Dhanalakshmi College Of Engineering

Manimangalam, Tambaram, Chennai –601 301

DEPARTMENT OF

MECHANICAL ENGINEERING

ME 6711 – SIMULATION AND ANALYSIS LABORATORY

VII SEMESTER - R 2013

LABORATORY MANUAL

Name : ________________________________

Reg. No. : ________________________________

Section : ________________________________

DHANALAKSHMI COLLEGE OF ENGINEERING

VISION

Dhanalakshmi College of Engineering is committed to provide highly disciplined, conscientious and enterprising professionals conforming to global standards through value based quality education and training.

MISSION

- To provide competent technical manpower capable of meeting requirements of the industry
- To contribute to the promotion of Academic Excellence in pursuit of Technical Education at different levels
- To train the students to sell his brawn and brain to the highest bidder but to never put a price tag on heart and soul

DEPARTMENT OF MECHANICAL ENGINEERING

VISION

Rendering the services to the global needs of engineering industries by educating students to become professionally sound mechanical engineers of excellent caliber.

MISSION

To produce mechanical engineering technocrats with a perfect knowledge intellectual and hands on experience and to inculcate the spirit of moral values and ethics to serve the society

ME6711-SIMULATION AND ANALYSIS LABORATORY

COURSE OBJECTIVES

• To give exposure to software tools needed to analyse engineering problems

• To expose the students to different applications of simulation and analysis tools

List of Experiments

A. Simulation

1. MAT LAB 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

B. Analysis

1. Force and stress analysis using link elements in Trusses, cables etc.
2. Stress and deflection analysis in beams with different support conditions.
3. Stress analysis of flat plates and simple shells.
4. Stress analysis of axi-symmetric components.
5. Thermal stress and heat transfer analysis of plate.
6. Thermal stress analysis of cylindrical shells.
7. Vibration analysis of spring-mass systems.
8. Model analysis of beams.
9. Harmonic, transient and spectrum analysis of simple systems.

COURSE OUTCOMES

- Upon completion of this course, the students can model, analyse and simulate experiments to meet real world system and evaluate the performance.

ME6711-SIMULATION AND ANALYSIS LABORATORY

CONTENTS

Expt. 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:

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

5. 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:

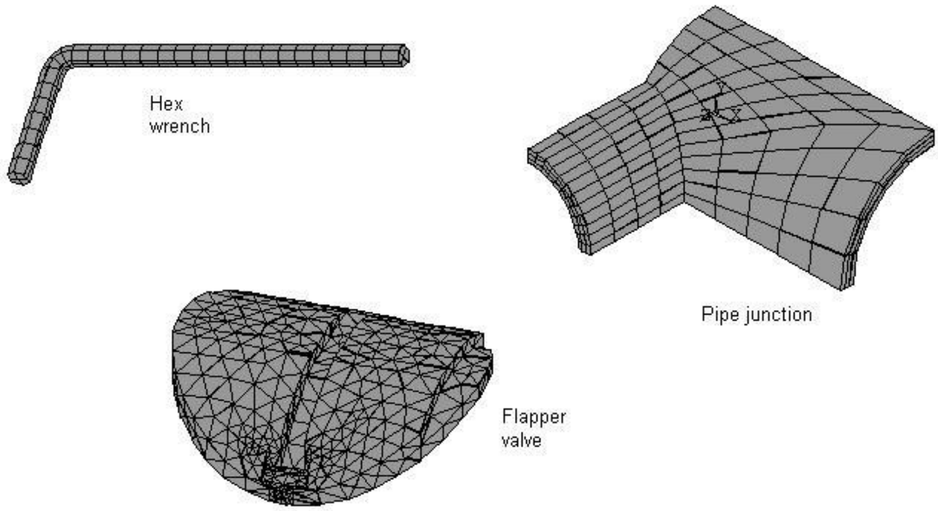
Most element types require material properties. Depending on the application, material properties can be linear (see linear material properties) or nonlinear (see nonlinear material properties).

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.
  a) Convection.
  b) Radiation.
  c) Heat flow rates.
  d) Heat fluxes.
  e) Heat generation rates.
  f) 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
  a) Design of concert house, where an even distribution of sound pressure is possible
  b) Noise cancellation in automobile
  c) Underground water acoustics
  d) Noise minimization in machine shop
  e) 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.

Result:

Thus the basics of ANSYS are studied.

Expt. 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 length 2/3/4/5– ok - Mesh Tool -Select TRI or QUAD - Free/Mapped – Mesh - Select the object –ok

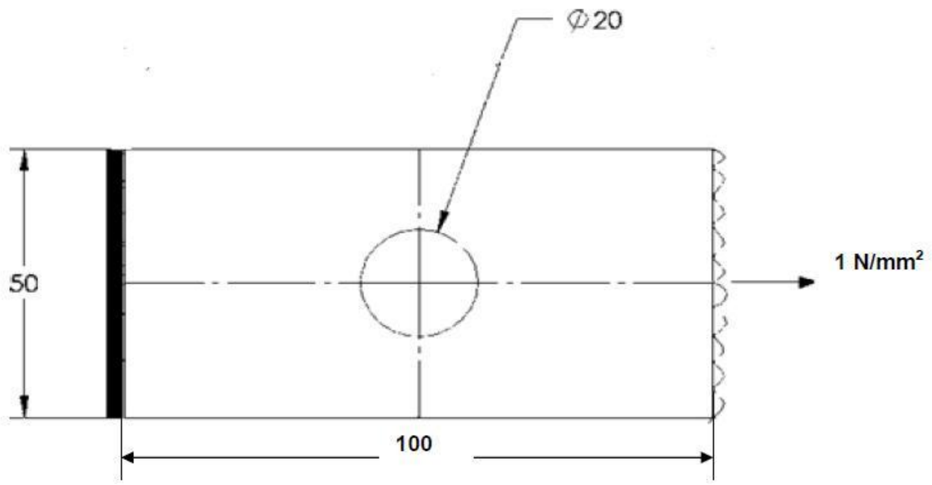
Solution:

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/mm^2 – 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	:	2×10^5 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:

Able to analyse a rectangular plate with a circular hole using the ANSYS software.

Viva-voce

1. What is meant by ANSYS?
2. What is meant by stress?
3. What is meant by stress concentration?
4. List out the different types of analysis in ANSYS.
5. What is meant by displacement?
6. What are ‘h’ and ‘p’ versions of finite element method?
7. State the methods of engineering analysis.
8. What is meant by finite element?
9. Write the stages of finite element analysis.
10. What is meant by structural problems?
11. What is meant by non-structural problems?
12. What is meant by preprocessing?
13. Write the FEA softwares available.
14. What is meant by post processing?
15. Write the applications of finite element method.
16. Define – Simple Element
17. Define – Complex Element
18. What is meant by one dimensional analysis?
19. What is meant by two dimensional analysis?
20. What are the differences between boundary value problem and initial value problem?

Expt. 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	:	Windows7
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 – 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 going to be arrested – ok - All DOF - ok. Pressure - On lines - Select the load applying area – ok - Load PRES valve = -10000 N (- Sign indicates the direction of the force i.e. downwards) – ok
8. Solve – Current LS – ok– Solution is done – close

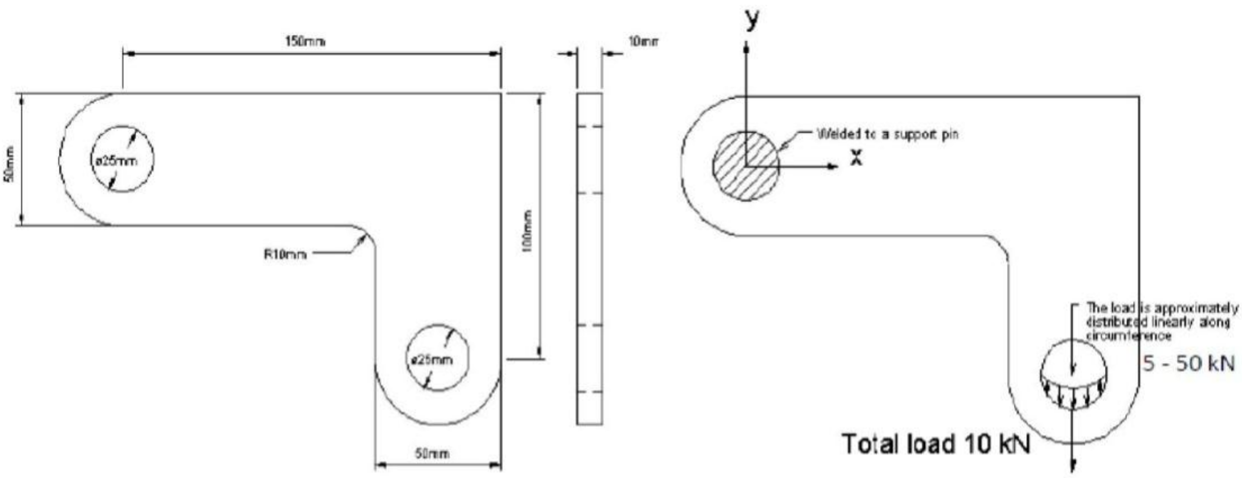
Post Processing:

9. General post proc - Plot Result - Contour plot - Nodal Solution – Stress –Von mises stress - ok

12. Plot control – Animates - Mode Shape – Stress - Von mises - ok

13. 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:

Able to analyse a rectangular L section bracket using the ANSYS software.

Viva-voce

1. What is meant by bracket?
2. List out any two types of bracket.
3. What is meant by stress distribution?
4. What is meant by degree of freedom?
5. List out any two types of meshing.
6. Define – Maximum Principal Stress Theory
7. Define – Maximum Shear Stress Theory
8. Define – Von-Mises Stress Theory
9. Differentiate between essential and natural boundary conditions.
10. What is meant by global co-ordinate system?
11. What is meant by local co-ordinate system?

12. Define – Factor of Safety
13. Write the material properties for steel.
14. Write the material properties for copper
15. What are the different types of loading acting on the structure?
16. Define – Body Force
17. Define – Traction Force
18. Define – Young's Modulus
19. What is meant by plane stress?
20. What is meant by plane strain?

Expt. No.04(a) STRESS ANALYSIS OF BEAM (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. 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 – 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

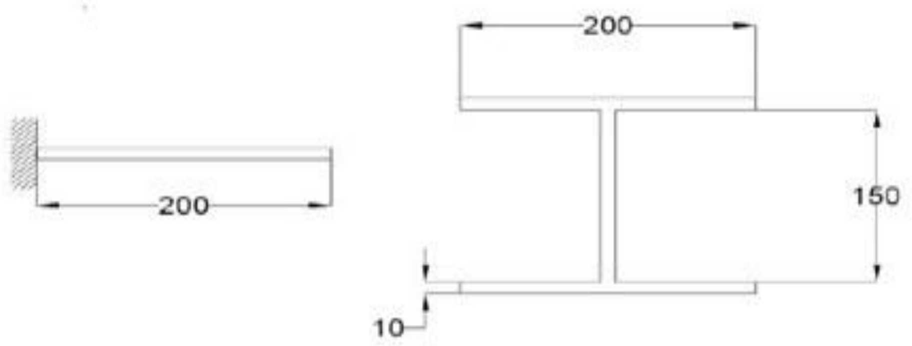
tool – Mesh – Pick all

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 Element Res – 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:

Able to analyse a cantilever beam using the ANSYS software.

Viva-voce

1. What is meant by beam?
2. What is meant by cantilever beam?
3. Define – Deflection
4. Define – Young's Modulus
5. What is the value for Young's modulus of steel?
6. Define – Shear Force Diagram
7. Define – Bending Moment Diagram
8. Define – Point of Contraflexture
9. Define – Slope
10. What is meant by moment of inertia?
11. Write the expression of moment of inertia for circular cross section.
12. At which point bending moment will be maximum?
13. What are sagging and hogging moments?
14. What is meant by flextural rigidity?
15. What is meant by torsional rigidity?
16. Write the types of beams.
17. Draw the shear force diagram for a cantilever beam, subjected to a uniformly distributed load.
18. Draw the shear force diagram for a cantilever beam, subjected to a point load acting at the center of the beam.
19. Draw the bending moment diagram for a cantilever beam, subjected to a uniformly distributed load.
20. Draw the bending moment diagram for a cantilever beam, subjected to a point load acting at the center of the beam.

Expt. No.04(b) STRESS ANALYSIS OF BEAM (SIMPLY SUPPORTED)

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. 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 - 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 (Table 2)

14. General postprocessor - contour plot - line element res. -ok

Table 1: Element Stresses

Sl.No.	SMAXI (N/mm^2)	SMAXJ (N/mm^2)	SMINI (N/mm^2)	SMINI (N/mm^2)
1				
2				
3				
4				
5				

Table 2: Displacement – Deflection

Nodes	UY
1	
2	
3	
4	

Result:

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

Outcome:

Able to analyse a simply supported beam using the ANSYS software.

Viva-voce

1. What are the types of beam?
2. Write the expression of deflection for a simply supported beam?
3. What are the properties of stiffness matrix?
4. What is the value of deflection and displacement of a cantilever beam?
5. What are the boundary conditions for 2D type of problems?
6. Define – Polar Moment of Inertia
7. Define – Section Modulus
8. Define – Polar Modulus
9. Write down the boundary conditions for a simply supported beam.

10. Draw the shear force diagram for a simply supported beam, subjected to a uniformly distributed load.
11. Draw the shear force diagram for a simply supported beam, subjected to a point load acting at center
12. Draw the bending moment diagram for a simply supported beam, subjected to a uniformly distributed load.
13. Draw the bending moment diagram for a simply supported beam, subjected to a point load acting at center.
14. Define – Theory of Bending
15. What are the assumptions made for analyzing the beams?
16. Define – Uniformly Distributed Load
17. Define – Uniformly Varying Load
18. What types of elements are used for analyzing the beam problems?
19. Define – Longitudinal Strain
20. Define – Lateral Strain

Expt. No.04(c) 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. 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 - 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 (Table 2)

14. General postprocessor - contour plot - line element res. -ok

Table 1: Element Stresses

Sl.No.	SMAXI (N/mm^2)	SMAXJ (N/mm^2)	SMINI (N/mm^2)	SMINI (N/mm^2)
1				
2				
3				
4				
5				

Table 2: Displacement – Deflection

Nodes	UY
1	
2	
3	
4	

Result:

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

Outcome:

Able to analyse a fixed beam using the ANSYS software.

Viva-voce

1. Define – Element and Node
2. What are the various types of numbering?
3. What are all the types of loads?
4. What are the types of co-ordinates?
5. Give the Stiffness matrix for two dimensional element (CST Element)
6. Define – Fixed Beam
7. Define – Poisson's Ratio
8. Write the expression of deflection for a fixed beam, subjected to a point load at center.
9. Write the expression of deflection for a fixed beam, subjected to a uniformly distributed load.

10. Write the expression of slope for a fixed beam, subjected to a point load at center.
11. Write the expression of slope for a fixed beam, subjected to a uniformly distributed load.
12. Draw the shear force diagram for a fixed beam, subjected to a point load acting at center.
13. Draw the shear force diagram for a fixed beam, subjected to a uniformly distributed load.
14. Draw the bending moment diagram for a fixed beam, subjected to a point load acting at center.
15. Draw the bending moment diagram for a fixed beam, subjected to a uniformly distributed load acting at center.
16. Write the various types of supports.
17. What is meant by shear centre?
18. Define – Rigid Body
19. Define – Deformable Body
20. What is meant by composite beam?

Expt. No.05 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. Pre Processing
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 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

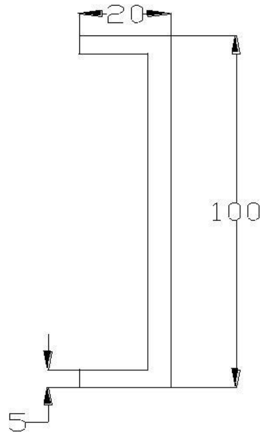
7. Preprocessor - meshing - mesh tool - size control - Areas - Element edge length = 2 mm -ok- mesh - Areas – free- pick all.

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, as shown 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:

Able to analyse an axi-symmetric component using the ANSYS software.

Viva-voce

1. Explain the basic steps of analysis.
2. What are the conditions for a problem to be axisymmetric?
3. What is meant by axisymmetric element?
4. Give the stiffness matrix for the four node quadrilateral axisymmetric component.
5. What are the properties of axisymmetric elements?
6. What is meant by element types?
7. What is meant by revolving option?
8. What is the basic component of ANSYS?
9. List out any four types of contour plot.
10. Define – Isotropic Elements
11. Define – Orthotropic Elements
12. Define – Anisotropic Elements
13. Define – Meshing
14. What types of elements are used to analyse an axisymmetric component?
15. Define – Radial Strain
16. Define – Circumferential Strain
17. Define – Longitudinal Strain
18. Define – Shear Strain
19. What is meant by plane stress analysis?
20. What is meant by plane strain analysis?

Expt. No.06 THERMAL STRESS ANALYSIS OF A 2D COMPONENT

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. 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

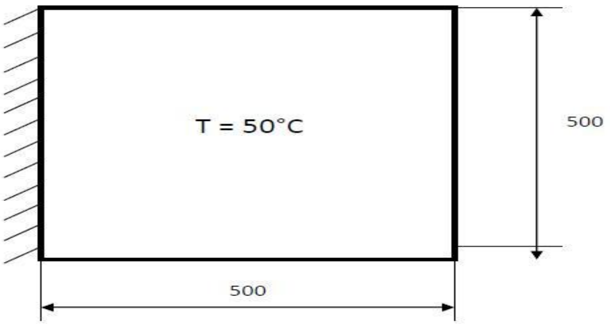
6. Meshing – Mesh tool – Areas, set – select the object – ok – Element edge length 10 - ok – Mesh tool- Tri, free - mesh – Select the object

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 = 12×10^{-6} /^{0}C

Result:

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

Outcome:

Able to analyse the thermal stress of a 2D component using the ANSYS software.

Viva-voce

1. What is meant by thermal analysis?
2. Define – Thermal Stress
3. Define – Thermal Strain
4. What are the modes of heat transfer?
5. What is meant by conduction?
6. What is meant by convention?
7. What is meant by radiation?
8. Define – Thermal Conductivity
9. What are the factors affecting the thermal conductivity?
10. State Fourier's law of conduction.
11. Write down the expression for conduction of heat through a slab or plane wall.
12. -----------commands are used to delete the force and pressure.
13. ----------- commands are used to save the image.
14. Define – Linear Element
15. Define – Non-Linear Element

Expt. No.07 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. 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

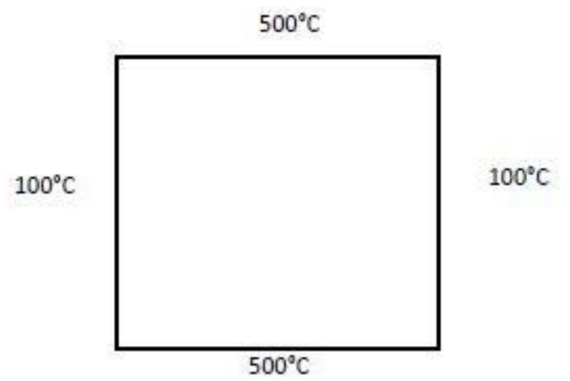
Solution:

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 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 ^{0}C

Dimension of the object = 2m × 2m

Result:

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

Outcome:

Able to analyse the conductive heat transfer of a 2D component using the ANSYS software.

Viva-voce

1. What is meant by thermal analysis?
2. What is meant by coordinate system?
3. Write the governing differential equation for a two dimensional heat transfer problem.
4. Define – Thermal Conductivity

5. What are the factors affecting the thermal conductivity?
6. State Fourier's law of conduction.
7. Write down the stiffness matrix equation for one dimensional heat conduction element.
8. Write down the expression of shape function for one dimensional heat conduction element.
9. Define – LST Element
10. Define – CST Element
11. Define – Isotropic Elements
12. Define – Orthotropic Elements
13. Define – Anisotropic Elements
14. What are the material properties required for thermal analysis?
15. What is meant by contour plot?

Expt. No.08 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. 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, 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:

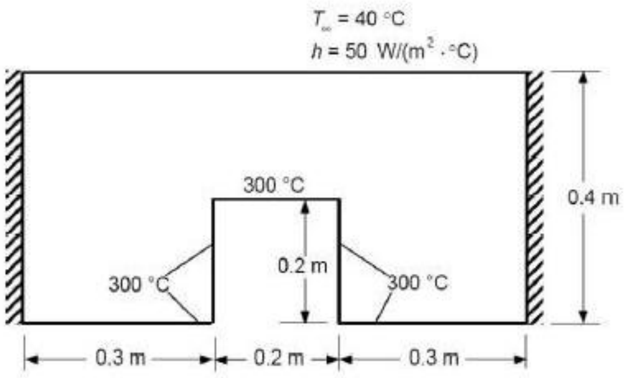
7. Solution – Define Loads – Apply – Thermal – Temperature - On lines – Select the lines–ok – Temp. Value 300 – ok – Convection – On lines – select the appropriate line – ok – Enter the values of film coefficient 50, bulk temperature 40 –ok

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}C

Result:

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

Outcome:

Able to analyse the convective heat transfer of a 2D component using the ANSYS software.

Viva-voce

1. Define – Convective Heat Transfer
2. What is meant by free or natural convection?
3. What is meant by forced convection?
4. What is meant by isotropic?
5. What is meant by anisotropic?
6. What is meant by orthotropic?
7. Define – Heat Transfer Coefficient
8. Define – Thermal Conductivity
9. What are the factors affecting the thermal conductivity?
10. What are the material properties required for thermal analysis?
11. Define – Key Point
12. Name the types of nodes.
13. Define –Meshing
14. What are the functions of post processor?
15. ------------- commands are used to apply the load in to object.

Expt. No.09(a) 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. 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 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:

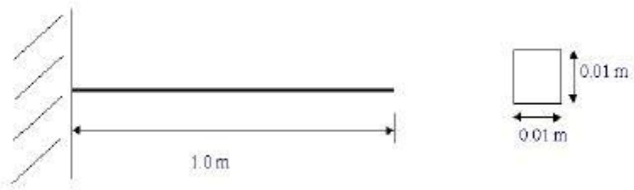
6. Solution – Define Loads – Apply – Structural – Displacement - On nodes – Select the node point

–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 = 200×10^9 N/m^2

Poisson's ratio = 0.25

Weight Density = 7.83×10^3 kg/m^3

Result:

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

Outcome:

Able to analyse the mode frequency of a cantilever beam using the ANSYS software.

Viva-voce

1. What is meant by static analysis?
2. What is meant by dynamic analysis?
3. What is meant by modal frequency?
4. What are the different types of vibrations?
5. Define – Frequency of Vibration

6. What is meant by longitudinal vibrations?
7. What is meant by transverse vibrations?
8. Define – Stiffness Matrix
9. Define – Mass Matrix
10. Define – Moment of Inertia
11. Define – Section Modulus
12. Define – Amplitude
13. What are the causes and effect of vibration?
14. What is meant by free vibration?
15. What is meant by forced vibration?
16. Define – Damped Vibration
17. Define – Undamped Vibration
18. Define – Lumped Mass Matrix
19. Define – Weight Density
20. Define – Young’s Modulus

Expt. No.09(b) MODE FREQUENCY ANALYSIS OF A SIMPLY SUPPORTED 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. 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 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:

Able to analyse the mode frequency of a simply supported beam using the ANSYS software.

Viva-voce

1. Define – Modal Analysis
2. Name the two types of mesh generators.
3. Define – Boundary Element Method
4. What is meant by vibration? What are the types of vibration?
5. Define –Magnification Factor
6. What are the types of eigen value problems?
7. State the principle of superposition
8. Define – Resonance
9. What are the methods used for solving transient vibration problems?
10. What is meant by plot controls?
11. Name the Boolean operations.
12. What are the steps to be followed for creating the finite elements?
13. What is meant by current LS?
14. Define – Natural Vibration
15. Define – Forced Vibration

<u>Expt. No.10</u> <u>HARMONIC ANALYSIS OF A 2D COMPONENT</u>

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. 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 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 –

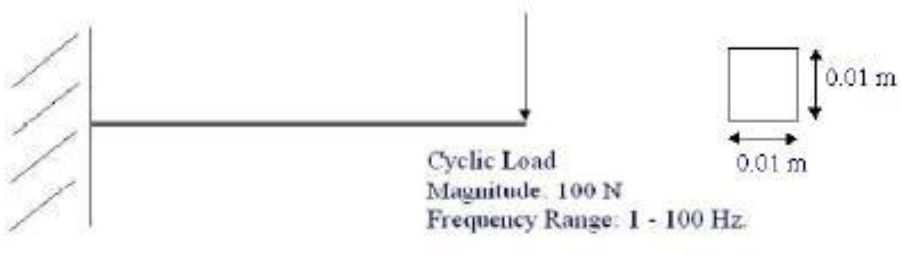
Time/Frequency – Freq and Sub stps – Enter the values of Harmonic freq range 1-100, Number of sub steps 100, Stepped – ok

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 the amplitude 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 = 206×10^9 N/m^2

Poisson’s ratio = 0.25

Weight Density = 7.83×10^3 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:

Able to analyse the harmonic analysis of 2D component using the ANSYS software.

Viva-voce

1. What is meant by harmonic analysis?
2. Distinguish between harmonic analysis and modal frequency analysis
3. -------------commands are used to joining the two link.
4. -------------commands are used to enter the material property.
5. What is meant by post processing?
6. What are real constants?
7. What are material properties required for harmonic analysis?
8. Define – Meshing
9. Define – Weight Density
10. What is meant by transient condition?
11. Name the three harmonic response analysis methods.
12. Define – Mode Superposition Method
13. Define – Reduced Method
14. Define – Full Method
15. What are the types of eigen value problems?

Expt. No.11 INTRODUCTION TO MATLAB

Aim:

To study the capabilities of MATLAB software

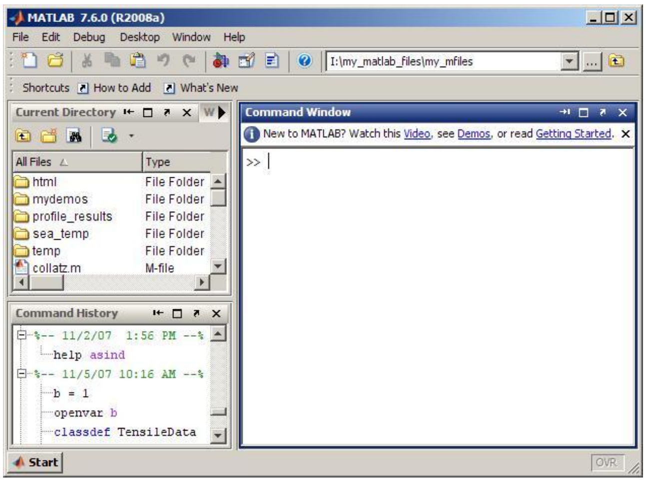
Introduction:

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 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 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.

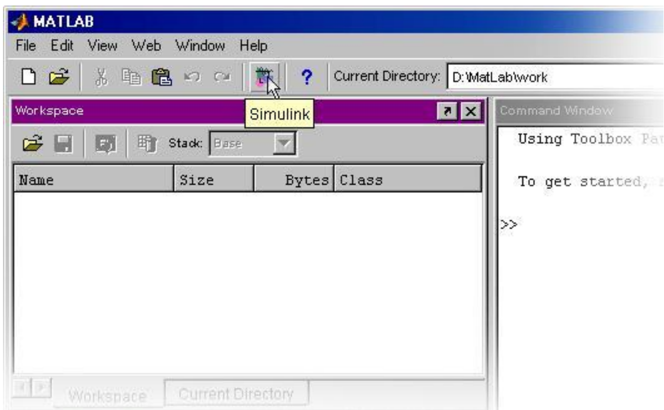
The idea behind these tutorials is that you can view them in one window while running Simulink in another window. System model files can be downloaded from the tutorials and opened in Simulink. You will modify and extend these systems while learning to use Simulink for system modeling, control, and simulation. 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

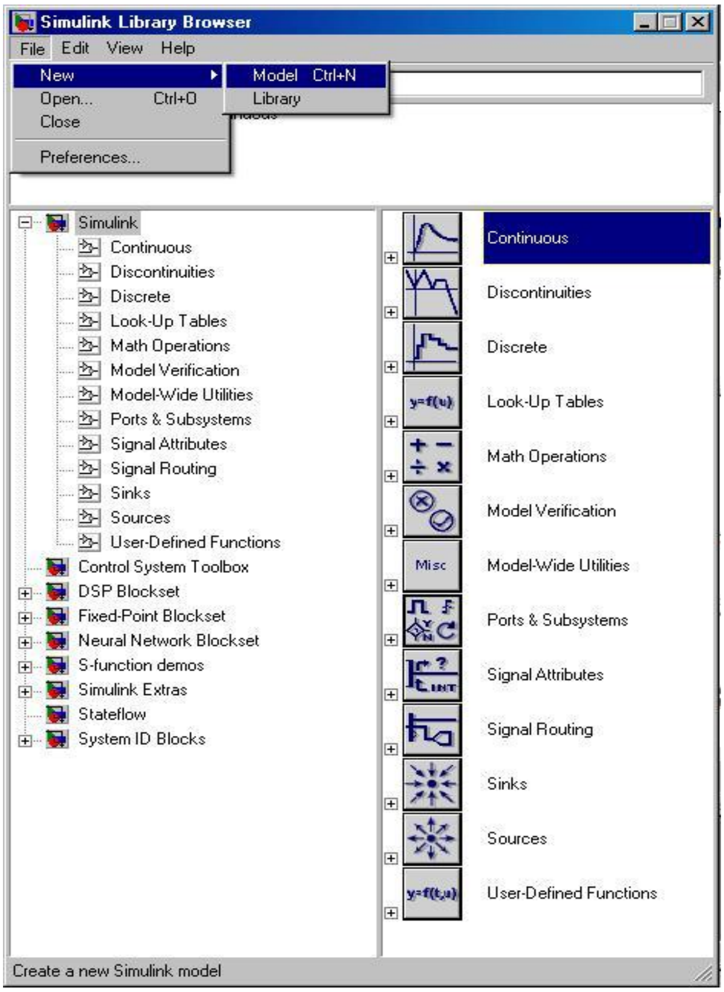
Starting Simulink

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

>> Simulink

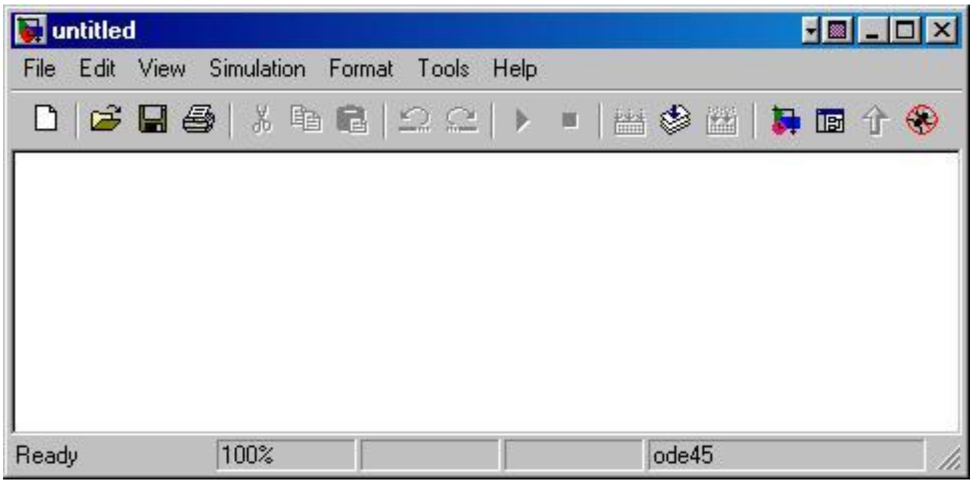


When it starts, Simulink brings up the Simulink Library browser.



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.



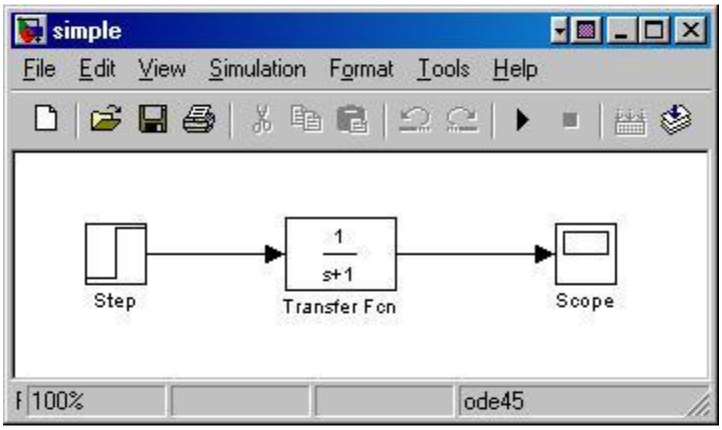
Model Files:

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.

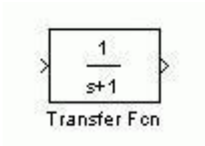
Blocks:

There are several general classes of blocks:

- Continuous
- Discontinuous
- Discrete
- Look-up tables
- Math operations
- Model verification

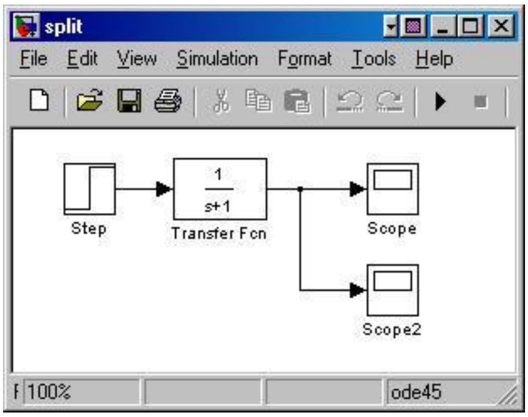
- Model-wide Utilities
- Ports & subsystems
- Signal attributes
- Signal routing
- Sinks: Used to output or display signals
- Sources: Used to generate various signals
- User-defined functions
- Discrete: Linear, discrete-time system elements (transfer functions, state-space models, etc.)
- Linear: Linear, 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.

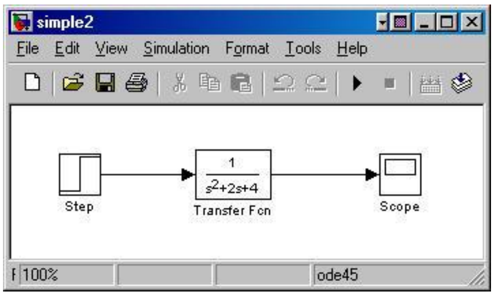


Lines can never inject a signal *into* another line; lines must be combined through the use of a block such as a summing junction.

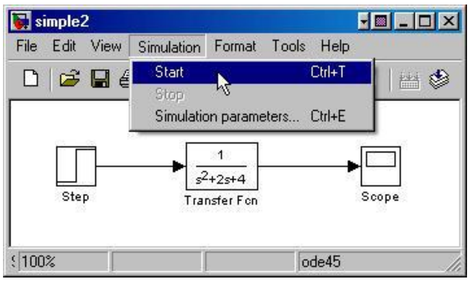
A signal can be either a scalar signal or a vector signal. For Single-Input, Single-Output systems, scalar signals are generally used. For Multi-Input, Multi-Output systems, vector signals are often used, consisting of two or more scalar signals. The lines used to transmit scalar and vector signals are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

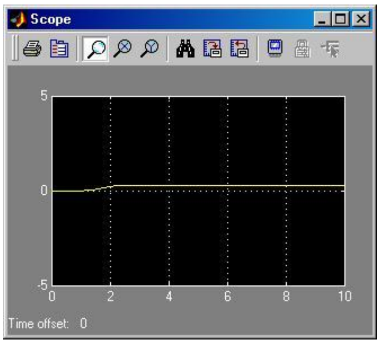
Running Simulations:

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



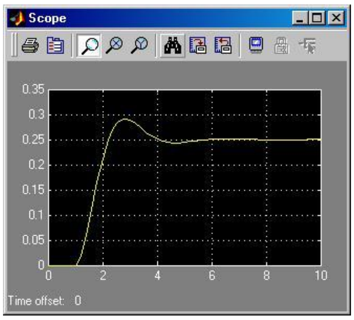
Before running a simulation of this system, first open the scope window by double-clicking on the scope block. Then, to start the simulation, either select Start from the Simulation menu (as shown below) or hit Ctrl-T in the model window.



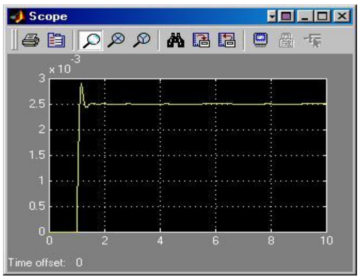


The simulation should run very quickly and the scope window will appear as shown below. If it doesn't, just double click on the block labeled "scope."

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

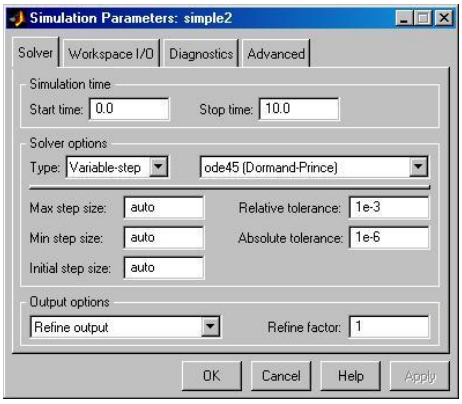


Note that the step response does not begin until t=1. This can be changed by double-clicking on the "step" block. Now, we will change the parameters of the system and simulate the system again. Double-click on the "Transfer Fcn" block in the model window and change the denominator to Re-run the simulation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



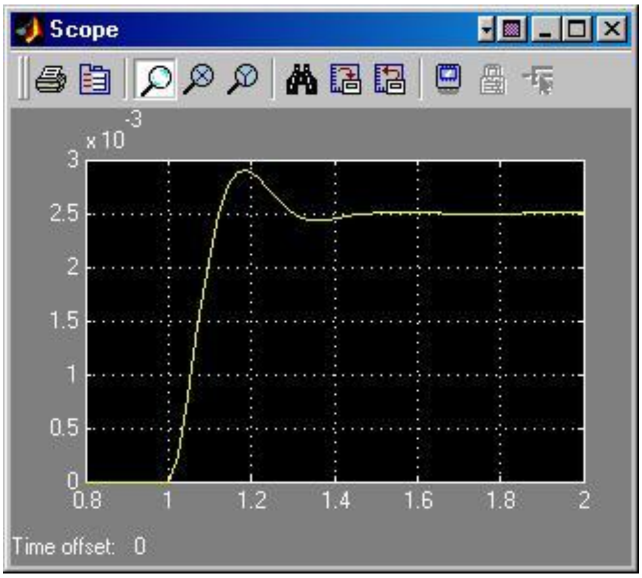
Notice that the autoscale button only changes the vertical axis. Since the new transfer function has a very fast response, it compressed into a very narrow part of the scope window. This is not really a problem with the scope, but with the simulation itself. Simulink simulated the system for a full ten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

Expt. No.12 SIMULATION OF SPRING-MASS SYSTEM USING MAT 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:

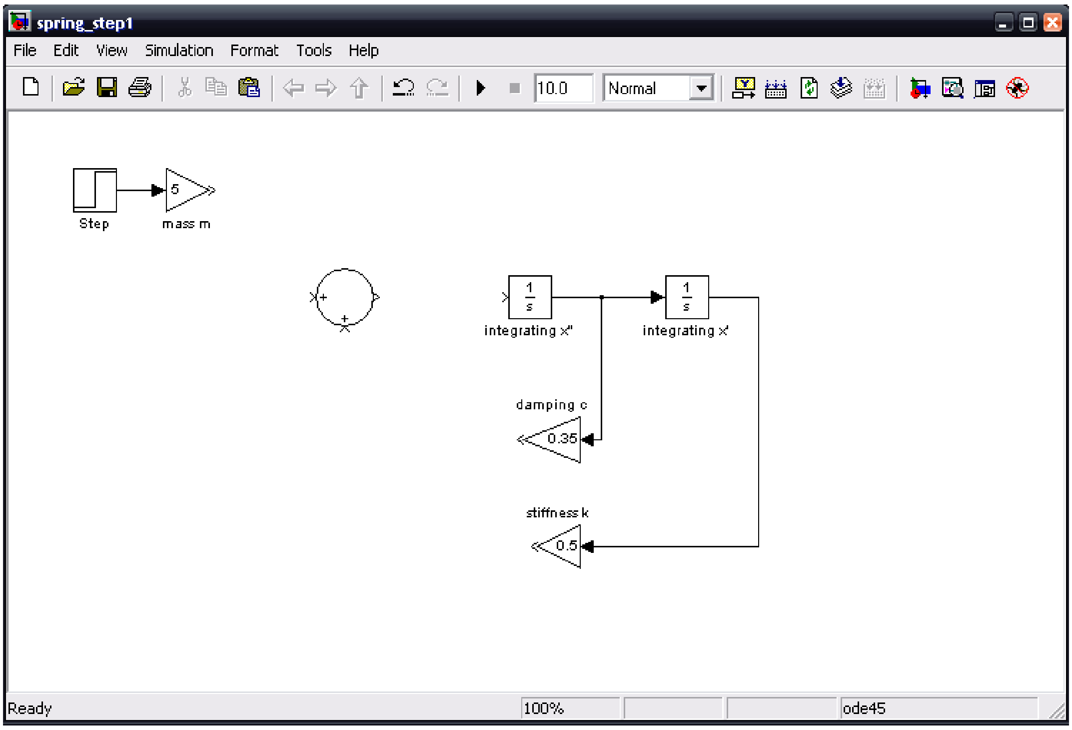
A mass on a spring with a velocity-dependent damping force and a time-dependent force acting upon it will behave according to the following equation:

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

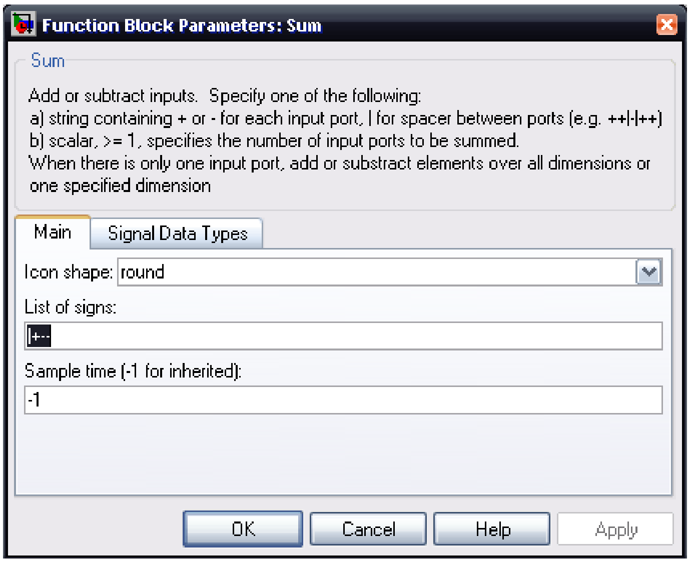
The model will be formed around this equation. In this equation, 'm' is the equivalent mass of the system; 'c' is the damping constant; and 'k' is the constant for the stiffness of the spring. First we want to rearrange the above equation so that it is in terms of acceleration; then we will integrate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

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

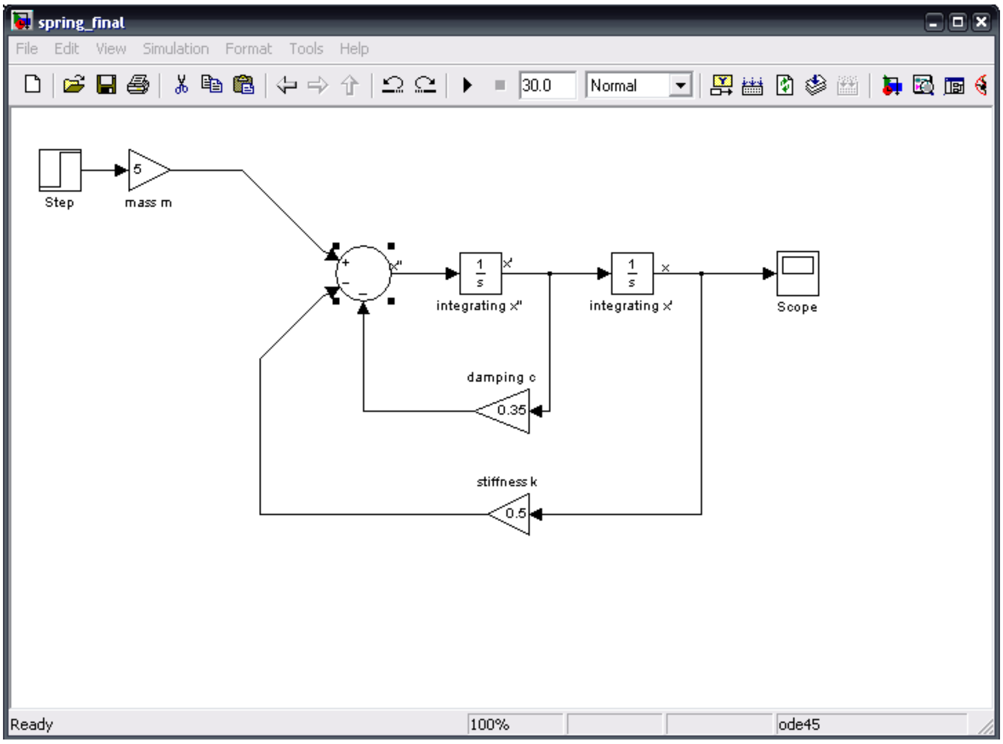
To build the model, we start with a 'step' block and a 'gain' block. The gain block represents the mass, which we will be equal to 5. We also know that we will need to integrate twice, that we will need to add these equations together, and that there are two more constants to consider. The damping constant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



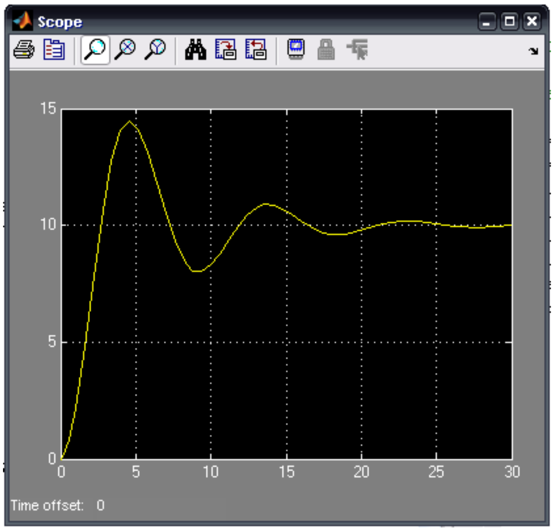
Once we have added places and corrected the signs for the sum block, we need only connect the lines to their appropriate places. To be able to see what is happening with this spring system, we add a 'scope' block and add it as follows:



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.



Outcome:

Able to simulate the spring-mass system using MATLAB software.

Viva-voce

1. What is meant by longitudinal vibration?
2. What is meant by transverse vibration?
3. Define – Degrees of freedom
4. Write the examples for free vibrations.
5. Write the examples for forced vibrations.
6. MATLAB stands for ___________________________
7. Define – One Degree of Freedom
8. Define – Stiffness
9. Define – Mode Shape
10. Define – Damping Factor
11. What is meant by eigen value?
12. What is meant by eigen vector?
13. Define – Natural Frequency
14. Define – Mass Matrix
15. Write the properties of stiffness matrix.

Expt. No.13 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	0 mm
End of first half of the cycle	180º	30 mm
End of the full cycle	360º	0 mm

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.

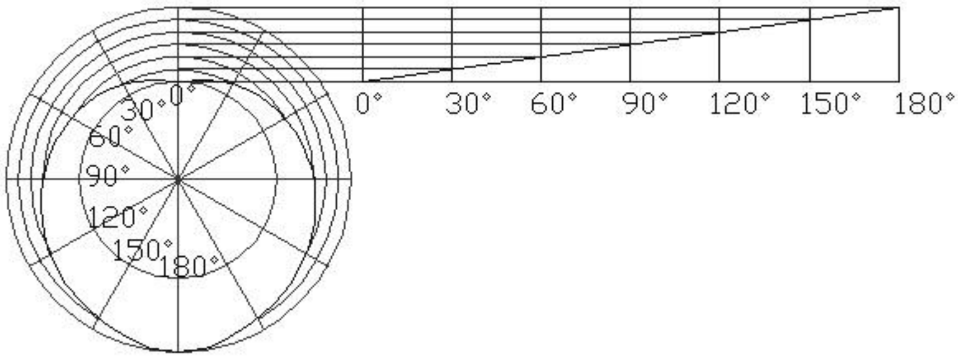
We shall take the cycle in steps. Firstly if the cam has to impart a displacement of 30mm on follower over half its cycle then it must impart a displacement of 30mm÷180º for every 1º turned by the cam i.e. it must move the follower 0.167mm per degree turn. This distance is to much to small to draw on 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 the length of these intervals will be decided on later).

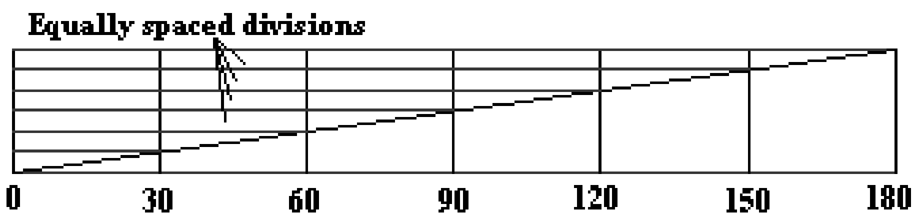
We shall consider this in terms of a displacement diagram:

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:

Able to simulate the cam and follower mechanism using MATLAB software.

Viva-voce

1. What are the functions of cam and follower?
2. What is meant by pressure angle?
3. Write the types of followers.
4. Define – Prime Circle
5. Define – Radial Follower
6. Define –Offset Follower
7. Write the types of motion of the follower.
8. Write the applications of cam and follower.
9. Define –Stroke
10. What is meant by knife edge follower?
11. What is meant by roller follower?
12. What are the advantages of roller follower than knife edge follower?
13. Define –Uniform Velocity Motion
14. Define –Simple Harmonic Motion
15. Define –Uniform Acceleration and Retardation

Expt. No.14 INTRODUCTION TO LS-DYNA

LS-DYNA is a highly advanced general purpose nonlinear finite element program that is capable of simulating complex real world problems. The distributed and shared memory solver provides very short turnaround times on desktop computers and clusters operated using Linux, Windows, and UNIX. With LS-DYNA, Livermore Software Technology Corporation (LSTC) aims to provide methods to seamlessly solve problems requiring

Multi-Physics Multi-Processing

Multiple Stages Multi-Scale

LS-DYNA is suitable to investigate phenomena involving large deformations, sophisticated material models and complex contact conditions for structural dynamics problems. LS-DYNA allows switching between explicit and different implicit time stepping schemes. Disparate disciplines, such as coupled thermal analyses, Computational Fluid Dynamics (CFD), fluid-structure interaction, Smooth Particle Hydrodynamics (SPH), Element Free Galerkin (EFG), Corpuscular Method (CPM), and the Boundary Element Method (BEM) can be combined with structural dynamics.

By determining product characteristics before a prototype is built, for many products LS-DYNA is the key to reducing time to market. Carrying out investigations with the aid of LS-DYNA supports the design of robust products with superior performance. F or pre- and post -processing, LS-DYNA comes with the LS-PrePost tool. LS-PrePost can be utilized to generate inputs and visualize numerical results. The software package LS-OPT for optimization and robust design is also supplied with LS-D YNA. With the option of multidisciplinary simulations, LS-D YNA significantly increases potentials for developing innovative products. These advantages contribute towards reducing development costs. All above-mentioned features and software packages are supplied as a single unit. LS-DYNA is not split for special applications, and the licensing scheme enables the different disciplines to be combined without limitations.

LS-DYNA has been developed in California for more than 20 years. It is the most frequently -used code for man y complex applications in structural nonlinear dynamics. Its usage is growing rapidly due to LS-DYNA's flexibility, enabling it to be applied to new disciplines. The new developments are driven in co-operation with leading universities from all over the world and new requirements requested by the vast customer base.

Analysis capabilities:

Different applications utilize one or a combination of the features listed below:

- Nonlinear dynamics
- Coupled rigid body dynamics
- Quasi-static simulations
- Normal modes
- Linear and nonlinear statics
- Eigen value analysis
- Thermal analysis
- Fluid analysis
- Eulerian capabilities
- Arbitrary Lagrangian Eulerian (ALE)
- Fluid-structure interactions
- Underwater Shock Analysis coupling (USA)
- Failure analysis
- Crack propagation
- Real-time acoustics
- Multi-physics coupling
- Structural-thermal coupling
- Adaptive re-meshing
- Smooth Particle Hydrodynamics (SPH)
- Element Free Methods (EFG)
- X-FEM
- CSE solver
- 2-D and 3-D formulations
- Arbitrary rigid to deformable

Applications:

1. LS-DYNA in metal forming:

The main application of LS-DYNA in metal forming is sheet metal stamping. The incremental approach of LS-DYNA allows the user to simulate multi-stage sheet metal stamping processes with a high degree of accuracy. The multiple core technology implemented, enables even large components with very high accuracy requirements to be simulated within just one hour. In addition, the simulation of the forming process can be complemented by simulating the trimming and spring back of the part.

The simulation may have different targets. One of them could be to determine the feasibility of a part in a forming process and its final geometry on completion of the various manufacturing processes. The information gained enables process parameters, forming sequences and optimal tool geometries to be determined. As a result, the part can be manufactured with a more accurate shape and better surface quality using fewer forming steps, thus lowering the cost per part.

Another target could be to design a hot forming process. The program allows the user to determine metal phase transformations encountered during cooling. Heating due to heat flow and radiation before, during, and after the forming process can all be analyzed. Thus, an analysis of the entire process can be carried out, from heating through forming right up to cooling, and only one model is required to predict the time needed for heating and cooling, the press requirements and part performance.

In addition to sheet metal forming, LS-DYNA is also capable of effectively analyzing other forming applications such as tube forming, cutting, extruding, impulse forming, forging, rolling, welding, hemming, flanging, electromagnetic forming and bending. Different disciplines are coupled for many of these applications. Features such as re-meshing, meshless methods, switching time-stepping schemes, ALE, thermal capacities, rigid body dynamics, among others, can be used simultaneously.

Many specific sheet metal stamping features come with LS-DYNA and the pre- and post-processing tool LS-Repost, which is included in the software package. The LS-DYNA solver has also been integrated into various forming simulation tools. These tools are provided by third party companies which supply highly effective support for specific forming applications and localization. A one step solver is currently under preparation and will soon be available in the standard LS-DYNA version.

2. LS-DYNA – a multi-purpose program for automotive suppliers:

LS-DYNA is essential for the virtual testing of various components in vehicles. The explicit and implicit time stepping schemes are capable of simulating static and dynamic tests using the same model. Component manufacturing processes are investigated by LS-D YNA using the metal forming and thermal capabilities. Hence, only one model is required to consider different problems. Ultimately, this results in lower costs for training and model creation compared to other solutions.

One example of the successful application of LS-DYNA is seat design. Seat manufacturers are able to consider static and dynamic load cases for seat frames as well as analyze the stability of belt anchor age points. LS-DYNA enables them to determine maximal locking mechanism loads or failure loads of seat tracks. LS-DYNA allows to investigate the influence of the seat on an occupant in a crash as well as the stamping process of a gear wheel. To enhance design and find a robust solution, this user group also often utilizes LS-OPT, a state-of-the-art optimization tool.

Other examples of similar beneficial applications of LS-DYNA are the design and manufacture of crash boxes, bumpers, front ends, dashboards, trimmings, and tires.

3. LS-DYNA in aerospace and defense:

LS-DYNA is a state-of-the-art program which can simulate high speed impacts, blasts and explosions. ALE and SPH methods are well suited for investigating high speed impacts on textiles, metal sheets and composites. The large library of constitutive equations with multiple options for material failure and non-localization complete the features required for many defense and aerospace applications.

Additionally, 2-D capabilities and automatic re-meshing and rezoning enable users to investigate axi-symmetric problems. The multi-physics capabilities of LS-D YNA, in conjunction with features developed for the automotive industry, facilitate the investigation of splashdown loads on tanks and rockets and emergency airplane landings. These features can also be used to optimize the design of airplane turbines and their blades with regard to bird collision.

4. LS-DYNA for drop test analysis:

LS-DYNA is used to investigate the behavior of products under impact conditions due to dropping. The application range includes consumer products, tools and also container design. For example, in the field of packaging design LS-DYNA helps to develop food containers capable of sustaining dynamic loading conditions during transport and storage.

Besides the broad spectrum of material models equipped with complex failure mechanisms, the flexible coupling and switching capabilities of LS-D YNA are essential for many applications. For instance, a liquid in a container can be modeled with the ALE or SPH method coupled with the structure. This allows the behavior of the liquid to be modeled accurately during impact.

To investigate cracks, the Element Free Galerkin (EFG) method can be used to eliminate mesh influence during crack propagation. To determine steady state deformation effectively, LS-DYNA provides the flexibility to switch time-stepping scheme arbitrarily between explicit and implicit. Furthermore, LS-DYNA also enables parts to be altered from rigid to deformable and vice versa. This feature is often used to determine the position of one part in relation to another during a falling phase before the main impact occurs.

5. LS-DYNA for containment:

In certain industries, there is a risk of accidents causing severe damage to communities or the environment. LS-DYNA acts as a tool to reduce such risks by generating knowledge about how a system may fail. Thus, design changes can be made to reduce or even eliminate risks associated with the load cases considered.

One example is containers utilized to transport nuclear fuel elements. It is essential that containers remain closed and tightly sealed in the case of any predictable accident which might occur during transport. LS-DYNA is used to design the transport container, its interior and the energy-absorbing buffers around the enclosing hull of the container.

Another example is the high-speed impact on objects, like a turbine blade separating from a turbine that ma y not damage the embankment dam. LS-DYNA allows the user to estimate the damage caused by a turbine blade hitting the turbine housing. As such types of simulation require the exact prediction of the post -failing phase, extensive preliminary material testing is needed. However, due to the high potential of severe damage, this effort is well spent.