St. Anne's college of Engineering and Technology

DEPARTMENT OF MECHANICAL ENGINEERING

ME8711 SIMULATION AND ANALYSIS LABORATORY

LABORATORY MANUAL

REGULATION - 2017

Prepared By: K.SHANMUGA ELANGO,

ASST. PROFESSOR / MECH

ME8711 SIMULATION AND ANALYSIS LABORATORY

OBJECTIVES:
 To give exposure to software tools needed to analyze engineering problems.
To expose the students to different applications of simulation and analysis tools.

LIST OF EXPERIMENTS

A. SIMULATION
1. MATLAB basics, Dealing with matrices, Graphing-Functions of one variable and two variables
2. Use of Matlab to solve simple problems in vibration
3. Mechanism Simulation using Multibody 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 plates.
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.

                                         TOTAL: 60 PERIODS

OUTCOMES:
Upon the completion of this course the students will be able to
CO1 simulate the working principle of air conditioning system, hydraulic and pneumatic cylinder and cam follower mechanisms using MATLAB.
CO2 analyze the stresses and strains induced in plates, brackets and beams and heat transfer problems. CO3 calculate the natural frequency and mode shape analysis of 2D components and

INTRODUCTION TO FEA AND ANSYS

WHAT IS FINITE ELEMENT ANALYSIS
Finite Element Analysis, commonly called FEA, is a method of numerical analysis. FEA is used for solving problems in many engineering disciplines such as machine design, acoustics, electromagnetism, soil mechanics, fluid dynamics, and many others. In mathematical terms, FEA is a numerical technique used for solving field problems described by a set of partial differential equations.

In mechanical engineering, FEA is widely used for solving structural, vibration, and thermal problems. However, FEA is not the only available tool of numerical analysis. Other numerical methods include the Finite Difference Method, the Boundary Element Method, and the Finite Volumes Method to mention just a few. However, due to its versatility and high numerical efficiency, FEA has come to dominate the engineering analysis software market, while other methods have been relegated to niche applications. You can use FEA to analyze any shape; FEA works with different levels of geometry idealization and provides results with the desired accuracy. When implemented into modern commercial software, both FEA theory and numerical problem formulation become completely transparent to users.

WHO SHOULD USE FINITE ELEMENT ANALYSIS
As a powerful tool for engineering analysis, FEA is used to solve problems ranging from simple to very complex. Design engineers use FEA during the product development process to analyze the design-in-progress. Time constraints and limited availability of product data call for many simplifications of the analysis models. At the other end of scale, specialized analysts implement FEA to solve very advanced problems, such as vehicle crash dynamics, hydro forming, or air bag deployment. This book focuses on how design engineers use FEA as a design tool. Therefore, we first need to explain what exactly distinguishes FEA performed by design engineers from "regular" FEA. We will then highlight the most essential FEA characteristics for design engineers as opposed to those for analysts.

WHY IS FEA NEEDED

➢ To reduce the amount of prototype testing.

➢ Computer Simulation allows multiple "what if"scenarios to be tested quickly and effectively.

➢ To simulate designs those are not suitable for prototype testing. E.g. Surgical Implants such as an artificial knee.

ABOUT ANSYS:

➢ ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering. ANSYS is a virtual Prototyping technique used to iterate various scenarios to optimize the product.

GENERAL PROCEDURE OF FINITE ELEMENT ANALYSIS:

➢ Creation of geometry or continuum using preprocessor.

➢ Discretization of geometry or continuum using preprocessor.

➢ Checking for convergence of elements and nodes using preprocessor.

➢ Applying loads and boundary conditions using preprocessor.

➢ Solving or analyzing using solver

➢ Viewing of Results using postprocessor.

Build Geometry:

➢ Construct a two (or) three dimensional representation of the object to be modeled and tested using the work plane co-ordinate system in Ansys.

Define Material Properties:

➢ Define the necessary material from the library that composes the object model which includes thermal and mechanical properties.

Generate Mesh:

➢ Now define how the model system should be broken down into finite pieces

Apply Loads:

➢ The last task in preprocessing is to restrict the system by constraining the displacement and physical loading.

Obtain Solution:

➢ The solution is obtained using solver available in ANSYS. The computer can understand easily if the problem is solved in matrices.

Present the Result:

➢ After the solution has been obtained there are many ways to present Ansys result either in graph or in plot.

SPECIFIC CAPABILITIES OF 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 the 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 iteration between two (or) more fields of engineering analysis. Pressure vessels, Induction heating and Micro electro mechanical systems are few examples
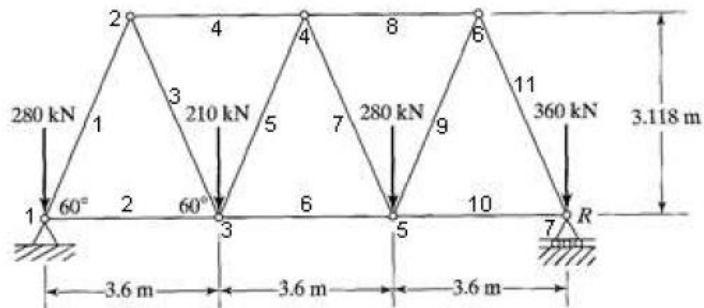
RESULT:
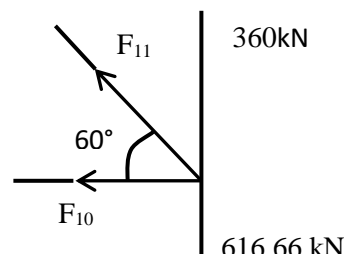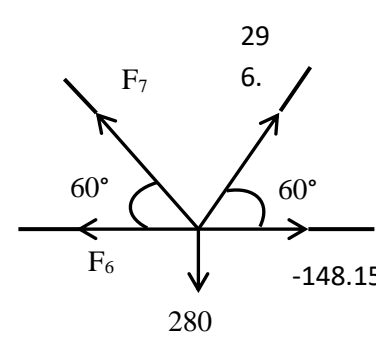Thus the basics of FEA and ANSYS are studied.

1. TRUSS ANALYSIS

Ex: No: 1

AIM: To determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm^2).

Analytical Solution:

$\Sigma H=0$ (sum of the horizontal force is equal to zero) $\Sigma v=0$ (sum of the vertical force is equal to zero) $\Sigma M=0$ (sum of the moment force is equal to zero) $\Sigma Moment=0$ $0=-210\times3.6-280\times7.2-360\times10.8+R_2\times10.8$ $R_2=616.66$ kN $\Sigma v=0$ $=280-210-280-360+617+R_1=0$ $R_1=513$ kN Node 7 Consider the force is away from the joints.	Node 5

Node 5 figure and Node 7 figure follow.

(Node 7 figure: F_{11}, 360kN, 60°, F_{10}, 616.66 kN)	(Node 5 figure: 29, F₇, 6., 60°, 60°, F₆, -148.15, 280)
$\Sigma v=0$ $-360+616+F_{11}\sin 60°=0$ $F_{11}=-296.3$ $\sigma_{11} = \frac{-296.3}{3250} = -91.8\ MPa$ $\Sigma H=0$ $F_{10}+F_{11}=\cos 60°=0$ $F_{10}=-F_{11}\cos 60°$ $F_{10}=-296.3\times\cos 60°$ $F_{10}=-148.15$ $\sigma_{10} = \frac{-148.15}{3250} = -45.58\ MPa$	$\Sigma v=0$ $-280+-296.3\sin 60°+F_7 \sin 60°=0$ $F_7 = \frac{23.3}{sin60°} = 27.016$ $\sigma_7 = \frac{27.016}{3250} = 8.312\ MPa$ $\Sigma H=0$ $148.15-F_6+296.3\cos 60°+27.016$ $\cos120°=0$ $148.15+134.642= F_6$ $F_6=282.792$ $\sigma_6 = \frac{282.792}{3250} = 87.012\ MPa$

ANSYS Analysis Procedure:

1. **Utility menu bar:** select **File > Change Title**:
    Enter the title (Give title of your choice) in the window appeared and **Click 'OK'.**
2. **ANSYS Main Menu: Preferences > Structural>** Click 'OK'
3. **ANSYS Main Menu: Preprocessor>Element Type > Add/Edit/Delete > in the** window click
    Add>select Link (sub window 1)> 2d spar (sub window 2)> Click 'OK'
4. **ANSYS Main Menu: Preprocessor>Real Constants > Add/Edit/Delete>** Click **Add...** and select
    'LINK1'> Click on 'OK'. The **Real Constants** window will appear> input **AREA** = 3250>Click on
    'OK'. 'Set 1' now appears in the dialog box. Click on 'Close' in the 'Real Constants' window.
5. **ANSYS Main Menu: Preprocessor>Material Props > Material
    Models>Structural>Linear>Elastic>Isotropic>**in the Linear elastic isotropic material model
    window, input **E=200000 and Poisson's ratio=0.3**
6. **ANSYS Main Menu: Preprocessor > Modeling > Create > Keypoints > In Active CS, in** the
    create key points in active coordinate system, **type key point no. and its coordinates. Here**
    KP   x   y   z   From the problem figure, calculate the key point values

1   0   0   0 (Click 'apply')
2 1800 3118   0 (Click 'apply')
3 3600   0   0 (Click 'apply')
4 5400 3118   0 (Click 'apply')
5 7200   0   0 (Click 'apply')
6 9000 3118   0 (Click 'apply')
7 10800   0   0 (Click 'ok')

7. **In the Command window bar, type :**
L,1,2 (enter)   L,1,3 (enter) L,2,3 (enter) L,2,4 (enter) L,3,4 (enter) L,4,5 (enter) L,3,5 (enter) L,5,6 (enter) L,4,6 (enter)   L,6,7 (enter) L,5,7 (enter)

8. **ANSYS Main Menu: Preprocessor>Meshing > Size Cntrls> Manual Size> Lines > All Lines,** in the Element sizes on All selected Lines window, **input in the 'NDIV' – No. of divisions field = 1, Click 'OK'**

9. **ANSYS Main Menu: Preprocessor>Meshing > Mesh > Lines,** Click 'Pick All' in the 'Mesh Lines' Window.

10. **ANSYS Main Menu: Solution >** select **Analysis Type > New Analysis> Tick Static, Click 'OK'**

11. **ANSYS Main Menu: Solution >Define Loads > Apply > Structural > Displacement** > **On Key points,** Type key point **1,7** in the Apply U, ROT on KPs window **click 'OK'**, then **select 'All dof'** in the Apply U, ROT on KPs window (Check Apply as field with **' Constant value'** and **VALUE = 0**), then **Click 'OK'**

12. **ANSYS Main Menu: Solution >Define Loads > Apply > Structural > Force/Moment > on Key points, Type key point 3** in the Apply F/M on KPs window **click 'OK',** in the Apply F/M on KPs window, set Direction of force field as **FY**, Apply as field as **Constant** and VALUE = **-280000,** then **Click 'OK'**

13. **Similarly, Apply loads in the Key points 1, 5, and 7 with load -210000, -280000 and -360000 respectively.**

14. **ANSYS Main Menu: Solution >Solve > Current LS> Click Solve**.

15. **Click 'Close' in Solution is Done window.**

16. **ANSYS Main Menu: General Postproc> List Results > Reaction Solution, in the List Reaction Solution window, select 'All Struc force F', Click 'OK'.**

17. Print Solution window will show **the Reaction results.**

18. **ANSYS Main Menu: General Postproc> Plot Results > Deformed Shape,** in the Plot Deformed shape window, **Tick deformed + undeformed edge (shows the deformed shape of the structure in the ANSYS main window).**

19. **ANSYS Main Menu: General Postproc> Plot results > Contour Plot > Nodal Solution, in the Contour Nodal Solution Data, click DOF solution > Displacement vector sum, Click 'OK'. (Shows the displacement contour of deformed structure in the ANSYS main window).**

20. **ANSYS Main Menu: General Postproc> Element Table > Define Table, in the Element table data window, Click 'ADD'.**

21. **In the Additional Element table items window, in the User defined label field, type 'SAXL', and select 'By sequence numb' in the Item, Comp Results data item field (sub window1) and 'LS' (sub window2), then type 'LS, 1' in the field below the sub window2 and Click 'OK'.**

22. **ANSYS Main Menu: General Postproc> Element Table > Plot Elem Table, in the Item to be plotted field, select 'SAXL' and click 'OK'. (Shows the Axial load contour of deformed structure in the ANSYS main window).**

23. **ANSYS Main Menu: General Postproc> Element Table > List Element Table.**

24. Print Solution window will show **the Axial load results on each element.**

MERITS:1.time saved

2.achived high accuracy.

APPLICATIONS: Beams are structural members, which are most commonly used in buildings. Beams have numerous other applications in case of bridges, automobiles or in mechanical systems. In this article we will see how we can do strength analysis of a beam

RESULT:
Thus the performance of nodal deflections, reaction forces, and stress for the truss system was analyzed and animated

Ex: No: 2 STRESS ANALYSIS OF PLATE WITH CIRCULAR HOLE
AIM:
To determine the stress acting on a plate with circular hole due to the applied external pressure.
SOFTWARE REQUIRED:
Ansys 14.0
PROCEDURE:

1. Preferences → structural → ok.

2. Preprocessor → element type → add → add → solid → quad 4 node 182 → ok.

Option → k1-full integration → k2-plane stress → k3-pure displacement → ok → close.

3. Material property → material models → structure → linear → elastic → isotropic → (enter the following values on the boxes) PX = 2e5 and PRXY = 0.3 → ok → close.

4. Modeling → create → area → rectangle → by 2 corners → (enter the following values on the boxes) X = 0; Y = 0; width = 100 height = 100 → ok.

5. Work plane → off set wp to → key points + → (select the two opposite corners of solid rectangle) → ok → (now axis move to centre of rectangle).

Create → solid circle → radius (10) → ok. (Small circle generated)

6. Operate → Booleans → subtract → area → (select larger area) → ok → ok (select smaller area) → next → ok → ok. (now hole is generated on the solid rectangle)

7. Meshing → mesh tool → global → set size (3) → ok → mesh → (select the area) → ok

8. Solution → define load → apply → structural → displacement → on lines → (select the line) → ok → ALL DOF → ok.

9. Pressure → on lines → (select the line) → ok. Load value → (-500) → ok.

(To see the number of pressure lines do the following procedure). Plot control → symbols → show pressures → arrows → ok. (Next move to) Plot → multi plot → type SBCT → (press enter)

10. Solve → current LS → ok

11. General post processor → plot result → counter plot → nodal solution → structural → Stress (or) Displacement sum → von misses stress (or) U sum → ok.

12. Plot control → animates → deformation results →stress (or) displacement sum → von misses stress (or) U sum → ok

MERITS: 1.time saved
2.achived high accuracy.

APPLICATIONS: Geometric irregularities such as holes, notches, keyways, shoulders provided on shafts etc. are common features provided in machine members. But such irregularities often lead to stress concentration near the irregularity due to which the stress near the irregularity is higher than the average stress in the whole member

RESULT:
Thus the performance of the stress analysis of a plate with a circular hole was analyzed and animated.

Ex: No: 4 STRESS ANALYSIS OF AXI – SYMMETRIC COMPONENT
AIM:
To determine the stress analysis on a axi symmetric component.
SOFTWARE REQUIRED:
Ansys 14.0
PROCEDURE:

1. Preferences → structural → ok

2. Preprocessor → element type → add → add → solid → quad 4 node 182 → ok.

Option → k1- full integration → k2 – plane stress k3 – axis symmetric → ok → close.

3. Material property → material models → structure → linear → elastic → isotropic → (enter the following values on the boxes) PX = 2.1e4 and PRXY = 0.3 → ok → close. And then select → density (type the value) 7.85e-6 → ok → close.

4. Modeling → create → area → rectangle → by 2 corners → (enter the following values on the boxes) → X = 500; Y = 0; width = 50 height = 100 → ok.

5. Meshing → mesh tool → global → set size (5) → ok → mesh → (select the area) → ok

6. Solution → define load → apply → structural → displacement → on lines → (select the bottom line of the rectangle) → ok → UY → ok.

7. Inertia → angular velocity → global → Y = (2*3.14*3000)/60 (ie ω = 2πN/60) → ok

8. Solve → current LS → ok

9. General post processor → plot result → counter plot → nodal solution → stress (or) displacement sum → von misses stress (or) U sum → ok

10. Plot control → animates → deformation results → stress (or) displacement sum → von misses stress (or) U sum → ok.

RESULT:
Thus the stress distribution of axi symmetric component is analyzed ,animated and plotted.

Ex: No: 5 (a) STRESS ANALYSIS OF BEAM
(Cantilever Beam)
AIM:
To determine the stress acting on a cantilever beam due to the applied external load.

SOFTWARE REQUIRED:
Ansys 10.0
PROCEDURE:

1. Preferences → structural → ok.

2. Preprocessor → element type → add → add → beam → 2 node 188 → ok.

3. Material property → material models → structure → linear → elastic → isotropic → (enter the following values on the boxes) → PX = 2e5 and PRXY = 0.3 → ok → close.

4. Sections → beam → common section → width = 100; height = 100 → ok.

5. Modeling → create → key points → In active CS

1st 0 0 0 → apply
2nd 1000 0 0 → ok
(1st) & (2nd) Key points Generated.

6. Modeling →Create→ line →straight line select 1st and 2nd point →ok

7. Meshing → mesh tool → global → set size (10) → ok → mesh → select the line → ok

8. Solution → define → load → apply → structural → displacement → on key points (select the 1st point) → ok → All DOF → ok.

9. Force → key points → (select the point where we apply the load) →FY (- 500) → ok.

10. Solve → current LS → ok.

11. General post processor → plot result → counter plot → nodal solution → structural → stress → von misses stress → ok.

12. Plot control → animates → deformation results → stress → von misses stress →ok.

13. Element table → Add → Select Sum Sequence all → Type 6, Apply, Select sum sequence all → Type 19, Apply Select sum sequence all → Type 2, Apply, Select sum sequence all, Type 15 → ok.

14. Plot result → counter polt → Line element result → select 6,19 to get shear force diagram, select 2,15 to get Bending moment diagram.

RESULT:

Thus the performance of the stress analysis of a cantilever beam was analyzed, animated, shear force and bending moment diagram was plotted

Ex: No: 5 (b) STRESS ANALYSIS OF BEAM
(Simply Supported Beam)
AIM:

To determine the stress acting on a simply supported beam due to the applied external load.

SOFTWARE REQUIRED:

Ansys 10.0

PROCEDURE:

1. Preferences → structural → ok

2. Preprocessor → element type → add → add → beam → 2 node 188 → ok → close.

3. Material property → material models → structure → linear → elastic →isotropic → (enter the following values on the boxes) PX = 2e5 and PRXY = 0.3 → ok → close.

4. Sections → beam → common section → width = 100; height = 100 → ok.

5. Modeling → create → key points → In active CS

1_{st} 0 0 0 → apply
2_{nd} 1000 0 0 → ok
(1_{st}) & (2_{nd}) Key points Generated.

6. Modeling → Create → line → straight line → select 1_{st} and 2_{nd} point →ok.

7. Meshing → mesh tool → global → set size (10) → ok → mesh select the line → ok.

8. Solution → define load → apply → structural → displacement → on key points (select the 1_{st} and 2_{nd} key points) → ok → UY & ROT UY→ ok.

9. Force → node → (select the node where we apply the load) → FY (-500) → ok.

10. Solve → current LS → ok.

11. General post processor → plot result → counter plot → nodal solution → structural → stress → von misses stress → ok.

12. Plot control → animates → deformation results → stress → von misses stress → ok.

13. Element table → Add → Select Sum Sequence all → Type 6, Apply, Select sum sequence all → Type 19, Apply Select sum sequence all → Type 2, Apply, Select sum sequence all, Type 15 → ok.

14. Plot result → counter polt → Line element result → select 6,19 to get shear force diagram, select 2,15 to get Bending moment diagram.

RESULT:
Thus the performance of the stress analysis of a simply supported beam was analyzed, animated, shear force and bending moment diagram was plotted.

Ex: No: 5(c) STRESS ANALYSIS OF BEAM
(Fixed Beam)
AIM :
To determine the stress acting on a fixed beam due to the applied external load.
SOFTWARE REQUIRED:
Ansys 10.0
PROCEDURE:

1. Preferences → structural → ok.

2. Preprocessor → element type → add → add → beam → 2 node 188 → ok → close.

3. Material property → material models → structure → linear → elastic isotropic → (enter the following values on the boxes)→ PX = 2e5 and PRXY = 0.3 → ok → close.

4. Sections → beam → common section → width = 100 ; height = 100 → ok.

5. Modeling → create → key points → in active → CS

1_{st} 0 0 0 → apply
2_{nd} 1000 0 0 → ok
(1_{st}) & (2_{nd}) key points Generated.

6. Modeling →Create → line →straight line →select 1_{st} and 2_{nd} point →ok

7. Meshing → mesh tool → global set → size (10) → ok → mesh →select the line→ ok.

8. Solution → define load → apply → structural → displacement → on key points (select the 1st and 2nd key points) → ok → All DOF → ok .

9. Force → node → (select the node where we apply the load) → FY (-500) → ok.

10. Solve → current LS → ok.

11. General post processor → plot result → counter plot → nodal solution → structural → stress → von misses stress → ok.

12. Plot control → animates → deformation results → stress → von misses stress → ok.

13. Element table → Add → Select Sum Sequence all → Type 6, Apply, Select sum sequence all → Type 19, Apply Select sum sequence all → Type 2, Apply, Select sum sequence all, Type 15 → ok.

14. Plot result → counter polt → Line element result → select 6,19 to get shear force diagram, select 2,15 to get Bending moment diagram.
MERITS: 1.time saved
2.achived high accuracy.

APPLICATIONS:dynamic analysis of machine elements

RESULT:
Thus the performance of the stress analysis of a simply supported beam was analyzed, animated, shear force and bending moment diagram was plotted.

Ex: No: 6(a) MODE FREQUENCY ANALYSIS OF BEAMS
(Cantilever Beam)
AIM:
To determine the stress due to mode frequencies on a cantilever beam due to the applied external load.
SOFTWARE REQUIRED:
Ansys 14.0
PROCEDURE:

1. Preferences → structural → ok.

2. Preprocessor → element type → add → add → beam → 2 node 188 → ok → close.

3. Material property → material models → structure → linear → elastic → isentropic → (enter the following values on the boxes) → PX = 2e5 and PRXY = 0.3 →ok →close and then select density (type the value) → 7850e-12 → ok → close.

4. Sections → beam → common section → width = 10; height = 10 → ok.

5. Modeling → create → key points → In active CS

1_{st} 0 0 0 → apply
2_{nd} 1000 0 0 → ok
(1_{st}) & (2_{nd}) key points Generated.

6. Modeling → Create → line → straight line → select 1_{st} and 2_{nd} point → ok

7. Meshing → mesh tool → global → set size (10) → ok → mesh → select the line → ok.

8. Solution → define load → apply → structural → displacement → on key points → (select the 1_{st} point) → ok → All DOF →ok.

9. Solution → analysis type → new analysis → modal → ok.

10. Analysis option → (enter the following value) → exact = 10 expand = 10 → ok → default → ok.

11. Solve → current LS → ok

12. General post processor → read result → by pick (from the set-frequency table) → close

13. Read result → first set→ plot result → contour plot → nodal solution → DOF solution → displacement vector sum → ok

14. Read result → next set → (right click on the screen → replot (difference mode will come on the screen)

15. Plot control → animates →mode shape →DOF solution → ok.

RESULT:
Thus the performance of the mode frequencies on a cantilever beam was analyzed and animated.

Ex.No.6 (b) MODE FREQUENCY ANALYSIS OF BEAMS
(Simply Supported Beam)
AIM:
To determine the stress due to mode frequencies on a simply supported beam due to the applied external load.
SOFTWARE REQUIRED:
Ansys 14.0
PROCEDURE:

1. Preferences → structural → ok.

2. Preprocessor → element type → add → add → beam → 2 node 188 → ok → close.

3. Material property → material models → structure → linear → elastic → isotropic → (enter the following values on the boxes) → PX = 2e5 and PRXY = 0.3 → ok → close. and then select → density (type the value) → 7850e-12 → ok → close.

4. Sections → beam → common section → width = 10; height = 10 → ok.

5. modeling → create → key points → in active → CS

1st 0 0 0 → apply
2nd 1000 0 0 → ok
(1st) & (2nd) key points Generated.

6. Modeling → Create → line → straight line → select 1st and 2nd point → ok.

7. Meshing → mesh tool → global → set → size (10) → ok → mesh select the line → ok.

8. Solution → define load → apply → structural → displacement → on key points → (select the 1st and 2nd key points) → ok →UY & ROT UY → ok.

9. Solution → analysis type → new analysis → modal → ok.

10. Analysis option (enter the following value) → exact = 10 expand = 10 → ok → default → ok

11. Solve → current LS → ok.

12. General post processor → read result → by pick → (from the set-frequency table) → close.

13. Read result → first set → plot result → contour plot → nodal solution → DOF solution → displacement vector sum → ok

14. Read result → next set → (right click on the screen → replot (difference mode will come on the screen)

15. Plot control → animates → mode shape → DOF solution →Displacement vector sum→ ok.

RESULT:
Thus the performance of the mode frequencies on a simply supported beam was analyzed and animated.

Ex.No.6(c) MODE FREQUENCY ANALYSIS OF BEAMS
(Fixed Beam)
AIM:
To determine the stress due to mode frequencies on a fixed beam due to the applied external load.
SOFTWARE REQUIRED:
Ansys 14.0
PROCEDURE:

1. Preferences → structural → ok.

2. Preprocessor → element type → add → add → beam → 2 node 188 → ok → close.

3. Material property → material models → structure → linear → elastic → isotropic → (enter the following values on the boxes)→ PX = 2e5 and PRXY = 0.3 → ok → close. and then select → density (type the value) → 7850e-12 → ok → close.

4. Sections → beam → common section → width = 10; height = 10 → ok.

5. Modeling → create → key points → In active CS

1_{st} 0 0 0 → apply
2_{nd} 1000 0 0 → ok
(1_{st}) & (2_{nd}) Key points Generated.

6. Modeling → Create → line → straight line → select 1_{st} and 2_{nd} point → ok.

7. Meshing → mesh tool → global → set → size (10) → ok → mesh select the line → ok.

8. Solution → define load → apply → structural → displacement → on key points → (select the 1_{st} and 2_{nd} key points) → ok → All DOF → ok

9. Solution → analysis type → new analysis → modal → ok

10. Analysis option → (enter the following value) → exact = 10 expand = 10 → ok → default → ok

11. Solve → current LS → ok

12. General post processor → read result → by pick (from the set-frequency table) → close

13. Read result → first set → plot result → contour plot → nodal solution → DOF solution → displacement vector sum → ok

14. Read result → next set → (right click on the screen → replot (difference mode will come on the screen)

15. Plot control → animates → mode shape → DOF solution → Displacement vector sum→ ok.

MERITS: 1.time saved
2.achived high accuracy.

APPLICATIONS: dynamic analysis of machine elements

RESULT:
Thus the performance of the mode frequencies on a fixed beam was analyzed and animated.

Ex: No: 7 HARMONIC ANALYSIS OF A 2D COMPONENT
AIM:
To determine the Harmonic response of a 2D component
SOFTWARE:
Ansys 10.0

PROCEDURE:

1. Start →Program → Ansys → Ansys product launcher

2. Preference → Structural →OK

3. Preprocessor → Element Type → Add→ Add→ Beam → 2 node 188 → Ok

4. Material Properties → Material Model → Structural → Linear → Elastic → Isontropic → E = 2e5, γ = 0.3 → Ok → Density = 7860e-12 → Ok

5. Section → Beam → Common Section → width = 10, Height = 10 → Ok

6. Modelling → Create → Keypoints → In Active CS → X1 = 0, Y1 = 0, Z1 = 0 → Apply → X2 = 1000, Y2 = 0, Z = 0 → Ok

7. Modelling → create → Line → Lines → Straight Line → Select the two points → Ok

8. Meshing → Mesh Tool → Global , Set → Element type = 10 → Ok → Mesh → Select the Line → Ok

9. Solution → Analysis Type → New Analysis → Harmonic → Ok

10. Solution → Define Load → Apply → Structural → Displacement → on key points → Select 1st key point → Ok → ALL DOF → Ok

11. Solution → Define Load → Apply → Structural → Force → on Key points → Select 2nd key points → Y= -100 → Ok

12. Solution → Load step opts → Time and Frequency → Frequency and substeps → Harmonic Frequency range = 0 , 100 ; No of substep = 100 → Ok

13. Solve → Current LS → Ok

14. Time Hist Postprocessor → Add Data → DOF → Y Displacement → Ok
Select the Maximum deflection point → Ok
At Top click Graph Data → Result comes for without damping condition

15. Solution → Load Step opts → Time and Frequency → Frequency and substeps → Harmonic Frequency range = 0 , 100 ; Substep = 100 → Ok
Damping → Constant Damping ratio → 0.1 → Ok

16. Solve → Current LS → Ok

17. Time Hist Postprocessor → Add Data → DOF → Y Displacement → Ok

Select the maximum Deflection point → Ok
At Top Click Graph Data → Result comes for with Damping condition
MERITS: 1.time saved
2. Achived high accuracy.

APPLICATIONS: Vibration, mechanical and acoustic measurements — Material testing and measurements with in-house testing stand

RESULT:
Thus the Harmonic response of the 2D component has been analyzed and animated.

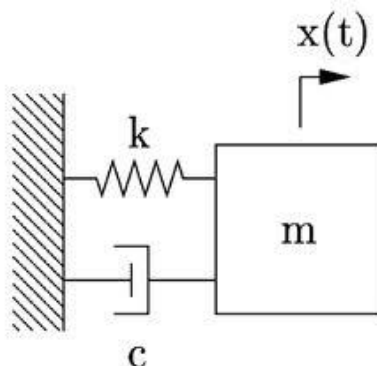# Ex: No: 9. TRANSIENT ANALYSIS OF A SPRING MASS DAMPER SYSTEM USING ANSYS

AIM:
To determine the transient analysis of a spring mass damper system by using ansys

SOFTWARE:
Ansys 10.0

Problem:

Show the effect of damper and its damped oscillation when the mass m is displaced from its equilibrium position and left free for its damped oscillation (as shown in Fig.7). Take spring stiffness (K)

= 100 N/m, and mass (m) = 10 kg. Show the response of the following cases, when damping coefficient (i) C = 0, (ii) C = 0.1 of C_c (iii) C = C_c, (iv) C = 1.2 of C_c where C_c = critical damping coefficient = $2\sqrt{K*m}$

ANSYS Analysis Procedure:

1.	**Utility menu bar:** select **File > Change Title**: Enter the title (Give title of your choice) in the window appeared and click 'OK'.
2.	**Main menu bar: Preferences > Structural>** Click 'OK'
3.	**Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the** window click **Add>select combination(sub window 1)> spring damper14(sub window 2)>** Click 'OK' **Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the** window select "**type1 COMBIN14**> click on "**OPTIONS**" > K2 options (enter) longitude UY DOF **Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the** window click **Add>selectsturcutral mass(sub window 1)>3D mass 21 (sub window 2)>** Click 'OK' **Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the** window select "**type1 COMBIN14**> click on "**OPTIONS**" > K3options (select) 2-D w/o rot iner> click **'OK'**
4.	**Main menu bar: Preprocessor>Real Constants > Add/Edit/Delete>** Click **Add...** and select '**COMBIN14**'> Click on 'OK'. The **Real Constants** window will appear> input **K** = 100>**CV1**=10> Click on 'OK'. 'Set 1' now appears in the dialog box. **Select Mass21**'> Click on 'OK'. The **Real Constants** window will appear> input **mass** = 10 Click on 'OK'. 'Set 2' now appears in the dialog box. Click on 'Close' in the 'Real Constants' window
5.	**Main menu bar: Preprocessor>Modeling> Create >nodes> In Active CS, in** the create nodes in active coordinate system**, type key point no. and its coordinates. Here** **node x y z** **1** 0 0 0 (Click 'apply') **2** 0 -1 0 (Click 'OK')
6.	**ANSYS Main Menu: Preprocessor>Modeling>Create>Nodes>In Active CS>** type the following in the Create Nodes in the Active Coordinate System window **Node number X Y Z** **1 0 0 0 (Click Apply)** **2 1 0 0 (Click OK)**
7.	**ANSYS Main Menu: Preprocessor>Modeling>Create>Element>Auto Numbered> Through Nodes,** in the Elements from Nodes window, type **'1' (Enter)** and type **'2' (Enter) and Click 'OK'.**
8.	**ANSYS Main Menu: Solution >Define Loads > Apply > Structural > Displacement >On nodes>**Type node **1** in the Apply U, ROT on Nodes window **click 'Apply'**, then **select 'UX'** in the Apply U, ROT on Nodes window (Check Apply as field with **' Constant value'** and **VALUE = 0**), then **Click 'OK'**
9.	**ANSYS Main Menu: Solution >Define Loads > Apply >Initial Condition>Define>**in the Define Initial Conditions window type **'2' and Click 'Apply'.** Then in the Define Initial Conditions window, in the DOF to be specified field select **'UX'**and in the Initial value of DOF type **'-1'.Click 'OK'.**
10.	**ANSYS Main Menu: Solution >Analysis Type>New Analysis>Transient>Click 'OK'>Click 'OK'.**
11.	**ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Transient>**in the Full Transient Options **Tick Transient effects** and in the Algorithm Field>select **HHT algorithm. Click 'OK'.**

12.	**ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Nonlinear>Click Set Convergence Criteria>Replace>Tolerance about VALUE>0.0001> OK>Close>Close>OK**
13.	**ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Basic>**in the Write Items to Result field, **Tick User selected>Nodal DOF solution** and in the Frequency field> select **Write every Nth Substep,** then in the Time Control field >in the Time at end of load step type **'0.0002'** and **Tick No. of Substeps and type 5 in the No. of Substeps field. Click 'OK'**
14.	**ANSYS Main Menu: Solution>Load step options>Output Ctrls> Solu Printout>**in the Solu Printout Controls window, in the Item for Printout Control field select **'Basic quantities'** and in the Print frequency field, **Tick None. Click 'OK'.**
15.	**ANSYS Main Menu: Solution>Load step options>Load step opts>Write LS file> Click 'OK'**
16.	**ANSYS Main Menu: Solution>Load step options >Output ctrls >Solu printout>>**in the Solu Printout Controls window, in the Item for Printout Control field select **'Basic quantities'** and in the Print frequency field, Tick **Last Substep. Click 'OK'.**
17.	**ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Basic>**in the Time Control field>in the Time at end of load step type**'10'** and **type 40 in the No. of Substeps field. Click 'OK'.**
18.	**ANSYS Main Menu: Solution>Load step options >Load step opts>Write LS file> Click 'OK'**
19.	**ANSYS Main Menu: Solution>Solve>From LS file,** in the Solve Load Step Files, in the LSMIN field, type **'1',** in the LSMAX field, type **'2'** and in the LSINC field, type **'1'. Click 'OK'.**
20.	**ANSYS Main Menu: Time Hist Postpro >Close the window**
21.	**ANSYS Main Menu: Time Hist Postpro >Settings>Data>**in the Data Settings window, in the TMIN field, type **'0.003',** in the TMAX field, type '**10'** and in the TINC field, type **'1'. Click 'OK'.**
22.	**ANSYS Main Menu: Time Hist Postpro >Define Variable>Add>** in the Add Time-History Variable window, **Tick Nodal Solution. Click 'OK',** then in the Define Nodal Data window, in the NVAR field, type **'2',** in the NODE field, type **'2',** and in the User specified Label field, type **'2UX'.** Then in the Item, Comp Data item, **Pick DOF solution (sub window 1) and Translation UX (sub window 2). Click 'OK',**
23.	**ANSYS Main Menu: Time Hist Postpro >Graph Variables>**in the Graph Time- History Variable window, in the NVAR1 field, type **'2'. Click 'OK',**
24.	**ANSYS main window shows the corresponding output graph.**
25.	**With the above said step, Change the Damping Coefficient (in Step 5) as C = 0 (no damper), 6.32 (under damped), 63.2 (critically damped), and 75.84 (over damped). See the corresponding output graph in the ANSYS main window.**

MERITS: 1.time saved
2.achived high accuracy

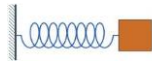
APPLICATIONS: dynamic analysis of machine elements

RESULT:
Thus the transient analysis of a spring mass damper system has been analyzed .

Ex: No: 10 MODAL ANALYSIS OF A SINGLE DEGREE SPRING MASS SYSTEM USING ANSYS

AIM: Determine the Natural frequency of the spring mass system shown in Fig.6 with spring stiffness (K) = 100 N/m, and mass (m) = 10 kg.

Problem:

Analytical Solution:

$$f_n = \frac{1}{2\pi}\sqrt{\frac{K}{m}} = \frac{1}{2\pi}\sqrt{\frac{100}{10}} = \textbf{0.50329 Hz}$$

ANSYS Analysis Procedure:

1. **Utility menu bar:** select **File > Change Title**:
   Enter the title (Give title of your choice) in the window appeared and **Click 'OK'.**
2. **ANSYS Main Menu: Preferences > Structural>Click 'OK'**
3. **ANSYS Main Menu: Preprocessor>Element Type > Add/Edit/Delete > in the** window click **Add>select Combination (sub window 1)> Spring Damper 14(sub window 2)>Click 'OK'**
4. In the Element type window, **click 'Options'**, in the element type options window, in theDOF select 1D behavior K2 KEY OPTION field select **' Longitude UY DOF'** and then **click 'OK'**and **Close**
5. **ANSYS Main Menu: Preprocessor>Element Type > Add/Edit/Delete > in the** window click **Add>select Structural Mass (sub window 1)>3D mass 21 (sub window 2)>Click 'OK'**
6. In the Element type window, **click 'Options'**, in the element type options window, in the Rotary Inertia options K3 KEY OPTION field select **' 2-D w/o rot iner'** and then **Click 'OK'**and **Close**
7. **ANSYS Main Menu: Preprocessor>Real constants**>Add>Click Combin14>ok>in the Real Constant Set Number 1, for COMBIN14 window, in the Spring constant K field type **'100'** and in the Damping coefficient CV1 field, type **'10'** then **Click 'OK'.**
8. **Now choose Mass21** in the Real constant window>**Click 'OK',** in the Real Constant Set Number 2 window, in the Mass field, type Mass = **'10',** then **Click 'OK'.**
9. **ANSYS Main Menu: Preprocessor>Modeling>Create>Nodes>In Active CS>** type the following in the Create Nodes in the Active Coordinate System window

Node number	X	Y	Z
1	**0**	**0**	**0 (Click Apply)**
2	**0**	**-1**	**0 (Click OK)**

10. **ANSYS Main Menu: Preprocessor>Modeling>Create>Element>Element Attributes>**In the Element Attributes window, in the Element Type field, select **1 COMBIN14** and in Real Constant Number type field, select **'1'** and **Click 'OK'**.

11. **In the Command window bar, type :**
    E,1,2

12. **ANSYS Main Menu: Preprocessor>Modeling>Create>Element>Element Attributes>**In the Element Attributes window, in the Element Type field, select **2 MASS21** and in Real Constant Number type field, select **'2'** and **Click 'OK'**

13. **In the Command window bar, type :**
    E, 2

14. **ANSYS Main Menu: Solution >Define Loads > Apply > Structural > Displacement >On nodes>**Type node **1** in the Apply U, ROT on Nodes window **click 'Apply'**, then **select 'All dof'** in the Apply U, ROT on Nodes window (Check Apply as field with **' Constant value'** and **VALUE = 0**), then **click 'OK'**

15. **In the Command window bar, type :**
    M,2,UY

16. **ANSYS Main Menu: Solution >Analysis Type>New Analysis**>Modal>Click**'OK'**.

17. **ANSYS Main Menu: Solution >Analysis Type>Analysis option >MODOPT set to reduced>No. of Modes to Extract= 1> Click 'OK'.**

18. **ANSYS Main Menu: Solution >Solve > Current LS**>Click Solve.

19. **Click 'Close' in Solution is Done window.**

20. **In the ANSYS output window we can see the Natural Frequency of the system defined.**

MERITS: 1.time saved
2.achived high accuracy

APPLICATIONS: • Bin or Hopper Agitation & Unloading

• Trailer Hopper Agitation & Unloading
• Bulk Material Settling & Packing
• Bulk Material Feeding & Conveying
• Bulk Material and Powder Screening & Scalping
• Chute & Pipe Material Flow
• Dust Collector Filter & Precipitator Cleaning

RESULT:
Thus the Natural frequency of the spring mass system has been analyzed and animated

Ex: No: 11. THERMAL STRESS ANALYSIS OF A 2D COMPONENT - STATIC
AIM:
To determine the thermal deformations at various points of the 2D components due to the temperature load applied over it.
SOFTWARE:
Ansys 14
PROCEDURE:

1. Start →Program →Ansys →Ansys product launcher

2. Preference → Select Structural and Thermal →OK

3. Preprocessor → Element Type → Add→ Add→ Coupled field → Vector Quad 13 → Ok

Option → K1= UX UY Temp AZ → Ok

4. Material Properties → Material Library → Import Library → SI → Ok

Browse → C: Program file → Ansys INC →V140 → Ansys → Matlib → select any one material → open → Ok → close

5. Modelling → create → Area → Rectangle → By 2 corners → Width =10, Height = 10 → Ok

6. Meshing → Mesh Tool → Global , Set = 0.5 → Ok

Mesh → select the object → Ok

7. Solution → Define load → Apply → Structural → Displacement → Onlines → Select the Base line → DOF → Ok

8. Solution → Define load → Apply → Thermal → Temperature → on lines → Select left side line → Temp = 40º → Apply → Select Right side line → Temp = 120º → Ok

9. Solve → Current LS → Ok

10. General postprocessor → Plot Result → Contour Plot → Nodal Solution → Select

Temperature or Stress → Ok

MERITS: 1.time saved
2.achived high accuracy

APPLICATIONS: .heat exchangers

2.air cooling and water cooling engines

RESULT:

Thus the thermal deformation over a 2D component is analyzed, animated and plotted.

Ex: No: 12. CONDUCTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

AIM:

To determine the conductive heat transfer of a 2D component

SOFTWARE:

Ansys 14

PROCEDURE:

1. Start →Program →Ansys →Ansys product launcher

2. Preference → Thermal →OK

3. Preprocessor → Element Type → Add→ Add→ Solid → Quad 4 node 55 → Ok

4. Material Properties → Material Library → Import Library → SI (MKS) → Ok

Browse → C: Program file → Ansys INC →V140 → Ansys → Matlib → select any one material → open → Ok → close

5. Modelling → create → Area → Rectangle → By 2 corners → Width =100, Height = 100 → Ok

6. Meshing → Mesh Tool → Global , Set → Element type = 10 → Click small size → Mesh → Select the object → Ok

7. Solution → Define load → Apply → Thermal → Temperature → On lines → Select the left line → Temp Value = 40º → Apply → select the right side line → Temp value = 120º → Ok

8. Solve → Current LS → Ok

9. General postprocessor → Plot Result → Contour Plot → Nodal Solution → DOF Solution → Nodal Temp → Ok

To check the Temperature at various point:

Query Result → Sub grid solution → DOF solution → Temperature → Ok → click any point on the object Temperature value displayed on the screen

MERITS: 1.time saved
2.achived high accuracy

APPLICATIONS: .heat exchangers
2.air cooling and water cooling engines
RESULT:
Thus the heat conduction over a 2D component is analyzed, animated and plotted.

Ex: No: 13.CONVECTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT
AIM:
To determine the convective heat transfer analysis of a 2D component
SOFTWARE:
Ansys 10.0
PROCEDURE:

1. Start →Program →Ansys →Ansys product launcher

2. Preference → Thermal →OK

3. Preprocessor → Element Type → Add→ Add→ Solid → Quad 4 node 55 → Ok

4. Material Properties → Material Library → Import Library → SI (MKS) → Ok

Browse → C: Program file → Ansys INC →V140 → Ansys → Matlib → select any one material → open → Ok → close

5. Modelling → create → Area → Rectangle → By 2 corners → Width =100, Height = 100 → Ok

6. Meshing → Mesh Tool → Global , Set → Element type = 10 → Click small size → Mesh → Select the object → Ok

7. Solution → Define load → Apply → Thermal → Temperature → On lines → Select the left line → Temp Value = 40º → Apply → select the right side line → Temp value = 120º → Ok

8. Solution → Define Load → Apply → Convection → on lines → Film coefficient = 20, Bulk Temp = 20º → Ok

9. Solve → Current LS → Ok

10. General postprocessor → Plot Result → Contour Plot → Nodal Solution → DOF Solution → Nodal Temp → Ok

To check the Temperature at various point:
Query Result → Subgrid solution → DOF solution → Temperature → Ok → click any point on the object Temperature value displayed on the screen

MERITS: 1.time saved
2.achived high accuracy

APPLICATIONS:1.heat exchangers
2.air cooling and water cooling engines

RESULT: Thus the heat convective over a 2D component is analyzed, animated and plotted.

STUDY OF MATLAB

AIM:
To study the mathematical computing simulation methods using MATLAB
MATLAB
MATLAB integrates mathematical computing, visualization, and a powerful language to provide a flexible environment for technical computing. MATLAB includes tools for:
Data acquisition

- ✓ Data analysis and exploration

- ✓ Visualization and image processing

- ✓ Algorithm prototyping and development

- ✓ Modeling and simulation

- ✓ Programming and application development

Category	Description
Desktop environment	The MATLAB desktop is a graphical user interface that transforms MATLAB in to an integrated environment for exploration and development

Matrices	MATLAB works with scalars, vectors and matrices. A scalar is really just a 1-by-1 matrix and a vector is nothing more than a long, thin matrix given as either a row or a column. Since matrix are the basis for all of MATLAB, this section contains demos that describes simple matrix operations and some of the basic MATLAB functions, the foundation upon which many of the more complex functions are built.
Numeric′s	This section contains some demos of numeric calculation in MATLAB. It includes demos illustrating curve, fitting, solving differential equations, and fast Fourier transform (FFT)
Graphics	These demos illustrate graphics functions that create 2-D and 3-D plots of data, and introduce visualization techniques. See also MATLAB

Language	MATLAB is both an environment and a programming language that allows you to build your own reusable tools. You can write programs as either scripts or functions using data types, operators, expressions, and statements that are similar to programming languages such as „C". You can also create object-oriented classes and objects in
Automation Client Interface (COM)	MATLAB or use its built- in Java interface to create and work with Java classes and objects. MATLAB provides an interface that enables communication with Automation servers and Automation controls. The demos in this section illustrate some of the features of this interface.

Category	Description

Features	Simulink provides many features for powerful and intuitive modeling. Some major features are illustrated in these demonstration models.
General	Simulink has the ability to simulate a large range of systems, from very simple to extraordinarily complex. The models and demonstrations that you will see in this section include both simple and complex systems. Although the complex systems are nowhere near the limits of what can be done, they hint at the level of sophistication that you can expect.
Automotive	Simulink, stateflow and the Real-Time workshop represent the industry standard toolset in the automotive field. The models that you will see in this area showcase the use of Math works tools in various automotive applications.
Aerospace	Simulink, statflow and the Real-Time Workshop represent the industry standard toolset in the aerospace field. The models that you will see in this area showcase the use of Math Works tools in various aerospace applications.
Real-Time Workshop	Generate C code from Simulink models

SimMechanics	Model and simulate mechanical systems
Simpower systems	Model and simulate electrical power systems
Simulink Report Generator	Automatically generate documentation for Simulink and stateflow models
State flow	Design and simulate event-driven system
XPC Target	Perform real-time rapid prototype using PC hardware

RESULT:
Thus the basics of MATLAB are studied.