POST26 (Main Menu> TimeHist Postpro), valid only at the Begin level. You can obtain graph plots of results data versus time (or frequency) and tabular listings. Other POST26 capabilities include arithmetic calculations and

Result:

Thus the basic steps to perform the analysis in ANSYS like

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

STRESS ANALYSIS OF A PLATE WITH CIRCULAR HOLE

ExNo : 02

Date :

AIM

To conduct the stress analysis in a plate with a circular hole using ANSYS software.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operating system:	Window XP Service Pack 3
Software:	ANSYS(Version12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference - Structural- h-Method -Ok.
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, 8 node 82 – Ok –Option – Choose Plane stress w/thk - Close.
3. Real constants - Add/Edit/Delete – Add – Ok – THK 0.5 – Ok -Close.
4. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX2e5, PRXY 0.3 -Ok.
5. Modeling – Create – Areas – Rectangle - by 2 corner - X=0, Y=0, Width=100, Height=50 - Ok. Circle - Solid circle - X=50, Y=25, Radius=10 - Ok. Operate – Booleans – Subtract – Areas - Select the larger area (rectangle) – Ok – Ok -Select Circle – Next –Ok -Ok.
6. Meshing - Mesh Tool – Area – Set - Select the object – Ok - Element edge length2/3/4/5 – Ok - Mesh Tool -Select TRI or QUAD - Free/Mapped – Mesh - Select the object - Ok

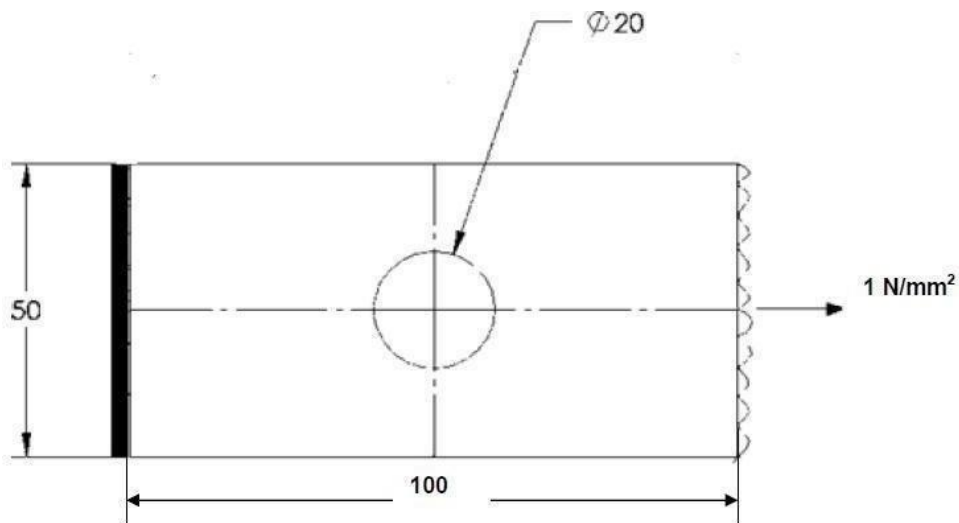
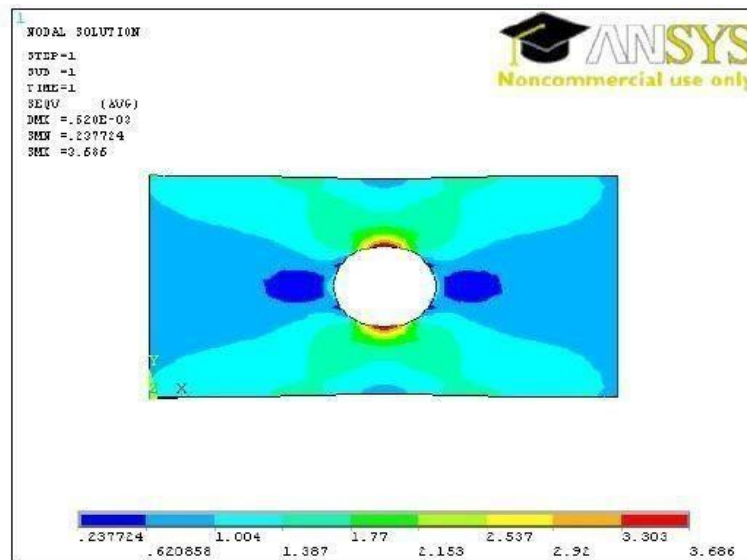
SOLUTION

7. Solution – Define Loads – Apply – Structural – Displacement - On lines -Select the boundary where is going to be arrested – Ok - All DOF -Ok.
Pressure - On lines - Select the load applying area – Ok - Load PRES valve = 1 N/mm²-Ok.

8. Solve – Current LS – Ok – Solution is done –Close.

Young's Modulus = 200 GPa

Poisson's Ratio = 0.3



RESULT

Thus the stress analysis of rectangular plate with a circular hole is done by using the ANSYS Software.

STRESS ANALYSIS OF RECTANGULAR L BRACKET

Ex No: 03

Date:

AIM

To conduct the stress analysis of a rectangular L section bracket using ANSYS software

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operating system:	Window XP Service Pack 3
Software:	ANSYS (Version 12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference - Structural- h-Method -Ok.
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, 8 node 82 – Ok –Option – Choose Planestress w/thk - Close.
3. Real constants - Add/Edit/Delete – Add – Ok – THK 0.5 – Ok -Close.
4. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX2e5, PRXY 0.3 -Ok.
5. Modeling – Create – Key points - In active CS – enter the key point number and X, Y, Z location for 6 key points to form the rectangular L-bracket. Lines – lines - Straight line - Connect all key points to form as lines. Areas – Arbitrary - by lines - Select all lines -ok. Lines - Line fillet - Select the two lines where the fillet is going to be formed – Ok – enter the Fillet radius=10- Ok. Areas – Arbitrary - through KPs - Select the key points of the fillet - Ok. Operate – Booleans – Add – Areas - Select the areas to be add (L Shape & fillet area) - ok. Create – Areas – Circle - Solid circle - Enter the co-ordinates, radius of the circles at the two ends(semicircles) -Ok. Operate – Booleans – Add – Areas - Select the areas to be add (L Shape & two circles) - Ok. Create – Areas – Circle - Solid circle – Enter the coordinates, radius of the two circles which are mentioned as holes -Ok. Operate –

Booleans – Subtract – Areas - Select the area of rectangle – Ok - Select the two circles - Ok.

5. Meshing - Mesh Tool – Area – Set - Select the object – Ok - Element edge length 2/3/4/5 – Ok - Mesh Tool -Select TRI or QUAD - Free/Mapped – Mesh - Select the object -Ok.

SOLUTION

6. Solution – Define Loads – Apply – Structural – Displacement - On lines - Select the boundary where is going to be arrested – Ok - All DOF - Ok. Pressure - On lines - Select the load applying area – Ok - Load PRES valve = -10000 N (- Sign indicates the direction of the force i.e. downwards) –Ok.

8. Solve – Current LS – Ok – Solution is done –Close.

POST PROCESSING

9. General post proc - Plot Result - Contour plot - Nodal Solution – Stress - Vonmises stress -Ok.

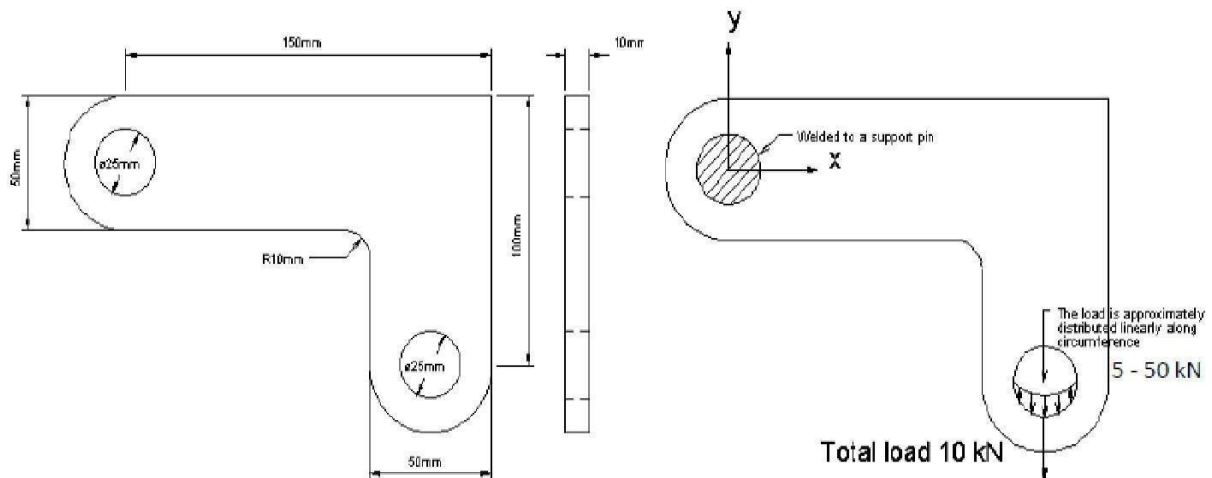
TO VIEW THE ANIMATION

10. Plot control – Animates - Mode Shape – Stress - Von mises -Ok.

11. Plot control – Animate - Save Animation - Select the proper location to save the file(E drive-user) -Ok.

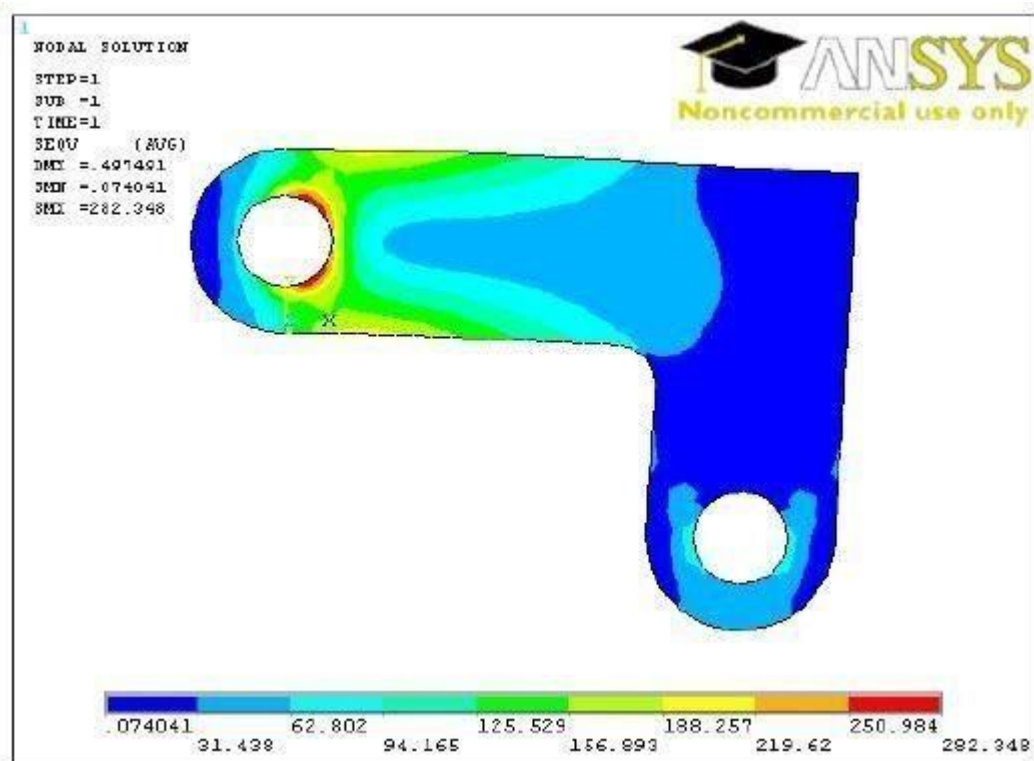
FOR REPORT GENERATION

12. File – Report Generator – Choose Append – OK – Image Capture – Ok -Close.



Young's Modulus = 200 GPa

Poisson's Ratio = 0.3



RESULT

Thus the stress analysis of rectangular L section bracket is done by using the ANSYS Software.

STRESS ANALYSIS OF BEAM

Ex No : 04

Date :

AIM

To conduct the stress analysis in a beam using ANSYS software.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operating system:	Window XP Service Pack 3
Software:	ANSYS (Version 12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference - Structural- h-Method -Ok.
2. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – Ok – Options – Ok -Close.
3. Sections – beam – Common sections – Select the correct section of the beam and input the of “w1, w2,w3” and “t1, t2, t3” – Preview – Note down the values of area,Iyy.
4. Real constants - Add/Edit/Delete – Add – Ok – Enter the values of area=5500, Izz=0.133e8, height=3 – Ok-Close.
5. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX2e5, PRXY 0.3 -Ok.
6. Modeling–Create–Keypoints–InactiveCS–EnterthevaluesofCSofeachkeypoints – Apply – Ok. Lines – Lines – Straight line – Pick the all points – Ok.
7. Meshing – Mesh attributes – All lines – Ok. Meshing – Size cntrls – Manual size – Lines–Alllines–Enterthevalueofelementedglength[or]Numberofelement

divisions – Ok. Mesh tool – Mesh – Pick all.

8. Meshing – Mesh attributes – All lines – Ok. Meshing – Size ctrls – Manual size – Lines – All lines – Enter the value of element edge length [or] Number of element divisions – Ok. Mesh tool – Mesh – Pickall.

SOLUTION

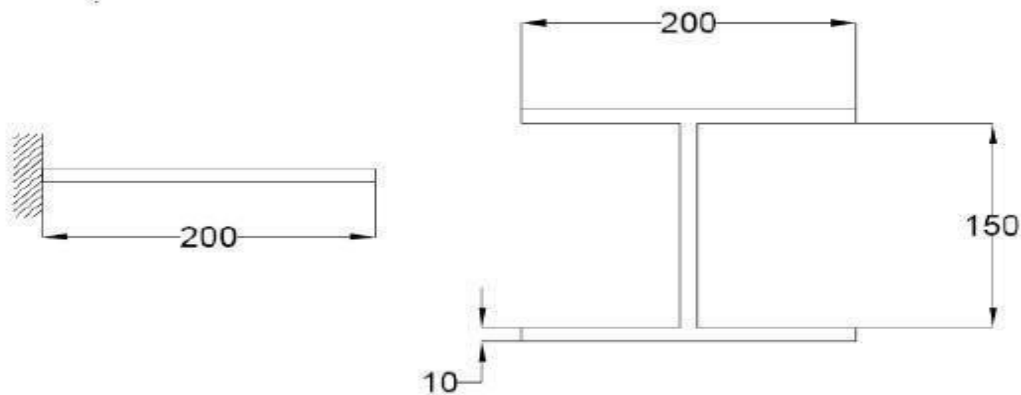
9. Solution – Define Loads – Apply – Structural – Displacement - On key points – Select the 1st key point – ALL DOF – Ok. On key points – select the 2nd key point– UY – Ok. Force/Moment – On key points – Select the key point – Ok – direction of force/moment FY, Value = -1,000 (- sign indicates the direction of the force) –Ok.
10. Solve – Current LS – Ok – Solution is done –Close.

POST PROCESSING

11. . General post proc – Element table – Define table – Add – By sequence num – SMISC,6 – Ok – SMISC,12 –Ok – LS,2 – Ok – LS,3 - Ok – Close. Plot results – Contour plot – Nodal solution – DOF solution – Y component of displacement – Ok. Contour plot – Line element Res – Node I SMIS 6, Node J SMIS 12 – Ok. Contourplot – Line element Res – Node I LS 2, Node J LS 3 – Ok

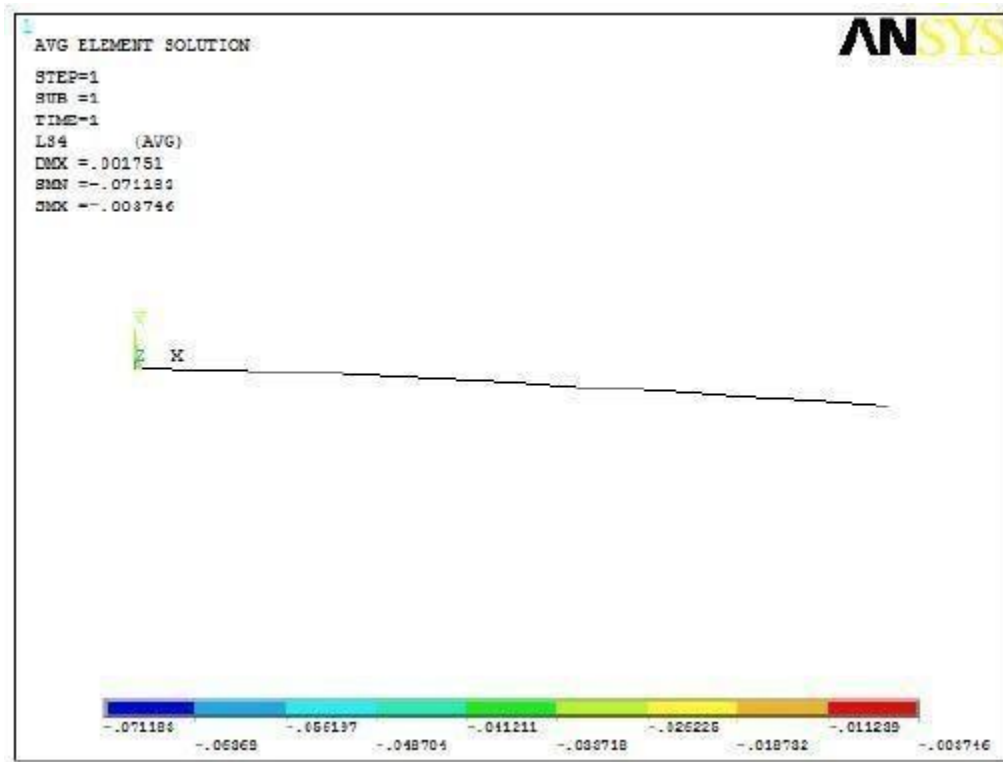
FOR REPORT GENERATION

12. . File – Report Generator – Choose Append – OK – Image Capture – Ok -Close.



All Dimensions are in "mm"

Young's Modulus = 200 GPa
Poisson's Ratio = 0.3



RESULT

Thus the stress analysis of a BEAM is done by using the ANSYS Software

MODE FREQUENCY ANALYSIS OF BEAM

Ex No : 05

Date :

AIM

To conduct the Mode frequency analysis of beam using ANSYS software.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operating system:	Window XP Service Pack 3
Software:	ANSYS (Version12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – Ok–Close.
2. Real constants - Add/Edit/Delete – Add – Ok – Area 0.1e-3, Izz 0.833e-9, Height 0.01–Ok –Close.
3. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX206e9, PRXY 0.25 – Ok –Density – DENS 7830 –Ok.
4. Modeling –Create–Keypoints–InactiveCS–Enterthecoordinatevalues-Ok.Lines -lines – Straight Line – Join the two key points – Ok.
5. Meshing – Size Cntrls – manual size – lines – all lines – Enter the value of no ofelement divisions 25 – Ok. Mesh – Lines – Select the line –Ok.

SOLUTION

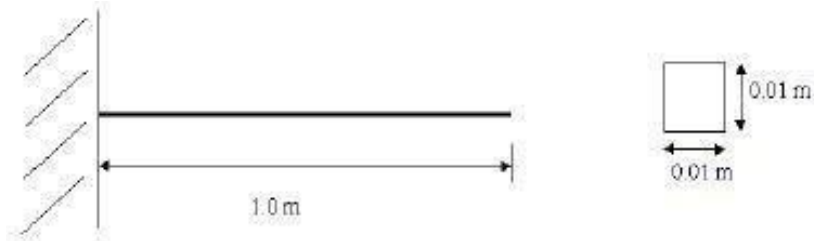
6. Solution – Define Loads – Apply – Structural – Displacement - On nodes – Select the node point –Ok – All DOF – Ok. Analysis type – New analysis – Modal – Ok. Analysis type – Analysis options – Block Lanczos – enter the value no of modes to extract as 3 or 4 or 5 – Ok – End Frequency 10000 –Ok.
7. Solve – Current LS – Ok – Solution is done –Close.

POST PROCESSING

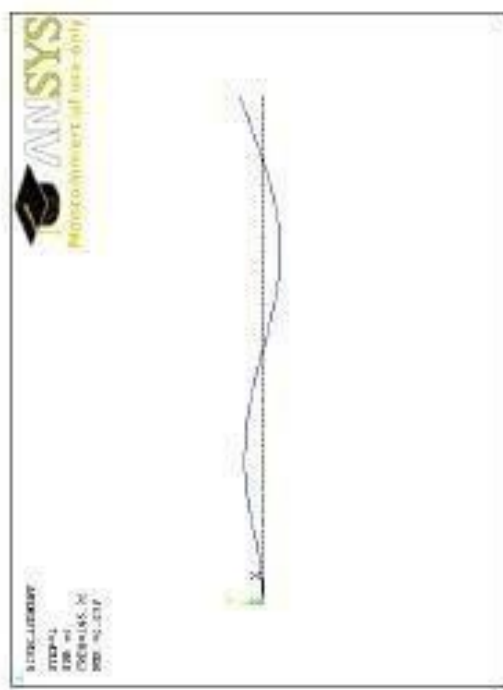
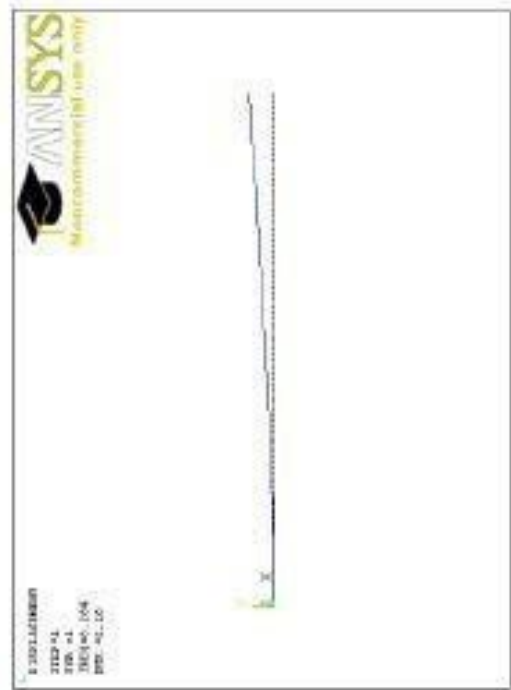
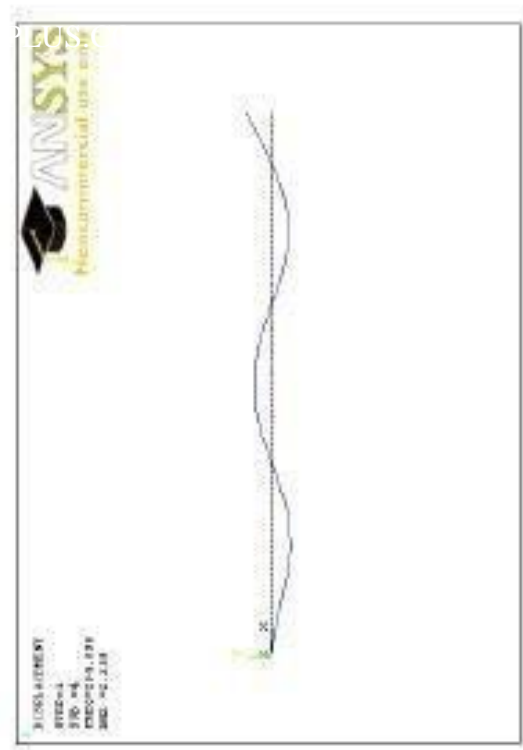
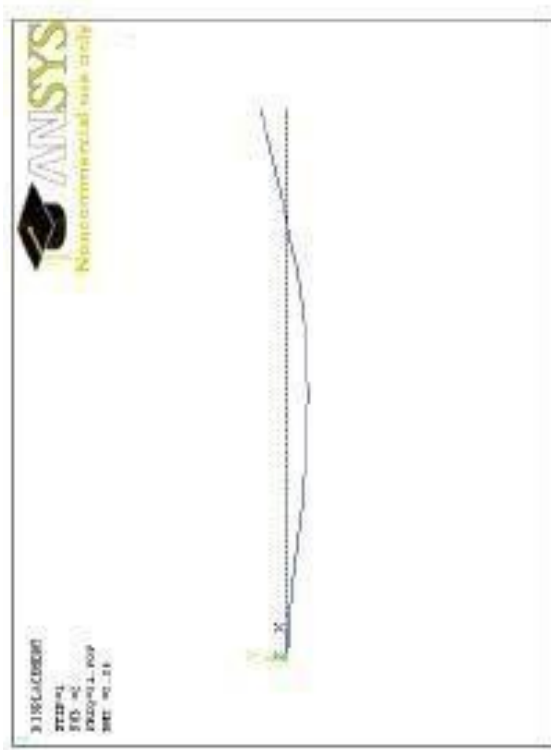
8. General post proc – Read results – First set - Plot results – Deformed shape – Choose Def+undeformed – Ok. Read results – Next set - Plot results – Deformed shape – Choose Def+undeformed – Ok and so on.

FOR REPORT GENERATION

9. File – Report Generator – Choose Append – OK – Image Capture – Ok - Close. (Capture all images)



Young's modulus	=	206 GPa
Poisson's Ratio	=	0.25
Weight Density	=	$7.83 \times 10^3 \text{ kg/m}^3$



RESULT

Thus the mode frequency analysis of a beam is done by using the ANSYS Software.

HARMONIC ANALYSIS OF A 2D COMPONENT

Ex No : 06

Date :

AIM

To conduct the harmonic analysis of a 2D component by using ANSYS software.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operatingsystem:	Window XP Service Pack 3
Software:	ANSYS(Version12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – Ok –Close.
2. Real constants - Add/Edit/Delete – Add – Ok – Area 0.1e-3, Izz 0.833e-9, Height 0.01 –Ok – Close.
3. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX206e9, PRXY 0.25 – Ok –Density – DENS 7830 –Ok.
4. Modeling – Create – Key points – Inactive CS – Enter the coordinate values - Ok. Lines– lines – Straight Line – Join the two key points –Ok.
5. Meshing – Size Cntrls – manual size – lines – all lines – Enter the value of no ofelement divisions 25 – Ok.Mesh – Lines – Select the line –Ok.

SOLUTION

6. Solution - Analysis type – New analysis – Harmonic – Ok. Analysis type – Analysis options – Full, Real+ imaginary – Ok– Use the default settings –Ok
7. Solution – Define Loads – Apply – Structural – Displacement - On nodes – Select the node point –Ok – All DOF – Ok. Force/Moment – On Nodes – select the node 2 – Ok – Direction of force/mom FY, Real part of force/mom -100 – Ok. Load step Opts–

Time/Frequency – Freq and Substps – Enter the values of Harmonic freq range 1-100, Number of sub steps 100, Stepped – Ok.

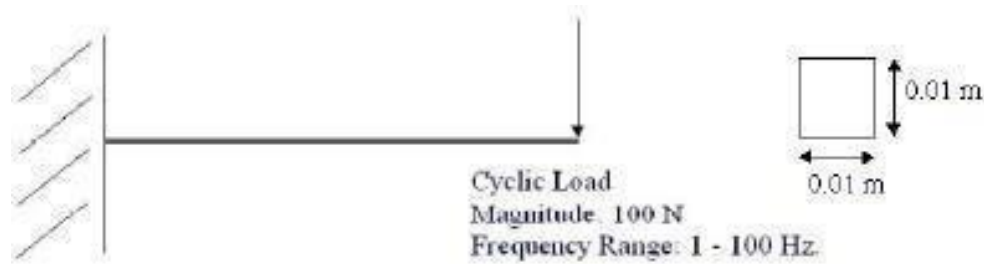
8. Solve – Current LS – Ok – Solution is done –Close.

POST PROCESSING

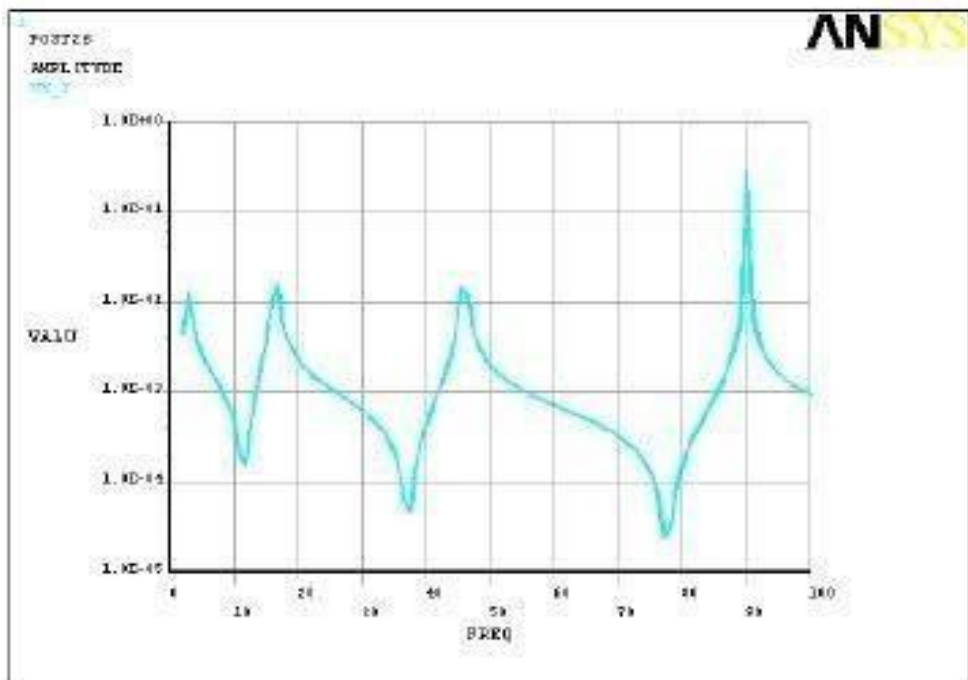
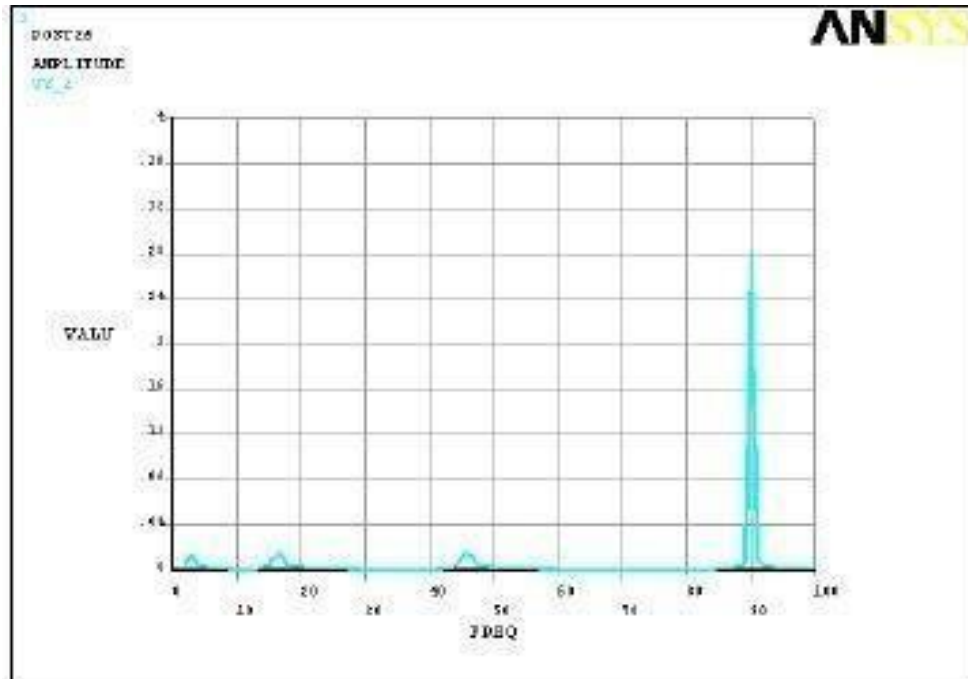
10. TimeHist postpro – Variable Viewer – Click “Add” icon – Nodal Solution – DOF Solution – Y-Component of displacement – Ok – Enter 2 – Ok. Click “List data” icon and view the amplitude list. Click “Graph” icon and view the graph. To get a better view of the response, view the log scale of UY. Plotctrls – Style – Graphs – Modify axes – Select Y axis scale as Logarithmic – Ok. Plot – Replot – Now we can see the better view.

FOR REPORT GENERATION

11. File – Report Generator – Choose Append – OK – Image Capture – Ok - Close.
(Capture allimages)



Young's modulus	=	206 GPa
Poisson's Ratio	=	0.25
Weight Density	=	$7.83 \times 10^3 \text{ kg/m}^3$
Length of the Beam	=	1 m



RESULT

Thus the harmonic analysis of 2D component is done by using the ANSYS Software

STRESS ANALYSIS OF AN AXI-SYMMETRIC COMPONENT

EX.NO:7

Date:

Aim:

To obtain the stress distribution of an axisymmetric component. The model will be that of a closed tube made from steel. Point loads will be applied at the centre of the top and bottom plate.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operating system:	Window XP Service Pack 3
Software:	ANSYS(Version12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

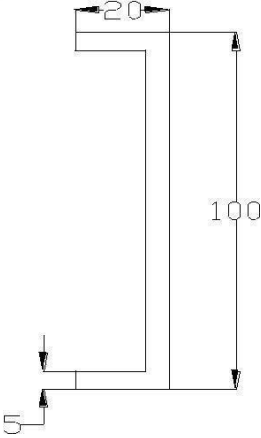
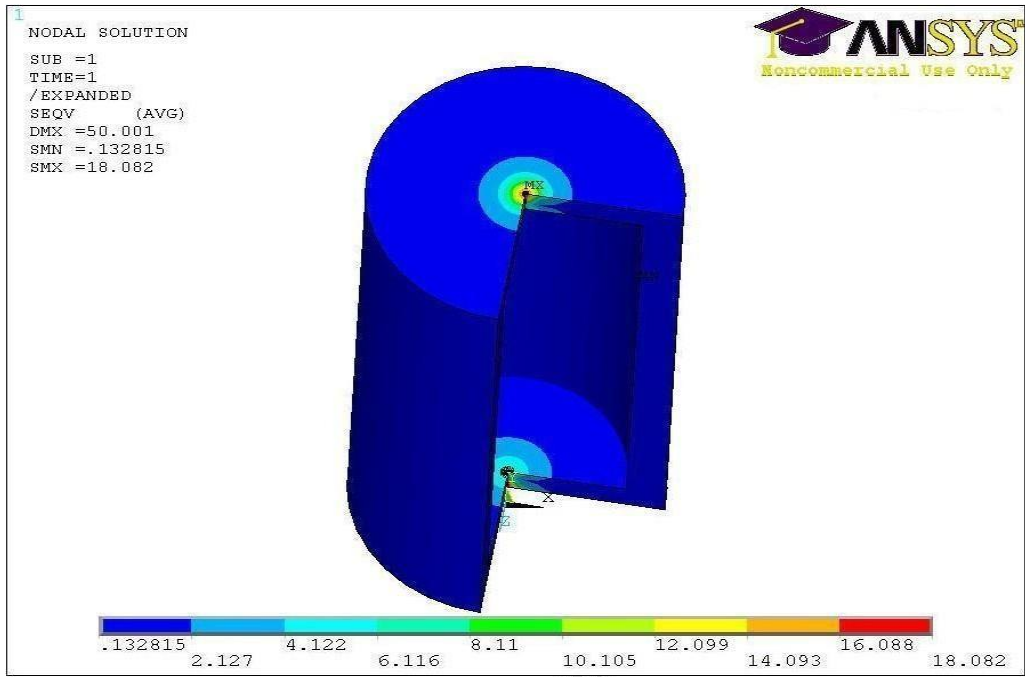
Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Utility Menu > Change Job Name > Enter Job Name. Utility Menu >File > Change Title > Enter NewTitle.
2. Preference > Structural >OK.
3. Preprocessor > Element type > Add/Edit/ delete > solid 8node 183 >options> axisymmetric.
4. Preprocessor > Material Properties > Material Model > Structural > Linear > Elastic> Isotropic > EX = 2E5, PRXY =0.3.
5. Preprocessor>Modeling>create>Areas>Rectangle> Bydimensions

Rectangle	X1	X2	Y1	Y2
1	0	20	0	5
2	15	20	0	100
3	0	20	95	100

6. Preprocessor > Modeling > operate > Booleans > Add > Areas > pick all >Ok.
7. Preprocessor > meshing > mesh tool > size control > Areas > Element edge length =2 mm > Ok > mesh > Areas > free> pickall.
8. Solution > Analysis Type>NewAnalysis>Static
9. Solution > Define loads > Apply .Structural > displacement > symmetry B.C > onlines. (Pick the two edger on the left at X =0)
10. Utility menu > select > Entities > selectall
11. Utility menu > select > Entities > by location > Y = 50>ok.
 - i. (Select nodes and by location in the scroll down menus. Click Y coordinates and type 50 in to the inputbox.)
12. Solution > Define loads > Apply > Structural > Force/Moment > on keypoints
 - a. FY > 100 > Pick the top left corner of the area > Ok.
13. Solution > Define Loads > apply > Structural > Force/moment > on key points > FY >- 100 > Pick the bottom left corner of the area >ok.
14. Solution > Solve > Current LS
15. Utility Menu > select >Entities
16. Select nodes >by location > Y coordinates and type 45, 55 in the min., max. box,as shown below and clickok.
17. General postprocessor > List results > Nodal solution > stress > componentsSCOMP.
18. Utilitymenu>plotcontrols>style>Symmetryexpansion>2DAxisymmetric>³/₄ expansion



Result:
 Thus the stress distribution of the axi symmetric component is studied

THERMAL STRESS ANALYSIS OF A 2D COMPONENT

Ex No: 08

Date:

AIM

To conduct the thermal c analysis of a 2D component by using ANSYS software.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operatingsystem:	Window XP Service Pack 3
Software:	ANSYS(Version12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Thermal - h-Method -Ok.
2. Preprocessor-Elementtype-Add/Edit/Delete–Add–Solid,Quad4node42–Ok
– Options – plane strsw/thk – Ok – Close.
3. Real constants - Add/Edit/Delete – Add – Ok – THK 100 – Ok –Close.
- 4 . Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 2e5,
PRXY 0.3 – Ok –Thermal expansion – Secant coefficient – Isotropic – ALPX 12e-6 – Ok.
4. Modeling – Create – Areas - Rectangle – by 2 corners – Enter thecoordinate
values, height, width -Ok.
5. Meshing – Mesh tool – Areas, set – select the object – Ok – Element edge length 10 - Ok –
Mesh tool- Tri, free - mesh – Select the object–Ok.

SOLUTION

7. Solution – Define Loads – Apply – Structural – Displacement - On lines – Select the boundary on the object –Ok – Temperature – Uniform Temp – Enter the temp. Value 50 –Ok.

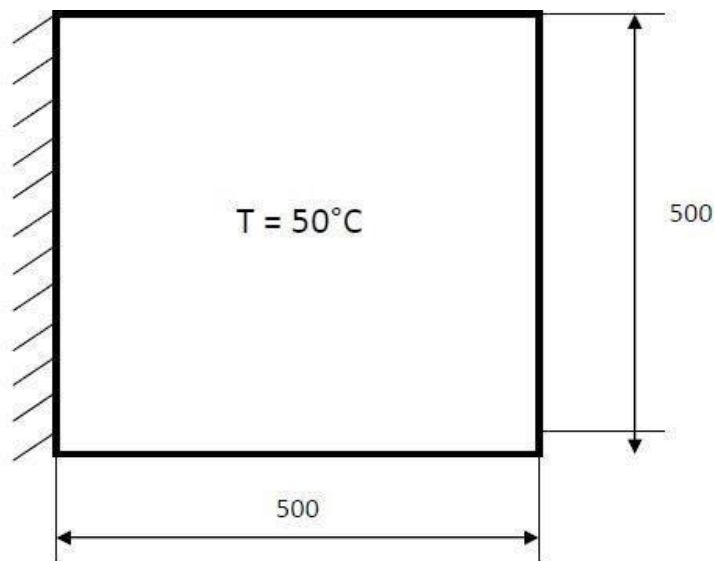
8. Solve – Current LS – Ok – Solution is done –Close.

POST PROCESSING

9. General post proc – Plot results – Contour plot – Nodal solution – Stress – 1st principal stress – Ok – Nodal solution – DOF Solution – Displacement vector sum -Ok.

FOR REPORT GENERATION

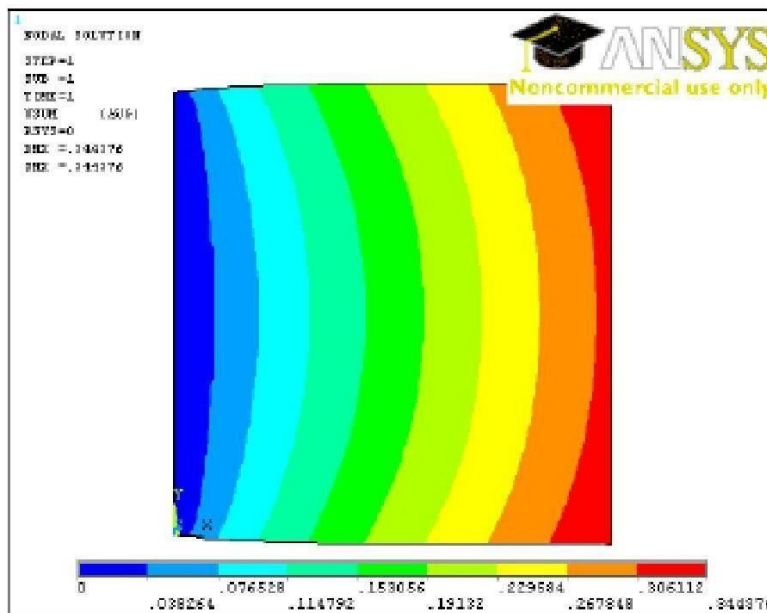
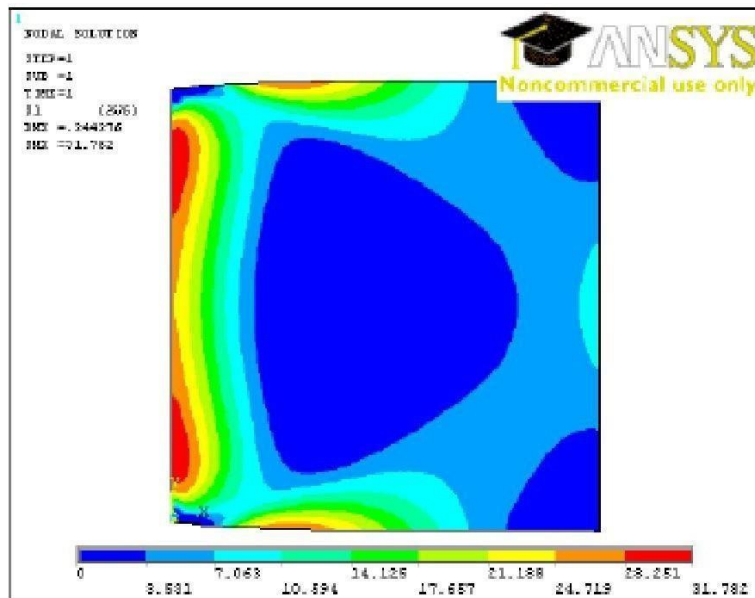
10. File – Report Generator – Choose Append – OK – Image Capture – Ok –Close



Young's modulus = 200 GPa

Poisson's Ratio = 0.3

Thermal expansion coefficient = 12×10^{-6}



RESULT

Thus the thermal stress analysis of a 2D component is done by using the ANSYS Software.

CONDUCTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

Ex No: 09

Date :

AIM

To conduct the conductive heat transfer analysis of a 2D component by using ANSYS software.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operating system:	Window XP Service Pack 3
Software:	ANSYS (Version 12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – Thermal - h-Method -Ok.
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 55 – Ok – Close – Options – plane thickness –Ok.
3. Real constants - Add/Edit/Delete – Add – Ok – THK 0.5 – Ok –Close.
4. Material props - Material Models –Thermal – Conductivity – Isotropic – KXX 10 –Ok.
5. Modeling – Create – Areas - Rectangle – by 2 corners – Enter the coordinate values, width -Ok.
6. Meshing – Mesh tool – Areas, set – select the object – Ok – Element edge length 0.05 - Ok – Mesh tool- Tri, free - mesh – Select the object–Ok.

SOLUTION

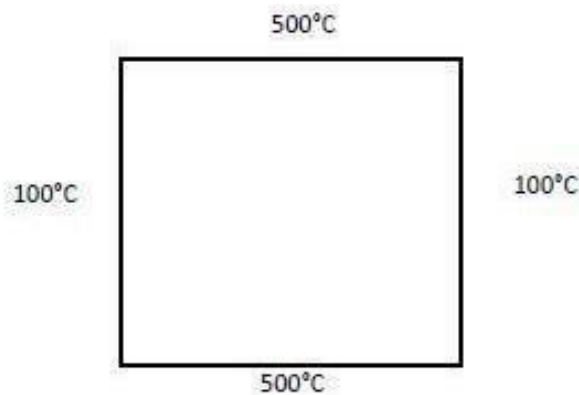
6. Solution – Define Loads – Apply – Thermal – Temperature - On lines – Select the right and left side of the object –Ok – Temp. Value 100 – On lines – select the top and bottom of the object – Ok –Temp 500 –Ok.
7. Solve – Current LS – Ok – Solution is done –Close.

POST PROCESSING

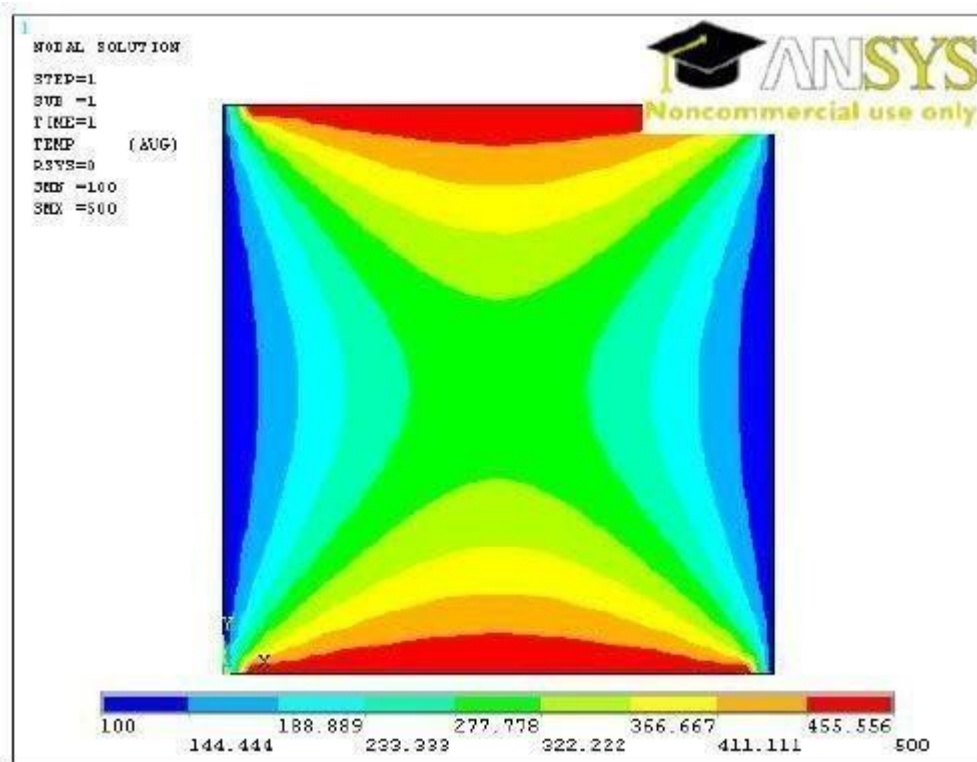
8. General post proc – Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature –Ok.

FOR REPORT GENERATION

9. File – Report Generator – Choose Append – OK – Image Capture – Ok -Close.



Thermal Conductivity of the material = 10 W/m.°C
Dimension of the object = 2 m x 2 m



RESULT

Thus the conductive heat transfer analysis of a 2D component by using ANSYS is studied.

CONVECTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

ExNo : 10

Date :

AIM

To conduct the convective heat transfer analysis of a 2D component by using ANSYS software.

SYSTEM CONFIGURATION

Ram:	2 GB
Processor:	Core 2 Quad / Core 2 Duo
Operating system:	Window XP Service Pack 3
Software:	ANSYS (Version 12.0/12.1)

PROCEDURE

The three main steps to be involved are

1. PreProcessing
2. Solution
3. PostProcessing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference – structural - h-Method -Ok.
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 55 – Ok– Close.
3. Real constants - Add/Edit/Delete – Add –Ok.
3. Material props - Material Models –Thermal – Conductivity – Isotropic – KXX 16 – Ok.
5. Modeling – Create – Key points - In active CS – enter the key point number and X, Y, Z location for 8 key points to form the shape as mentioned in the drawing. Lines – lines - Straight line - Connect all the key points to form as lines. Areas – Arbitrary -by lines - Select all lines - ok. [We can create full object (or) semi-object if it is a symmetrical shape]

6. Meshing – Mesh tool – Areas, set – select the object – Ok – Element edge length 0.05 - Ok – Mesh tool- Tri,free mesh – Select the object–Ok.

SOLUTION

7. Solution – Define Loads – Apply – Thermal – Temperature - On lines – Select the lines –Ok – Temp. Value 300 – Ok – Convection – On lines – select the appropriate line – Ok – Enter the values of film coefficient 50, bulk temperature 40 – Ok.

8. Solve – Current LS – Ok – solution is done –Close.

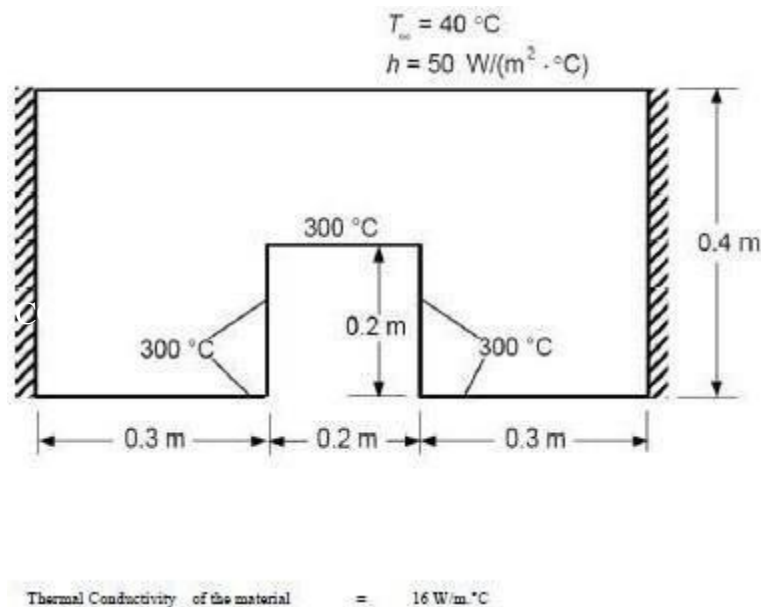
POST PROCESSING:

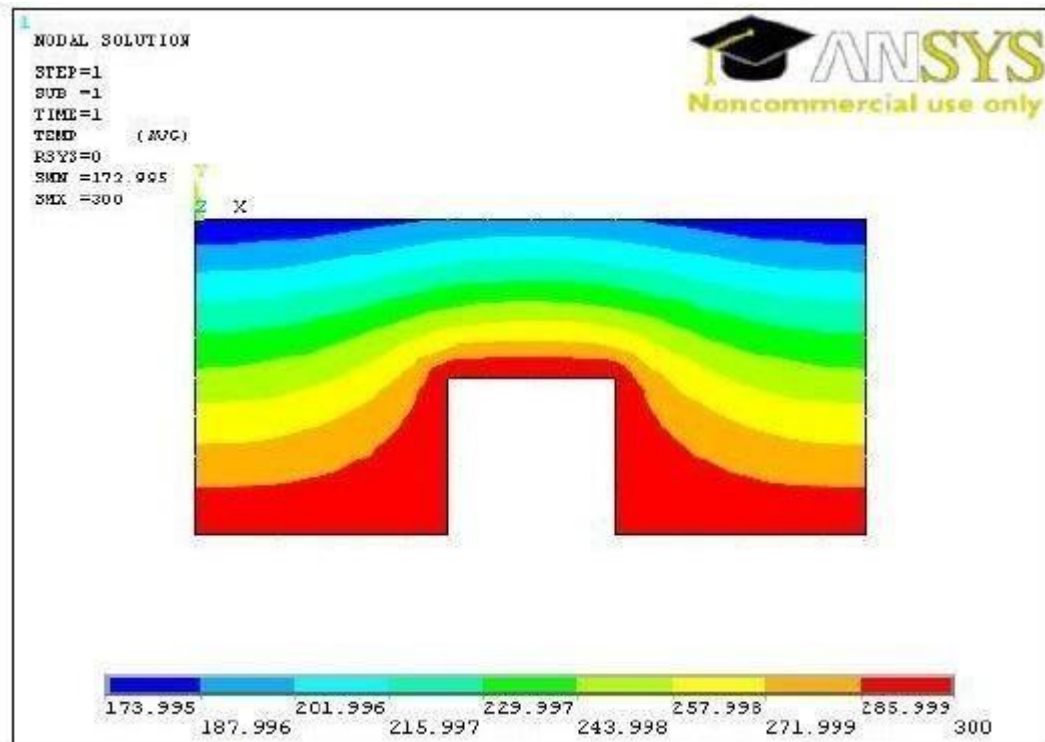
10. General post proc – List results – Nodal Solution – DOF Solution – Nodal temperature– Ok.

11. Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature –Ok.

FOR REPORT GENERATION:

12. File – Report Generator – Choose Append – OK – Image Capture – Ok -Close.





RESULT

Thus the convective heat transfer analysis of a 2D component is done by using the ANSYS Software

Introduction to MATLAB

EX: 11

Date:

Aim :

To Study the capabilities of MatLab Software.

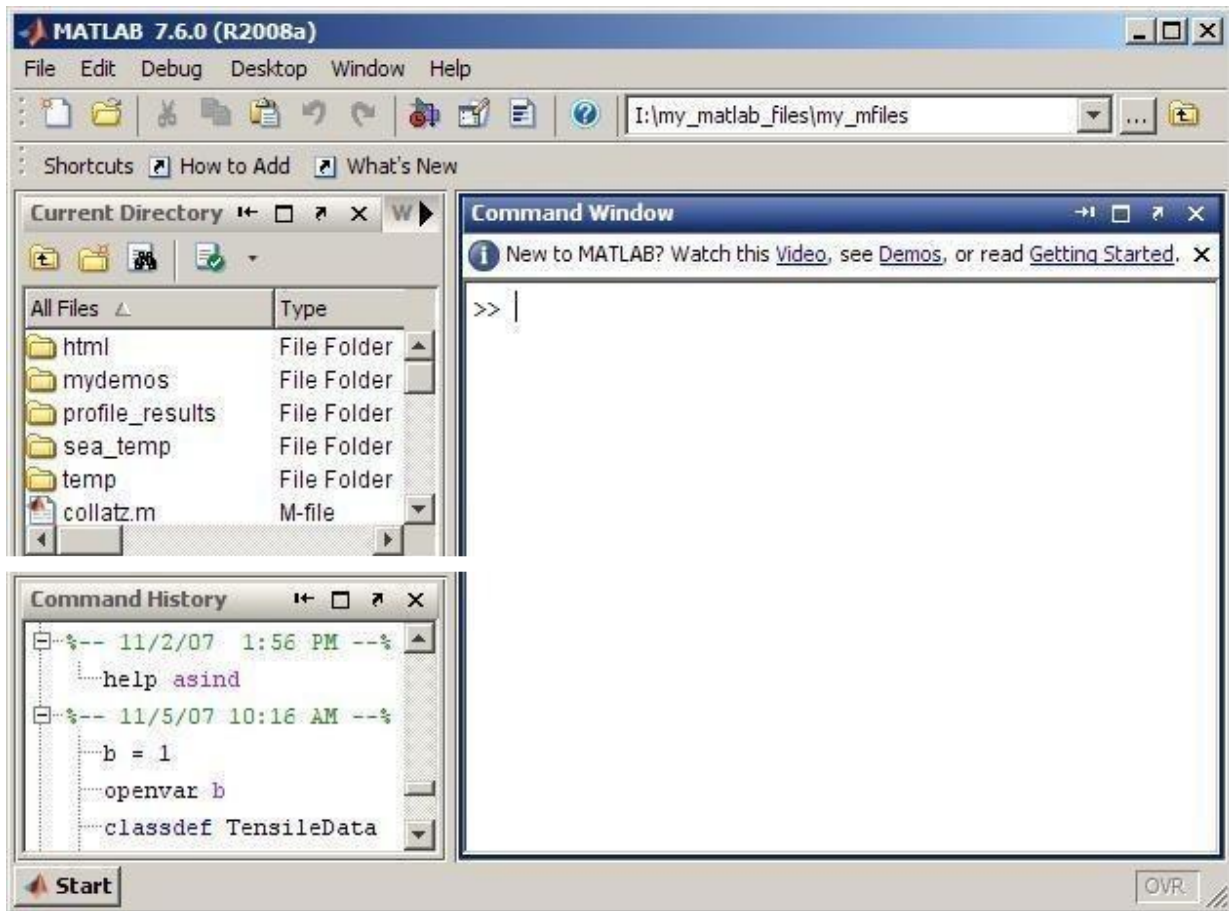
Introduction

The MATLAB is a high-performance language for technical computing integrates computation, visualization, and programming in an easy-to-use environment where problems and solutions are expressed in familiar mathematical notation. Typical uses include

- Math and computation
- Algorithm development
- Data acquisition
- Modeling, simulation, and prototyping
- Data analysis, exploration, and visualization
- Scientific and engineering graphics
- Application development,

Including graphical user interface building MATLAB is an interactive system whose basic data element is an array that does not require dimensioning. It allows you to solve many technical computing problems, especially those with matrix and vector formulations, in a fraction of the time it would take to write a program in a scalar noninteractive language such as C or FORTRAN.

The name MATLAB stands for *matrix laboratory*. MATLAB was originally written to provide easy access to matrix software developed by the LINPACK and EISPACK projects. Today, MATLAB engines incorporate the LAPACK and BLAS libraries, embedding the state of the art in software for matrix computation.



SIMULINK INTRODUCTION:

Simulink is a graphical extension to MATLAB for modeling and simulation of systems. In Simulink, systems are drawn on screen as block diagrams. Many elements of block diagrams are available, such as transfer functions, summing junctions, etc., as well as virtual input and output devices such as function generators and oscilloscopes. Simulink is integrated with MATLAB and data can be easily transferred between the programs. In these tutorials, we will apply Simulink to the examples from the MATLAB tutorials to model the systems, build controllers, and simulate the systems. Simulink is supported on Unix, Macintosh, and Windows environments; and is included in the student version of MATLAB for personal computers.

The idea behind these tutorials is that you can view them in one window while running Simulink in another window. System model files can be downloaded from the tutorials and opened in Simulink. You will modify and extend these system while learning to use Simulink for system modeling, control, and simulation for your actual Simulink windows. Most images in these tutorials are not live - they simply display what you should see in your own Simulink windows. All Simulink operations should be done in your Simulink windows.

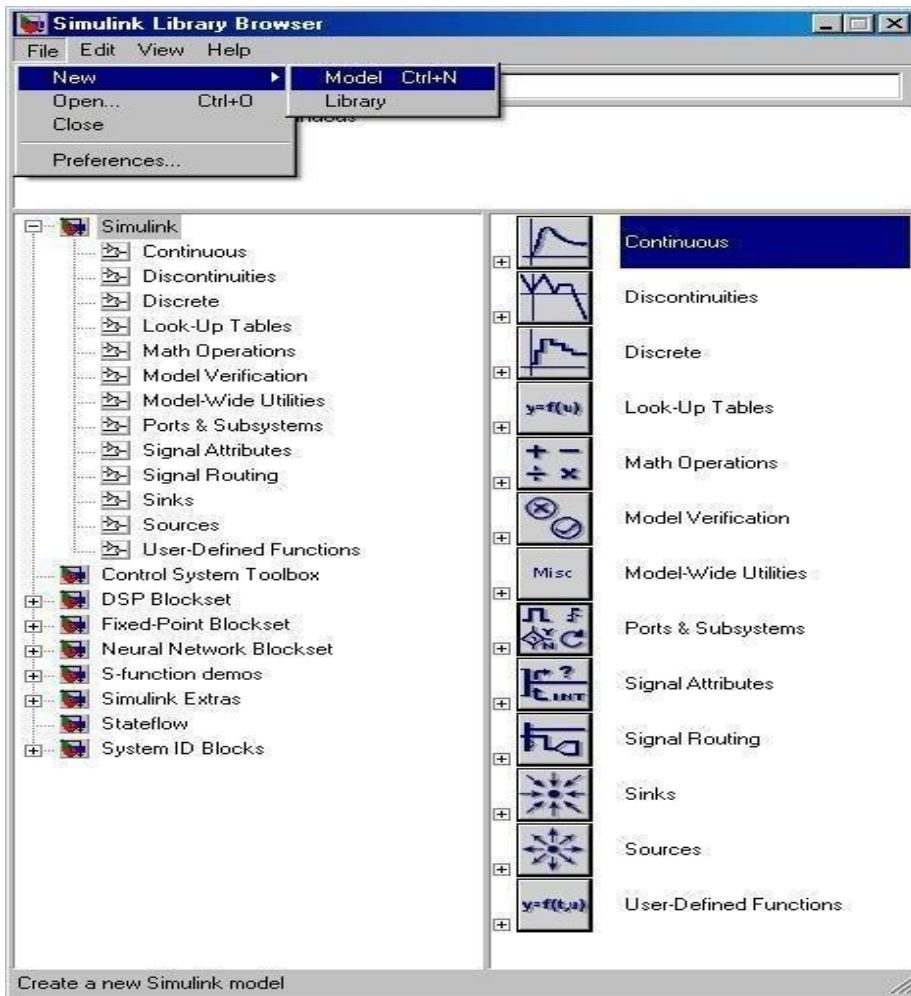
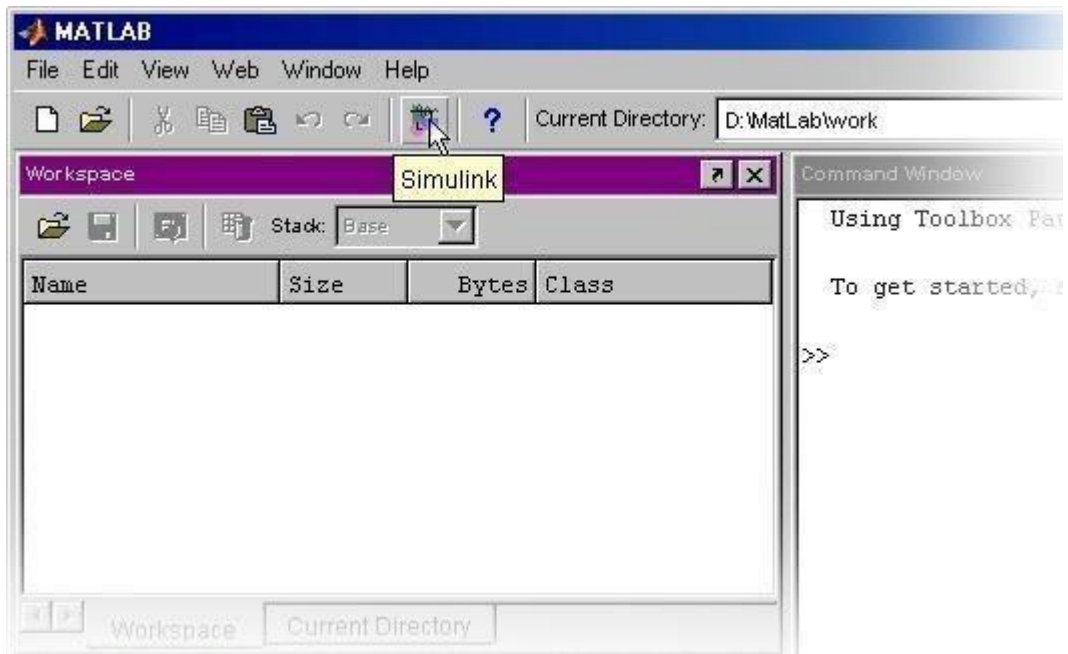
1. Starting Simulink
2. Model Files
3. Basic Elements
4. Running Simulations
5. Building Systems

Starting Simulink

Simulink is started from the MATLAB command prompt by entering the following command:

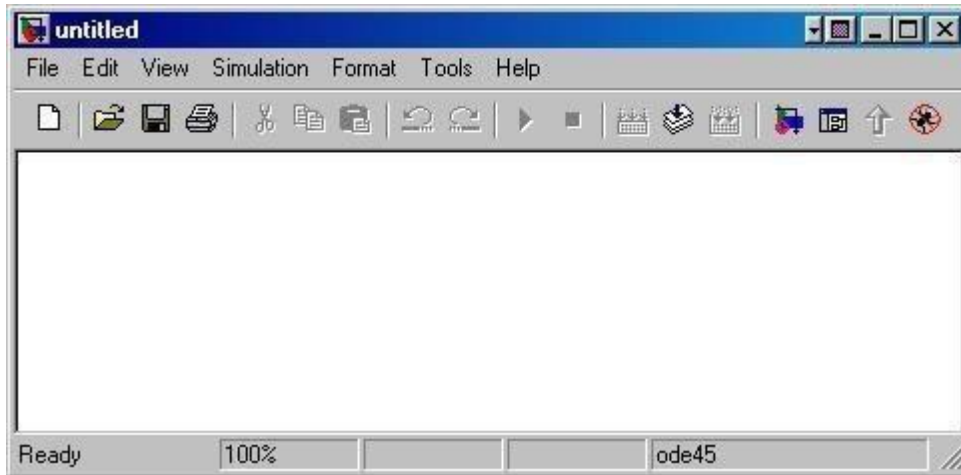
```
>> Simulink
```

Alternatively, you can hit the Simulink button at the top of the MATLAB window as shown below:



. Open the modeling window with New then Model from the File menu on the Simulink Library Browser as shown above.

This will bring up a new untitled modeling window shown below



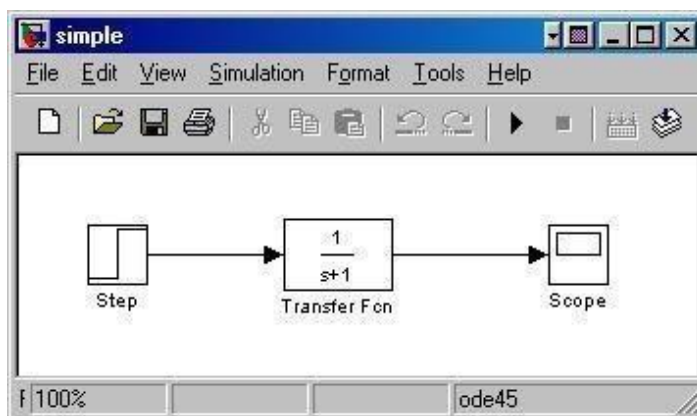
Model Files

In Simulink, a model is a collection of blocks which, in general, represents a system. In addition to drawing a model into a blank model window, previously saved model files can be loaded either from the File menu or from the MATLAB command prompt.

You can open saved files in Simulink by entering the following command in the MATLAB command window. (Alternatively, you can load a file using the Open option in the File menu in Simulink, or by hitting Ctrl+O in Simulink.)

```
>> filename
```

The following is an example model window.



A new model can be created by selecting New from the File menu in any Simulink window (or by hitting Ctrl+N).

Basic Elements

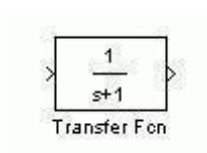
There are two major classes of items in Simulink: blocks and lines. Blocks are used to generate, modify, combine, output, and display signals. Lines are used to transfer signals from one block to another.

Blocks

There are several general classes of blocks:

- Continuous
- Discontinuous
- Discrete
- Look-UpTables
- MathOperations
- ModelVerification
- Model-WideUtilities
- Ports &Subsystems
- SignalAttributes
- SignalRouting
- Sinks: Used to output or displaysignals
- Sources: Used to generate various signals
- User-DefinedFunctions
- Discrete: Linear, discrete-time system elements (transfer functions, state-space models,etc.)
- Linear: Linear, continuous-time system elements and connections (summing junctions, gains, etc.)
- Nonlinear: Nonlinear operators (arbitrary functions, saturation, delay,etc.)
- Connections: Multiplex, Demultiplex, System Macros,etc.

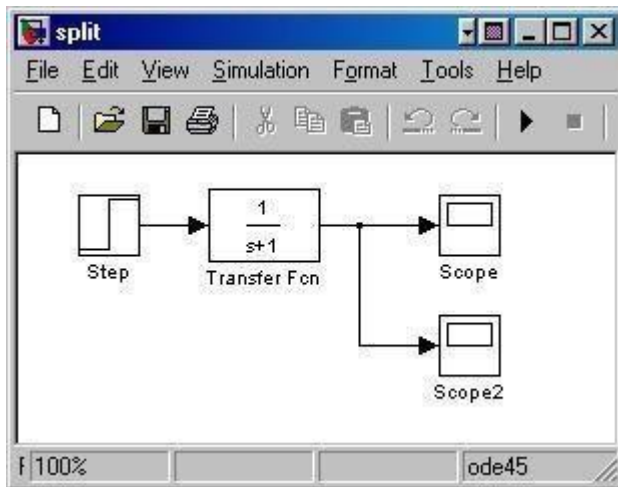
Blocks have zero to several input terminals and zero to several output terminals. Unused input terminals are indicated by a small open triangle. Unused output terminals are indicated by a small triangular point. The block shown below has an unused input terminal on the left and an unused output terminal on the right.



Lines

Lines transmit signals in the direction indicated by the arrow. Lines must always transmit signals from the output terminal of one block to the input terminal of another block. One exception to this

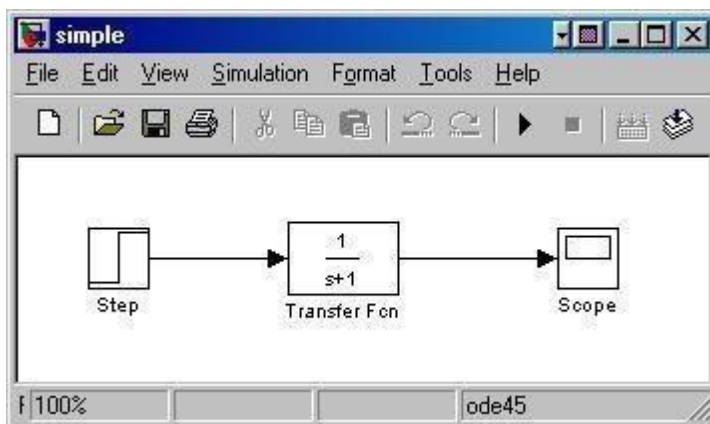
is a line can tap off of another line, splitting the signal to each of two destination blocks, as shown below.



Lines can never inject a signal *into* another line; lines must be combined through the use of a block such as a summing junction.

A signal can be either a scalar signal or a vector signal. For Single-Input, Single-Output systems, scalar signals are generally used. For Multi-Input, Multi-Output systems, vector signals are often used, consisting of two or more scalar signals. The lines used to transmit scalar and vector signals are identical. The type of signal carried by a line is determined by the blocks on either end of the line.

Simple Example



The *simple* model (from the [model files](#) section) consists of three blocks: Step, Transfer Fcn, and Scope. The Step is a source block from which a step input signal originates. This signal is transferred through the line in the direction indicated by the arrow to the Transfer Function linear block. The Transfer Function modifies its input signal and outputs a new signal on a line to the Scope. The Scope is a sink block used to display a signal much like an oscilloscope.

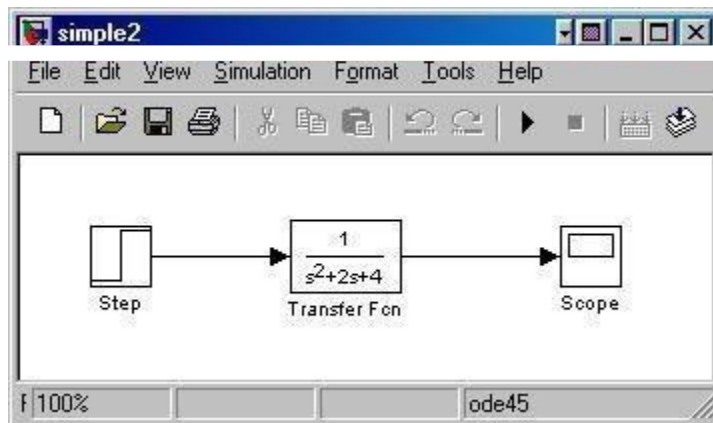
There are many more types of blocks available in Simulink, some of which will be discussed later. Right now, we will examine just the three we have used in the simple model.

Running Simulations

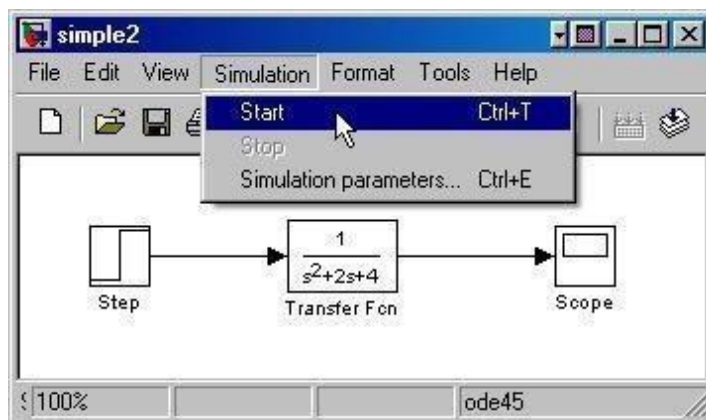
To run a simulation, we will work with the following model file:

simple2.mdl

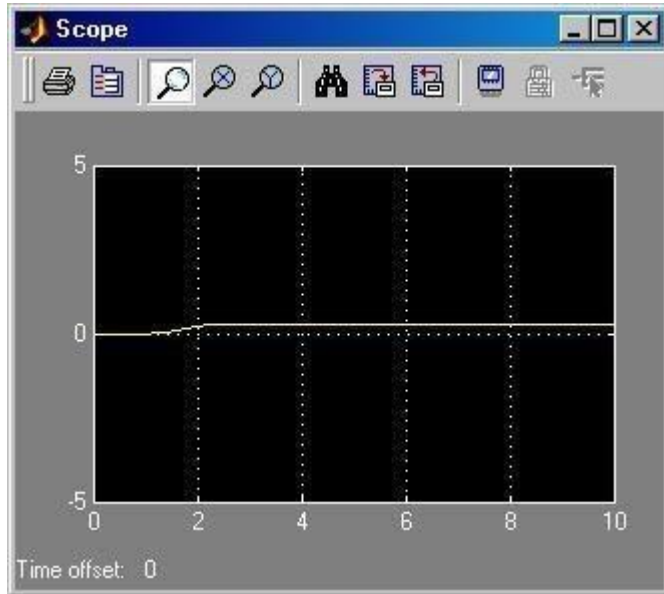
Download and open this file in Simulink following the previous instructions for this file. You should see the following model window.



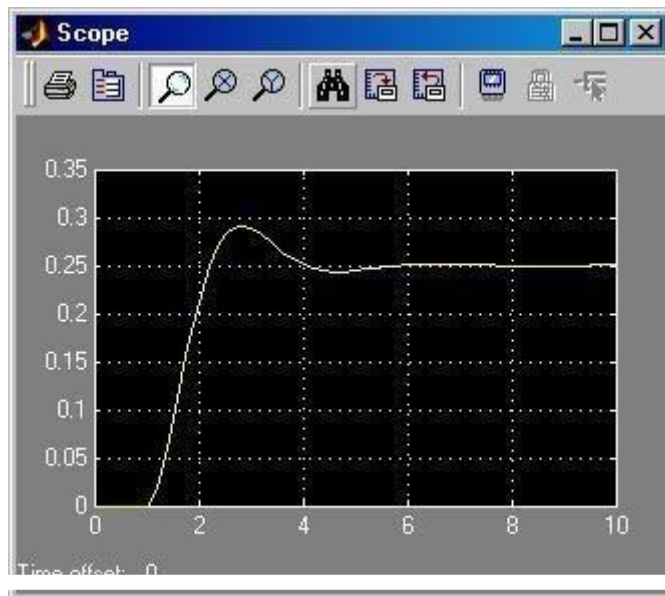
Before running a simulation of this system, first open the scope window by double-clicking on the scope block. Then, to start the simulation, either select Start from the Simulation menu (as shown below) or hit Ctrl-T in the model window.



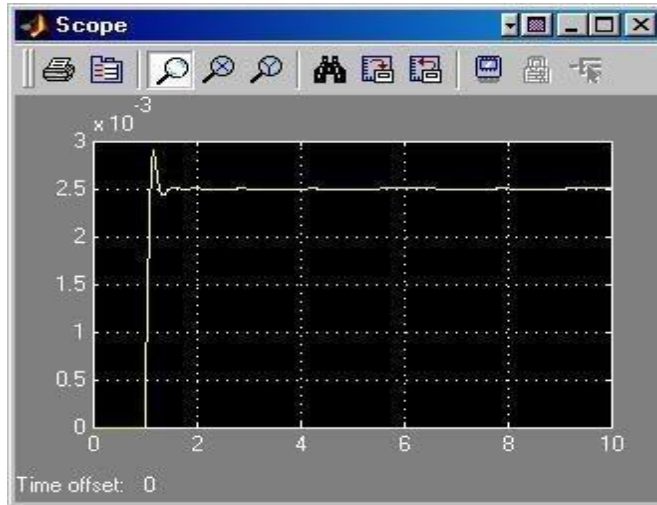
The simulation should run very quickly and the scope window will appear as shown below. If it doesn't, just double click on the block labeled "scope."



Note that the simulation output (shown in yellow) is at a very low level relative to the axes of the scope. To fix this, hit the autoscale button (binoculars), which will rescale the axes as shown below.

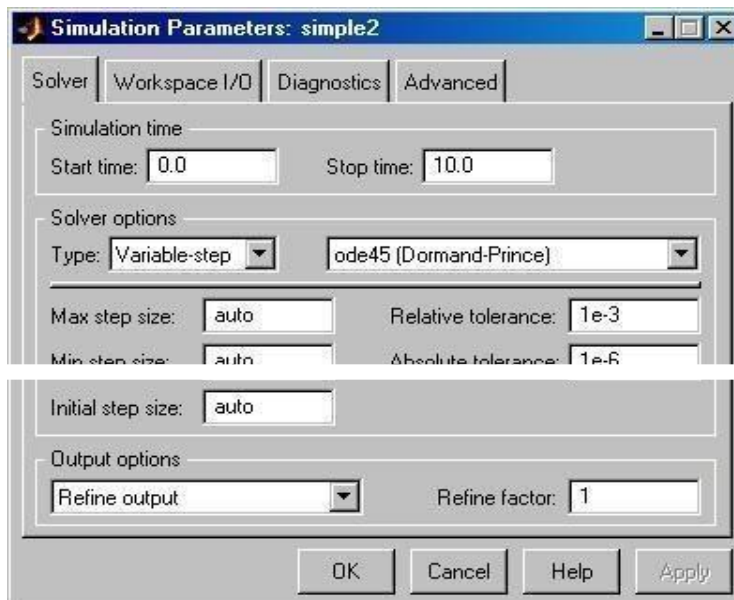


Note that the step response does not begin until $t=1$. This can be changed by double-clicking on the "step" block. Now, we will change the parameters of the system and simulate the system again. Double-click on the "Transfer Fcn" block in the model window and change the denominator to Re-run the simulation (hit Ctrl-T) and you should see what appears as a flat line in the scope window. Hit the autoscale button, and you should see the following in the scope window.



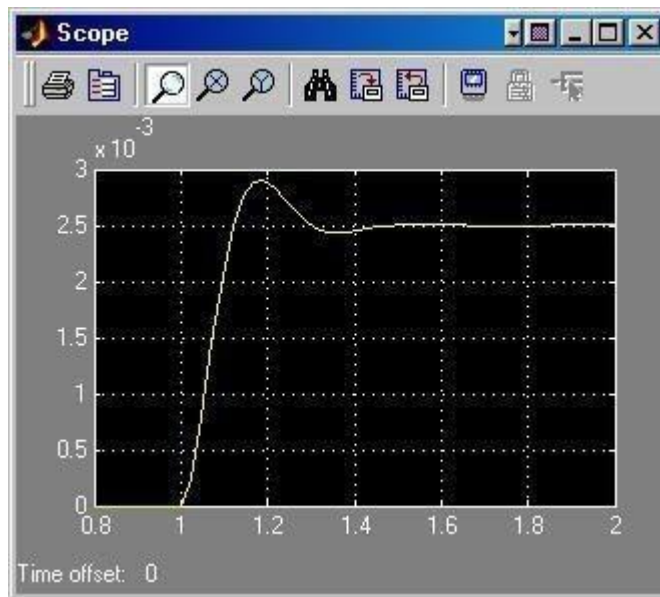
Notice that the autoscale button only changes the vertical axis. Since the new transfer function has a very fast response, it compressed into a very narrow part of the scope window. This is not really a problem with the scope, but with the simulation itself. Simulink simulated the system for a full ten seconds even though the system had reached steady state shortly after onesecond.

To correct this, you need to change the parameters of the simulation itself. In the model window, select Parameters from the Simulation menu. You will see the following dialogbox.



There are many simulation parameter options; we will only be concerned with the start and stop times, which tell Simulink over what time period to perform the simulation. Change Start time from 0.0 to 0.8 (since the step doesn't occur until $t=1.0$). Change Stop time from 10.0 to 2.0, which should be only shortly after the system settles. Close the dialog box and rerun the simulation.

After hitting the autoscale button, the scope window should provide a much better display of the step response as shown below.



RESULT

Thus the features of MATLAB are studied

EX.NO: FORCE AND STRESS ANALYSIS USING LINK ELEMENTS IN TRUSSES AND CABLES

AIM:

To conduct the force and stress analysis using link elements in trusses and cables using ANSYS software.

SYSTEM CONFIGURATION

Ram : 2 GB
Processor : Core 2
Quad / Core 2 Duo Operating
system: Window
XP Service Pack 3 Software :
ANSYS
(Version 12.0/12.1)

PROCEDURE:

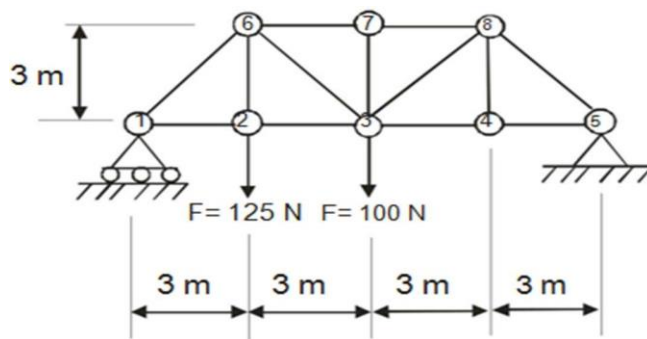
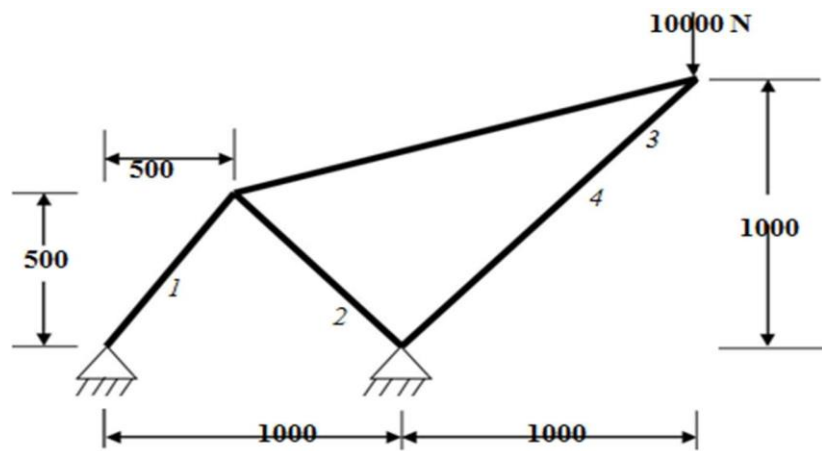
The three main steps to be involved are

1. Pre Processing
2. Solution
3. Post Processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. File – clear and start new – do not read file – ok – yes.
2. Ansys Main Menu – Preferences select – STRUCTURAL - ok
3. **Element type** – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.
4. **Real constants** – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
5. **Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – ok – close.
6. **Modeling** – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 4 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 4, 3 (x, y value w.r.t first node) – apply (third node is created) – 0, 3 (x, y value w.r.t first node) – ok (fourth node is created).
7. Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 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).



8. Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 4 – apply– DOFs to be constrained – All DOF – ok – on Nodes – pick node 2 – apply – DOFs to be constrained – UY – ok.
9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.

SOLUTION

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

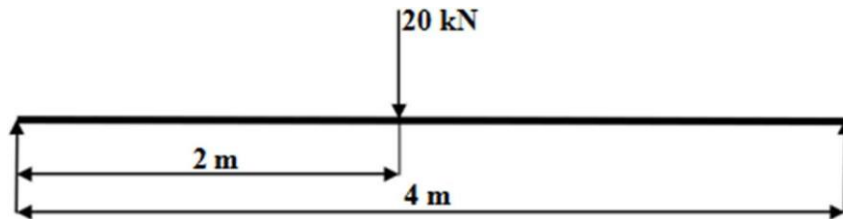
GENERAL POST PROCESSOR

11. Element table – Define table – Add – ‘Results data item’ – By Sequence num – LS – LS1 – ok.
12. Plot Results – Deformed Shape – def+undeformed – ok. Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).
13. Plot results – contour plot – Nodal solution – DOF solution – displacement vector sum – ok. List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers). List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).
14. **PlotCtrls** – Animate – Deformed shape – def+undeformed-ok

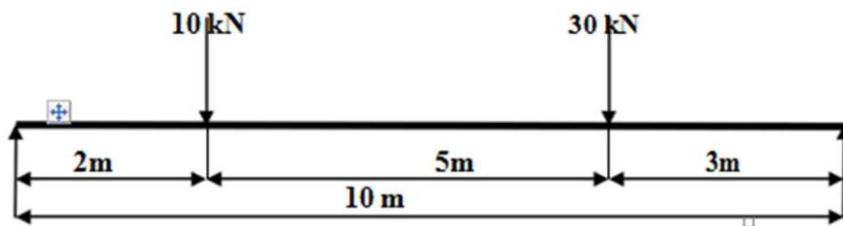
INFERENCE

RESULT

Thus the force and stress analysis using link elements in trusses and cables is done by using the ANSYS Software.



Rectangular *c/s* area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



Rectangular *c/s* area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.

EX.NO: 2D-THERMAL STATIC ANALYSIS OF CHIMNEY

AIM :

To analyse the temperature distribution of chimney.

SYSTEM CONFIGURATION

Ram : 2 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system: Window XP Service Pack 3
Software : ANSYS (Version12.0/12.1)

PROCEDURE:

The three main steps to be involved are

1. Pre Processing
2. Solution
3. Post Processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference → tick → thermal → select 'h' method → ok
2. Title: Utility menu → file change title → chimney → ok
3. Utility menu → plot → replot.
4. Elements: Main menu → preprocessor → element type → add/edit/delete → add → thermal mass → solid → quad. 4node 55 → ok
5. Material Properties: Main menu → preprocessor → Material props → material models → material model 1 → thermal → conductivity → isotropic → enter $KXX1 = 0.037$ → ok
6. Material → new model → material ID 2 → material model 2 → thermal → conductivity → isotropic → enter $KXX2 = 0.012$ → ok.
7. Modeling: Main menu → preprocessor → modeling → Create → Areas → Rectangle → by center and corner → enter 0,0,26,26 → ok.

8. Main menu → preprocessor → Modeling → Create → Areas → Rectangle → By center and corner → enter 0,0,13,13 → ok.
9. Main menu → preprocessor → Modeling → Operate → boolean → subtract → area → select base area → apply → select area to be subtracted → ok.
10. Main menu → preprocessor → Modeling → Create → Areas → Rectangle → By center and corner → enter 0,0,13,13 → ok.
11. Main menu → preprocessor → Modeling → Create → Areas → Rectangle → By center and corner → enter 0,0,12,12 → ok.
12. Main menu → preprocessor → Modeling → Operate → boolean → subtract → area → select base area → apply → select area to be subtracted → ok.
13. Main menu → preprocessor → Modeling → Operate → boolean → glue → area → pick all → ok.
14. Utility menu → plot cntrl → hard copy → to file → jpeg → save to: chimney. jpg → ok
15. Main menu → preprocessor → Meshing → size cntrl → manual size → global → size → enter SIZE as 0.25 → ok
16. Main menu → preprocessor → meshing → mesh attributes → difficult attributes → select element 1; material 1 → ok.
17. Main menu → preprocessor → Meshing → mesh tool → mesh → pick brick area (3) → ok.
18. Main menu → preprocessor → Meshing → mesh attributes → difficult attributes → select element 1 ; material 2 → ok.
19. Main menu → preprocessor → Meshing → mesh tool → mesh → pick concrete area (1) → ok
20. Utility menu → plot cntrl → hard copy → to file → jpeg → save to: chimney mesh. jpg → ok
21. Boundary conditions and Loads
22. Main menu → solution → define loads → apply → thermal → convection → on lines → select by mouse; all outer lines (1, 2, 3, 4) → ok → enter ValI 0.012 → enter Val2I 10 → ok.
23. Main menu → solution → define loads → apply → thermal → convection → on lines → select by mouse; all inner lines (13, 14,15, 16) → ok → enter ValI 0.037 → enter Val2I 140 → ok.

24. Utility menu → plot controls → hard copy → to file → jpeg → save to : chimney load.jpg →ok

SOLUTION

25. Solve – Current LS – Ok – Solution is done – Close.

POST PROCESSING

26. General post proc – Plot results – Contour plot – Nodal solution – Stress – 1st principal stress – Ok – Nodal solution – DOF Solution – Displacement vector sum - Ok.

FOR REPORT GENERATION

27. File – Report Generator – Choose Append – OK – Image Capture – Ok - Close.

INFERENCE

RESULT:

Thus the temperature distribution of chimney is done by using the ANSYS Software.

EX.NO: **MODAL ANALYSIS OF BEAMS**

AIM :

To conduct the Modal Analysis of beams by using ANSYS software.

SYSTEM CONFIGURATION

Ram : $E_x = 210 \text{ Gpa} = 210000 \text{ N/mm}^2$,PRXY=0.3 , Density= $7.85 * 10^{-9} \text{ N/mm}^3$
Processor :
Operating syster
Software :



PROCEDURE:

The three main steps to be involved are

1. Pre Processing
2. Solution
3. Post Processing

Start - All Progr $E_x = 210 \text{ Gpa} = 210000 \text{ N/mm}^2$,PRXY=0.3 ,Density= $7.85 * 10^{-9} \text{ N/mm}^3$ the
Working Director



PREPROCESSING

1. Preference – structural - n-Method - Ok.
2. **Element type** – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.
3. **Real constants** – Add – ok – real constant set no – 1 – c/s area – $0.01 * 0.01$ moment of inertia – $0.01 * 0.01 ** 3 / 12$ – total beam height – 0.01 – ok.
4. **Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – $200e9$ – PRXY – 0.27 – Density – 7800 – ok – close.
5. **Modeling** – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations – 1,0 – ok (Keypoints created). Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.
6. **Meshing** – Size Cntrl – ManualSize – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

SOLUTION

7. Solution – Analysis Type – New Analysis – Modal – ok.
8. Solution – Analysis Type – Subspace – Analysis options – no of modes to extract – 5 – no of modes to expand – 5 – ok – (use default values) – ok.
9. Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick first keypoint – apply – DOFs to be constrained – ALL DOF – ok. Solve – current LS – ok (Solution is done is displayed) – close.
10. Solve – Current LS – Ok – solution is done – Close.

GENERAL POST PROCESSOR:

11. Read Results – First Set. Plot Results – Deformed Shape – def+undeformed – ok.
PlotCtrls – Animate – Deformed shape – def+undeformed-ok.
12. Read Results – Next Set. Plot Results – Deformed Shape – def+undeformed – ok. **PlotCtrls** – Animate – Deformed shape – def+undeformed-ok.

INFERENCE

RESULT:

Thus the Modal Analysis beams is done by using the ANSYS Software.

EX.NO: SIMULATION OF AN ACCUMULATOR

AIM :

To perform simulation of an accumulator using matlab.

PROBLEM STATEMENT

An accumulator is loaded with 400KN weight. The diameter of ram is 300mm and has a stroke length of 6m. if friction is taken as 5%, it takes two minutes to fall through full stroke. Find the total work supplied and power delivered using matlab the flow rate of the fluid is $0.0075\text{m}^3/\text{sec}$ while accumulator descends with stated velocity. Take density of fluid as $1000\text{N}/\text{m}^3$.

PROGRAM:

```
#include<stdio.h>
#include<conio.h>
#include<stdlib.h>
#include<math.h> int main()
{
Float sl=0.0,w=0.0,d=0.0,f=0.0,a=0.0,p=0.0,h=0.0,ds=0.0,wd=0.0,fl=0.0;
Float ls=0.0,t=0.0,nl=0.0,ws=0.0,tw=0.0,pd=0.0,ns=0.0,l=0.0;
Printf("\nEnter stroke length<m>");
Scanf("%f",&sl);
Printf("\nEnter time taken for completion of stroke<s>");
Scanf("%f",&t);
Printf("\nEnter the piston force<kN>");
Scanf("%f",&w);
Printf("\nEnter friction value<%>");
Scanf("%f",&fl);
Printf("\nEnter the diameter of ram<m>");
Scanf("%f",&d);
Printf("\nEnter the flow rate<m3/s>");
```

```
scanf("%f",&ls);
f=fl/100;
a=(3.14/4)*(d*d); nl=w*(1-f);
wd=nl*(sl/t); p=(nl/a); h=p/(1000*9.81); ws=(9810*ls)*h; tw=wd+ws;
printf("\nTotal work supplied to the hydraulic machine in kNm=%f",tw); pd=tw/t;
printf("\nPower delivered in kw=%f",pd); getch();
}
```

INPUT PARAMETERS

Stroke length

Time taken for completion of a stroke

Piston force friction value

Diameter of ram flow rate

OUTPUT PARAMETERS

Total work supplied to the hydraulic machine

Power delivered

RESULT

Thus simulation of an accumulator has been done using matlab.

EX.NO: **SIMULATION OF CAM AND FOLLOWER**

AIM :

To perform simulation of cam and follower using matlab.

PROBLEM STATEMENT

A cam is to operate a flat faced follower having uniform acceleration and deceleration. The least radius of cam is 50mm. During decent the deceleration period is half of the acceleration period. The ascent lift is 37.5 mm. The ascent is for one fourth period, dwell for one third and remaining is descent. The cam rotates at 600rpm. Simulate using matlab to determine the maximum velocity during ascent and descent.

PROGRAM:

```
#include<stdio.h>

#include<conio.h>

#include<stdlib.h>

int main()

{

int a1, a2;

Float n,w,s1,s,outr,inr,vo,ao,vr,ar;

Printf("\nEnter speed of rotation<rpm>");

Scanf("%f",&n); w=(2*3.14*n)/60;

Printf("\nEnter the cam lift <mm>");

Scanf("%f",&s1); s=s1/1000;

Printf("\nEnter cam ascent period<degrees>");

Scanf("%f",&out);

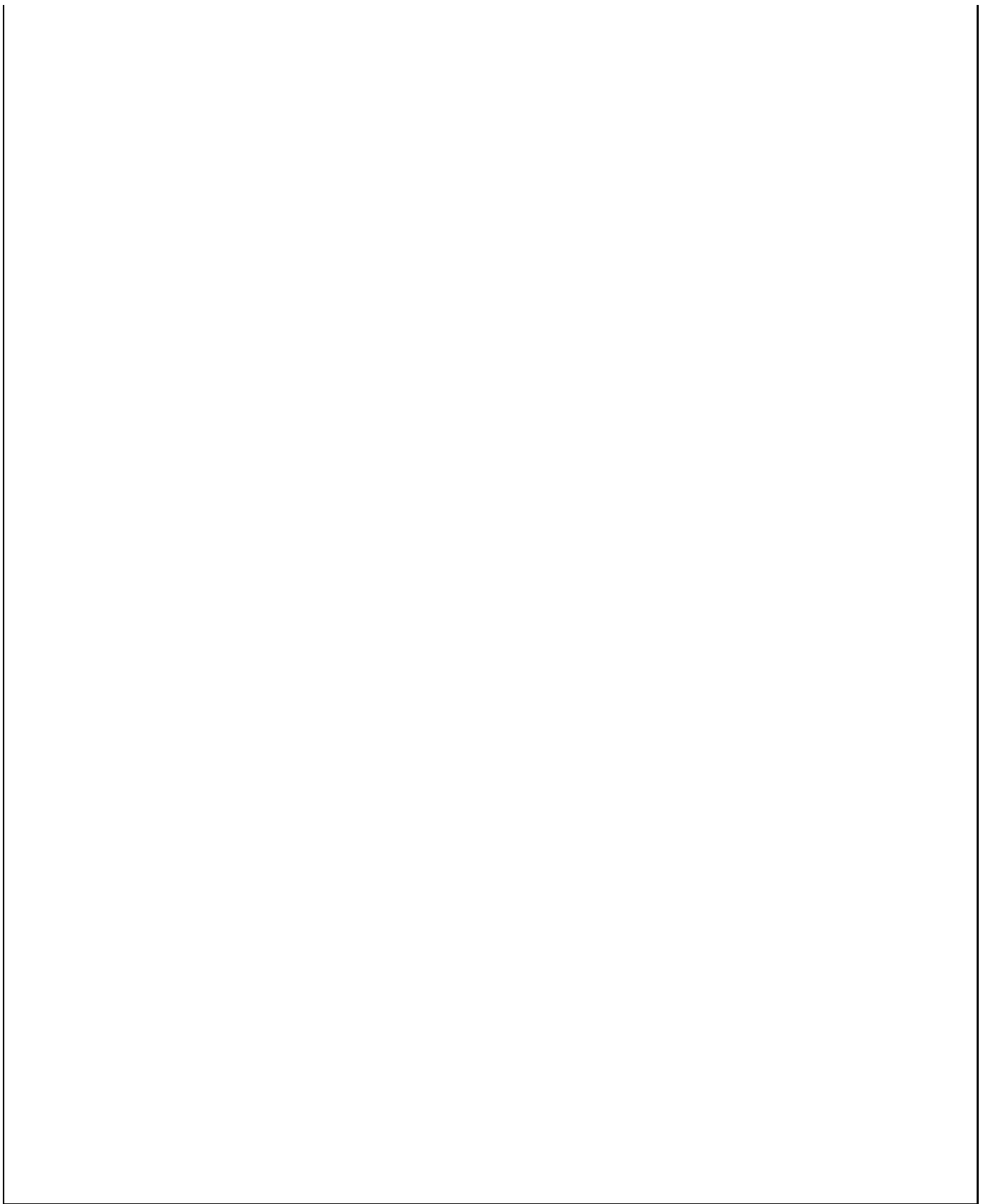
Printf("\nEnter cam descent period<degrees>");

Scanf("%f",&in);

Outr=(3.14/180)*out;

Inr=(3.14/180)*in;

Vo=(2*w*s)/out;
```

```

ao=(4*w*w*s)/(outr*outr);
vr=(2*w*s)/inr;
ar=(4*w*w*s)/(inr*inr);
Printf("\nAngular velocity = %f<rad/s>",w);
Printf("\nOUTWARD STROKE");
Printf("\nMaximum velocity =%f<m/s>",vo);
Printf("\nMaximum acceleration =%f<m/s^2>",ao);
Printf("\nRETURN STROKE");
Printf("\nMaximum velocity =%f<m/s>",vr);
Printf("\nMaximum acceleration =%f<m/s^2>",ar);
getch()
}

```

INPUT PARAMETERS

Speed of rotation

Cam lift

Cam ascent period

Cam descent period

OUTPUT PARAMETERS

Angular velocity =

OUTWARD STROKE

Maximum velocity =

Maximum acceleration =

RETURN STROKE

Maximum velocity =

Maximum acceleration =

RESULT

Thus simulation of an cam and follower has been done using Matlab.



EX.NO: **DESIGN OF SLIDER CRANK MECHANISM USING ADAMS**

AIM :

To design and develop slider crank mechanism using Adams

SYSTEM CONFIGURATION

Ram : 2 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system: Window XP Service Pack 3
Software : Adams View, Adams Post Processor

PROCEDURE:

Step 1. Creating the Model

- a. Start Adams/View
- b. Create a new model.
- c. Resize the working grid.
- d. Open Coordinate Window
- e. Create crank part from point (60, 0, 0) to (150, 0, 0)
- f. Rename. slider crank. crank

Step 2. Creating Revolution

- a. Select Rigid Body: Revolution
- b. Click the points
- c. Right-click too close
- d. Rename. slider crank. cylinder.

Step 3. Creating Joints

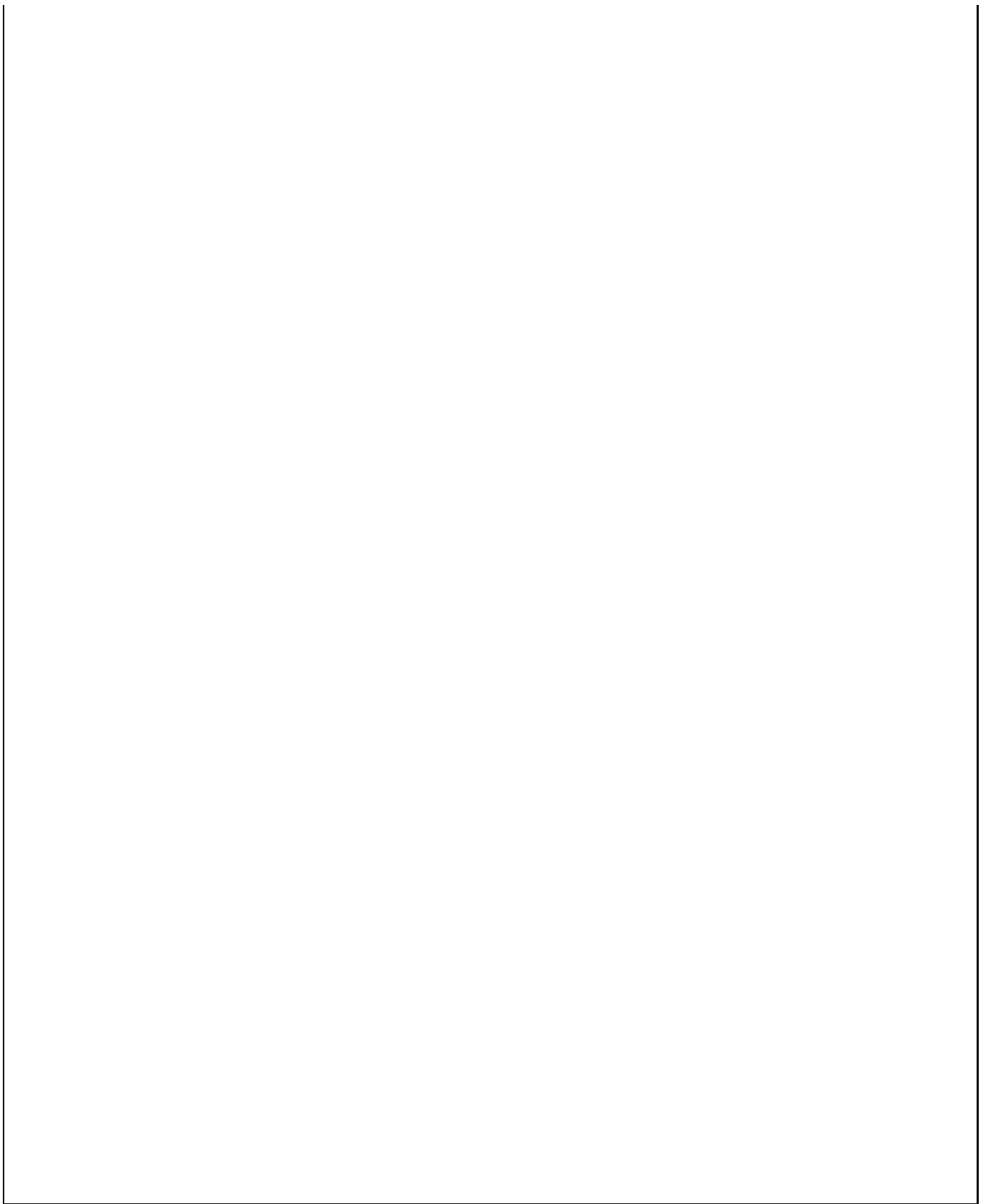
- a. Select Rigid body: Cylinder
- b. Create piston part. Rename. slider crank. piston
- d. Create revolute joints between crank and ground
- e. Create spherical joint between cylinder and ground
- f. Create translational joint between piston and cylinder.
- g. Create Hooke joint between crank and piston
- h. Add Rotational joint motion to revolute joint with function = $-30\text{deg} - 60 * \text{time}$.

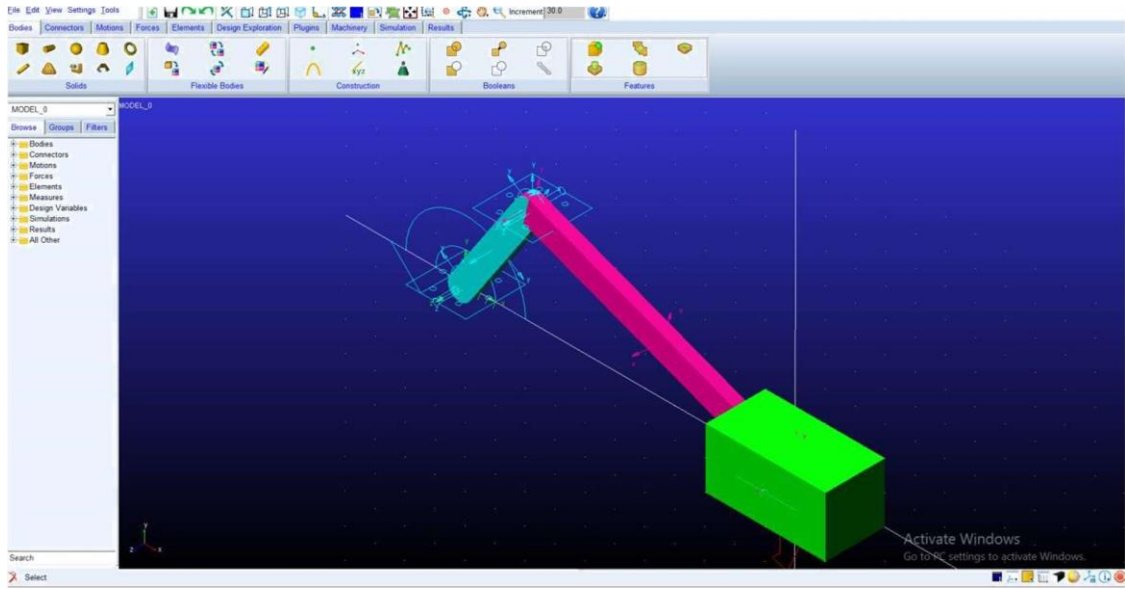
Step 4. Create Point-to-Point Measure

- a. From Design Exploration ribbon, select Point to-Point Measure
- b. Select Displacement as Characteristic
- c. Select Global as Component.
- d. Select the Marker at the left end of cylinder and the Marker at the left end of crank
- e. Rename it as MEA_PT2PT_R

Step 5. Create Point Measure

- a. Under the piston tree in the Model Browser, rightsticks and select Measure
- b. Select Translational Velocity and select Component.
- c. Enter cylinder.cm as Represent coordinates





- d. Select any Marker belongs to the ground as DO time derivatives in.
- e. Repeat the above steps to create a translational acceleration

Step 6. Create Angle about Axis Measure

- a. In the Bodies tree, right-click the spherical joint between cylinder and the ground
- b. Select info and remember the name of I Marker and Marker.
- c. Close the info window.
- d. Select Function Measure
- e. Select Angle about Z under Displacement and enter the marker name in Step b.
- f. Select angle as units

Step 7. Testing the Model

- a. From Simulation ribbon, select Run an Interactive Simulation
- b. Set End Time to 0.01 and Step Size to 0.001, and then click Start
- c. Click Plotting
- d. Use the Plot tracking tool
- e. Follow the plot curve. Find the size measurement at $X= 0.0$

Theory:

The Slider-crank mechanism is used to transform rotational motion into translational motion by means of a rotating driving beam, a connection rod and a sliding body. In the present example, a flexible body is used for the connection rod. The sliding mass is not allowed to rotate and three revolute joints are used to connect the bodies. While each body has six degrees of freedom in space, the kinematical conditions lead to one degree of freedom for the whole system.

A slider crank mechanism converts circular motion of the crank into linear motion of the slider. In order for the crank to rotate fully the condition $L > R+E$ must be satisfied where R is the crank length, L is the length of the link connecting crank and slider and E is the offset of slider. A slider crank is a RRRP type of mechanism i.e. It has three revolute joints and 1 prismatic joint. The total distance covered by the slider between its two extreme positions is called the path length. Kinematic inversion of slider crank mechanisms produces ordinary a white work quick return mechanism

RESULT :

Slider crank mechanism is designed using ADAMS