# SRM VALLIAMMAI ENGINEERING COLLEGE

**(An Autonomous Institution)**

**SRM NAGAR, KATTANKULATHUR – 603 203**

# ME8711- SIMULATION AND ANALYSIS LABORATORY MANUAL

**(Seventh Semester B.E/B.Tech Students for the Academic Year 2021-2022)**

**R2017**

## Prepared By


## Faculty Members

## Department of Mechanical Engineering


**(Private Circulation Only)**


**SRM VALLIAMMAI ENGINEERING COLLEGE, SRM NAGAR,**

**KATTANKULATHUR-603203**

# Department of Mechanical Engineering

**Instructions to the students**

**The following instructions must be followed by the students in their laboratory classes.**

1. Students are expected to be punctual to the lab classes. If they are late, they will be considered absent for that particular session.
2. Students should strictly maintain the dress code (i.e.) they have to wear the lab uniform with black shoes.
3. Students must bring their observation note, record note (completed with previous experiment) and the calculator to every lab class without fail.
4. Students are advised to come with full preparation for their lab sessions by
    a. Reading the detailed procedure of the experiment from the laboratory manual.
    b. Completion of observation note book (i.e.) Aim, Apparatus required, Formula (with description), least count calculation, diagrams and the tabular column should be written in the observation note before entering into the laboratory.
5. Data enter in the observation note book must be by pen only.
6. Students must get attestations immediately for their observed readings.
7. Students are advised to get their results evaluated in the observation note book on the same day of that experiment.
8. Class assessment marks for each experiment is based only on their performance in the laboratory.
9. Record note has to be completed then and there and get corrected when the students are coming or the next lab class.
10. Students must strictly maintain silence during lab classes.
11. If any of the students is absent for the lab class for genuine reasons, he/she will be permitted to do the experiment during the repetition class only.
12. Students are advised to perform their experiments under safety care.
13. If any student is found causing damage to the lab equipments, he/she shall replace the same with a new.

# Department of Mechanical Engineering

**VII Semester Mechanical Engineering**

## ME8711- SIMULATION AND ANALYSIS LABORATORY

## SYLLABUS

**Course Objective:** The main learning objective of this course is to prepare the students for:

1. Analyzing the force and stress in mechanical components.
2. Analyzing deflection in mechanical components.
3. Analyzing thermal stress of mechanical components.
4. Analyzing heat transfer in mechanical components..
5. Analyzing the vibration of mechanical components.

**LIST OF EXPERIMENTS**

1. Study of Basics in ANSYS
2. Stress analysis of a plate with a circular hole
3. Stress analysis of rectangular L bracket
4. Stress analysis of cantilever beam
5. Stress analysis of simply supported beam
6. Stress analysis of fixed beam
7. Stress analysis of an axi-symmetric component
8. Thermal stress analysis of a 2D component
9. Conductive heat transfer analysis of a 2D component
10. Convective heat transfer analysis of a 2D component
11. Mode frequency analysis of cantilever beam
12. Mode frequency analysis of simply supported beam
13. Harmonic analysis of a 2D component
14. Stress analysis of a truss
15. Introduction to MAT LAB
16. Simulation of Spring-mass system using MAT LAB
17. Simulation of cam and follower mechanism using MATLAB

**Course Outcomes:** Upon completion of this course, the students will be able to:

1. Find out the effect of force and impact of stress on the mechanical components.
2. Calculate the deflection occuring on the mechanical components.
3. Get a detailed understanding of the thermal stress creation and its mechanism of spreading in mechanical components.
4. Gain knowledge regarding the mechanism of heat transfer in mechanical components.
5. Find out the vibration effects on mechanical components.

# Department of Mechanical Engineering

### VII Semester Mechanical Engineering

## ME8711- SIMULATION AND ANALYSIS LABORATORY

## Course Information Sheet

**COURSE DESCRIPTION:**

ME8711 Simulation And Analysis Laboratory is the seventh semester practical subject covering stress, thermal, harmonic analysis of mechanical components. This laboratory designed for fourth year Mechanical students with two credits.

**PROGRAM SPECIFIC OUTCOMES (PSO):**

1. **PSO1-** Ability to analyze and solve problems related to mechanical engineering
2. **PSO2-** Can be able to apply thermal, fluid, design, electrical & electronics engineering principles to mechanical engineering applications.
3. **PSO3-** Would be able to apply the tools and techniques of quality management of manufacturing service processes.
4. **PSO4-** Ability to use the concept of process planning and cost estimation to manufacture different products.

**CONTRIBUTION OF COURSE TO MEETING THE REQUIREMENTS:**

| Mathematics and Basic Sciences | Engineering Topics | General Education | Engineering Design |
|---|---|---|---|
| ✔ | - | - | - |

**PROGRAM OUTCOMES (PO):**

| PO | Description | |
|---|---|---|
| A | Graduates will demonstrate basic knowledge in mathematics, science and engineering. | ✔ |
| B | Graduates will demonstrate the ability to design and conduct experiments, interpret and analyze data, and report results. | ✔ |
| C | Graduates will demonstrate the ability to design a mechanical system or a thermal | - |

| | | |
|---|---|---|
| | system or a process that meets desired specifications and requirements. | |
| D | Graduates will demonstrate the ability to function on engineering and science laboratory teams, as well as on multidisciplinary design teams. | ✔ |
| E | Graduates will demonstrate the ability to identify, formulate and solve mechanical engineering problems. | - |
| F | Graduates will demonstrate an understanding of their professional and ethical responsibilities. | - |
| G | Graduates will be able to communicate effectively in both verbal and written forms. | ✔ |
| H | Graduates will have the confidence to apply engineering solutions in global and societal contexts. | - |
| I | Graduates should be capable of self-education and clearly understand the value of lifelong learning. | ✔ |
| J | Graduates will be broadly educated and will have an understanding of the impact of engineering on society and demonstrate awareness of contemporary issues. | - |
| K | Graduates will be familiar with modern engineering software tools and equipment to analyze mechanical engineering problems. | ✔ |
| L | Competent enough to succeed in National level competitive examinations in the field of Mechanical Engineering. | - |
| M | Ability to be a successful Entrepreneur in the fields of Mechanical, Automobile, Mechatronics Engineering and Industrial Automation. | ✔ |

| **co** | **PO** | | | | | | | | | | | | **PSO** | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | **1** | **2** | **3** | **4** | **5** | **6** | **7** | **8** | **9** | **10** | **11** | **12** | **1** | **2** | **3** | **4** |
| **1** | 3 | 3 | | | 3 | | | 3 | | | | 3 | 3 | | | |
| **2** | 3 | 3 | | | 3 | | | 3 | | | | 3 | 3 | | | |
| **3** | 3 | 3 | | | 3 | | | 3 | | | | 3 | 3 | | | |
| **4** | 3 | 3 | | | 3 | | | 3 | | | | 3 | 3 | | | |
| **5** | 3 | 3 | | | 3 | | | 3 | | | | 3 | 3 | | | |

## ME 8711 - SIMULATION AND ANALYSIS LABORATORY
## CONTENTS

# EXP. NO.01 STUDY OF BASICS IN ANSYS

## 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:

1. Build the model.
2. Apply loads and obtain the solution.
3. Review the results.

## Build the model:

1. Defining the Job name:

The job name is a name that identifies the ANSYS job. When you define a job name for an analysis, the job name 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 job name 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:

| | |
|---|---|
| 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.

4.Defining Element Real Constants:

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 Material Properties:

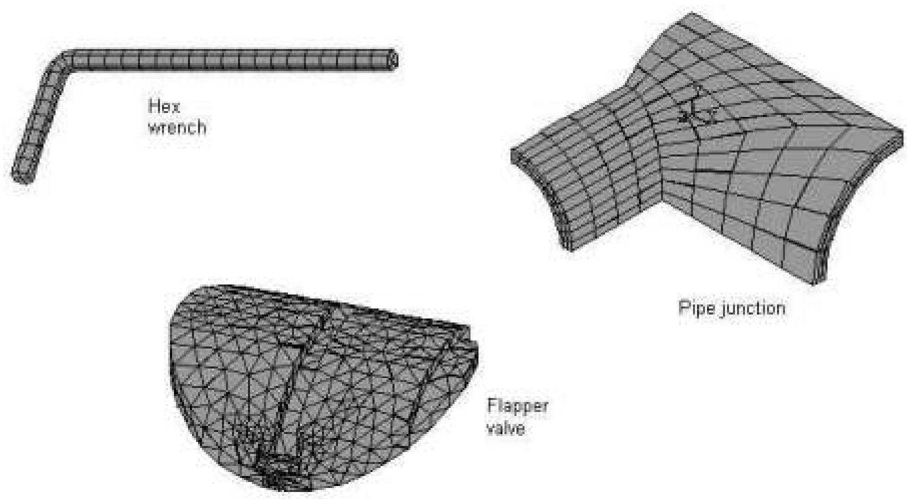
Most element types require material properties. Depending on the application, material properties can be linear (see linear material properties) or nonlinear (see nonlinear material properties).

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 model geometry:

Once you have defined material properties, the next step in an analysis is generating a finite element model i.e., 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.




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 PREP7 preprocessor.

1. Defining the analysis types and analysis options

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 sub structuring.

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 sub steps, 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

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 results file.

## 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 Post proc), valid only at the beginning 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 path operations.

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>Time Hist Post pro), valid only at the beginning level. You can obtain graph plots of results data versus time (or frequency) and tabular listings. Other POST26 capabilities include arithmetic calculations and complex algebra.

Specific Capabilities of ANSYS Structural Analysis:

Structural analysis is probably the most the common application of the finite element method such as piston, machine parts and tools.

Static Analysis:

It is the used to determine displacement, stress etc. under static loading conditions. ANSYS can compute linear and non-linear types (e.g. the large strain hyper elasticity and creep problems).

Transient Dynamic Analysis:

It is used to determine the response of a structure to time varying loads.

Buckling Analysis:

It is used to calculate buckling load and to determine the shape of the component after applying the buckling load. Both linear buckling and non - linear buckling analysis are possible.

Thermal Analysis:

The steady state analysis of any solid under thermal boundary conditions calculates the effect of steady thermal load on a system (or) component that includes the following.

1. Convection.
2. Radiation.
3. Heat flow rates.
4. Heat fluxes.
5. Heat generation rates.
6. Constant temperature boundaries.

Fluid Flow:

The ANSYS CFD offers comprehensive tools for analysis of two-dimensional and three dimensional fluid flow fields.

Magnetic:

Magnetic analysis is done using ANSYS / Electromagnetic program. It can calculate the magnetic field in device such as power generators, electric motor etc. Interest in magnetic analysis is finding magnetic flux, magnetic density, power loss and magnetic forces.

Acoustic / Vibrations:

Ansys is capable of modeling and analyzing vibration system. Acoustic is the study of the generation, absorption and reflection of pressure waves in a fluid application.

Few examples of acoustic applications are

1. Design of concert house, where an even distribution of sound pressure is possible
2. Noise cancellation in automobile
3. Underground water acoustics
4. Noise minimization in machine shop
5. Geophysical exploration

Coupled fields:

A coupled field analysis is an analysis that takes into account the interaction between two (or) more fields of engineering analysis. Pressure vessels, Induction heating and micro electro mechanical systems are few examples.

# EXP. NO.02 STRESS ANALYSIS OF A PLATE WITH A CIRCULAR HOLE

## Aim:

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

## System configuration:

| | |
|---|---|
| Ram | : 8 GB |
| Processor | : Core 2 Quad / Core 2 Duo |
| Operating system | : Windows 7 |
| 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 – Select Circle – Next –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 theobject –ok

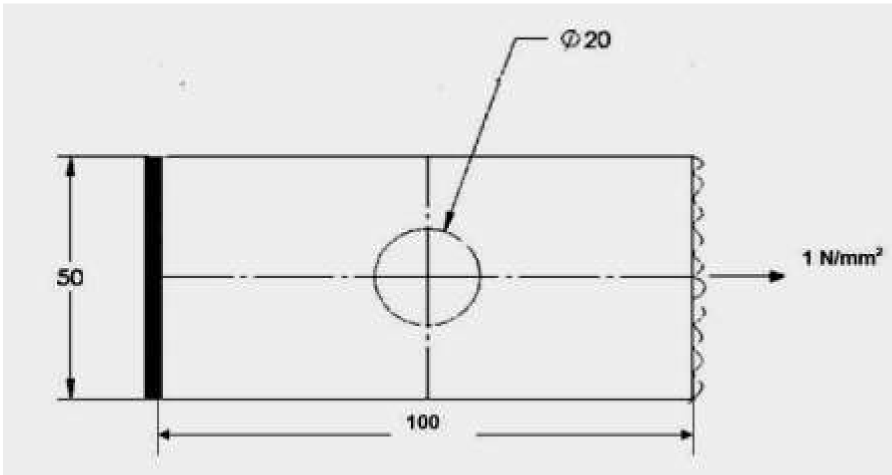
## Solution:

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 = 1N/mm – ok

8. Solve – Current LS – ok– Solution is done – close

## Post processing:

9. General post proc - Plot Result - Contour plot - Nodal Solution – Stress –Von mises stress – ok
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
12. File – Report Generator – Choose Append – ok – Image Capture – ok - close




Young"s Modulus : $2\times10^5$ N/mm $^2$

Poisson"s ratio : 0.3

## Result:

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

## Outcome:

From this experiment, students will be able to analyse a rectangular plate with a circular hole using the ANSYS software for structural applications.

## Application:

1. Air Craft Industry
2. Automobile Industry

# EXP. NO.03 STRESS ANALYSIS OF A RECTANGULAR L - BRACKET

## Aim:

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

## System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
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 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 - EX 2e5, 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

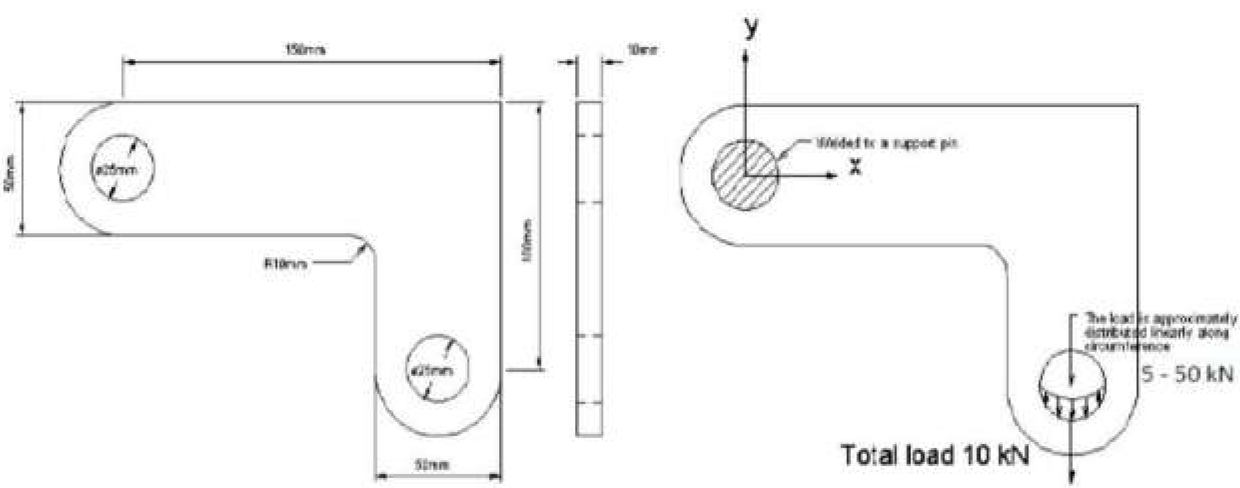
6. Meshing - Mesh Tool – Area – Set - Select the object – ok - Element edge length 2/3/4/5 – ok - Mesh Tool - Select TRI or QUAD - Free/Mapped – Mesh - Select the object - ok

## Solution:

7. Solution – Define Loads – Apply – Structural – Displacement - On lines - Select the boundary where is goingto be arrested – ok - All DOF - ok.Pressure - On lines - Select the load applying area – ok - Load PRES valve = -10000 N (- Sign indicates thedirection 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
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
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.

## Outcome:

From this experiment, students will be able to analyse a rectangular L section bracket using the ANSYS software for structural applications.

## Application:

1. Structural Application
2. Military Application

# EXP. NO.04 STRESS ANALYSIS OF CANTILEVER BEAM

## Aim:

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

## System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
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:

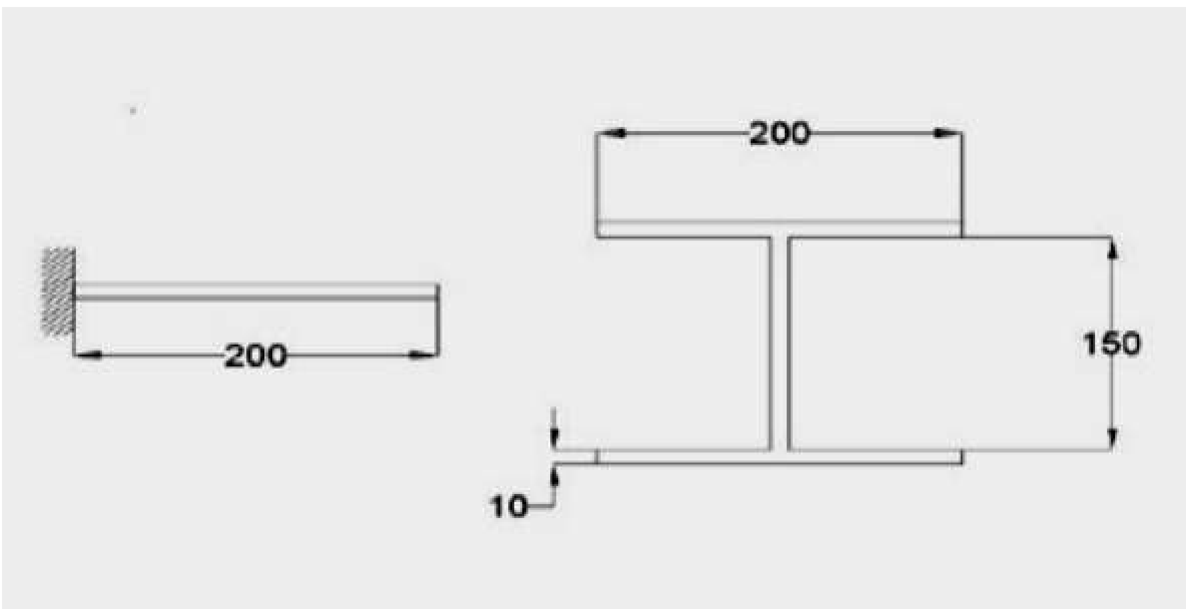
1. Preference - Structural- h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – ok – Options – ok - close
3. Sections – beam – Common sections – Select the correct section of the beam and input the of “w1, w2,w3” and “t1, t2, t3” – Preview – Note down the values 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 - EX 2e5, PRXY 0.3 - ok
6. Modeling – Create – Key points – In active CS – Enter the values of CS of each key points – Apply – ok. Lines – Lines – Straight line – Pick the all points – ok.
7. Meshing – Mesh attributes – All lines – ok. Meshing – Size cntrls – Manual size –Lines – All lines – Enter the value of element edge length [or] Number of element divisions – ok. Mesh tool – Mesh – Pick all

## Solution:

8. 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
9. Solve – Current LS – ok – Solution is done – close

## Post processing:

10. 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 ElementRes – Node I SMIS 6, Node J SMISC, 12 – ok. Contour plot– Line element Res – Node I LS 2, Node J LS 3 – ok
11. 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 a cantilever beam is done by using the ANSYS Software.

## Outcome:

From this experiment, students will be able to analyse a cantilever beam using the ANSYS software.

Application:

1. Structural Application
2. Construction Application

# Expt.No.05 STRESS ANALYSIS OF SIMPLY SUPPORTED BEAM

## Aim:

To conduct the stress analysis in a simply supported beam using ANSYS software

## System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
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 - beam 2D elastic 3 –Options – ok– close – real constant- Add/Edit/Delete- Add- area = 100, $I_{zz}$ = 833.33 & height =10- ok
3. Preprocessor – Material Properties – Material Model – Structural – Linear – Elastic – Isotropic – EX 2e5, PRXY0.3 – ok
4. Preprocessor – Modeling – create – nodes – inactive CS

   Node 1 X=0 Y=0

   Node 2 X= 25 Y=0

   Node 3 X= 50 Y=0

   Node 4 X= 75 Y=0

   Node 5 X= 100 Y=0
5. List - nodes - coordinate only -ok
6. Preprocessor- modeling- create- elements- Auto numbered through nodes- select

Node 1 & 2

Node 2 & 3

Node 3 & 4

Node 4 & 5

Node 5 & 6 - ok

## Solution:

7. Solution - define loads- apply- structural - displacement - on nodes - select node 1 & node 5 - apply - UY - displacement = 0 -ok
8. Solution - Force/moment - on nodes - node 3 - apply - FY = -100 -ok
9. Solution - solve - current LS -ok

## Post Processing:

10. General post processor - plot result - deform shape - Deformed + Undeformed-ok
11. General post processor - element table - define table - add - user table for item

    Smax I > by sequence num> NMISC 1 > apply

    Smax J > by sequence num> NMISC 3 >apply

    Smin I > by sequence num> NMISC 2 > apply

    Smin J > by sequence num> NMISC 4 >ok

12. Plot result - line element result -Smax I-Smax J - first result -Evaluate table data –
    Smax I, Smax J, Smin I, Smin J -ok
13. General postprocessor - list result - nodal solution - DOF solution - UY-displacement result
14. General postprocessor - contour plot - line element res. –ok

## Result:

Thus the stress analysis of a simply supported beam is done by using the ANSYS Software.

## Outcome:

From this experiment, students will be able to analyse a simply supported beam using the ANSYS software.

Application:

1. Structural Application
2. Construction Application

# EXP. NO.06 STRESS ANALYSIS OF FIXED BEAM

## Aim:

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

## System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
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 - beam 2D elastic 3 –Options – ok – close – real constant- Add/Edit/Delete- Add- area = 100, $I_{zz}$ = 833.33 & height =10- ok
3. Preprocessor – Material Properties – Material Model – Structural – Linear – Elastic – Isotropic – EX 2e5, PRXY0.3 – ok
4. Preprocessor – Modeling – create – nodes – inactive CS

   Node 1 X=0 Y=0

   Node 2 X= 25 Y=0

   Node 3 X= 50 Y=0

   Node 4 X= 75 Y=0

   Node 5 X= 100 Y=0
5. List - nodes - coordinate only -ok
6. Preprocessor- modeling- create- elements- Auto numbered through nodes- select

Node 1 & 2

Node 2 & 3

Node 3 & 4

Node 4 & 5

Node 5 & 6 -ok

## Solution:

7. Solution - define loads- apply- structural - displacement - on nodes - select node 1 & node 5 - apply - UY - displacement = 0 -ok
8. Solution - Force/moment - on nodes - node 3 - apply - FY = -100 -ok
9. Solution - solve - current LS -ok

## Post processing:

10. General post processor - plot result - deform shape - Deformed + Undeformed -ok
11. General post processor - element table - define table - add - user table for item

    Smax I > by sequence num> NMISC 1 > apply

    Smax J > by sequence num> NMISC 3 >apply

    Smin I > by sequence num> NMISC 2 > apply

    Smin J > by sequence num> NMISC 4 >ok
12. Plot result - line element result - Smax I- Smax J - first result -Evaluate table data –Smax I, Smax J, Smin I, Smin J -ok

    General postprocessor - list result - nodal solution - DOF solution - UY - displacement result
13. General postprocessor - contour plot - line element res. -ok

## Result:

Thus the stress analysis of a fixed beam is done by using the ANSYS Software.

## Outcome:

From this experiment, students will be able to analyse a fixed beam using the ANSYS software for structural applications.

Application:

1. Structural Application
2. Construction Application

# EXP. NO.07 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 | : 8 GB |
| Processor | : Core 2 Quad / Core 2 Duo |
| Operating system | : Windows 7 |
| Software | : ANSYS (Version12.0/12.1) |

## Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

## Preprocessing:

1. Utility Menu - Change Job Name - Enter Job Name. Utility Menu - File - Change Title - Enter New Title
2. Preference - Structural –h method - ok
3. Preprocessor - Element type - Add/Edit/ delete - solid 8node 183 – options-axisymmetric
4. Preprocessor - Material Prop - 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.

## Solution:

8. Solution - Analysis Type-New Analysis-Static
9. Solution - Define loads - Apply .Structural - displacement - symmetry BC - on lines.

(Pick the two edger on the left at X = 0)

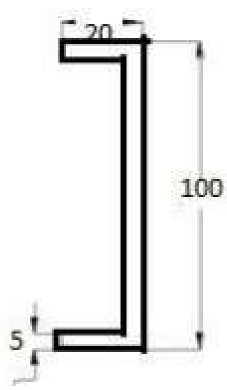
10. Utility menu - select - Entities - select all
11. Utility menu - select - Entities - by location - Y = 50 -ok.
12. Solution - Define loads - Apply - Structural - Force/Moment - on key points - FY =100 - Pick the

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, asshown below and click ok

## Post processing:

17. General postprocessor - List results - Nodal solution - stress - components SCOMP
18. Utility menu - plot controls - style - Symmetry expansion - 2D Axisymmetric - ¾ expansion




Young‟s Modulus : 200 GPa

Poisson‟s ratio : 0.3

## Result:

Thus the stress analysis of an axi-symmetric component done by using the ANSYS software.

## Outcome:

From this experiment, students will be able to analyse an axi-symmetric component using the ANSYS software.

## Application:

1. Structural Application
2. Automobile Application

# EXP. NO.08 THERMAL STRESS ANALYSIS OF A 2DCOMPONENT

## 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 (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
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:

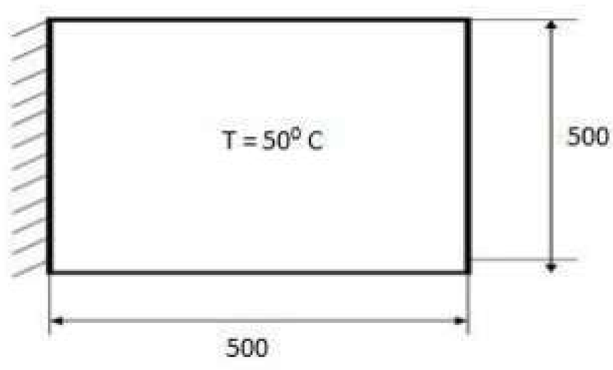
1. Preference – Thermal - h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 42 – ok – Options – plane strsw/thk – ok – Close
3. Real constants - Add/Edit/Delete – Add – ok – THK 100 – ok – Close
4. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 2e5,PRXY 0.3 – ok –Thermal expansion – Secant coefficient – Isotropic – ALPX 12e-6 – ok
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 = $12 \times 10^{-6}$ °/ C

## Result:

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

## Outcome:

From this experiment, students will be able to analyse the thermal stress of a 2D component using the ANSYS software for heat transfer applications.

## Application:

1. Structural Application
2. Heat Exchange Application

# EXP. NO.09 CONDUCTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

## Aim:

To conduct the conductive heat transfer 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 (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

## Preprocessing:

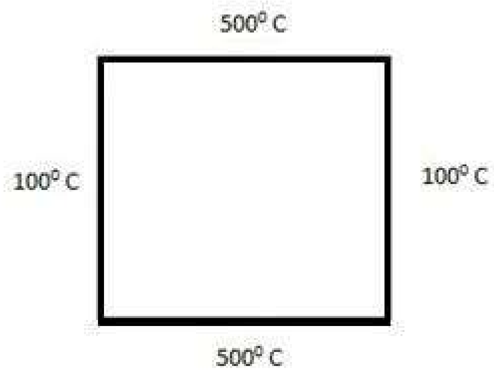
1. Preference – Thermal - h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 55 – ok – close – options – plane thickness – ok
3. Real constants - Add/Edit/Delete – Add – ok – THK 0.5 – ok – Close
4. Material props - Material Models –Thermal – Conductivity – Isotropic – 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 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
10. File–Report Generator–Choose Append–ok–Image Capture–ok - close




Thermal Conductivity of the material = 10 W/m ℃

Dimension of the object = 2m × 2m

## Result:

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

## Outcome:

From this experiment, students will be able to analyse the conductive heat transfer of a 2D component using the ANSYS software.

## Application:

1. Boiler Application
2. Heat Transfer Application

# EXP. NO.10 CONVECTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

## Aim:

To conduct the convective heat transfer 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 (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

## 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
4. 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.
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:

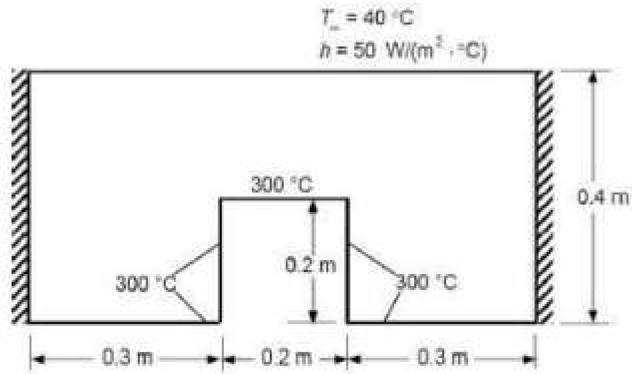
7. Solution – Define Loads – Apply – Thermal – Temperature - On lines – Select the lines–ok – Temp. Value 300 – ok – Convection – On lines – select the appropriate line – ok – Enter the values of film coefficient 50, bulk temperature 40 –ok
8. Solve – Current LS – ok – solution is done – Close

## Post Processing:

9. General post proc – List results – Nodal Solution – DOF Solution – Nodal temperature – ok

10. Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature – ok
11. File – Report Generator – Choose Append – ok – Image Capture – ok - Close

.




Thermal Conductivity of the material = 16 W/m $^0$

## Result:

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

## Outcome:

From this experiment, students will be able to analyse the convective heat transfer of a 2D component using the ANSYS software.

## Application:

1. Structural Application
2. Heat Exchanger Application

# EXP. NO.11 MODE FREQUENCY ANALYSIS OF A CANTILEVER BEAM

## Aim:

To conduct the Mode frequency analysis of a cantilever beam using ANSYS software

## System configuration:

| | |
|---|---|
| Ram | : 8 GB |
| Processor | : Core 2 Quad / Core 2 Duo |
| Operating system | : Windows 7 |
| Software | : ANSYS (Version12.0/12.1) |

## Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

## Preprocessing:

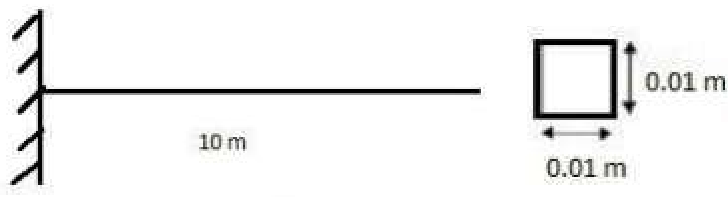
1. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – ok – close
2. Real constants - Add/Edit/Delete – Add – ok – Area 0.1e-3, Izz 0.833e-9, Height 0.01 – ok – close
3. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 2e9, PRXY 0.25 – ok – Density – 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 of element 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
7. 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.
8. Solve – Current LS – ok – Solution is done – close

## Post processing:

9. 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
10. File–Report Generator–Choose Append–ok–Image Capture–ok – Close (Capture all images)




Young‟s Modulus = $200\times10^{9}$ N/$m^{2}$

Poisson‟s ratio = 0.25

Weight Density = $7.83 \times10^{3}$ kg/$m^{3}$

## Result:

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

## Outcome:

From this experiment, students will be able to analyse the mode frequency of a cantilever beam using the ANSYS software.

## Application:

1. Structural Application
2. Construction Application

# EXP. NO.12 MODE FREQUENCY ANALYSIS OF A SIMPLYSUPPORTED BEAM

## Aim:

To conduct the Mode frequency analysis of a simply supported beam using ANSYS software

## System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

## 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 2e9, 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 of element divisions 25 – ok. Mesh – Lines – Select the line – ok

## Solution:

6. Solution – Define Loads – Apply – Structural – Displacement - On nodes – Select the first point and second point –ok – apply –UY- displacement = 0-ok. Analysis type – New analysis – Modal – ok
7. 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
8. Solve – Current LS – ok – Solution is done – close

## Post processing:

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

## Result:

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

## Outcome:

From this experiment, students will be able to analyse the mode frequency of a simply supported beam using the ANSYS software.

## Application:

1. Structural Application
2. Construction Application

# EXP. NO.13 HARMONIC ANALYSIS OF A 2D COMPONENT

## Aim:

To conduct the harmonic 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 (Version12.0/12.1)

## Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

## 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 Cntrls – manual size – lines – all lines – Enter the value of no of element divisions 25 – ok. Mesh – Lines – Select the line – ok

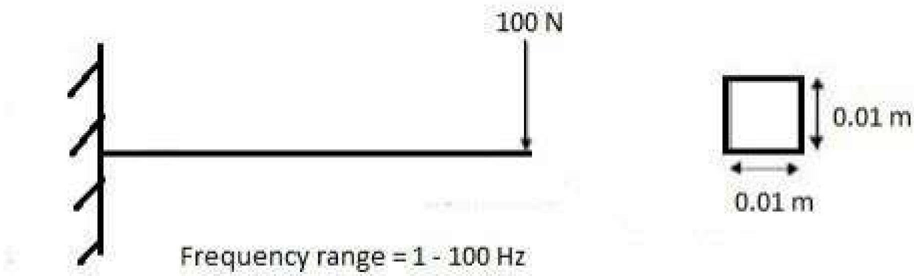
## Solution:

6. Solution - Analysis type – New analysis – Harmonic – 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 Sub stps – 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:

9. Time Hist post pro – Variable Viewer – Click “Add” icon – Nodal Solution – DOF Solution –Y-Component of displacement – ok – Enter 2 – ok. Click “List data” icon and view theamplitude list. Click “Graph” icon and view the graph. To get a better view of the response, view the log scale of UY. Plot ctrls – Style – Graphs – Modify axes – Select Y axis scale as Logarithmic – ok. Plot – Replot – Now we can see the better view.
10. File–Report Generator–Choose Append–ok–Image Capture–ok - close




Young‟s Modulus = $206\times10^{9}$ N/m$^{2}$

Poisson‟s ratio = 0.25

Weight Density = $7.83 \times 10^{3}$ kg/m$^{3}$

Length of the beam = 1 m

## Result:

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

## Outcome:

From this experiment, students will be able to analyse the harmonic analysis of 2D component using the ANSYS software.

## Application:

1. Structural Application
2. Construction Application

# EXP. NO.14 STRESS ANALYSIS OF A TRUSS

## Aim:

To conduct the stress analysis of a truss by using ANSYS software

## System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

Procedure:

Step 1: Ansys Utility Menu

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

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL - ok

Step 3: Preprocessor

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

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – ok – close.

Step 4: Preprocessor

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).

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).

Step 5: Preprocessor

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.

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.

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.

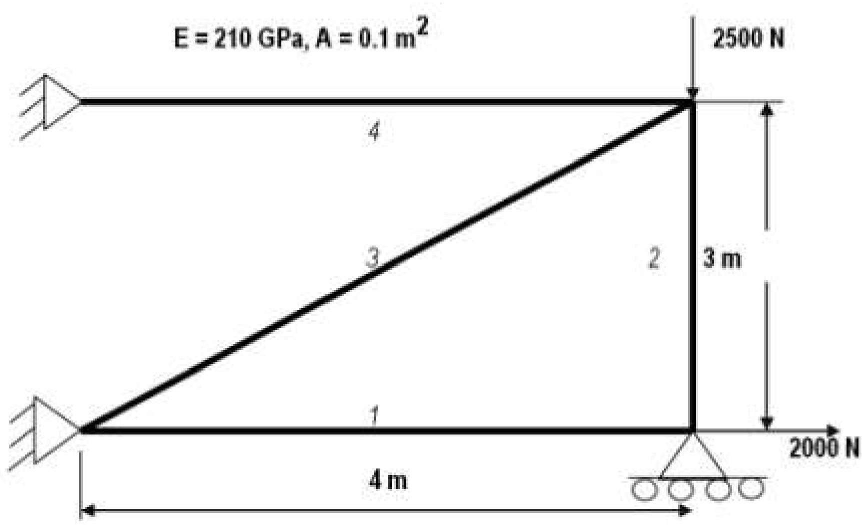
Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

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).

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



# EXP. NO.15 INTRODUCTION TO MATLAB

## 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 mathematicalnotation. Typical uses include

Math and computation

Algorithm development

Data acquisition

Modeling, simulation, and prototyping

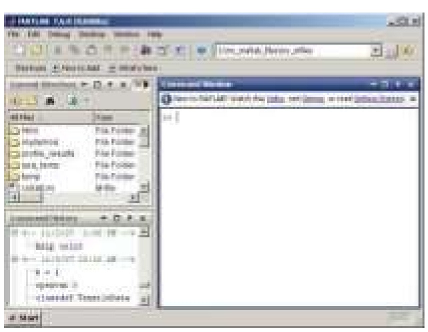
Data analysis, exploration, and visualization

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 non - interactive 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.

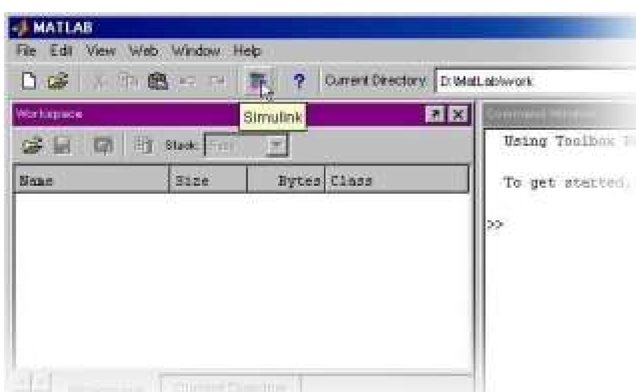
The idea behind these tutorials is that you can view them in one window while running Simulink in another window. System model files can be downloaded from the tutorials and opened in Simulink. You will modify and extend these systems while learning to use Simulink for system modeling, control, and simulation. Do not confuse the windows, icons, and menus in the tutorialsfor 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

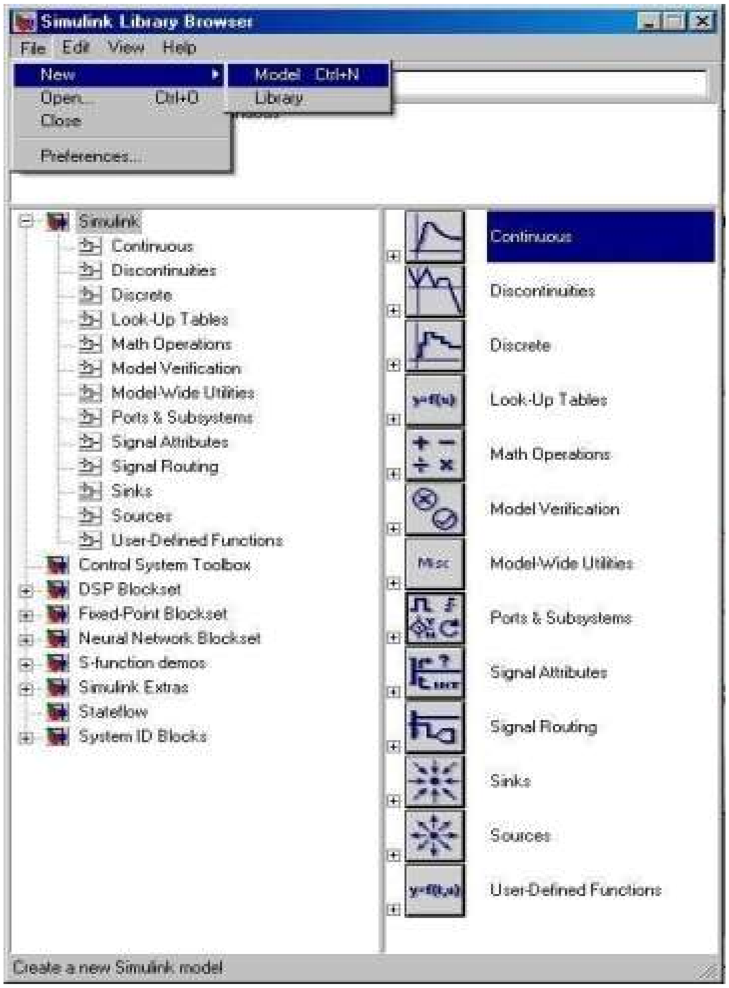
Starting Simulink

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

>> Simulink

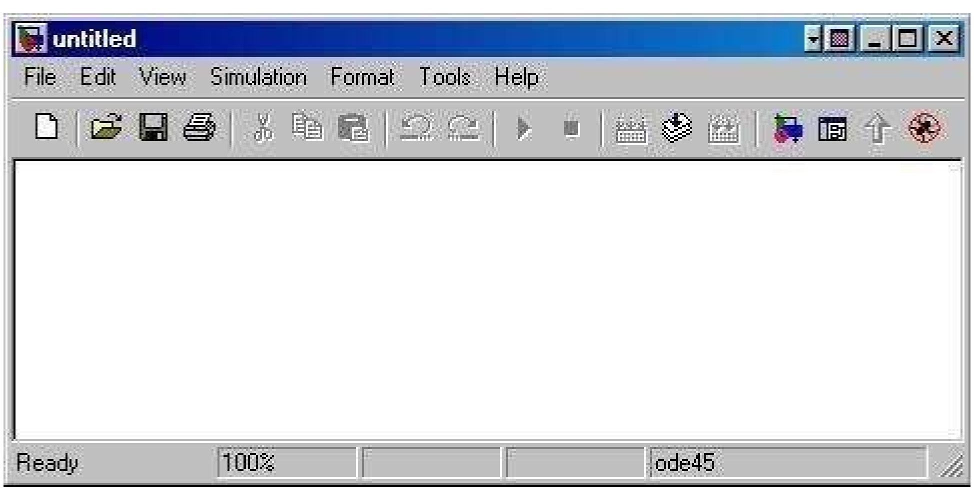



When it starts, Simulink brings up the Simulink Library browser.




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 untitiled modeling window shown below.



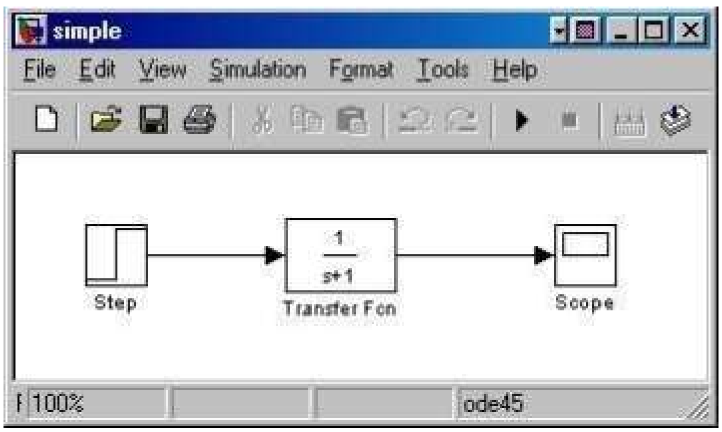
## Model Files:

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:

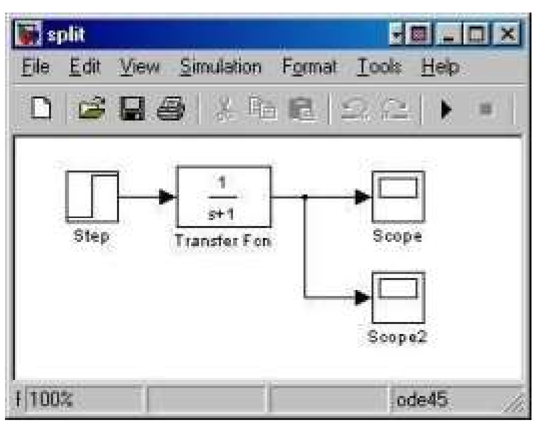
1. Continuous
2. Discontinuous
3. Discrete
4. Look-up tables
5. Math operations
6. Model verification

7. Model-wide Utilities
8. Ports & subsystems
9. Signal attributes
10. Signal routing
11. Sinks: Used to output or display signals
12. Sources: Used to generate various signals
13. User-defined functions
14. Discrete: Linear, discrete-time system elements (transfer functions, state-space models, etc.)
15. Linear: Linear, continuous-time system elements and connections (summing junctions, gains, etc.)
16. Nonlinear: Nonlinear operators (arbitrary functions, saturation, delay, etc.)
17. 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.

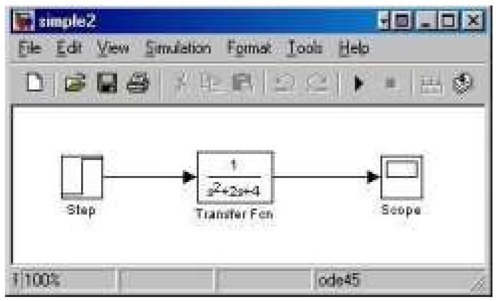



Lines can never inject a signal *into* another line; lines must be combined through the use of a block such as a summing junction.

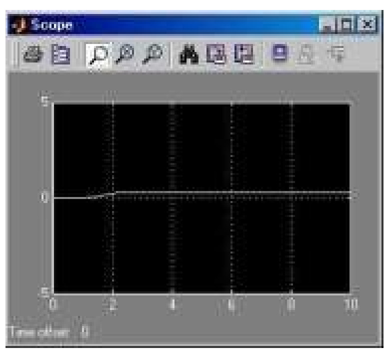
A signal can be either a scalar signal or a vector signal. For Single-Input, Single-Output systems, scalar signals are generally used. For Multi-Input, Multi-Output systems, vector signals are often used, consisting of two or more scalar signals. The lines used to transmit scalar and vector signals are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

## Running Simulations:

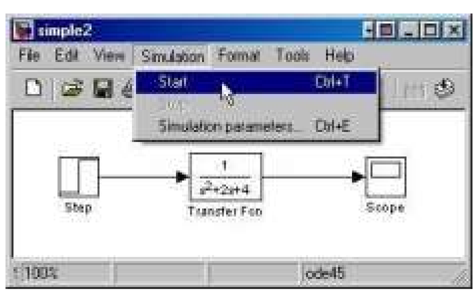
To run a simulation, we will work with the following model file:
simple2.mdl
Download and open this file in Simulink following the previous instructions for this file. You should see the following model window.




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)

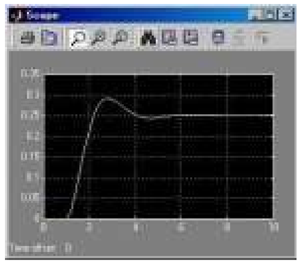



or hit Ctrl-T in the model window.



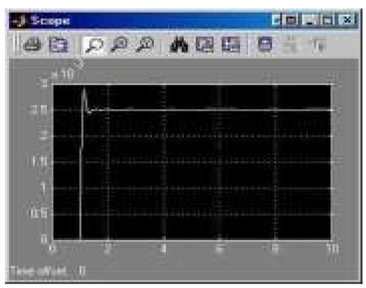

The simulation should run very quickly and the scope window will appear as shown below. If it doesn't, just double click on the block labeled "scope."

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.




Notethat 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

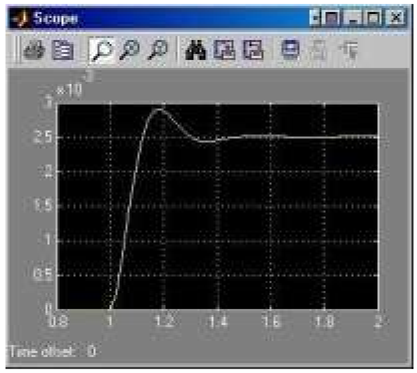



very fast response, it compressed into a very narrow part of the scope window. This is not really a problem with the scope, but with the simulation itself. Simulink simulated the system for a full ten seconds even though the system had reached steady state shortly after one second.

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.



# EXP. NO.16 SIMULATION OF SPRING-MASS SYSTEM USINGMAT LAB

## 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:

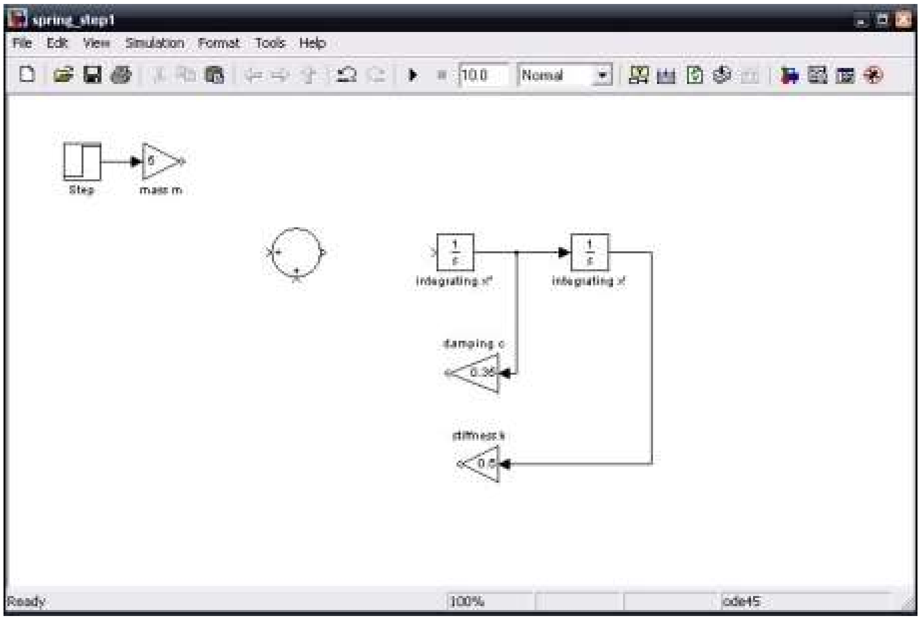
$$m\ddot{x}+c\dot{x}+kx=f(t)$$

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:

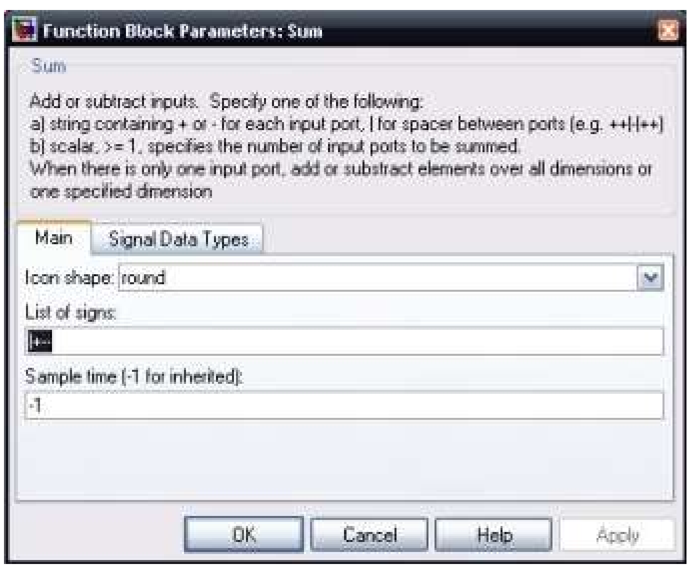
$\chi$ =

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

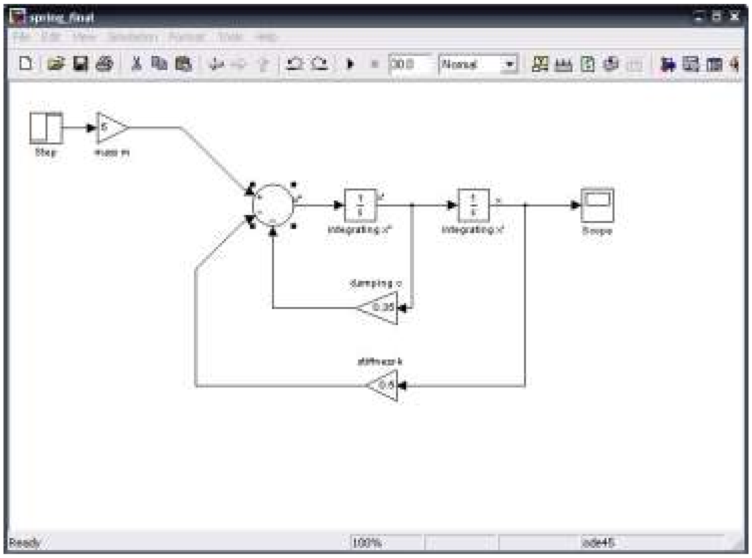
To build the model, we start with a 'step' block and a 'gain' block. The gain block represents the mass, which we will be equal to 5. We also know that we will need to integrate twice, that we will need to add these equations together, and that there are two more constants to consider. The damping constant 'c' will act on the veloc ity, 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:



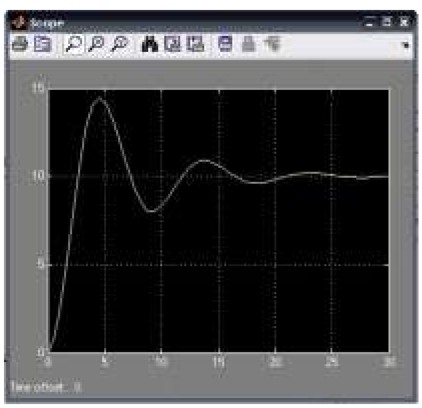

Once we have added places and corrected the signs for the sum block, we need only connect the lines to their appropriate places. To be able to see what is happening with this spring system, we add a 'scope' block and add it as follows:



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.



## Outcome:

From this experiment, students will be able to simulate the spring-mass system using MATLAB software.

## Application:

1. Structural Application
2. Automobile Application

# EXP. NO.17 SIMULATION OF CAM AND FOLLOWER MECHANISM USING MAT LAB

## Aim:

To simulate the cam and follower mechanism using MATLAB software

## System configuration:

| | |
|---|---|
| Ram | :8 GB |
| Processor | :Core 2 Quad / Core 2 Duo |
| Operating system | :Windows 7 |
| Software | :MATLAB |

## Procedure:

A cam and follower system is system/mechanism that uses a cam and follower to create a specific motion. The cam is in most cases merely a flat piece of metal that has had an unusual shape or profile machined onto it. This cam is attached to a shaft which enables it to be turned by applying a turning action to the shaft. As the cam rotates it is the profile or shape of the cam that causes the follower to move in a particular way. The movement of the follower is then transmitted to another mechanism or another part of the mechanism.

Examining the diagram shown above we can see that as some external turning force is applied to the shaft (for example: by motor or by hand) the cam rotates with it. The follower is free to move in the Y plane but is unable to move in the other two so as the lobe of the cam passes the edge of the follower it causes the follower to move up. Then some external downward force (usually a spring and gravity) pushes the follower down making it keep contact with the cam. This external force is needed to keep the follower in contact with the cam profile.

## Displacement Diagrams:

Displacement diagrams are merely a plot of two different displacements (distances). These two dispalcements are:

1. The distance travelled up or down by the follower and
2. The angular displacement (distance) rotated by the cam

If we examine the diagram shown below we can see the relationship between a displacement diagram and the actual profile of the cam. Note only half of the displacement diagram is drawn because the second half of the diagram is the same as the first. The diagram is correct from a theoretical point of view but would have to change slightly if the cam was to be actually made and used. We will consider this a little more in the following section - Uniform Velocity.

| | Angle the cam has rotated through | Distance moved by the follower |
|---|---|---|
| Start of the cycle | 0º | 0mm |
| End of first half of the cycle | 180º | 30mm |
| End of the full cycle | 360º | 0mm |

## Uniform Velocity:

Uniform Velocity means travelling at a constant speed in a fixed direction and as long as the speed or direction don't change then its uniform velocity. In relation to cam and follower systems, uniform velocity refers to the motion of the follower.

Now let's consider a typical displacement diagram which is merely a plot of two different displacements (distances). These two displacements are:

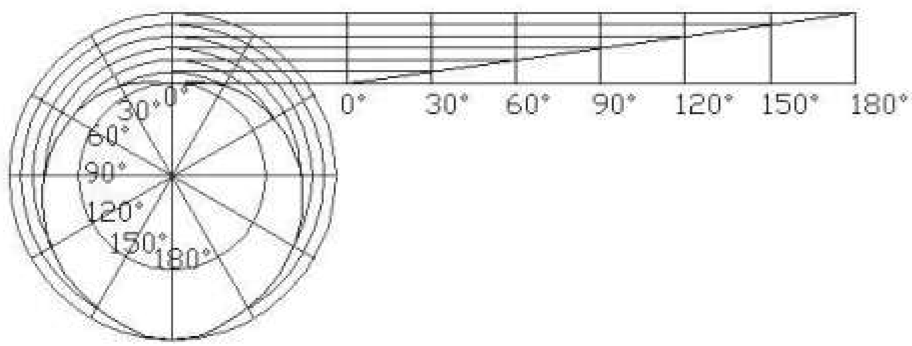
1. The distance travelled up or down by the follower and
2. The angular displacement (distance) of the cam

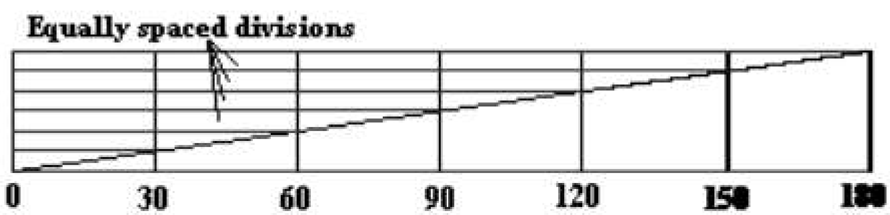
Let us consider the case of a cam imparting a uniform velocity on a follower over a displacement of 30mm for the first half of its cycle.

We shall take the cycle in steps. Firstly if the cam has to impart a displacement of 30mm on follower over half its cycle then it must impart a displacement of 30mm÷180º for every 1º turned by the cam i.e. it must move the follower 0.167mm per degree turn. This distance is to much to small to draw on a displacement diagram so we will consider the displacement of the follower at the start, at the end of the half cycle, the end of the full cycle and at certain other intervals (these intervals or thelength of these intervals will be decided on later).We shall consider this in terms of a displacement diagram:

First we will plot the graph. Before doing this we must first consider the increments that we will use. We will use millimeters for the follower displacement increments and because 1º is too small we will use increments of 30º for the angular displacement.Once this is done then we can draw the displacement diagram as shown below. Note a straight line from the displacement of the follower at the start of the motion to the displacement of the follower at the end of the motion represents uniform velocity.

Displacement Diagram - Uniform Velocity Motion:




## Result:

The displacement diagram for uniform velocity motion of the cam and follower is simulated using MATLAB software.

## Outcome:

From this experiment, students will be able to simulate the cam and follower mechanism using MATLAB software.

## Application:

1. Structural Application
2. Aitomobile Application

## SIMULATION AND ANALYSIS LAB VIVA QUESTIONS

1) What are the different approximate solution methods.
Different approximate solution methods are
1.Functional approximation
2.Finite Difference method
3. Finite Element method

2) What do you mean by continuum.
Structure which is considered for analysis is called continuum.

3) Define term node
The element which is connected with another element at junction is called node.

4) Define term element
Descritised structure is called an element .

5) What is convergence
Process of achieving value to actual solution.

6) What are the types convergence?
p-convergence
h-convergence

7) What is p- convergence?
Convergence by increasing the elements

8) What is h convergence?
Convergence by increasing nodes

9) What is higher order elements?
The element which contain more no. of nodes are called higher order element .

10) Give example for higher order elements
Higher order elements are CST, Quadrilateral element.

11) What do you mean by compatible elements?
Elements, which are compatible with adjacent element, like no discontinuity, overlap or sudden slope.

12) What is geometric invariance?
The property in which the shape of the element will not change with change in local coordinates is called geometric invariance.

13) Why do we use Pascal‟s triangle in FEA?
If the displacement equation doesn‟t contain all the required terms then balancing is done by the Pascal‟s triangle.

14) What are the steps involved in FEA?
Steps involved in FEA:
1.Modelling
2.Descritization of structure

3.Derivation of elemental stiffness matrix.
4.Assembly of elemental equation
5.Applying boundary conditions
6.Computation of stress and strain
7.Interpretaion of results

15) What is stiffness matrix?
The matrix when contain parameters like E, A, displacement and applied force is stiffness matrix

16) How to obtain stiffness matrix?
Stiffness matrix can be obtained by applying condition of minimum potential energy to potential energy equation.

17) What is displacement function?
Displacement function is the assumed polynomial equation, which satisfies boundary conditions.

18) How to identify order of elements?
Order of elements depends on the no. of nodes.

19) Mention different types of elements.
Different types of elements are bar elements, beam element, truss element, shell element, axis symmetric.

20) Mention some application of FEA.
Mechanical, Aerospace, Civil, structure analysis, biomedical, geo -mechanic, electromagnetic.

21) What is connectivity?
Relation between the connected elements is connectivity.

22) What are the methods to improve problem solution?
Problem solution can be improved by increasing no. of elements or no. of nodes .

23) Define symmetry in matrix.
It is the square matrix in which the element of the row are same as that of element of column.

24) What is plane stress?
Stress acting on 2-D element.

25) What is plane strain?
Strain occurring in 2-D element.

26) Compare FEA with solid mechanics.
complicated irregular structures are difficult in solid mechanics but FEA it"s easier with greater accuracy.

27) What are the packages available for FEA?
packages available for FEA: ANSYS, I-DEAS,NASTRAN,ABAQUS, COSMOS,ALGOL,PATRAN.

28) Define potential energy.
Energy possessed by the body due to it"s position

29) Define minimum potential energy.
for consertive system of all kinematically admissible displacement field those corresponding to equilibrium extremise the total potential energy if extreme condition is a minimum the equilibrium state is stable

30) Write potential energy equation for cantilever beam
= EI/2 (d2y/dx2) dx – pie

31) Mention 2 different methods to approach the model of physical system .
Discrete system , continuum

33) What is local coordinate?
local coordinate contains 1-D.O.F. at each node.

34) What is global co ordinate?
Global coordinate contains 2-D.O.F. at each node.

35) General assumption made in stress
Assumptions:
1.Truss elements are connected by fracture less pin.
2.Load is applied on the load.
3.Only two forces compressive and tensile are considered.

36)What is shape function?
It is mathematical polynomial, which gives displacement within the element.

37)What are two general natural coordinate?
Two general co-ordinates are (eeta) and zeeta.

38) Mention the range of natural co-ordinate.
Range of natural coordinates is – 0 to 1 and –1 to 1.

39) What is the number of shape function in CST?
Number of shape function in CST are three.

40) What is the number of shape function in quadrilateral?
Number of shape function in quadrilateral are 4.

41) Why we are using natural integration?
we are using natural integration to simplify the problem.

42) Explain one point formula.
1-point shape function contains one term f(x)dx=w.f(x)

43) Explain two point formula.
2-point shape function contains two term f(x).dx=w1f(x1) w2f(x2)

44) Why we are using polynomial equation in FEA?
Polynomial equation gives continuous solution & it is simple to solve problem.

45) What are the two important characteristics in stiffness matrix?
Characteristics of stiffness matrix is symmetric and bonded

47) Mention two schemes to represent band width.
2-Schemes are
Horizontal numbering
Vertical numbering

48)What are forces involved in work potential?
Forces involved are Body force, Traction force and Point force

49) What is isoparametric elements?
These are those the S.F. used to define variables of displacement equal to S.F. used to represent geometry.

50) What is orthotropic elements?
Material which has three orthogonal planes of symmetry said to be orthotropic elements. Only nine constants are required to describe constituent equation

51) What is anisotropic elements?
The material which doesn‟t contain any plane of symmetry

52) What is isotropic elements?
isotropic material is one in which every plane is plane of symmetry only two constants are enough to describe constituent equation

53) What is super parametric elements?
GSF > DSF (geometric shape function, displacement shape function
54) What is sub parametric elements
GSF < DSF

55) Different coordinates involved in chain rule.
Different coordinates involved in chain rule are normal , local and displacement coordinates .

56) What are the 2 different approaches to study elasticity ?
Strength of material , Theory of elasticity

57) Mention any two methods to solve continuum problems.
1. Raleighritz method
2. Galerkin method

58) List the properties of shape functions.
SFfor 1D bar element N1=0, N2= 0 at node 1
N1=0, N2= 1 at node 2 .
Diff. Of S.F. S.F are constant

59) Define truss.
Structural member which is subjected to either tensile or compression

60) What is weighted residual methods.
It‟s a method in FEA for accurate solution avoiding error (residue = error)

61) Different methods to solve weighed residual problem.
Point allocation, Sub domain , Galerkein,least square.

62)Explain the principle of virtual work.
If the force and displacement are unrelated by cause effect relation then the work is said to be virtual work.

63) Explain the principle of virtual displacement.
Actual displacement is considered without bothering amount of force is called virtual displacement .

64)What are different types of shape functions.
Diff types are Lagarangian and Hermite Shape function

65) Differentiate two types of shape functions.
Lagarangian shape function only for variable
Hermite shape function is for both variable and its derivative

66) Mention some advantages of FEA over solid mechanics.
1. Applied for complicated structure
2. Analysis is simple
3. More accurate solution

67) Define Young"s Modulus and Poisson"s Ratio.
E - It is the ratio of stress and strain
μ -It is the ratio of lateral strain and longitudinal strain

68) Mention different types of elastic constants.
Young"s modulus, shear modulus, bulk modulus

69)Specify the terms required to solve FEA problem.
Meshing , properties of material, boundry condition and initial condition.

70) What are the assumptions made in linear static problems.
all displacement is small, material is isotrpic, linear, elastic solid with E and

71) Which is the most accepted form of numerical integration in FEM?
Gaussian Quadrature

72)List the different approaches to derive integral equation
Direct method , variation method, weighed residual method, energy method

73)What are the advantages of symmetrical matrix.
symmetrical matrix simplifies the calculation

74) What are the different types of errors in FEA?
Modeling error , Descritized and Numerical error

75) What are the advantages of isoparametric elements
Useful in modeling structure with curved edges
They are versatile & they are used in 2-D and 3-D elasticity problems

76). Define frontal method for finite element matrices.
In 3- dimensional problems, the size of the stiffness matrix increases rapidly even with the banded method of modeling. An alternative direct method that results in considerable saving in the use of computer memory is called frontal method.

77) What is the another name of the 3-dimensional frames
Space frames.

78). Define beam elements.
Beam elements are slender members that are used for supporting transverses loading.

79) Explain preprocessor steps.
Determining the Nodal coordinates, connectivity, boundary condition, material information

80) Explain processing steps.
Stiffness generation, modification, solutions to the equation resulting in evolution.

81) Explain post processing steps.
Deformation confirmation, mode shapes, temperature and stress distribution, interpretation

82) What are the difference b/w beams and plane frames.
It is similar to beams expect that axial loads &axial deformations are present. The elements also have different orientation.

83) Mention some common material properties.
Isotropic, orthrotropic, ductility, brittleness

84) What are the different types of analysis.
Thermal, structural (load), fluid, electromagnetic analysis

85) Define steady state analysis.
The analysis carried out at constant temperature.

86) What is the advantage of subjecting solids to axisymmetric loading.
Axisymmetric loading reduces the 3-D problem into 2-D problems because of total symmetry about the z-axis.

87) Define CST elements.
The constant strain triangle is that where the displacement inside an element is represented by 3-nodal displacement (3-shape functions).

88) How to generate the data files for larger problems in FEA. Using
MESHGEN program generates data files.

89) Define mesh plotting
It is the convenient way of reviewing the coordinate and connectivity data is by plotting it using computer.

90) Explain lumped mass matrices.
It is the total element mass in each direction is distributed equally to the nodes of the element, and the masses are associated with translational degrees of freedom.

91) Briefly explain steps involved in Lagrangian method

Formulation of potential energy function.
Assuming displacement function
Checking displacement function considering boundary condition
Substitute differential function in potential energy equation
Potential energy function is minimized
Unknown parameters are determined and substituted in assumed equation

92) Explain steps involved in Gallerkins method.
Formulate differential equation of the equilibrium
Assume trial function, which satisfies boundary conditions
Substitute displacement function in differential equation then assume the difference due to approx. function be „R" (residue)
Use gallerkin formula
Determine unknown terms and then substitute in differential function

93) Define jacobian matrix.
The matrix which is defined explicitly in terms of the local coordinate is known as jacobian(J).

94) Mention six components of stress.
3 linear stress along x, y, z direction
3 lateral stress along x, y, z direction

95) Mention six components of strain.
3 linear strain along x, y, z direction
3 lateral strain along x, y, z direction

96) Define Winkler foundations.
Large beams are supported on soil form a class of applications known as winkler foundations.

97) Define variational principle
The problem which specifies a scalar quantity potential energy is defined in an integral form.

98) How to solve the prismatic problems.
The coefficient of the ordinary differential equation are independent of one of the coordinate and the solution of the system can frequently carried out efficiently by standard analytical methods.

99) Define stress & strain.
Stress is the ratio of applied load to its area.
Strain is the ratio of change in length to its original length.

100) Mention the two distinct procedures available for obtaining the approximation in the integral forms.
Method weighed residuals. Method of variation functional.

# MATLAB VIVA QUESTIONS

1) Explain what is MatLab? Where MatLab can be applicable?
MatLab is a high-level programming language with an interactive environment for visualization, numerical computation and programming function.
Matlab can be applicable at numerous instances like

· Allows matrix manipulations
· Plotting of functions and data
· Implementation of algorithms
· Creation of user interfaces
· Analyze data
· Develop algorithm
· Create models and applications
· Interfacing with programs written in other languages ( C++, C, Java and Fortran)

2) What does MatLab consist of?
MatLab consists of five main parts
· MatLab Language
· MatLab working environment
· Handle Graphics
· MatLab function library
· MatLab Application Program Interface (API)

3) Explain MatLab API (Application Program Interface)?
MatLab API is a library that enables you to write Fortran and C programs that interact with MatLab. It contains the facilities for calling routines from MatLab, for reading and writing Mat files and calling Matlab as a computational engine.

4) What are the types of loops does Matlab provides?
Matlab provides loops like
· While Loop
· For Loop
· Nested Loops

5) List out the operators that MatLab allows?
Matlab allows following Operators
· Arithmetic Operators
· Relational Operators
· Logical Operators
· Bitwise Operations
· Set Operations

6) Explain what is Simulink?
Simulink is an add-on product to MatLab, it provides an interactive, simulating, graphical environment for modeling and analyzing of dynamic systems.

7) In MatLab is it possible to handle multi-dimensional arrays?
Yes, it is possible in MatLab to handle multi-dimensional arrays. Matlab‟s internal data structure is limited to a two-dimensional matrix. But to handle multi-dimensional arrays in Matlab, you can create your own functions in Matlab language.

8) Mention what is the sign convention used in MatLab‟s fft routines?
The sign convention used in MatLab‟s fft routines are defined as sum(x(i)*exp (-j*i*k/N)) and not sum (x(i)exp(j*i*k/N)). The first version is used by engineers, and the second is used by mathematician.

9) What are the four basic functions to solve Ordinary Differential Equations (ODE)?
The four basic functions that MatLab has to solve ODE‟s are
· Quad
· Quad8
· ODE23
· ODE45

10) Explain how polynomials can be represented in MatLab?
A polynomial in MatLab is denoted by a vector. To create a polynomial in MatLab enter each co-efficient of the polynomial into the vector in descending order

11) What is the type of program files that MatLab allows to write?
Matlab allows two types of program files
· Scripts: It is a file with .m extension. In these files, it writes series of command that you want to execute together. It does not accept inputs and do not return any outputs
· Functions: They are also files with .m extension. Functions can accept inputs and return outputs.

12) Explain how to modify the MatLab Path?
To modify the MatLab Path use the PathTool GUI. Also, you can use add path directories from the command line and add the path to rc to write the current path back to „pathdef.m.‟ In the case if you don‟t have permission to write for „pathdef.m‟ then pathrc can be written into a different file, you can execute from your „startup.m.‟

13) Explain what is LaTex in MatLab?
MatLab handles naturally simple LaTex encoding which allows introducing greek letters or modifying the font size and appearance in plots.

14) Explain how you can pre-allocate a Non-Double Matrix?
Pre-allocating a block of memory for holding a non-double matrix is memory efficient. While allocating blocks of memory for a matrix, zeros are pre-allocated to a matrix.
The functions to pre allocate memory is int8(), example matrix =int8(zeros(100));
Repmat function is used to create a single double matrix, example matrix2=repmat(int8(0), 100, 100)

15) What is Xmath-Matlab? Mention the Xmath features?
For Xwindow workstations, Xmath is an interactive scripting and graphics environment.
Following are the X-math features
· Scripting language with OOP features
· Libraries that are LNX and C language compatible
· A debugging tools with GUI features
· Color graphics can be pointed and clickable

16) Name the graphic system used in MatLab?
Graphic system used in MatLab is known as handle graphics. It has a high level and low- level commands.
· High Level Commands: High level command performs image processing, data visualization and animation for 2D and 3D presentation graphics
· Low Level Commands: Full customization of the appearance of graphics and building of complete graphical user interface

17) Explain what is M-file and MEX files in MatLab?
M files: They are just a plain ASCII text that is interpreted at run time. They are like sub-programs stored in text files with .m extensions and are called M-files. For most of the MatLab, development M-files are used.
MEX files: They are basically native C or C++ files which are linked directly into the MatLab application at runtime. MEX files have efficiency to crash the MatLab application.

18) Explain what is Interpolation and Extrapolation in Matlab? What are their types?
· Interpolation: Taking out function values between different data points in an array is referred as Interpolation
· Extrapolation: Finding function values beyond the endpoints in array is referred as Extrapolation
The two types of Interpolation and Extrapolation are
· Linear Interpolation and Extrapolation
· Quadratic Interpolation and Extrapolation

19) List out some of the common toolboxes present in Matlab?
Some of the common toolboxes in Matlab are
· Control System
· Fuzzy Logic
· Image Processing
· LMI control
· Neural Networks
· Robust Control
· System Identification

20) What is Get and Set in Matlab?
Get and Set are referred as getter and setter functions. For assigning properties, setter functions are used while for accessing properties getter functions are used.