

ME8711 – SIMULATION AND ANALYSIS LABORATORY

VII SEMESTER - MECHANICAL

Name of the Student :

Register Number :

Year / Semester / Section :

Batch :

Department of Mechanical Engineering



**CHENNAI
INSTITUTE OF TECHNOLOGY**

EXP. NUMBER	DATE OF THE EXPERIMENT	TITLE	DATE OF COMPLETION	SIGNATURE OF THE FACULTY
1		MATLAB BASICS, DEALING WITH MATRICES, GRAPHING- FUNCTIONS OF ONE VARIABLE AND TWO VARIABLES		
2		USE OF MATLAB TO SOLVE SIMPLE PROBLEMS IN VIBRATION		
3		MECHANISM SIMULATION USING MULTI BODY DYNAMIC SOFTWARE		
4		FORCE AND STRESS ANALYSIS USING LINK ELEMENTS IN TRUSSES, CABLES ETC.		
5		STRESS AND DEFLECTION ANALYSIS IN BEAMS WITH DIFFERENT SUPPORT CONDITIONS		
6		STRESS ANALYSIS OF FLAT PLATES AND SIMPLE SHELLS		
7		STRESS ANALYSIS OF AXI-SYMMETRIC COMPONENTS		
8		THERMAL STRESS AND HEAT TRANSFER ANALYSIS OF PLATE.		
9		THERMAL STRESS ANALYSIS OF CYLINDRICAL SHELLS.		
10		VIBRATION ANALYSIS OF SPRING-MASS SYSTEMS.		
11		MODEL ANALYSIS OF BEAMS.		
12		HARMONIC, TRANSIENT AND SPECTRUM ANALYSIS OF SIMPLE SYSTEMS		

CONTENT BEYOND THE SYLLABUS:

EXP. NUMBER	DATE OF THE EXPERIMENT	TITLE	DATE OF COMPLETION	SIGNATURE OF THE FACULTY
1)		STRESS ANALYSIS OF A BICYCLE SPANNER		
2)		2D-THERMAL STATIC ANALYSIS OF CHIMNEY		

OPEN ENDED EXPERIMENT:

EXP. NUMBER	DATE OF THE EXPERIMENT	TITLE	DATE OF COMPLETION	SIGNATURE OF THE FACULTY
1		VERIFY THE CALCULUS USING MATLAB		

DESIGN EXPERIMENT:

EXP. NUMBER	DATE OF THE EXPERIMENT	TITLE	DATE OF COMPLETION	SIGNATURE OF THE FACULTY
1		STEPPED BARS		

EX.NO:	INTRODUCTION TO MAT LAB

Aim :

To Study the capabilities of MatLab Software.

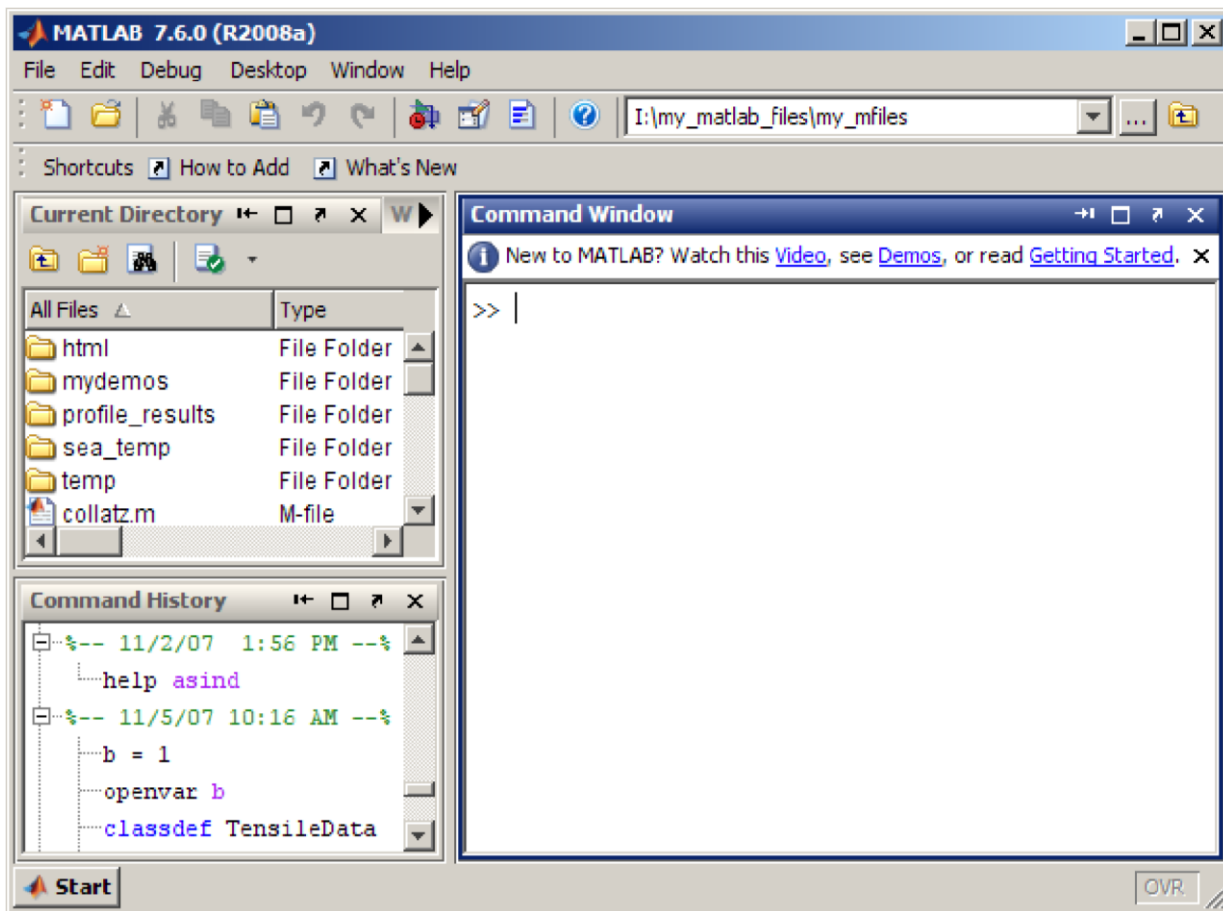
Introduction

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. Do not confuse the windows, icons, and menus in the tutorials 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

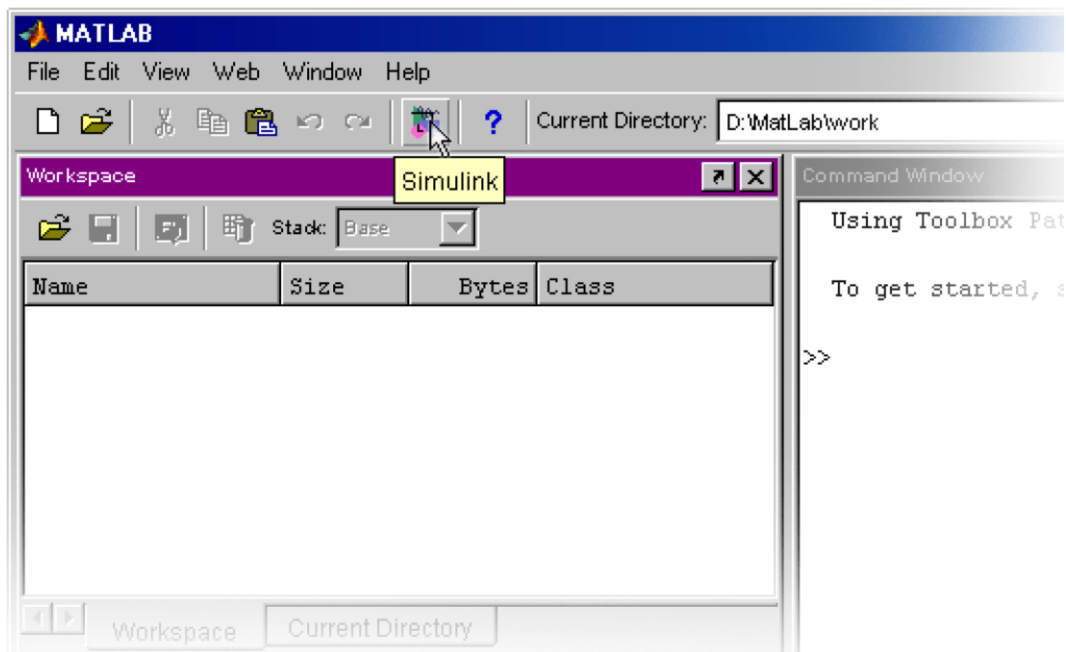
3. Basic Elements
4. Running Simulations
5. Building Systems

Starting Simulink

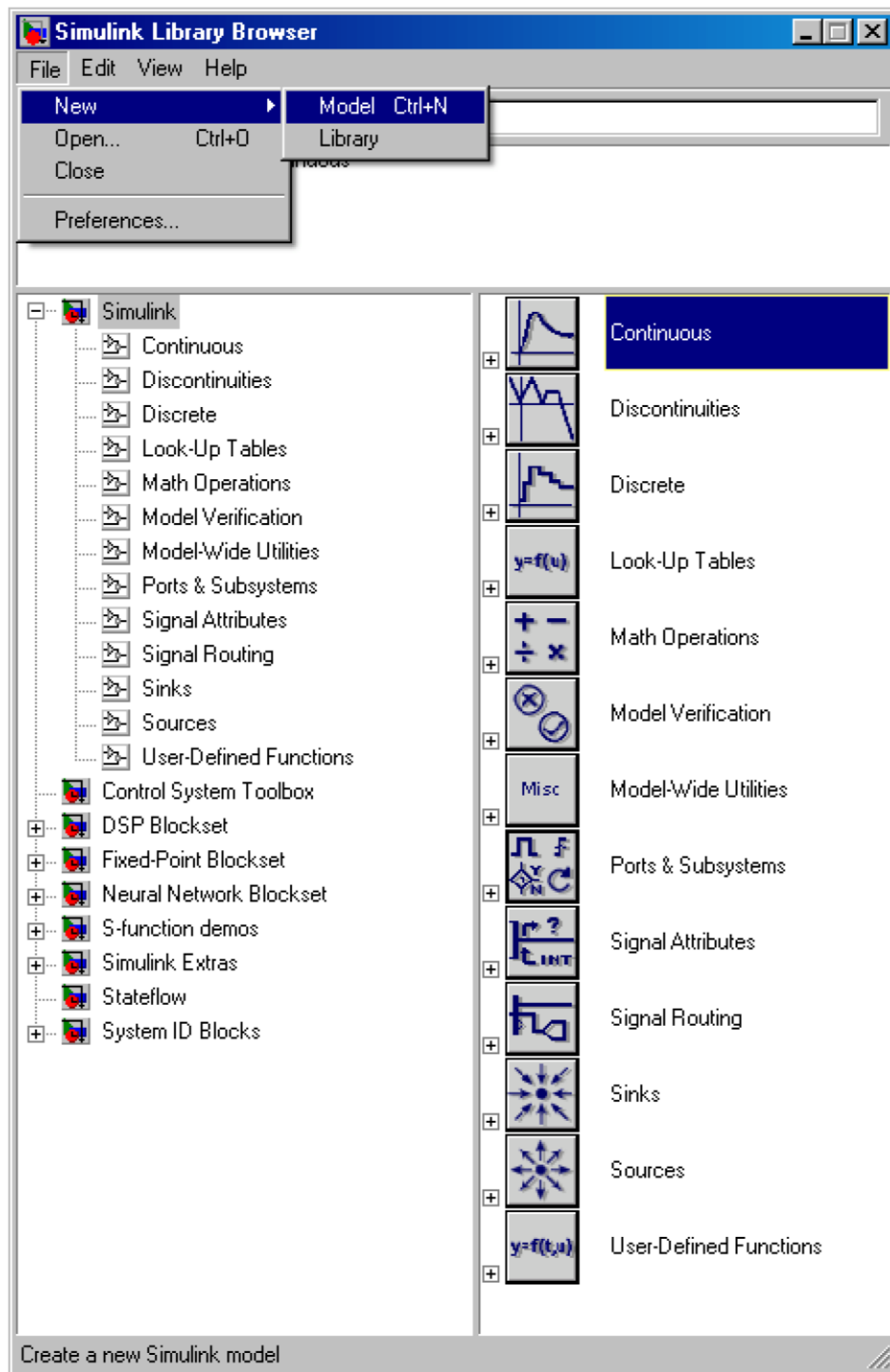
Simulink is started from the MATLAB command prompt by entering the following command:

```
>> Simulink
```

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

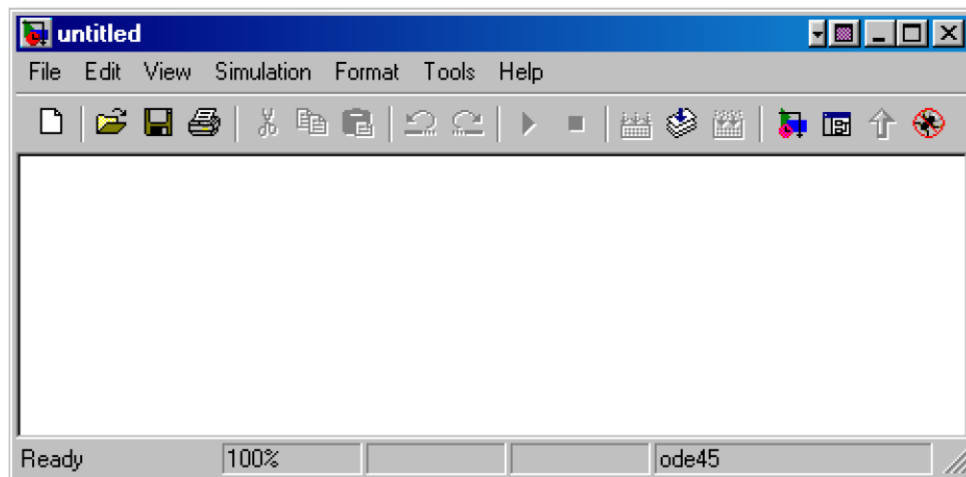


When it starts, Simulink brings up the Simulink Library browser.



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.



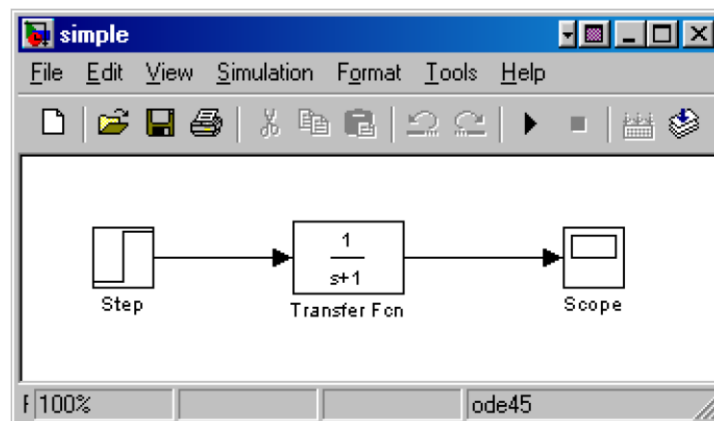
Model Files

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

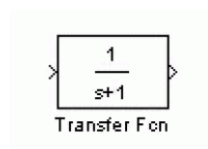
There are two major classes of items in Simulink: blocks and lines. Blocks are used to generate, modify, combine, output, and display signals. Lines are used to transfer signals from one block to another.

Blocks

There are several general classes of blocks:

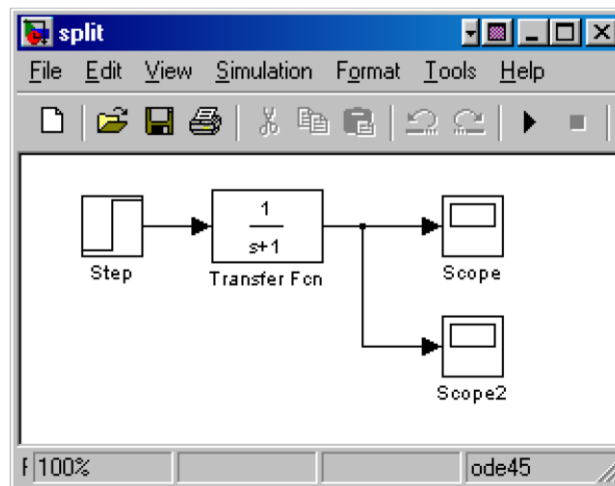
- Continuous
- Discontinuous
- Discrete
- Look-Up Tables
- Math Operations
- Model Verification
- Model-Wide Utilities
- Ports & Subsystems
- Signal Attributes
- Signal Routing
- Sinks: Used to output or display signals
- Sources: Used to generate various signals
- User-Defined Functions
- Discrete: 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

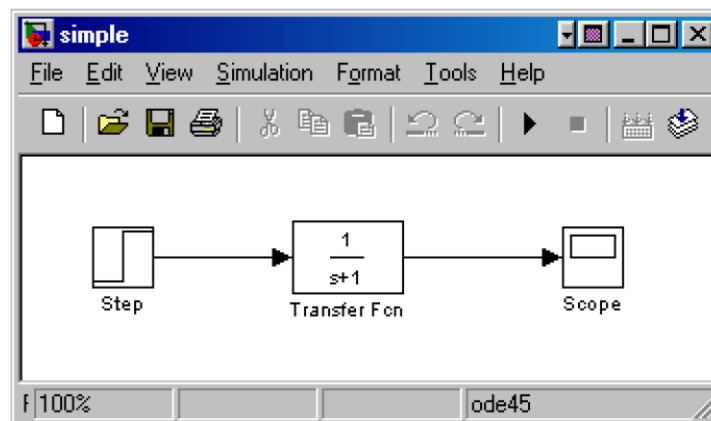
Lines transmit signals in the direction indicated by the arrow. Lines must always transmit signals from the output terminal of one block to the input terminal of another block. One exception to this is a line can tap off of another line, splitting the signal to each of two destination blocks, as shown 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.

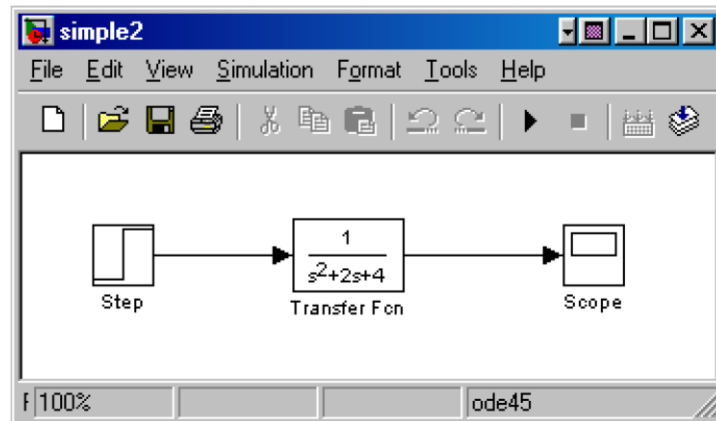
There are many more types of blocks available in Simulink, some of which will be discussed later. Right now, we will examine just the three we have used in the simple model.

Running Simulations

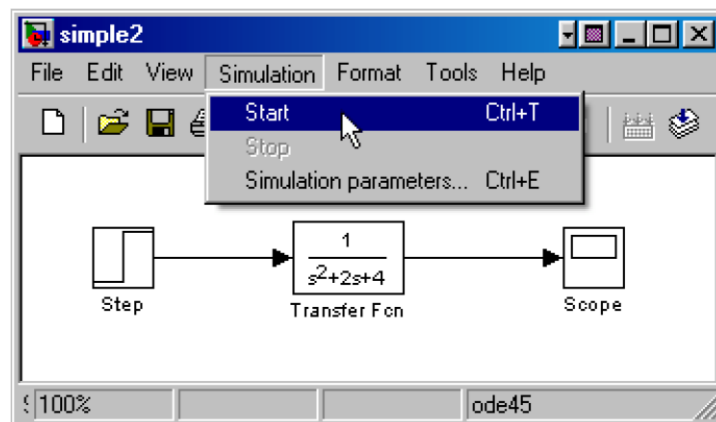
To run a simulation, we will work with the following model file:

simple2.mdl

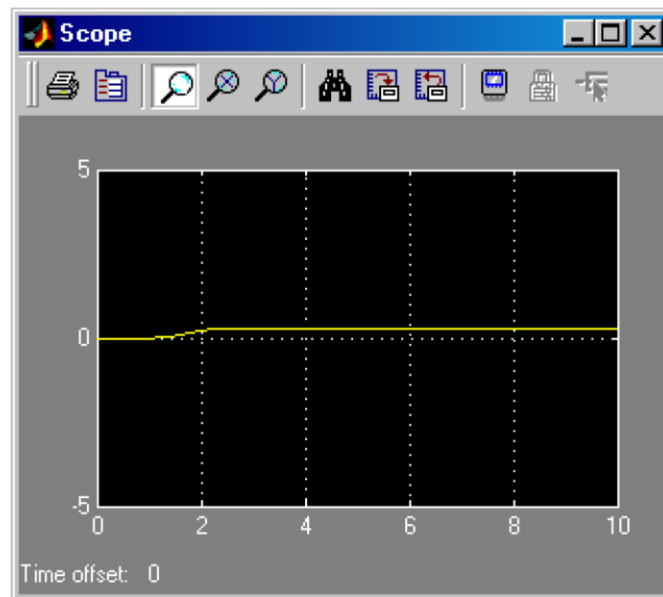
Download and open this file in Simulink following the previous instructions for this file. You should see the following model window.



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 doubleclicking 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

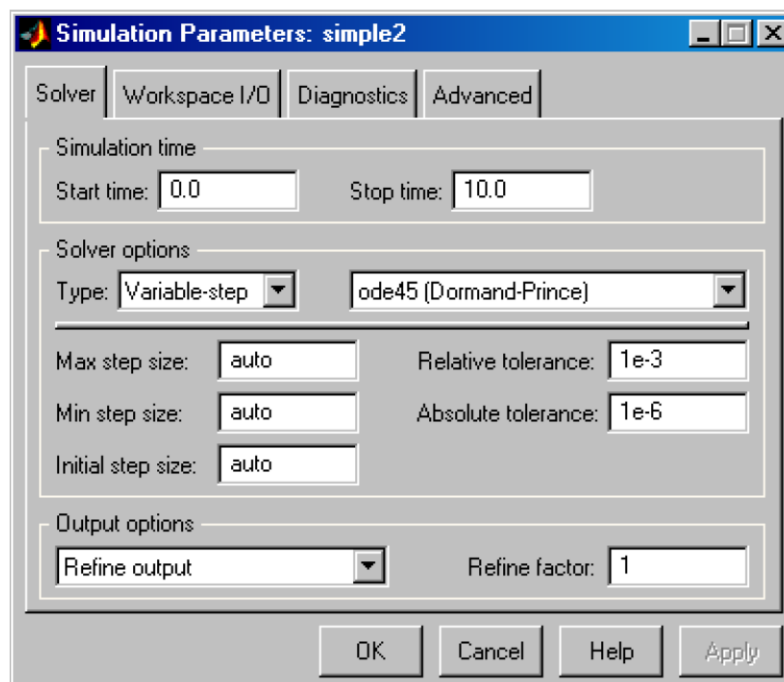
$[1 \ 20 \ 400]$

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.



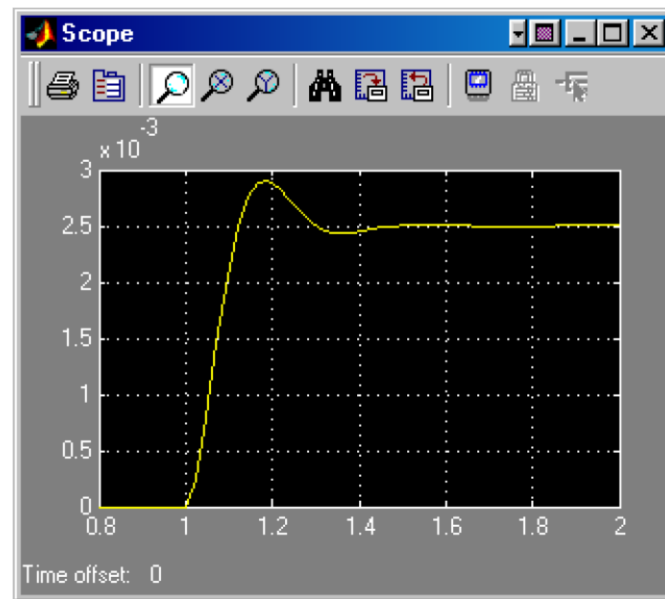
Notice that the autoscale button only changes the vertical axis. Since the new transfer function has a very fast response, it compressed into a very narrow part of the scope window. This is not really a problem with the scope, but with the simulation itself. Simulink simulated the system for a full ten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

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 dialog box.



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:	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 1000N/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 in kNm =

Power delivered in kw =

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

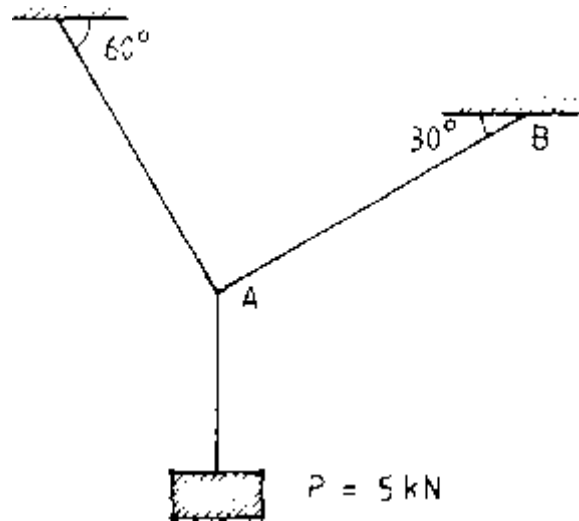
Maximum velocity =

Maximum acceleration =

RESULT

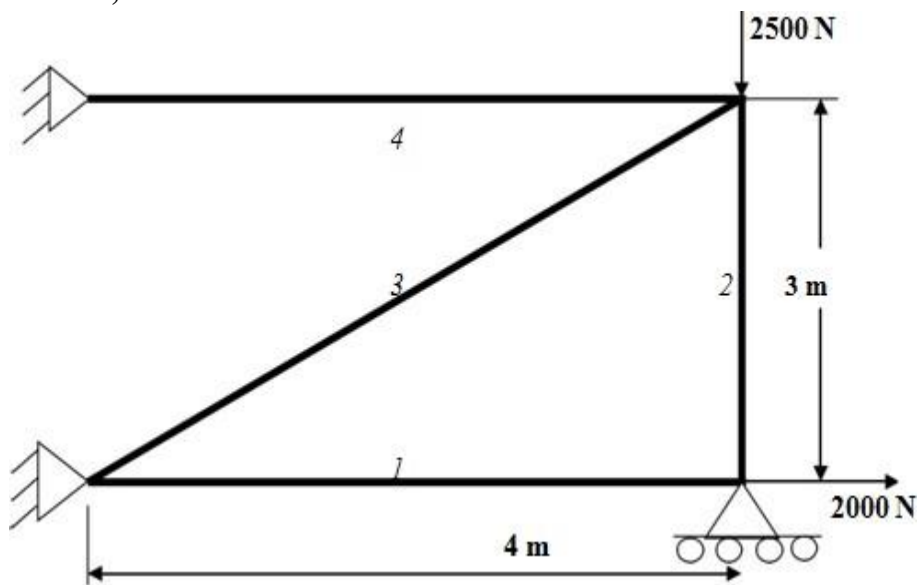
Thus simulation of an cam and follower has been done using Matlab.

CABLE



TRUSS

$E = 210 \text{ GPa}$, $A = 0.1 \text{ m}^2$



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:

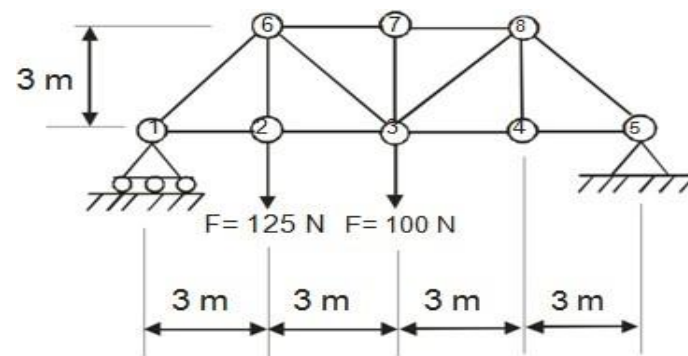
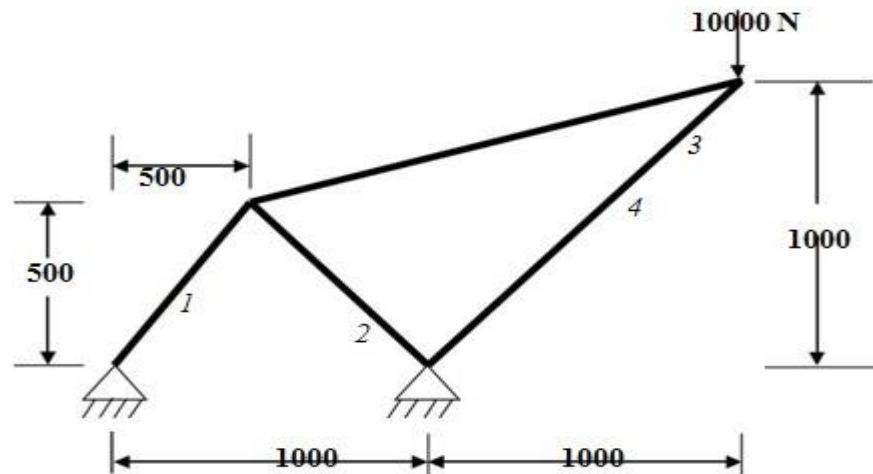
The three main steps to be involved are

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 – 210×10^9 – 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 (forth 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.

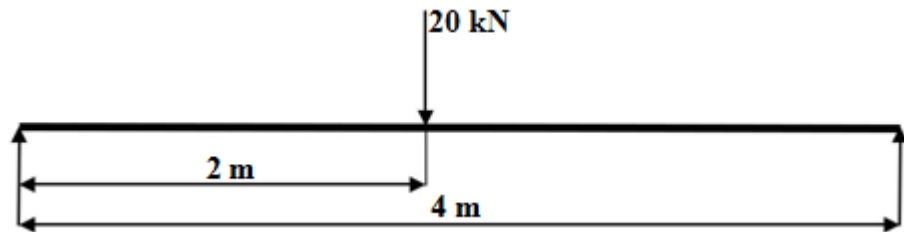
GENERAL POST PROCESSOR

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

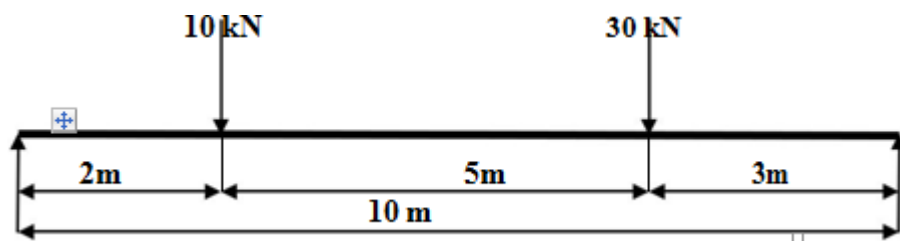
INFERENCE

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:	STRESS AND DEFLECTION ANALYSIS IN SIMPLY SUPPORTED BEAM

AIM:

To conduct the stress Stress and deflection analysis in Simply supported 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. Pre Processing
2. Solution
3. Post Processing

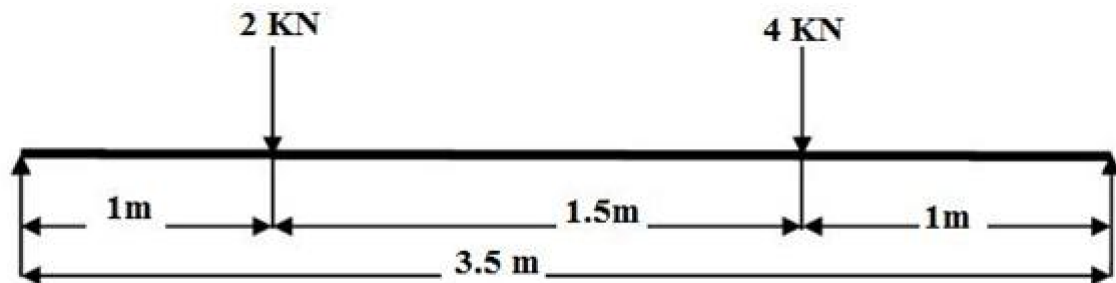
Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

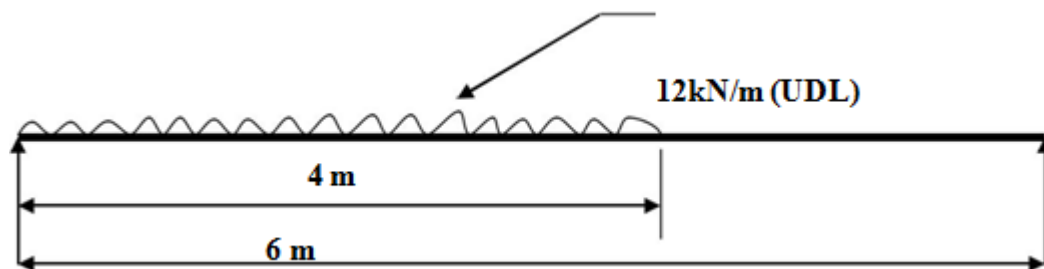
1. Preference - Structural- h-Method - Ok.
2. **Element type** – Add/Edit/Delete – Add – BEAM – 2D elastic 3 – ok- close.
3. **Real constants** – Add – ok – real constant set no – 1 – c/s area – 0.2×0.3 moment of inertia $0.2 \times 0.3^3 / 12$ – total beam height – 0.3 – ok.
4. **Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 210×10^9 – PRXY – 0.27 – ok – close.
5. **Modeling** – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 2 (x value w.r.t first node) – apply (second node is created) – 4 (x value w.r.t first node) – ok (third node is created). Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 apply – pick 2 & 3 – ok (elements are created through nodes).
6. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – -20000 (-ve value) – ok.

SOLUTION

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



Rectangular c/s area of 100 mm * 100mm, Young's modulus of 210 MPa, Poisson's ratio 0.27.



Rectangular c/s area of 100 mm * 100mm, Young's modulus of 210 MPa, Poisson's ratio 0.27.

GENERAL POST PROCESSOR

8. Plot Results – Deformed Shape – def+undeformed – ok.
9. Plot Results – Contour plot – Nodal solu – DOF solution – displacement vector sum – ok.
Element table – Define table – Add – ‘Results data item’ – By Sequence num – SMISC – SMISC, 2 – apply, By Sequence num – SMISC – SMISC, 8 – apply, By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 12 – ok – close.
10. 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).
11. **PlotCtrls** – Animate – Deformed results – DOF solution – USUM – ok.

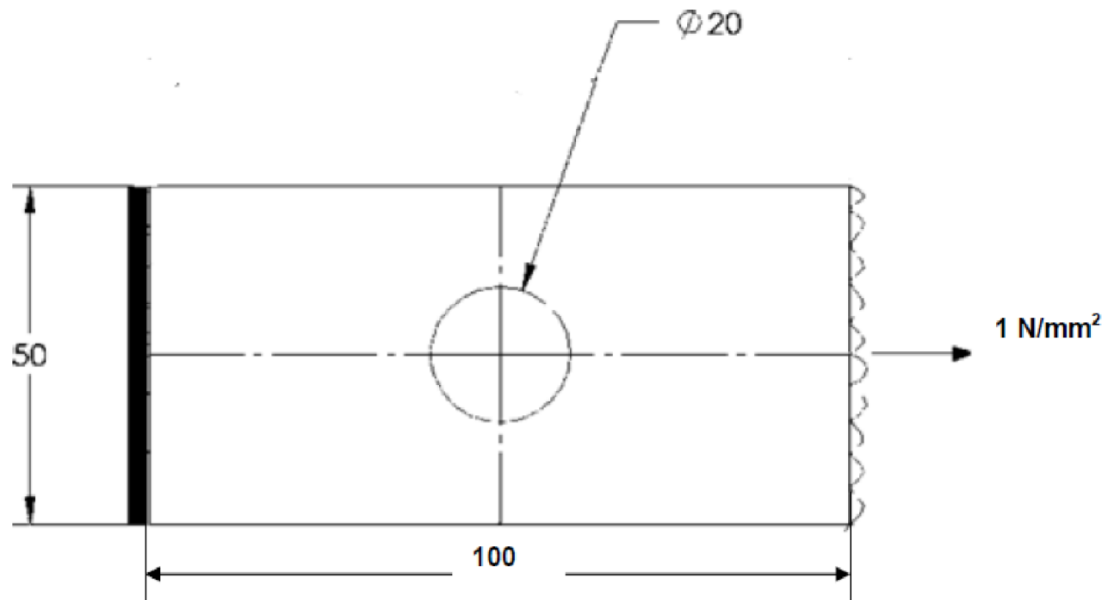
INFERENCE

RESULT

Thus the Stress and deflection analysis in Simply supported beam is done by using the ANSYS Software.

Young's Modulus = 200 GPa

Poisson's Ratio = 0.3



EX.NO:	STRESS ANALYSIS OF FLAT PLATES

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. Pre Processing
2. Solution
3. Post Processing

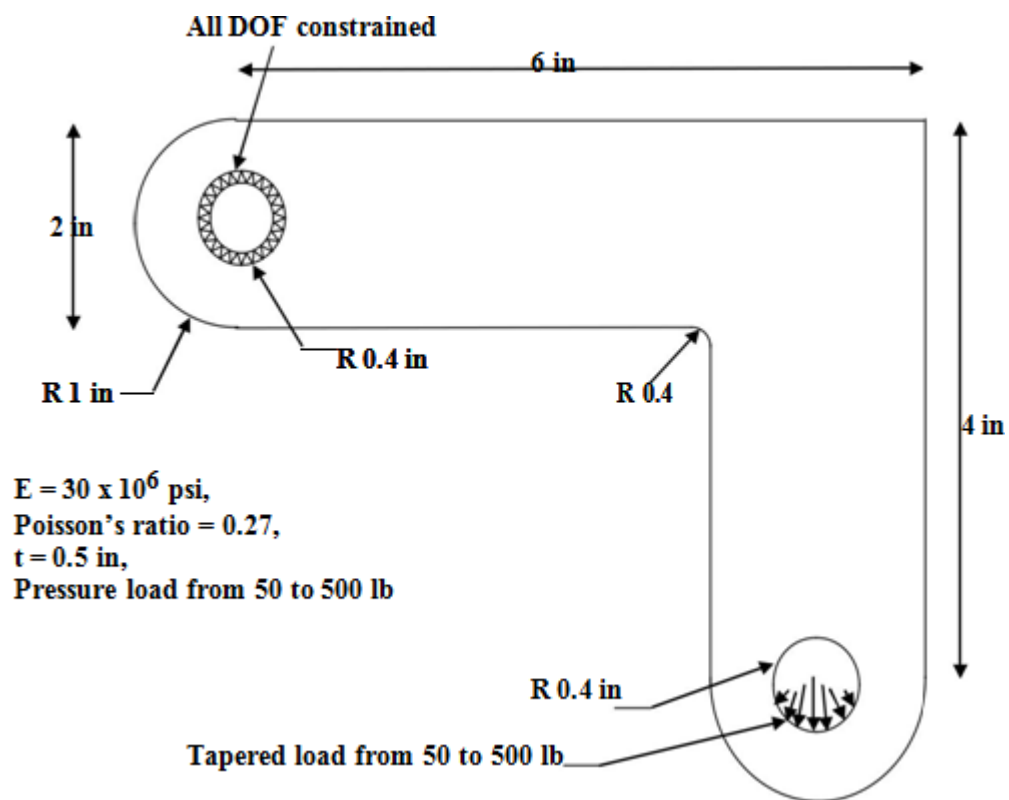
Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

1. Preference - 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 - EX 2e5, 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 length 2/3/4/5 – 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.



POST PROCESSING

1. 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.

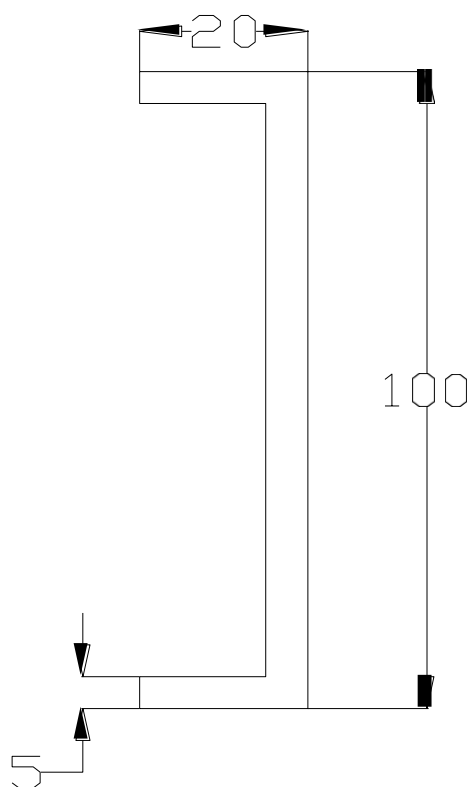
FOR REPORT GENERATION

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

INFERENCE

RESULT:

Thus the stress analysis in a plate is done by using the ANSYS Software.



EX.NO:	STRESS ANALYSIS OF AN AXI – SYMMETRIC COMPONENT

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. 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. Utility Menu > Change Job Name > Enter Job Name.
 Utility Menu > File > Change Title > Enter New Title.
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> By dimensions

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 > pick all.
8. Solution > Analysis Type > New Analysis > Static
9. Solution > Define loads > Apply .Structural > displacement > symmetry B.C > on lines. (Pick the two edges on the left at $X = 0$)
10. Utility menu > select > Entities > select all
11. Utility menu > select > Entities > by location > $Y = 50$ > ok.

(Select nodes and by location in the scroll down menus. Click Y coordinates and type 50 in to the input box.)
12. Solution > Define loads > Apply > Structural > Force/Moment > on key points > 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 click ok.
17. General postprocessor > List results > Nodal solution > stress > components SCOMP.
18. Utility menu > plot controls > style > Symmetry expansion > 2D Axisymmetric > $\frac{3}{4}$ expansion

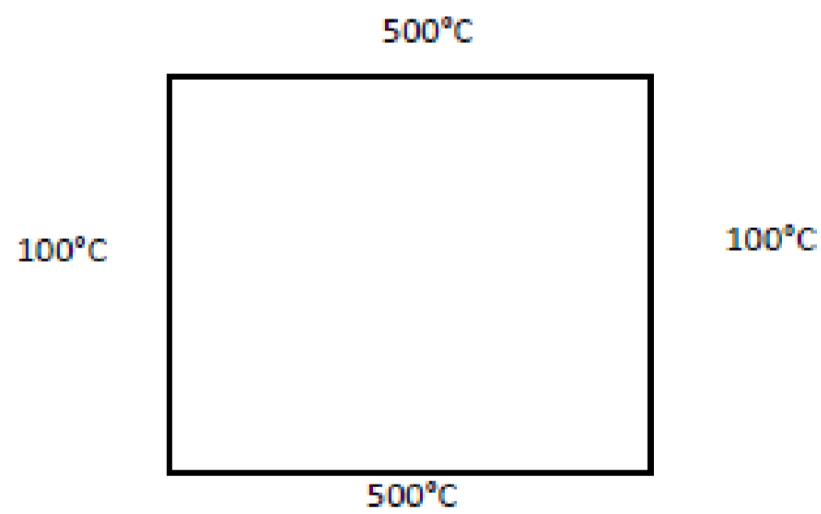
FOR REPORT GENERATION

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

INFERENCE

RESULT:

Thus the stress distribution of the axisymmetric component is done by using the ANSYS Software.



Thermal Conductivity of the material = 10 W/m. $^{\circ}\text{C}$

Dimension of the object = 2 m x 2 m

EX.NO:	THERMAL STRESS AND HEAT TRANSFER ANALYSIS OF PLATES

AIM:

To conduct the Thermal stress and heat transfer analysis of plates 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 (Version12.0/12.1)

PROCEDURE:

The three main steps to be involved are

1. Pre Processing
2. Solution
3. Post Processing

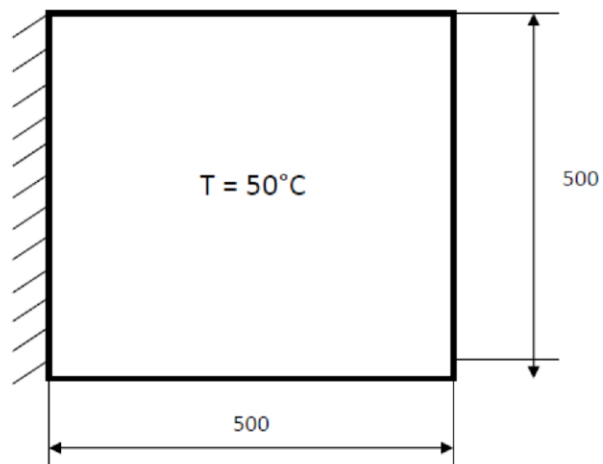
Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

PREPROCESSING

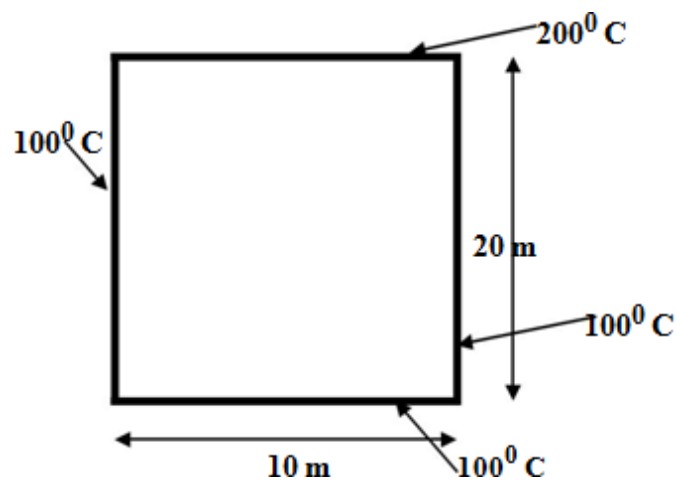
1. Preference – Thermal - h-Method - Ok.
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 55 – Ok – Close – Options – plane thickness – Ok.
3. Real constants - Add/Edit/Delete – Add – Ok – THK 0.5 – Ok – Close.
4. Material props - Material Models –Thermal – Conductivity – Isotropic – 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

7. 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



Young's modulus	=	200 GPa
Poisson's Ratio	=	0.3
Thermal expansion coefficient	=	12×10^{-6}



object – Ok –Temp 500 – Ok.

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

POST PROCESSING

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

FOR REPORT GENERATION

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

INFERENCE

RESULT:

Thus the Thermal stress and heat transfer analysis of plates is done by using ANSYS software.

EXP.NO:	THERMAL STRESS ANALYSIS OF CYLINDRICAL SHELLS

Aim:

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

System Configuration:

Ram: 8 GB

Processor: Core 2 Quad / Core 2 Duo

Operating system: Windows 7

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. Preference – Thermal - h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 42 – ok – Options – plane strs w/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
5. Modeling – Create – Areas - Rectangle – by 2 corners – Enter the coordinate values, height, width - ok
6. Meshing – Mesh tool – Areas, set – select the object – ok – Element edge length 10 - ok – Mesh tool- Tri, free - mesh – Select the object

Solution:

7. Solution – Define Loads – Apply – Structural – Displacement - On lines – Select the boundary

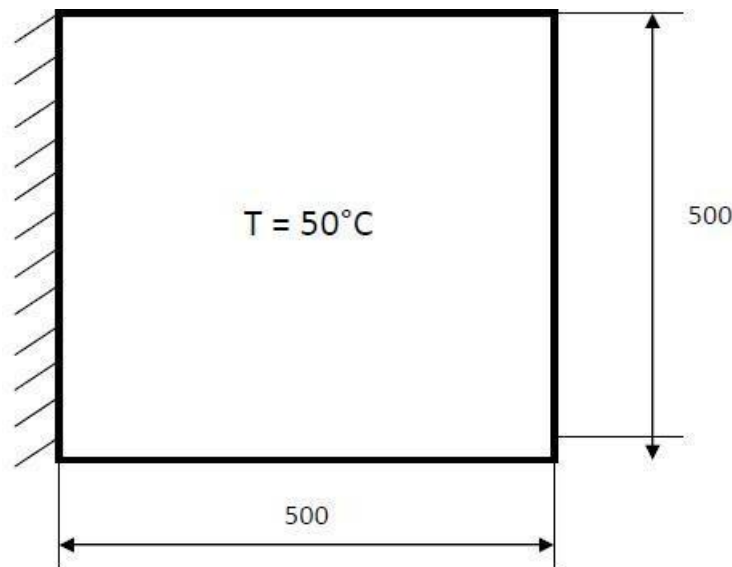
on the object –ok – Temperature – Uniform Temp – Enter the temp. Value 50 –ok.

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

10. File – Report Generator – Choose Append – ok – Image Capture – ok – close



Young's Modulus = 200 GPa

Poisson's ratio = 0.3

Thermal expansion coefficient = $12 \times 10^{-6} / ^\circ\text{C}$

Result:

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

EXP.NO:	VIBRATIONAL ANALYSIS OF SPRING MASS SYSTEM USING MATLAB

Aim:

To create a Simulink model for a mass attached to a spring with a linear damping force.

System Configuration:

Ram: 8 GB

Processor: Core 2 Quad / Core 2 Duo

Operating system: Windows 7

Software: MATLAB

Procedure:

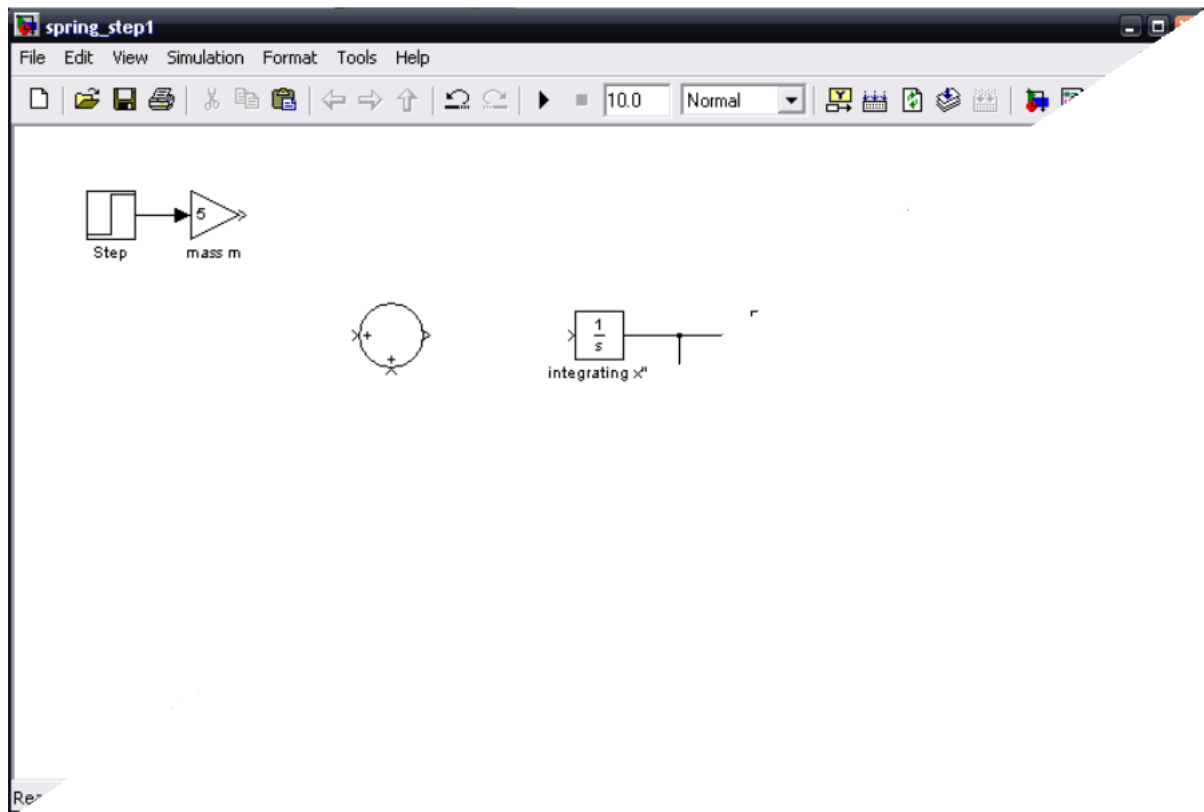
A mass on a spring with a velocity-dependent damping force and a time-dependent force acting upon it will behave according to the following equation:

$$m\ddot{x} + c\dot{x} + kx = f(t)$$

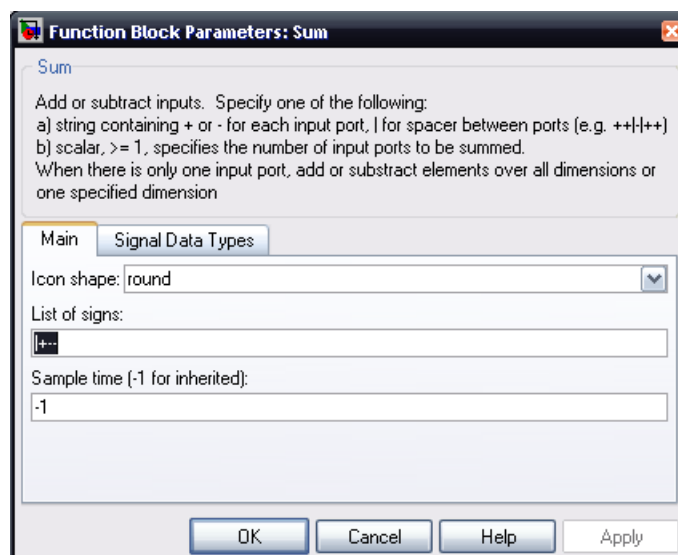
The model will be formed around this equation. In this equation, 'm' is the equivalent mass of the system; 'c' is the damping constant; and 'k' is the constant for the stiffness of the spring. First we want to rearrange the above equation so that it is in terms of acceleration; then we will integrate to get the expressions for velocity and position. Rearranging the equation to accomplish this, we get:

$$\ddot{x} = \frac{1}{m}(f(t) - c\dot{x} - kx)$$

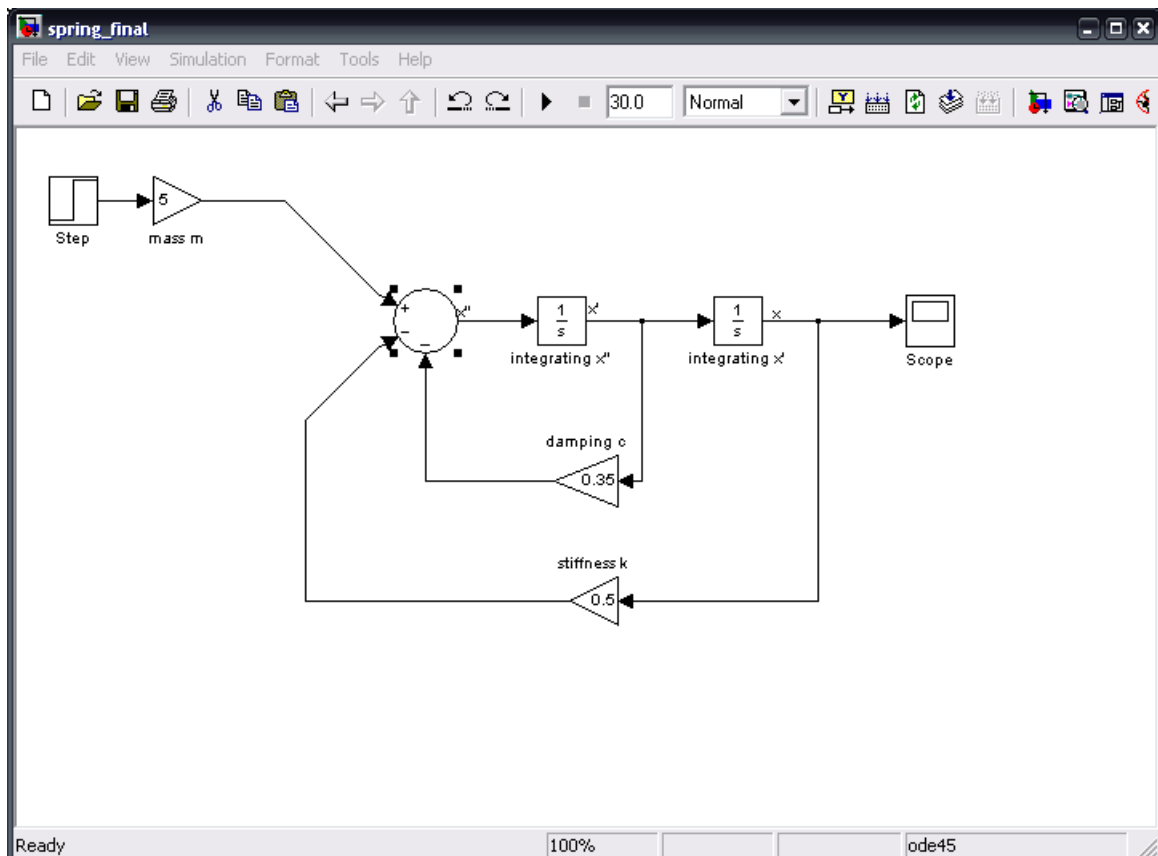
To build the model, we start with a 'step' block and a 'gain' block. The gain block represents the mass, which we will be equal to 5. We also know that we will need to integrate twice, that we will need to add these equations together, and that there are two more constants to consider. The damping constant 'c' will act on the velocity, that is, after the first integration, and the constant 'k' will act on the position, or after the second integration. Let $c = 0.35$ and $k = 0.5$. Laying all these block out to get an idea of how to put them together, we get:



By looking at the equation in terms of acceleration, it is clear that the damping term and spring term are summed negatively, while the mass term is still positive. To add places and change signs of terms being summed, double-click on the sum function block and edit the list of signs:



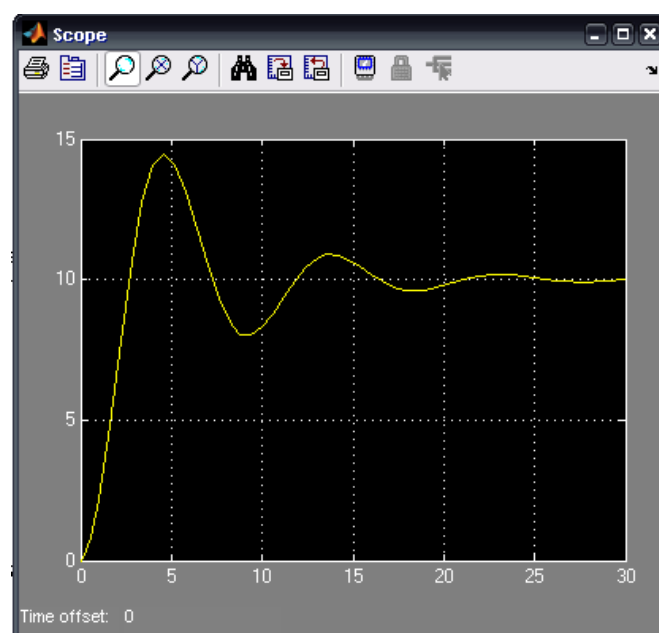
Once we have added places and corrected the signs for the sum block, we need only connect the lines to their appropriate places. To be able to see what is happening with this spring system, we add a 'scope' block and add it as follows:

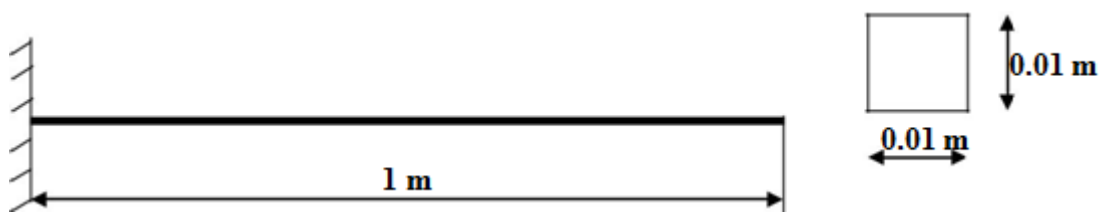


The values of 'm', 'c' and 'k' can be altered to test cases of under-damping, critical-damping and over-damping. To accurately use the scope, right-click the graph and select "Autoscale". The mdl-file can now be saved.

Result:

Then the simulation is verified for spring-mass system using MATLAB software, when the model is run for 30 iterations.





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

EX.NO:	MODAL ANALYSIS OF BEAMS

AIM:

To conduct the Modal Analysis of beams 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

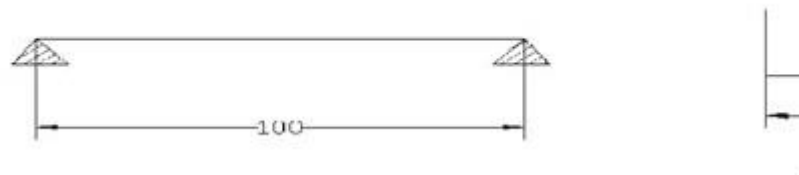
1. Pre Processing
2. Solution
3. Post Processing

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 – structural - h-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 – $0.01 \times 0.01 \times \frac{3}{12}$ – total beam height – 0.01 – ok.
4. **Material Properties** – material models – Structural – Linear – Elastic – Isotropic – EX – 200×10^9 – 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 – Manual Size – Lines – All Lines – element edge length – 0.1 – ok. Mesh – Lines – Pick All – ok.

$E_x = 210 \text{ Gpa} = 210000 \text{ N/mm}^2$, $\nu_{xy}=0.3$, $\text{Density}=7.85 * 10^{-9} \text{ N/mm}^3$



Simply supported beam

$E_x = 210 \text{ Gpa} = 210000 \text{ N/mm}^2$, $\nu_{xy}=0.3$, $\text{Density}=7.85 * 10^{-9} \text{ N/mm}^3$



Fixed beam

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.

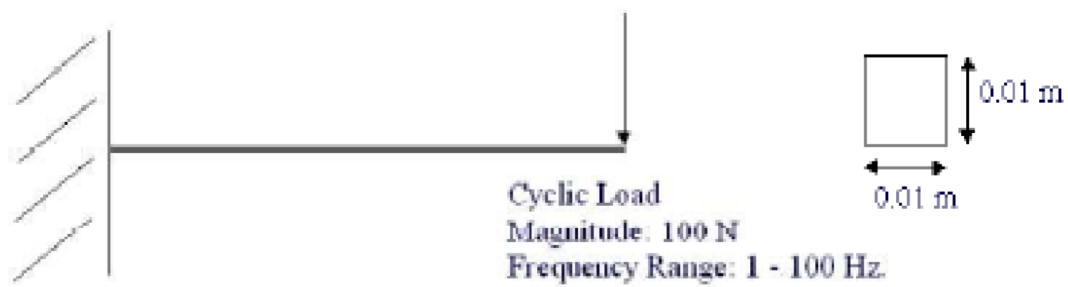
PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

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.



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

EX.NO:	HARMONIC, TRANSIENT AND SPECTRUM ANALYSIS OF 2D COMPONENT

AIM:

To conduct the harmonic, transient and spectrum 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 (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. 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 – EX 206e9, 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 Cntrl – manual size – lines – all lines – Enter the value of no of element divisions 25 – Ok.Mesh – Lines – Select the line – Ok.

SOLUTION

6. Solution - Analysis type – New analysis – Harmonic/transient/spectrum – 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/transient/spectrum freq range 1-100, Number of sub steps 100, Stepped – Ok.

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

GENERAL POST PROCESSING

10. Time Hist 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.

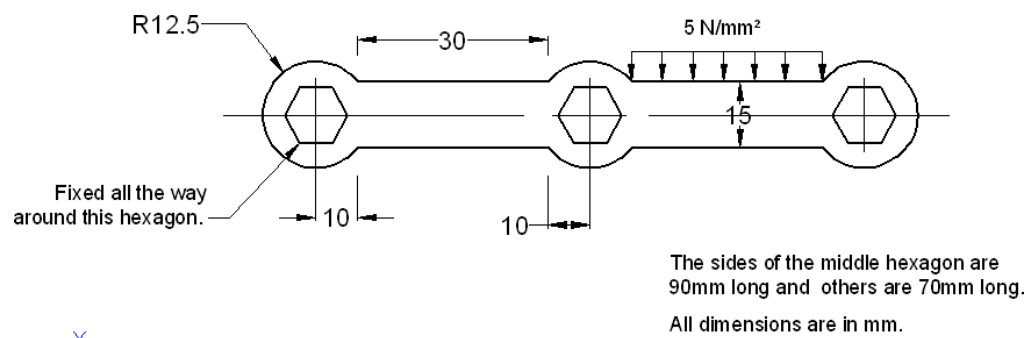
FOR REPORT GENERATION

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

INFERENCE

RESULT:

Thus the harmonic, transient and spectrum analysis of 2D component is done by using the ANSYS Software.



$$E = 200\text{GPa}$$

$$\text{Poisson's ratio } \nu = 0.32$$

EX.NO:	STRESS ANALYSIS OF A BICYCLE SPANNER

AIM:

To analyse the stress distribution of a bicycle spanner.

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 → structural → select 'h' method → ok
2. Title: Utility menu → file change title → spanner → ok
3. Utility menu → plot → replot.
4. Elements: Main menu → preprocessor → element type → add/edit/delete → add → structural mass → solid → Quad 8 node (Plane 82) → ok → options → select plane stress w/thk. in K3 option → ok → close.
5. Real constants: Main menu → preprocessor → Real constants → add/edit/delete → add → ok → enter the thickness as 3 → ok → close.
6. Material Properties: Main menu → preprocessor → Material props → material models → structural → linear → elastic → isotropic → Young's modulus ($E = 200 \times 10^3$) and Poissons ratio ($\nu = 0.32$) → ok.

7. Modeling: Main menu → preprocessor → modeling → create → areas → rectangle → by centre & corner → enter 0,0,1000,150 → ok
8. Main menu → preprocessor → modeling → create → areas → solid circle → enter -500, 0,125 → ok.
9. Main menu → preprocessor → modeling → create → areas → solid circle → enter 0,0,125 → ok
10. Main menu → preprocessor → modeling → create → areas → solid circle → enter 500,0,125 → ok
11. Main menu → preprocessor → modeling → operate → Boolean → add → areas → pick all → ok
12. Main menu → preprocessor → modeling → create → areas → polygon → hexagon → enter 0,0,90,blank space → apply → - 500,0,70, blank → apply → 500,0,70, blank → ok
13. Main menu → preprocessor → modeling → operate → Boolean → subtract → area → select → base area → ok → select subtracting area (all hexagons) → ok.
14. Utility menu → plot ctrls → hard copy → to file → jpeg → save to : spanner.jpg → ok
15. Main menu → preprocessor → Meshing → size ctrl → manual size → global → size → enter size as 5 → ok.
16. Main menu → preprocessor → meshing → mesh tool → mesh → areas → pick all → ok.
17. Utility menu → plot ctrls → hard copy → to file → jpeg → save to: spanner mesh → ok.
18. Boundary conditions and Loads: Utility menu → plot → keypoints → keypoints → ok.
19. Utility points → select → entites → keypoints → by num/pick → ok → select left hexagon keypoints → ok
20. Utility points → plot → keypoints → keypoints → ok
21. Main menu → solution → define loads → apply → structural → displacement → on keypoints → pick all → select all DOF → ok
22. Utility menu → select → everything Utility menu → plot → multiplot
Main menu → solution → define loads → apply → structural → pressure → on lines → select by cursor top right horizontal line → ok → enter pres = 5 → ok Utility menu → plot ctrls → hard

copy → to file → jpeg → save to : spanner load.jpg → ok

SOLUTION

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

POST PROCESSING

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

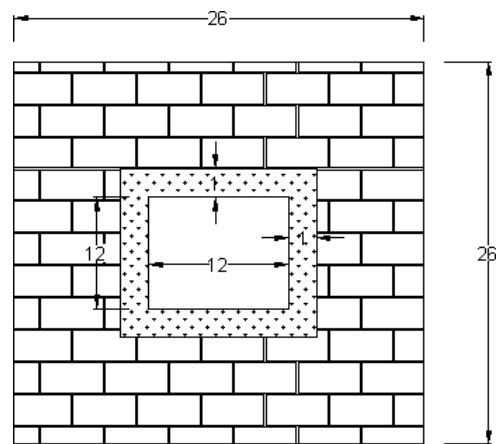
FOR REPORT GENERATION

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

INFERENCE

RESULT:

Thus the stress distribution of a bicycle spanner is done by using the ANSYS Software.



All dimensions are in 'm'

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.

EXP.NO:	VERIFICATION OF CALCULUS USING MATLAB

OBJECTIVES

- Verification of basic properties of limits for the functions $f(x) = (3x + 5)/(x - 3)$ and $g(x) = x^2 + 1$ as x tends to 4.
- Find the derivative of $(x+2)(x^2+3)$
- Calculate the area enclosed between the x-axis, and the curve $y=x^3 - 2x+5$ and the ordinates $x = 1$ and $x = 2$.

SOFTWARE REQUIRED

- MATLAB R2013a.
- Windows 7/XP SP2.

PROCEDURE

- Open MATLAB
- Open new M-file
- Type the program
- Save in current directory
- Compile and Run the program
- For the output see command window\ Figure window

PROGRAM

Properties of limits

```
f = (3*x + 5)/(x-3);
g = x^2 + 1;
l1 = limit(f, 4)
l2 = limit (g, 4)
lAdd = limit(f + g, 4)
lSub = limit(f - g, 4)
lMult = limit(f*g, 4)
lDiv = limit (f/g, 4)
```

Derivative

```
syms x
f=(x+2)*(x^2+3)
diff(f)
diff(ans)
diff(ans)
diff(ans)
```

Integration

```
syms x
f = x^3 - 2*x + 5;
a = int(f, 1, 2)
display('Area: '), disp(double(a));
```

OUTPUT

Properties of limits

$$l1 = 17$$

$$l2 = 17$$

$$lAdd = 34$$

$$lSub = 0$$

$$lMult = 289$$

$$lDiv = 1$$

Derivative

$$f = (x+2)*(x^2+3)$$

$$ans = x^2+3+2*(x+2)*x$$

$$ans = 6*x+4$$

$$ans = 6$$

$$ans = 0$$

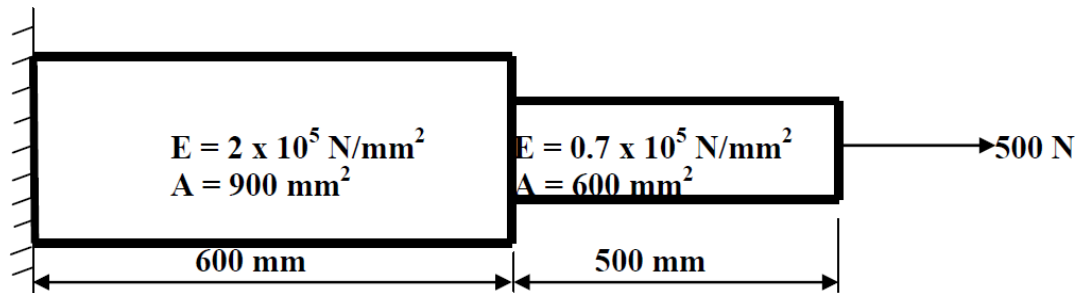
Integration

$$a = 23/4$$

$$Area: 5.7500$$

EXP.NO:	STEPPED BARS

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



Step 1: Ansys Utility Menu

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

Step 2: Ansys Main Menu – Preferences

select – STRUCTURAL - ok

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

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

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

Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 600 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 1100 (x value w.r.t first node) – ok (third node is created).

Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Create – Elements – Elem Attributes – Material number – 2 – Real constant set number – 2 – ok Auto numbered – Thru Nodes – pick 2 & 3 – ok (elements are created through nodes).

Step 5: Preprocessor

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

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

Step 6: Solution

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

Step 7: General Post Processor

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

Step 8: General Post Processor

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

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

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

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

Step 9: Plot Ctrls – Animate – Deformed shape – def+undeformed-ok