Varuvan Vadivelan
Institute of Technology

Dharmapuri – 636 703

LAB MANUAL

Regulation : 2013

Branch : *B.E.* – Mechanical Engineering

Year & Semester : IV Year / VII Semester

ME6711-SIMULATION AND ANALYSIS

ANNA UNIVERSITY: CHENNAI

SYLLABUS (R - 2013)

ME6711-SIMULATION AND ANALYSIS LABORATORY

List of Experiments

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

Simulation:

10. MAT LAB basics, dealing with matrices, Graphing-functions of one variable and two variables

11. Use of MATLAB to solve simple problems in vibration

12. Mechanism Simulation using multi body dynamic software

INDEX

Ex. No	Date	Name of the Experiment	Staff Signature	Remarks
		INTRODUCTION		
1		Force and stress analysis using four link elements in trusses		

2	Force and stress analysis using two link elements in trusses
3	Stress and deflection analysis in simply supported beam with point load
4	Stress and deflection analysis in simply supported beam with uniformly varying load
5	Stress and deflection analysis in simply supported beam with uniformly distributed load
6	Stress and deflection analysis in beam with moment and overhanging
7	stress and deflection analysis in cantilever beam with point load
8	Stress and deflection analysis in beam with angular loads
9	Stress analysis of a rectangular plate with circular hole
10	Stress analysis of a the corner angle bracket
11	Stress analysis of an axis-symmetric component
12	Thermal stress analysis within the rectangular plate
13	Convective heat transfer analysis of a 2D component
14	Model analysis of cantilever beam without load
15	Model analysis of cantilever beam with load
16	Harmonic analysis of a 2d component
17	MATLAB basics, Dealing with matrices, Graphing-Functions of one variable and two variables
18	Simulation of spring-mass system using MAT LAB
19	Simulation of cam and follower mechanism using MAT LAB

INTRODUCTION

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 very 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.

FEA for Design Engineers: another design tool:

For design engineers, FEA is one of many design tools among CAD, Prototypes, spreadsheets, catalogs, data bases, hand calculations, text books,

etc. that are all used in the design process.

FEA for Design Engineers: based on CAD models:

Modern design is conducted using CAD tools, so a CAD model is the starting point for analysis. Since CAD models are used for describing geometric information for FEA, it is essential to understand how to design in CAD in order to produce reliable FEA results, and how a CAD model is different from FEA model. This will be discussed in later chapters.

FEA for Design Engineers:

Since FEA is a design tool, it should be used concurrently with the design process. It should keep up with, or better yet, *drive* the design process. Analysis iterations must be performed fast, and since these results are used to make design decisions, the results must be reliable even when limited input is available.

Limitations of FEA for Design Engineers:

As you can see, FEA used in the design environment must meet high requirements. An obvious question arises: would it be better to have dedicated specialist perform FEA and let design engineers do what they do best - design new products? The answer depends on the size of the business, type of products, company organization and culture, and many other tangible and intangible factors. A general consensus is that design engineers should handle relatively simple types of analysis, but do it quickly and of course reliably. Analyses that are very complex and time consuming cannot be executed concurrently with the design process, and are usually better handled either by a dedicated analyst or contracted out to specialized consultants.

Objectives of FEA for Design Engineers:

The ultimate objective of using the FEA as a design tool is to change the design process from repetitive cycles of "design, prototype, test" into streamlined process where prototypes are not used as design tools and are only needed for final design verification. With the use of FEA, design iterations are moved from the physical space of prototyping and testing into the virtual space of computer simulations (figure 1-1).

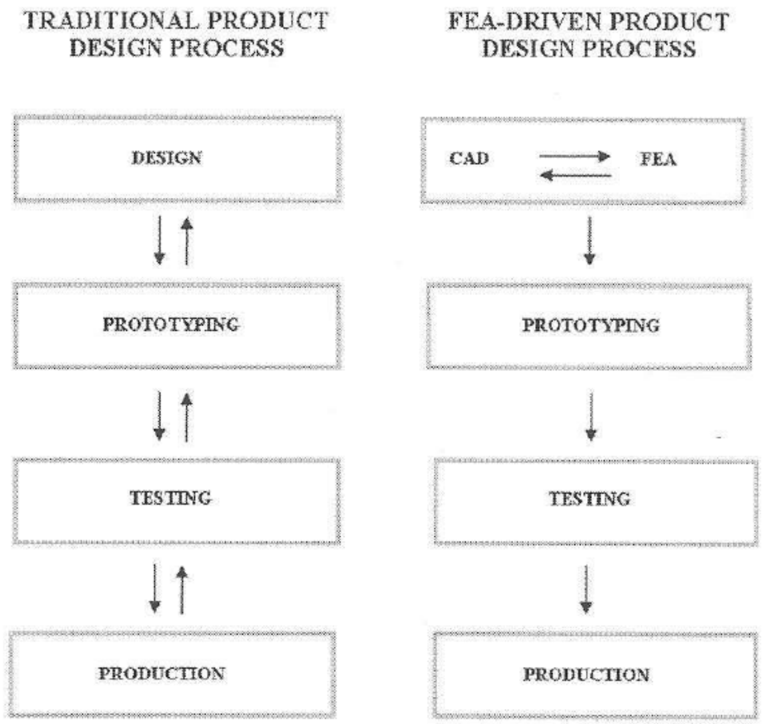


Figure 1-1: Traditional and. FEA- driven product development

Traditional product development needs prototypes to support design in progress. The process in FEA-driven product development uses numerical models, rather than physical prototypes to drive development. In an FEA driven product, the prototype is no longer a part of the iterative design loop.

What is Solid Works Simulation?

Solid Works Simulation is a commercial implementation of FEA, capable of solving problems commonly found in design engineering, such as the analysis of deformations, stresses, natural frequencies, heat flow, etc. Solid Works Simulation addresses the needs of design engineers. It belongs to the family of engineering analysis software products developed by the Structural Research & Analysis Corporation (SRAC). SRAC was established in 1982 and since its inception has contributed to innovations that have had a significant impact on the evolution of FEA. In 1995 SRAC partnered with the Solid Works Corporation and created Solid Works Simulation, one of the first Solid Works Gold Products, which became the top-selling analysis solution for Solid Works Corporation. The commercial success of Solid Works Simulation integrated with Solid Works CAD software resulted in the acquisition of SRAC in 2001 by Desalt Systems, parent of Solid Works Corporation.

In 2003, SRA Corporations merged with Solid Works Corporation. Solid Works Simulation is tightly integrated with Solid Works CAD software and uses Solid Works for creating and editing model geometry. Solid Works is a solid, parametric, feature-driven CAD system. As opposed to many other CAD systems that were originally developed in a UNIX environment and only later ported to Windows, Solid Works CAD was developed specifically for the Windows Operating System from the very beginning. In summary, although the history of the family of Solid Works FEA products dates back to 1982, Solid Works Simulation has been specifically developed for Windows and takes full advantage this of deep integration between Solid Works and Windows, representing the state-of-the-art in the engineering analysis software.

Fundamental steps in an FEA project:

The starting point for any Solid Works Simulation project is a Solid Works model, which can be one part or an assembly. At this stage, material properties, loads and restraints are defined. Next, as is always the case with using any FEA based analysis tool, we split the geometry into relatively small and simply shaped entities, called finite elements. The elements are called "finite" to emphasize the fact that they are not infinitesimally small, but only reasonably small in comparison to the overall model size. Creating finite elements is commonly called meshing. When working with finite elements, the Solid Works Simulation solver approximates the solution being sought (for example, deformations or stresses) by assembling the solutions for individual elements.

From the perspective of FEA software, each application of FEA requires three steps:

1. Preprocessing of the FEA model, which involves defining the model and then splitting it into finite elements

2. Solution for computing wanted results

3. Post-processing for results analysis

We will follow the above three steps every time we use Solid Works Simulation. From the perspective of FEA methodology, we can list the following FEA steps:

1. Building the mathematical model

2. Building the finite element model

3. Solving the finite element model

4. Analyzing the results

Building the mathematical model:

The starting point to analysis with Solid Works Simulation is a Solid Works model. Geometry of the model needs to be mesh able into a correct and reasonably small element mesh. This requirement of mesh ability has very important implications. We need to ensure that the CAD geometry will indeed mesh and that the produced mesh will provide the correct solution of the data of interest, such as displacements, stresses, temperature distribution, etc. This necessity often requires modifications to the CAD geometry, which can take the form of

De featuring, idealization and/or clean-up, described below:

Term	Description
Defeaturing	The process of removing geometry features deemed insignificant for analysis, such as external fillets, chamfers, logos, etc.
Idealization	A more aggressive exercise that may depart from solid CAD geometry by for example, representing thin walls with surfaces
Clean-up	Sometimes needed because for geometry to be meshable, it must satisfy much higher quality demands than those required for Solid Modeling. To cleanup, we can use CAD quality-control tools to check for problems like sliver faces, multiple entities, etc. that could be tolerated in the CAD model, but would make subsequent meshing difficult or impossible

It is important to mention that we do not always simplify the CAD model with the sole objective of making it mesh able. Often, we must simplify a model even though it would mesh, correctly "as is", but the resulting mesh would be too large and consequently, the analysis would take too much time. Geometry modifications allow for a simpler mesh and shorter computing times. Also, geometry preparation may not be required at all; successful meshing depends as much on the quality of geometry submitted for meshing as it does on the Sophistication of the meshing tools implemented in the FEA software.

Having prepared a mesh able, but not yet meshed geometry we now define material properties. These can also be imported from a Solid Works model, loads and restraints, and provide information on the type of analysis that we wish to perform. This procedure completes the creation of the mathematical model (figure 1-2). Notice that the process of creating the mathematical model is not FEA-specific. FEA has not yet entered the picture.

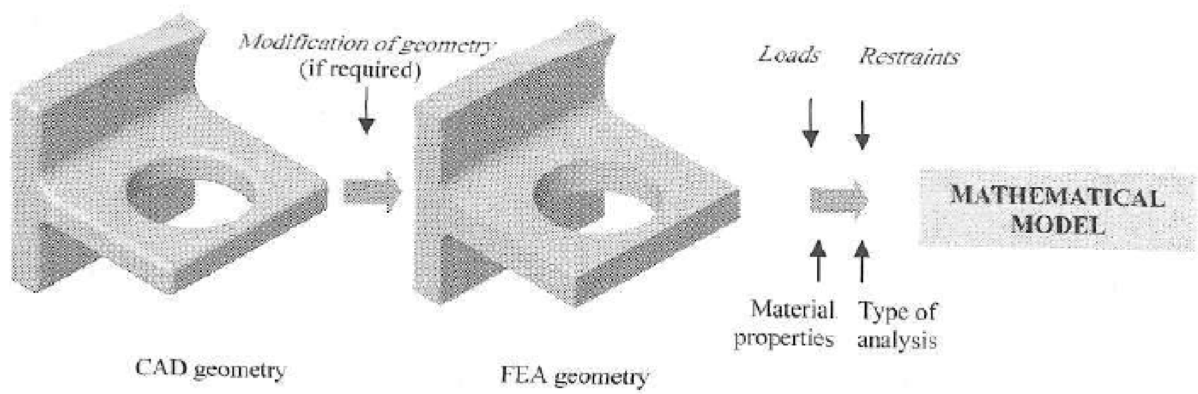


Figure 1-2: Building the mathematical model

The process of creating a mathematical model consists of the modification o CAD geometry (here removing external fillets), definition of loads, restraint material properties, and definition of the type of analysis (e.g., static) that we wish to perform.

Building the finite element model:

The mathematical model now needs to be split into finite elements through a process of discretization, more commonly known as meshing (figure 1-3).Loads and restraints are also discretized and once the model has been meshed the discretized loads and restraints are applied to the nodes of the finite element mesh.

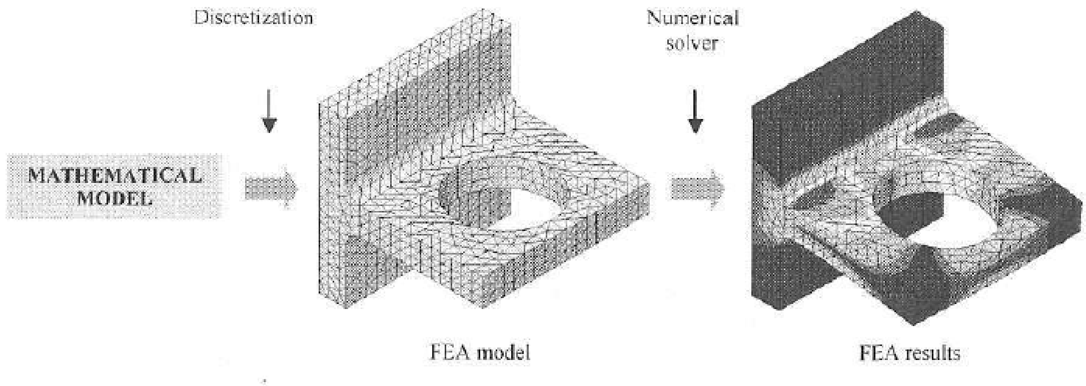


Figure 1-3: Building the finite element model

The mathematical model is discretized into a finite element model. This completes the pre-processing phase. The FEA model is then solved with one of the numerical solvers available in Solid Works Simulation Solving the finite element model having created the finite element model, we now use a solver provided in Solid Works Simulation to produce the desired data of interest (figure 1-3).

Analyzing the results:

Often the most difficult step of FEA is analyzing the results. Proper interpretation of results requires that we understand all simplifications (and errors they introduce) in the first three steps: defining the mathematical model, meshing its geometry, and solving.

Errors in FEA:

The process illustrated in figures 1-2 and 1-3 introduces unavoidable errors. Formulation of a mathematical model introduces modeling errors (also called idealization errors), discretization of the mathematical model introduces discretization errors, and solving introduces numerical errors. Of these three types of errors, only discretization errors are specific to FEA. Modeling errors affecting the mathematical model are introduced before FEA is utilized and can only be controlled by using correct modeling techniques. Solution errors caused by the accumulation of round-off errors are difficult to control, but are usually very low.

A closer look at finite elements:

Meshing splits continuous mathematical models into finite elements. The type of elements created by this process depends on the type of geometry meshed, the type of analysis, and sometimes on our own preferences. Solid Works Simulation offers two types of elements: tetrahedral solid elements (for meshing solid geometry) and shell elements (for meshing surface geometry).Before proceeding we need to clarify an important terminology issue. In CAD terminology "solid" denotes the type of geometry: solid geometry (as opposed to surface or wire frame geometry), in FEA terminology it denotes the type of element.

Solid elements:

The type of geometry that is most often used for analysis with Solid Works Simulation is solid CAD geometry. Meshing of this geometry is accomplished with tetrahedral solid elements, commonly called "test" in FEA jargon. The tetrahedral solid elements in Solid Works Simulation can either be first order elements (draft quality), or second order elements (high quality).

The user decides whether to use draft quality or high quality elements for meshing. However, as we will soon prove, only high quality elements should be used for an analysis of any importance. The difference between first and second order tetrahedral elements is illustrated in figure 1-4.

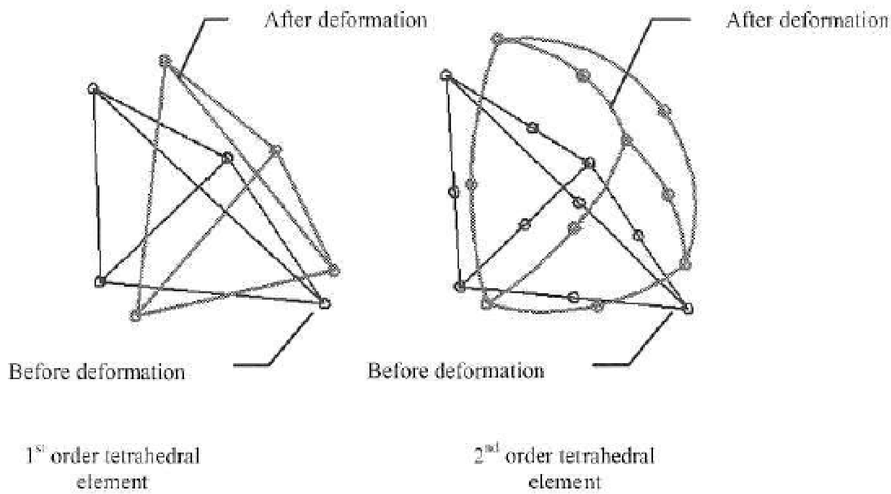


Figure 1 -4: Differences between first and second order tetrahedral elements

First and the second order tetrahedral elements are shown before and after deformation. Note that the deformed faces of the second order element may assume curvilinear shape while deformed faces of the first order element must remain fiat.

First order tetrahedral elements have four nodes, straight edges, and flat faces. These edges and faces remain straight and flat after the element has experienced deformation under the applied load. First order tetrahedral elements model the linear field of displacement inside their volume, on faces, and along edges. The linear (or first order) displacement field gives these elements their name: first order elements. If you recall from the Mechanics of Materials, strain is the first derivative of displacement. Therefore, strain and consequently stress, are both constant in first order tetrahedral elements. This situation imposes a very severe limitation on the capability of a mesh constructed with first order elements to model stress distribution of any real complexity. To make matters worse, straight edges and flat faces cannot map properly to curvilinear geometry, as illustrated in figure 1-5.

Figure 1-5: Failure of straight edges and flat faces to map to curvilinear geometry

A detail of a mesh created with first order tetrahedral elements. Notice the imprecise element mapping to the hole; flat faces approximate the face of the cylindrical hole.

Second order tetrahedral elements have ten nodes and model the second order (parabolic) displacement field and first order (linear) stress field in their volume, along laces, and edges. The edges and faces of second order tetrahedral elements before and after deformation can be curvilinear. Therefore, these elements can map precisely to curved surfaces, as illustrated in figure 1-6. Even though these elements are more computationally demanding than first order elements, second order tetrahedral elements are used for the vast majority of analyses with Solid Works Simulation.

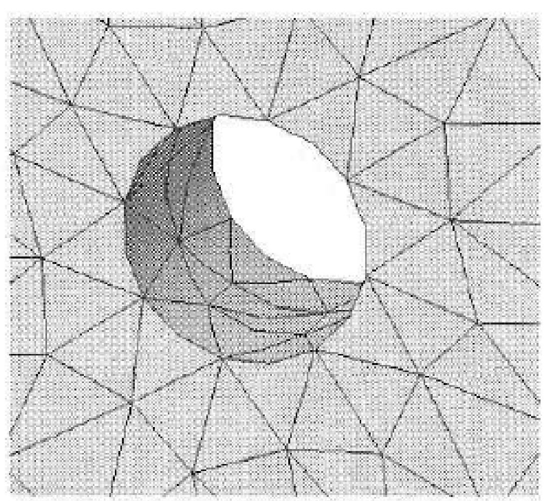


Figure 1-6: Mapping curved surfaces

A detail is shown of a mesh created with second order tetrahedral elements. Second order elements map well to curvilinear geometry.

Shell elements:

Besides solid elements, Solid Works Simulation also offers shell elements. While solid elements are created by meshing solid geometry, shell elements are created by meshing surfaces. Shell elements are primarily used for analyzing thin-walled structures. Since surface geometry does not carry information about thickness, the user must provide this information. Similar to solid elements, shell elements also come in draft and high quality with analogous consequences with respect to their ability to map to curvilinear geometry, as shown in figure 1-7 and figure 1-8. As demonstrated with solid elements, first order shell elements model the linear displacement field with constant strain and stress while second order shell elements model the second order (parabolic) displacement field and the first order strain and stress field.

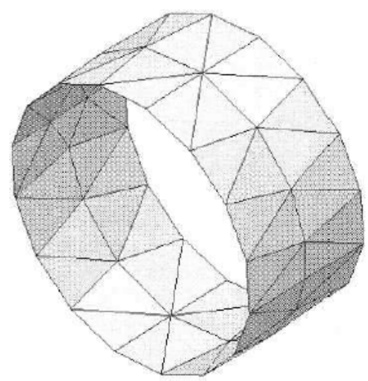


Figure 1-7: First order shell element

This shell element mesh was created with first order elements. Notice the imprecise mapping of the mesh to curvilinear geometry.

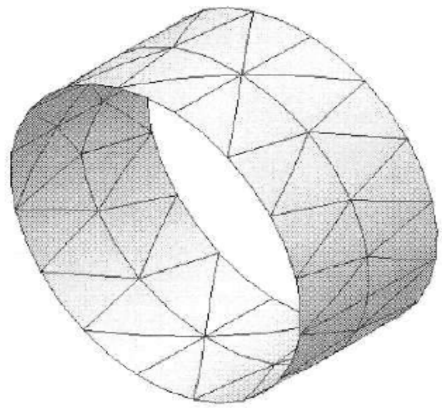


Figure 1-8: Second order shell element

Shell element mesh created with second order elements, which map correctly to curvilinear geometry.

Certain classes of shapes can be modeled using either solid or shell elements, such as the plate shown in figure 1-9. The type of elements used depends then on the objective of the analysis. Often the nature of the geometry dictates what type of element should be used for meshing. For example, parts produced by casting are meshed with solid elements, while a sheet metal structure is best meshed with shell elements.

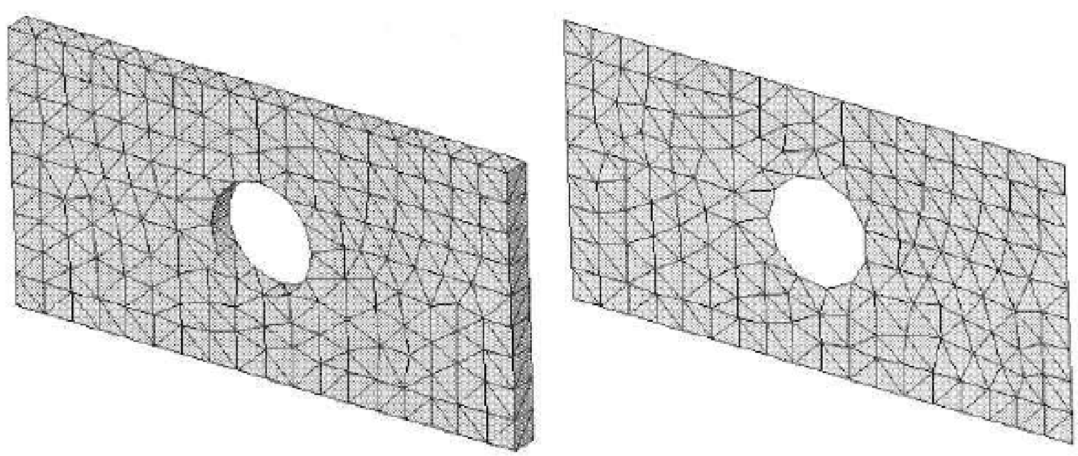


Figure 1-9: Plate modeled with solid elements (left) and shell elements

The plate shown can be modeled with either solid elements (left) or shell elements (right). The actual choice depends on the particular requirements of analysis and sometimes on personal preferences Figure 1-10, below, presents the basic library of elements in Solid Works Simulation. Elements like a hexahedral solid, quadrilateral shell or other shapes are not available in Solid Works Simulation.

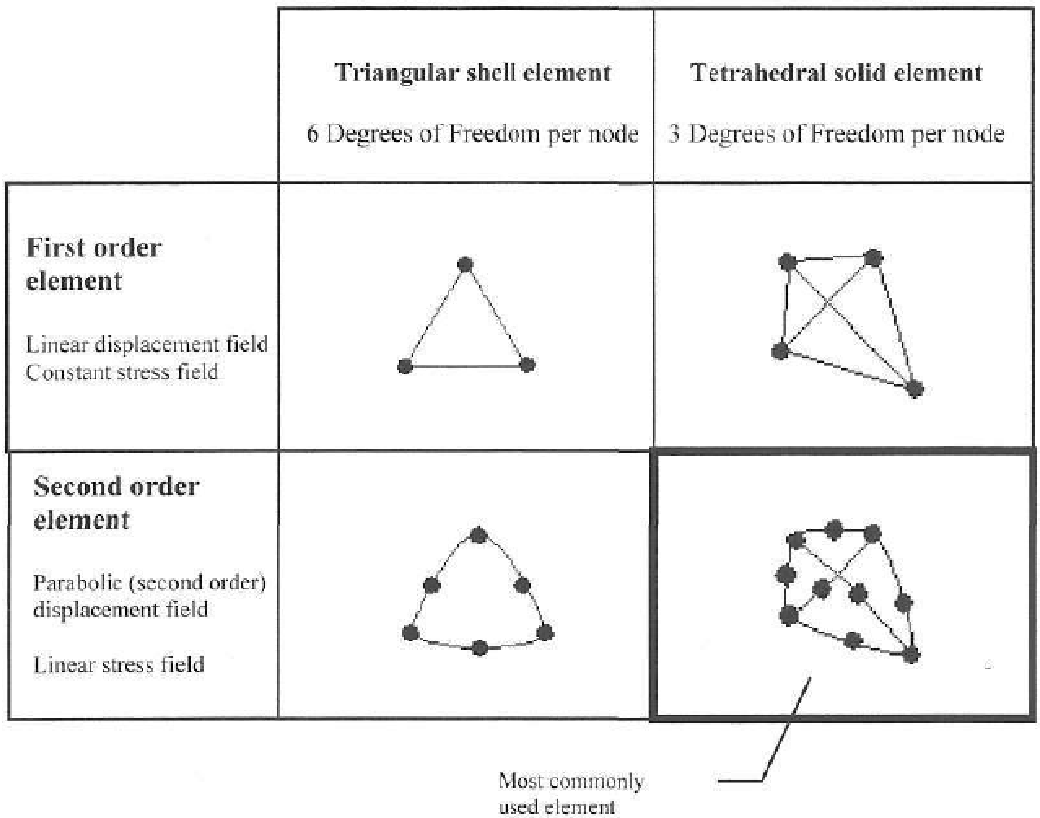


Figure 1-10: Solid Works Simulation element library

Four element types are available in the Solid Works Simulation element library. The vast majority of analyses use the second order tetrahedral element. Both solid and shell first order elements should be avoided.

The degrees of freedom (DOF) of a node in a finite element mesh define the ability of the node to perform translation or rotation. The number of degrees of freedom that a node possesses depends on the type of element that the node belongs to. In Solid Works Simulation, nodes of solid elements have three degrees of freedom, while nodes of shell elements have six degrees of freedom. This means that in order to describe transformation of a solid element from the components of nodal displacement, most often the x, y, z displacements. In the case of shell elements, we need to know not only the translational components of nodal displacements, but also the rotational displacement components.

What is calculated in FEA?

Each degree of freedom of every node in a finite element mesh constitutes an unknown. In structural analysis, where we look at deformations and stresses, nodal displacements are the primary unknowns. If solid elements are used, there are three displacement components (or 3 degrees of freedom) per node that must be calculated. With shell elements, six displacement components (or6 degrees of freedom) must be calculated. Everything else, such as strains and stresses, are calculated based on the nodal displacements. Consequently, rigid restraints applied to solid elements require only three degrees of freedom to be constrained. Rigid restraints applied to shell elements require that all six degrees of freedom be constrained. In a thermal analysis, which finds temperatures and heat flow, the primary unknowns are nodal temperatures. Since temperature is a scalar value (unlike the vector nature of displacements), then regardless of what type of element is used, there is only one unknown (temperature) to be found for each node. All other results available in the thermal analysis are calculated based on temperature results. The fact that there is only one unknown to be found for each node; rather than three or six, makes thermal analysis less computationally intensive than structural analysis.

How to interpret FEA results:

Results of structural FEA are provided in the form of displacements and stresses. But how do we decide if a design "passes" or "fails" and what does it take for alarms to go off? What exactly constitutes a failure?

To answer these questions, we need to establish some criteria to interpret FEA results, which may include maximum acceptable deformation, maximum stress, or lowest acceptable natural frequency.

While displacement and frequency criteria are quite obvious and easy to establish, stress criteria are not. Let's assume that we need to conduct a stress analysis in order to ensure that stresses are within an acceptable range. To judge stress results, we need to understand the mechanism of potential failure, if a part breaks, what stress measure best describes that failure? Is it vonMises stress, maximum principal stress, shear stress, or something else? COSMOS Works can present stress results in any form we want. It is up to us to decide which stress measure to use for issuing a "pass" or "fail" verdict.

Any textbook on the Mechanics of Materials provides information on various failure criteria. Interested readers can also refer to books. Here we will limit our discussion to outlining the differences between two commonly used stress measures: Von Mises stress and the principal stress.

Von Mises stress:

Von Mises stress, also known as Huber stress, is a stress measure that accounts for all six stress components of a general 3-D state of stress (figure 1-11).

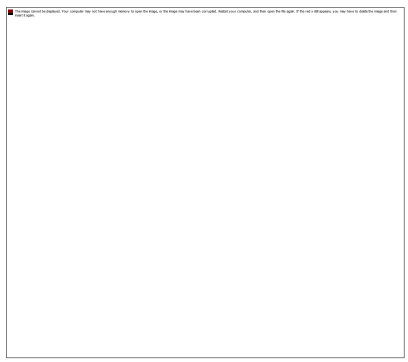
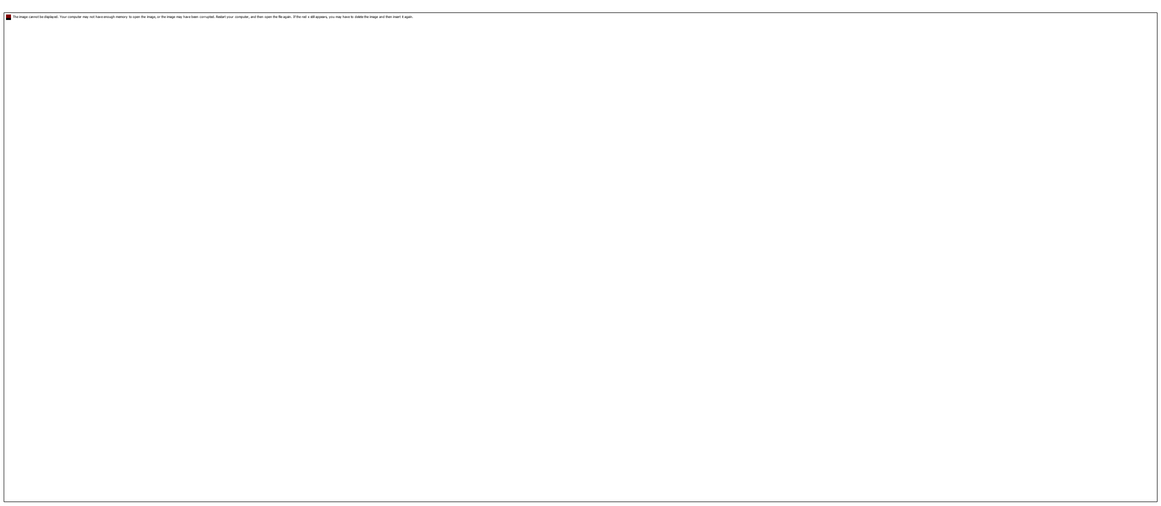


Figure 1-11: General state of stress represented by three normal stresses: σ_x, σ_y, σ_z and six shear stresses $\tau_{xy} = \tau_{yx}$, $\tau_{yz} = \tau_{zy}$, $\tau_{zx} = \tau_{xz}$

Two components of shear stress and one component of normal stress act on each side of an elementary cube. Due to equilibrium requirements, the general 3-Dstate of stress is characterized by six stress components: σ_x, σ_y, σ_z and $\tau_{xy} = \tau_{yx}$, $\tau_{yz} = \tau_{zy}$, $\tau_{zx} = \tau_{xz}$.



Note that von Mises stress is a non-negative, scalar value. Von Mises stress is commonly used to present results because structural safety for many engineering materials showing elasto-plastic properties (for example, steel) can be evaluated using von Mises stress.

The magnitude of von Mises stress can be compared to material yield or to ultimate strength to calculate the yield strength or the ultimate strength safety factor.

Principal stresses:

By properly adjusting the angular orientation of the stress cube in figure 1-11, shear stresses disappear and the state of stress is represented only by three principal stresses: o:, o2, and o3, as shown in figure 1-12. In Solid Works simulation, principal stresses are denoted as σ_1, σ_2, and σ_3.

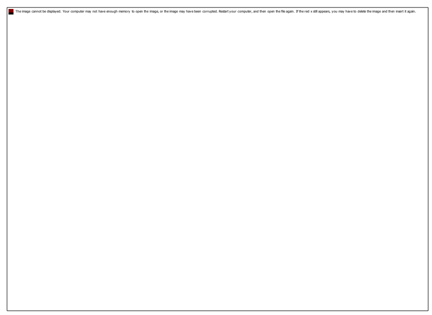


Figure 1-12: General state of stress represented by three principal stresses: σ_{1}, σ_{2}, σ_{3}

ANSYS HISTORY:

1. A general purpose Finite Element Analysis (FEA) package
2. Founded in 1970
3. Founder: Dr. John Swanson
4. 1969: STASYS (Structural Analysis System)
5. 1970: ANSYS

6. 1975: PC version

7. 2004: ANSYS 8.1 multi-physics

8. 2010: ANSYS 12.1

WHAT CAN ANSYS DO?

1. Structural analysis-Stress / strain, deformation, etc.

2. Thermal analysis

3. Fluid flow (CFD)

4. Electro magnetic

5. Multi-physics (solid/fluid/electromagnetic.)

MODELLING APPROACH:

1. Bottom-up approach:
    1. Creation of model by defining the geometry of the structure with nodes and elements

2. Top-down approach:
   Building a solid model using a 3D CAD program and then dividing the model into nodes and elements

APDL (ANSYS Parametric Design Language):

1. **h - Method**

It is used for regular surface

Developed in 1970s

The name is derived from the field of numerical analysis where the 'h' is used for step size,to achieve convergence in the analysis.The h-element is always of low order, usually linear or quadratic.

2. **p – Method**

3. No restrictions to shape and size

4. Developed in late 1980s

5. This provides option of optimizing a structure.

p – Elements can have edge polynomial as high as 9^{th} order, unlike the low order polynomials of h-elements

UNIT SYSTEM IN ANSYS:

1. ANSYS had no build-in unit system
2. The unit must be consistent
3. The precision of results depends on the model type and mesh type

ANSYS BENEFITS:

1. Simulate how a design behaves under real-world conditions without having to build costly physical prototypes.

2. Obtain accurate results on design behavior

3. Increase your confidence in your final product design by virtually testing a design under all

   Conceivable loading conditions

ANSYS APPLICATIONS:

1. Strength, durability and vibration assessment of various structures such as aircrafts, cars, trucks, or a train

2. Structural dynamic response simulation of loads that vary with time or frequency

3. Modal based analysis of large systems such as in automotive and aerospace vehicle systems

4. Simulation of interior acoustics for sound pressure inside a bounded domain

5. Static and transient analysis of structures involving material and geometric nonlinear behavior and nonlinear boundary condition

6. Advanced heat transfer analysis with contact including conduction, convection and radiation to understand the effect of temperature fluctuations in consumer electronic devices such as television or cell phone.

ANSYS Graphical User Interface:

The image cannot be displayed. Your computer may not have enough memory to open the image, or the image may have been corrupted. Restart your computer, and then open the file again. If the red x still appears, you may have to delete the image and then insert it again.

After starting ANSYS, two windows will appear. The first is the ANSYS 12.1 Output Window. This window displays a listing of every command that ANSYS executes. If you encounter problems, this is a good place to look to see what ANSYS is doing or has done.

This is one location where you will find all of the warnings and error messages that appear and the command that generated the warning/error.

The second window is the ANSYS Research FS graphical user interface. This is divided into 4 sections.

1. ANSYS Utility Menu

2. ANSYS Toolbar Menu

3. ANSYS Main Menu

4. Display window

ANSYS utility menu:

Within this menu, you can perform file operations, list and plot items, and change display options

File Drop-down Menu:

The File drop-down menu includes the options to clear the database, change, resume, and save the current model.

List Pull-Down Menu:

The list pull-down menu allows you to view the log and error files, obtain a listing of geometric entities, elements and their properties, nodes, and boundary conditions and loads applied to the model

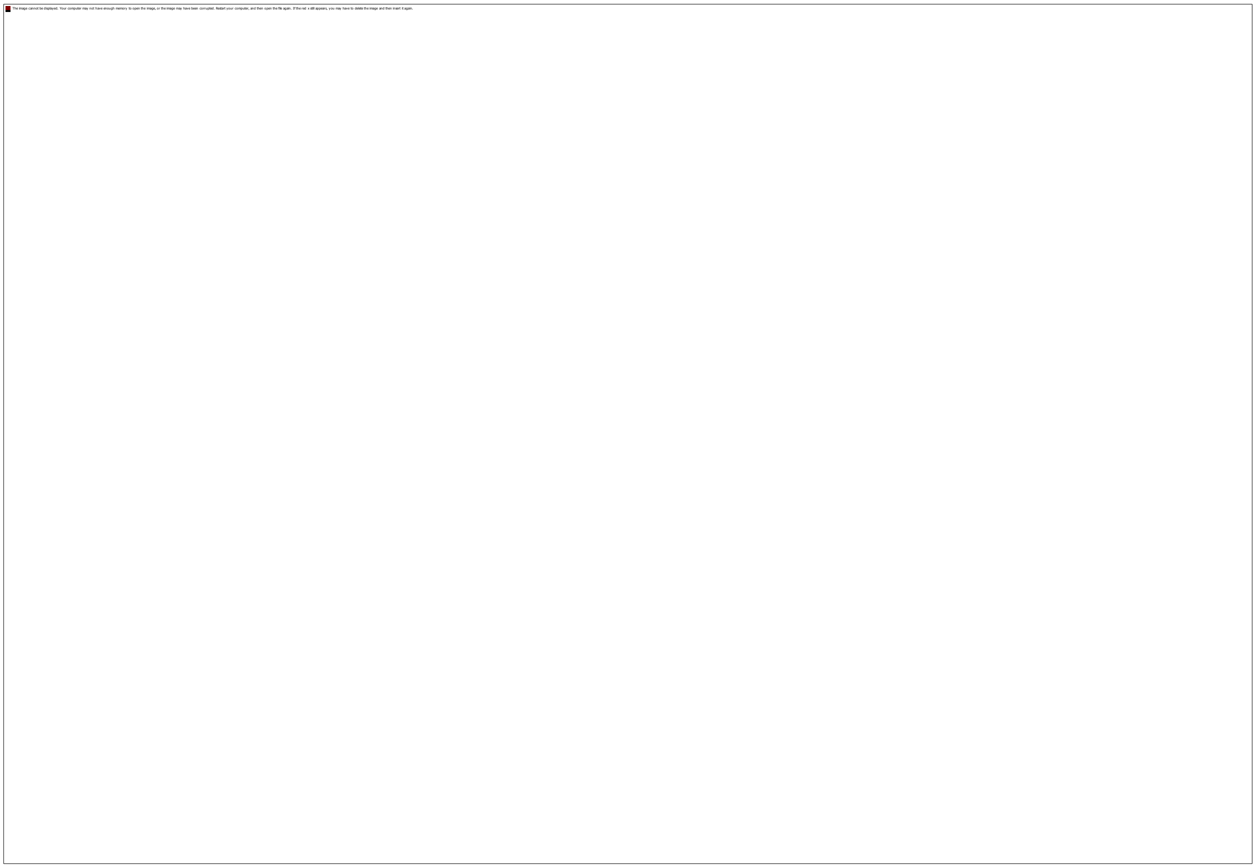
Plot Pull-Down Menu:

This pull-down menu allows you to plot the various components of the model such as key points, areas, volumes and elements.

The image cannot be displayed. Your computer may not have enough memory to open the image, or the image may have been corrupted. Restart your computer, and then open the file again. If the red x still appears, you may have to delete the image and then insert it again.

Plot Ctrls Pull-Down Menu:

This menu includes the controls to pan/zoom/rotate your model, select the numbering options, change styles and generate hard copies of the plots.



ANSYS Main Menu:

The ANSYS Main Menu contains all of the commands to create, mesh, apply loads, solve, and view results of the FE analysis. The Main Menu is divided into sections that sequentially follow the steps involved in an analysis

Preprocessor:

The image cannot be displayed. Your computer may not have enough memory to open the image, or the image may have been corrupted. Restart your computer, and then open the file again. If the red x still appears, you may have to delete the image and then insert it again.

Solution:

The image cannot be displayed. Your computer may not have enough memory to open the image, or the image may have been corrupted. Restart your computer, and then open the file again. If the red x still appears, you may have to delete the image and then insert it again.

Post processing:

GENERAL STEPS

Step 1: Ansys Utility Menu

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

File – change job name – enter new job name – xxxx – ok

File – change title – enter new title – yyy – ok

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

Step 3: Preprocessor

Element type – select type of element from the table and the required options

Real constants – give the details such as thickness, areas, moment of inertia, etc. required depending on the nature of the problem.

Material Properties – give the details such as Young's modulus, Poisson's ratio etc. depending on the nature of the problem.

Step 4: Modeling – create the required geometry such as nodes elements, area, and volume by using

the appropriate options**.**

Step 5: Generate – Elements/ nodes using Mesh Tool if necessary (in 2D and 3D problems)

Step 6: Apply boundary conditions/loads such as DOF constraints, Force/Momentum, Pressure etc.

Step 7: Solution – Solve the problem

Step 8: General Post Processor – plot / list the required results.

Step 9: Plot ctrls – animate – deformed shape – def+undeformed-ok

Step 10: to save the solution

FORCE AND STRESS ANALYSIS USING FOUR LINK ELEMENTS IN TRUSSES

EX.NO. 1

DATE:

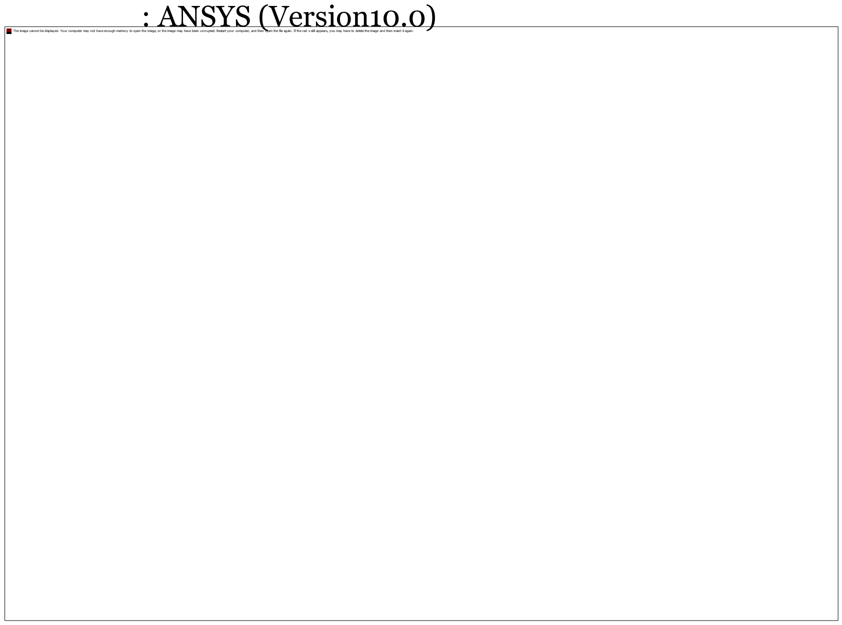
AIM:

To perform a force and stress analysis of four link elements in trusses using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: Intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM



DESCRIPTION:

Consider the four bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. E = 210 GPa, A = 0.1 m^2.

PROCEDURE:

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok
2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.
3. Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.
4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9– Ok – close.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 4 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 4, 3 (x, y value w.r.t first node) – apply (third node is created) – 0, 3 (x, y value w.r.t first node) – ok (forth node is created).

6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number

   – 1 – ok

7. Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 1

   – apply pick 3 & 4 – ok (elements are created through nodes).

8. Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 4 – apply – DOFs to be constrained – All DOF – ok – on Nodes – pick node 2 – apply – DOFs to be constrained – UY – ok.

9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok – Structural –

10. Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.

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

12. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.

13. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.

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

15. Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.

16. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok

RESULT:

Thus the force and stress analysis of four link element in trusses is done by using the ANSYS Software.

FORCE AND STRESS ANALYSIS USING TWO LINK ELEMENTS IN TRUSSES

EX.NO. 2

DATE:

AIM:

To perform a force and stress analysis of two link elements in trusses using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Consider the two bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. E = 210 GPa, A = 0.1 m^2.

PROCEDURE:

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – Link – 3D Finit stn 180 – ok – close.

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

4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX
   1. 210e9– Ok – close.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 0.75 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS –(0, -0.5),(x, y value w.r.t first node) – ok (third node is created

6. Create–Elements–Elem Attributes – Material number – 1 – Real constant set number
   1. 1 – ok

7. Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3–– ok (elements are created through nodes).

8. Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 &3 – apply – DOFs to be constrained – All DOF – ok

9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – 5000 (-ve value)

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

11. Element table – Define table – Add –'Results data item' – By Sequence num – LS – LS1 – ok.

12. Plot results – contour plot –Element table – item to be plotted LS,1, avg common nodes- yes average- ok.

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

14. Plot results- nodal solution-ok-DOF solution- Y component of displacement-ok.

15. Animation: PlotCtrls – Animate – Deformed shape – def+undeformed-ok.

RESULT:

Thus the force and stress analysis of two link element in trusses is done by using the ANSYS Software.

STRESS AND DEFLECTION ANALYSIS IN SIMPLY SUPPORTED BEAM WITH POINT LOAD

EX.NO. 3

DATE:

AIM:

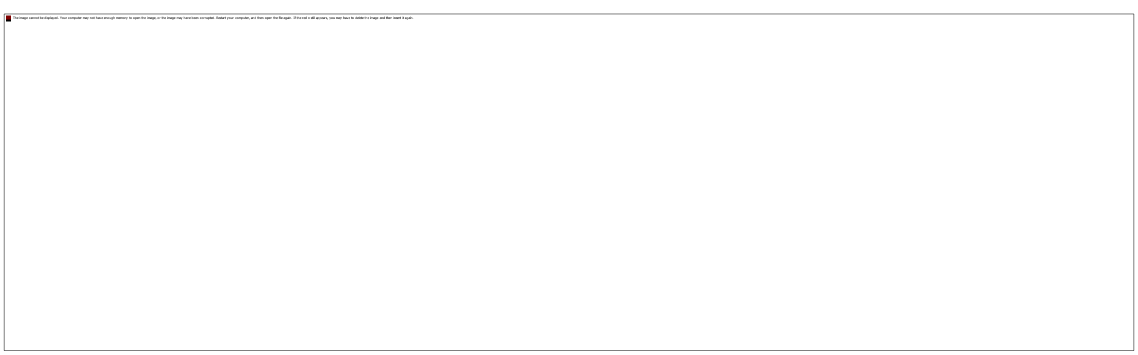
To perform a stress and deflection analysis in simply supported beam with point load using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100 mm * 100mm, Young's modulus of210 MPa, Poisson's ratio 0.27.



PROCEDURE:

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node BEAM 188– ok- close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 2.10e5– PRXY – 0.27 – ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) -enter b=100, h=100-preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 1000 (x value w.r.t first node) – apply (second node is created) – 2500 (x value w.r.t first node) – apply(third node is created)- x,y,z location in CS-3500 (x value w.r.t first node)-ok.

6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 apply – pick 2 & 3 apply – pick 3 & 4 – ok (elements are created through nodes).

7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply –DOFs to be constrained – all DOF – ok.

8. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value – -2000(-ve value) – ok-Force/Moment –

on Nodes- pick node 3 – apply –direction of For/Mom – FY – Force/Moment value – -4000(-ve value) – ok.

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

10. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

11. Stress: Plot Results – Contour plot – Nodal solution – stress – vonmises stress – ok.

12. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

13. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

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

NOTE: For Shear Force Diagram use the combination SMISC 6 & SMISC 19, for Bending Moment Diagram use the combination SMISC 3 & SMISC 16.

16. Animation: PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

RESULT:

Thus the stress and deflection analysis in simply supported beam with point load is done by using the ANSYS Software.

STRESS AND DEFLECTION ANALYSIS IN SIMPLY SUPPORTED BEAM WITH UNIFORMLY VARYING LOAD

EX.NO. 4

DATE:

AIM:

To perform a stress and deflection analysis in simply supported beam with UVL.

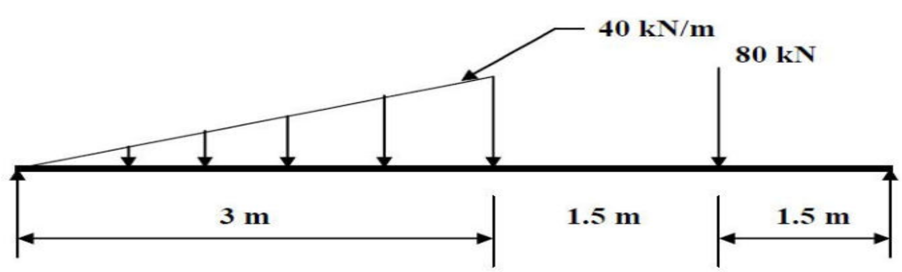
using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 100mm * 100m m, Young's modulus of 2.1×10^5 N/mm², Poisson's ratio= 0.27.



PROCEDURE:

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method- ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 nodes Beam 188 – ok – close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 2.1e5– PRXY – 0.27 –ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=100, h=100-preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 3000 (x value w.r.t first node) – apply (second node is created) – 4500 (x value w.r.t first node) –apply (third node is created) – 6000 (x value w.r.t first node) – ok (forth node is created).

6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3

   – apply –pick 3 & 4 – ok (elements are created through nodes).

7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 – apply –DOFs to be constrained – all DOF – ok.

8. Loads – Define loads – apply – Structural – Pressure – on Beams – pick element between nodes 1 & 2–apply–pressure value at node I– 0 (value)– pressure value at node J – 40000–ok.

9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – (-80000) (-ve value) – ok.

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

11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.

13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

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

17. Animation: PlotCtrls – Animate – Deformed results – DOF solution – deformed + undeformed – ok.

RESULT:

Thus the stress and deflection analysis in simply supported beam with UVL is done by using the ANSYS Software.

STRESS AND DEFLECTION ANALYSIS IN SIMPLY SUPPORTED BEAM WITH UNIFORMLY DISTRIBUTED LOAD

EX.NO. 5

DATE:

AIM:

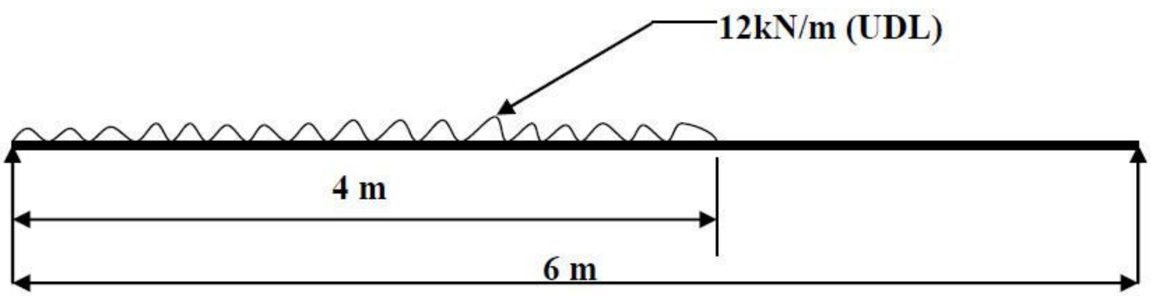
To perform a stress and deflection analysis in simply supported beam with uniformly distributed load using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.1 m * 0.1 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



PROCEDURE:

1. Ansys Main Menu – Preferences-select – STRUCTURAL – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188 – ok- close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 210e9– PRXY – 0.27 –ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=100, h=100-preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 4 (x value w.r.t first node) – apply (second node is created) – 6 (x value w.r.t first node) – ok (third node is created).

6. Create – Nodes – Fill between Nds – pick 1 & 2 – apply – number of nodes to fill 7 – startingnode no – 4 – ok.

7. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 4 apply– pick 4 & 5 apply– pick 5 & 6 apply– pick 6 & 7 apply– pick 7 & 8 apply– pick 8 & 9 apply – pick 9 & 10 apply– pick 10 & 2 apply – pick 2 & 3 – ok (elements are created through nodes).

8. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 3 – apply – DOFs to be constrained – UY – ok.

9. Loads – Define loads – apply – Structural – Pressure – on Beams – pick all elements betweennodes 1 & 2 – apply – pressure value at node I – 12000 – pressure value at node J – 12000 –ok.

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

11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.

13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS 6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS 3 – Element table item at node J – SMIS 16 – ok (bending moment diagram will be displayed).

16.Reaction forces: List Results – reaction solution – items to be listed – All items – ok

(reaction forces will be displayed with the node numbers).

17. PlotCtrls – Animate – Deformed results – DOF solution – USUM – ok.

RESULT:

Thus the stress and deflection analysis in simply supported beam with uniformly distributed load done by using the ANSYS Software.

STRESS AND DEFLECTION ANALYSIS IN BEAM WITH MOMENT AND OVERHANGING

EX.NO. 6

DATE:

AIM:

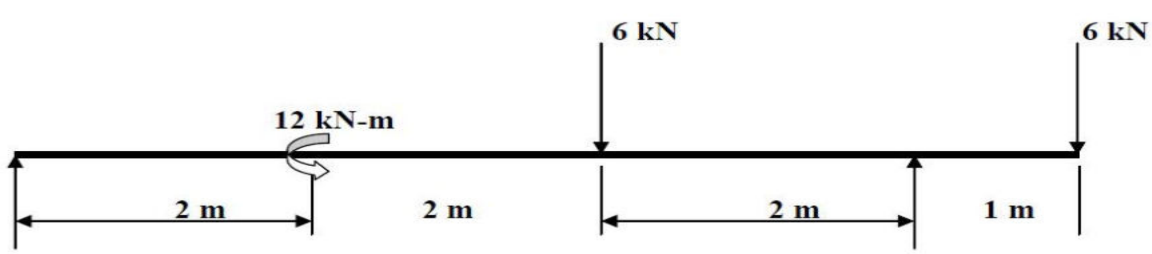
To perform a stress and deflection analysis in beam with moment and over hanging using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



PROCEDURE:

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method- ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188 – ok – close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 210e9– PRXY – 0.27 –ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1^{st} element) - enter b=200, h=300- preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 2 (x value w.r.t first node) – apply (second node is created) – 4 (x value w.r.t first node) – apply (third node is created) – 6 (x value w.r.t first node) – apply (forth node is created) – 7 (x value w.r.t first node) – ok (fifth node is created).

6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3

   – apply –pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).

7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 & 4 –apply –DOFs to be constrained – UY – ok.

8. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply direction of For/Mom – MZ – Force/Moment value - 12000 (anticlockwise, +ve value) – apply –pick node 3 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) –apply – pick node 5 – apply – direction of For/Mom – FY – Force/Moment value - -6000 (-ve value) – ok.

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

10. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

11. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.

12. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

13. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

14.Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 –

Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

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

2. Animation: Plot Ctrls – Animate – Deformed results – DOF solution – deformed + undeformed – ok.

RESULT:

Thus the stress and deflection analysis in beam with moment and over hanging is done by using the ANSYS Software.

STRESS AND DEFLECTION ANALYSIS IN CANTILEVER BEAM WITH POINT LOAD

EX.NO. 7

DATE:

AIM:

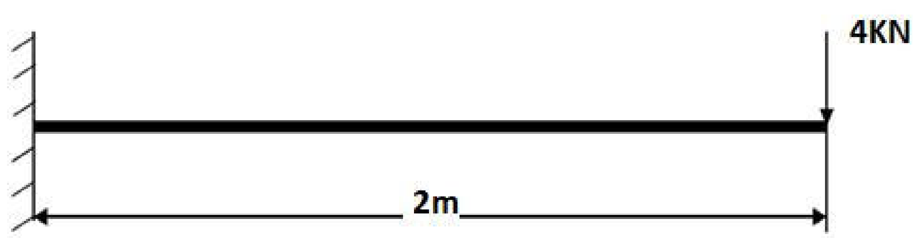
To perform a stress and deflection analysis in cantilever beam with point load using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Compute the Shear force and bending moment diagrams for the beam shown and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa, Poisson's ratio 0.27.



PROCEDURE:

1. Ansys Main Menu – Preferences-Select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node Beam 188 – ok- close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 210e9– PRXY – 0.27 –ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=200, h=300-preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 2 (x value w.r.t first node) – ok (second node is created).

6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

7. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply –

8. DOFs to be constrained – ALL DOF – ok.

9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FY – Force/Moment value –(-40000) (-ve value) – ok.

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

11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.

13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

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

RESULT:

Thus the stress and deflection analysis in cantilever beam with point load is done by using the ANSYS Software.

STRESS AND DEFLECTION ANALYSIS IN BEAM WITH ANGULAR LOADS

EX.NO. 8

DATE:

AIM:

To perform a stress and deflection analysis in beam with angular load using analysis software ANSYS.

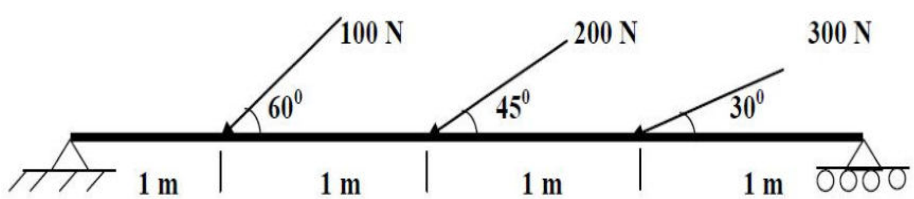
SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Compute the Shear force and bending moment diagrams for the beam shown in fig such that one end hinged and at the other end is having roller support and find the maximum deflection. Assume rectangular c/s area of 0.2 m * 0.3 m, Young's modulus of 210 GPa,

Poisson's ratio 0.27.



PROCEDURE:

1. Ansys Main Menu – Preferencesselect – STRUCTURAL – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2node 188 – ok – close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 210e9– PRXY – 0.27 –ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=200, h=300-preview-ok.

5. Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS– 1 (x value w.r.t first node) – apply (second node is created) – 2 (x value w.r.t first node) – apply (third node is created) – 3 (x value w.r.t first node) – apply (forth node is created) – 4 (x value w.r.t first node) – ok (fifth node is created).

6. Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3

   – apply – pick 3 & 4 – apply – pick 4 & 5 – ok (elements are created through nodes).

7. Create – Nodes – Rotate nodes CS – by angles – pick node 2 – apply – about nodal z-axis – 60 – apply – pick node 3 – apply about nodal z- axis – 45 – apply – pick node 4

   – apply – about nodal z –axis – 30 – ok.

8. Loads – Define loads – apply – Structural – Displacement – on Nodes- pick node 1 – apply – DOFs to be constrained – UX & UY – apply – pick node 5 – apply – DOFs to be constrained – UY – ok.

9. Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply direction of For/Mom – FX – Force/Moment value - -100 (-ve value) – apply – pick node 3 – apply – direction of For/Mom – FX – Force/Moment value - -200 (-ve value) – apply – pick node 4 – apply – direction of For/Mom – FX – Force/Moment value - 300 (-ve value) – ok.

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

11. Displacement: Plot Results – Contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

12. Stress: Plot Results – Contour plot – Nodal solution – stress – von mises stress – ok.

13. Element table – Define table – Add – 'Results data item' – By Sequence num – SMISC – SMISC, 6 – apply, By Sequence num – SMISC – SMISC, 19 – apply, By Sequence num – SMISC – SMISC, 3 – apply, By Sequence num – SMISC – SMISC, 16 – ok – close.

14. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS6 – Elem table item at node J – SMIS19 – ok (Shear force diagram will be displayed).

15. Plot results – contour plot – Line Element Results – Elem table item at node I – SMIS3 – Elem table item at node J – SMIS16 – ok (bending moment diagram will be displayed).

RESULT:

Thus the stress and deflection analysis in beam with angular load is done by using the ANSYS Software.

STRESS ANALYSIS OF A RECTANGULAR PLATE WITH CIRCULAR HOLE

EX.NO. 9

DATE:

AIM:

To perform a stress analysis of a rectangular plate with circular hole using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram : 2 GB

Processor : intel CORE i3

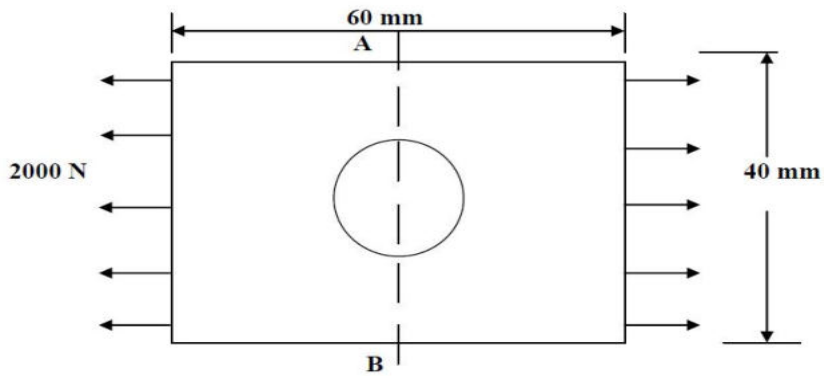
Operating system : Window XP Service Pack 3

Software : ANSYS (Version10.0)

PROBLEM DESCRIPTION:

In the plate with a hole under plane stress, find deformed shape of the hole and determine the maximum stress distribution along A-B (you may use t = 1 mm). E = 210GPa, t = 1 mm, Poisson's ratio = 0.3, Dia of the circle = 10 mm, Analysis assumption

plane stress with thickness is used.



PROCEDURE:

1. Ansys Main Menu – Preferences-Select – STRUCTURAL-h method – ok

2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 42 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

3. Real constants – Add – ok – real constant set no – 1 – Thickness – 1 – ok.

4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2.1e5 –PRXY – 0.3 – ok – close.

5. Modeling –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 60, 0, 40 – ok.

6. Create – Area – Circle – solid circle – X, Y, radius – 30, 20, 5 – ok.

7. Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (rectangle) – apply – pick area which is to be deleted (circle) – ok.

8. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

9. Loads – Define loads – apply – Structural – Displacement – on Nodes – select box – drag the left side of the area – apply – DOFs to be constrained – ALL DOF – ok.

10. Loads – Define loads – apply – Structural – Force/Moment – on Nodes – select box – drag the right side of the area – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok.

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

12. Deformed shape-Plot Results – Deformed Shape – def+undeformed – ok.

13. Plot results – contour plot – Element solution – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

RESULT:

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

STRESS ANALYSIS OF A THE CORNER ANGLE BRACKET

EX.NO. 10

DATE:

AIM:

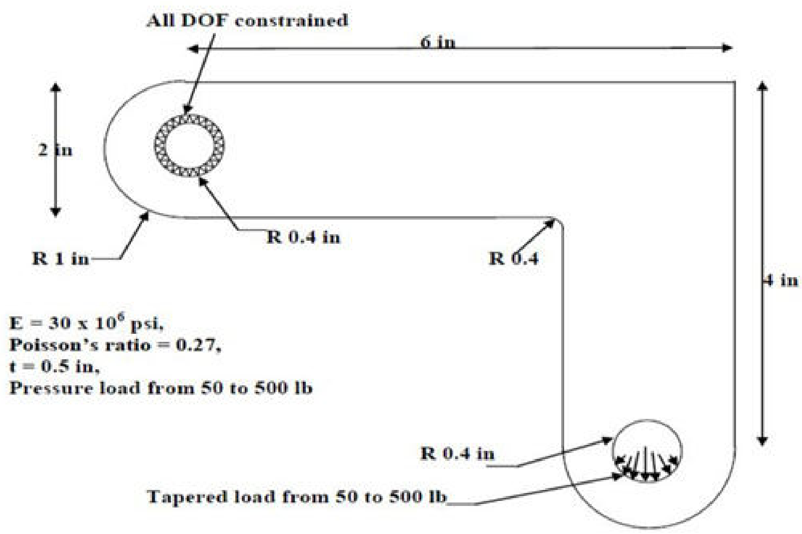
To perform a stress analysis of a corner angle bracket using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

The corner angle bracket is shown below. The upper left hand pin-hole is constrained around its entire circumference and a tapered pressure load is applied to the bottom of lower right hand pin-hole. Compute Maximum displacement, Von-Mises stress.



PROCEDURE:

1. Ansys Main Menu – Preferencesselect – STRUCTURAL – ok

2. Element type – Add/Edit/Delete – Add – Solid – Quad 8 node – 82 – ok – option – element behavior K3 – Plane stress with thickness – ok – close.

3. Real constants – Add – ok – real constant set no – 1 – Thickness – 0.5 – ok.

4. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 30e6 –PRXY – 0.27 – ok – close.

5. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 6, 0, 2 – apply –Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 4, 6, -2, 2 – ok. Create – Area –Circle – solid circle – X, Y, radius – 0, 1, 1 – apply – X, Y, radius – 5, -2, 1 – ok.

6. Operate – Booleans – Add – Areas – pick all.

7. Create – Lines – Line fillet – pick the two lines where fillet is required – apply – fillet radius – 0.4 – ok. Create – Areas – Arbitrary – by lines – pick filleted lines – ok. Operate – Booleans –

8. Add – Areas – pick all. Create – Area – Circle – solid circle – X, Y, radius – 0, 1, 0.4 – apply –X, Y, radius – 5, -2, 0.4 – ok.

9. Operate – Booleans – Subtract – Areas – pick area which is not to be deleted (bracket) – apply – pick areas which is to be deleted (pick two circles) – ok.

10. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine– pick all – Level of refinement – 3 – ok.

11. Loads – Define loads – apply – Structural – Displacement – on Lines – select the inner lines of the upper circle – apply – DOFs to be constrained – ALL DOF – ok.

12. Loads – Define loads – apply – Structural – Pressure – on Lines – Pick line defining bottom left part of the circle – apply – load PRES value – 50 – optional PRES value – 500 – ok. Structural – Pressure – on Lines – Pick line defining bottom right part of the circle – apply – load PRES value – 500 – optional PRES value – 50 – ok.

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

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

1. Plot results – contour plot – Element solu – Stress – Von Mises Stress – ok (the stress distribution diagram will be displayed).

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

RESULT:

Thus the stress analysis of a corner angle bracket is done by using the ANSYS Software.

STRESS ANALYSIS OF AN AXI-SYMMETRIC COMPONENT

EX.NO. 11

DATE:

AIM:

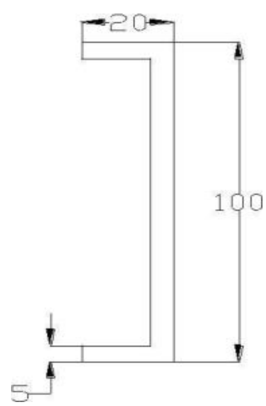
To perform a stress analysis of an axis symmetric component using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

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.



Young"s Modulus	: 200 GPa
Poisson"s ratio	: 0.3

PROCEDURE:

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.

1. Solution - Analysis Type-New Analysis-Static

1. Solution - Define loads - Apply .Structural - displacement - symmetry BC - on lines. (Pick the two edger on the left at X = 0)

2. Utility menu - select - Entities - select all

3. Utility menu - select - Entities - by location - Y = 50 -ok.

4. Solution - Define loads - Apply - Structural - Force/Moment - on key points - FY = 100 - Pick the top left corner of the area -ok

5. Solution - Define Loads - apply - Structural - Force/moment - on key points - FY = -100- Pick the bottom left corner of the area -ok

1. Solution - Solve - Current LS

2. Utility Menu - select - Entities

3. Select nodes - by location - Y coordinates and type 45, 55 in the min., max. box, as Shown below and click ok

1. General postprocessor - List results - Nodal solution - stress - components SCOMP

2. Utility menu - plot controls - style - Symmetry expansion - 2D Axisymmetric - ¾ expansion.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.

RESULT:

Thus the stress analysis of an axis symmetric component is done by using the ANSYS Software

THERMAL STRESS ANALYSIS WITHIN THE RECTANGULAR PLATE

EX.NO. 12

DATE:

AIM:

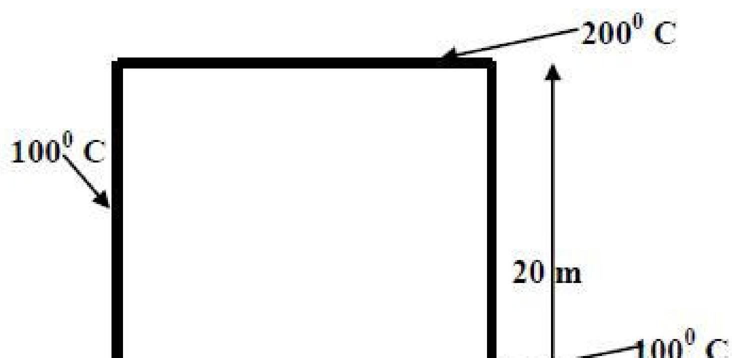
To perform a thermal stress analysis of a rectangular plate using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

2-D heat conduction problem for the temperature distribution within the rectangular plate. Thermal conductivity of the plate, KXX=401 W/(m-K).



PROCEDURE:

1. Ansys Main Menu – Preferences-select – THERMAL- h method– ok

2. Element type – Add/Edit/Delete – Add – Solid – Quad 4 node – 55 – ok – option – elementbehavior K3 – Plane stress with thickness – ok – close.

3. Material Properties – material models – Thermal – Conductivity – Isotropic – KXX – 401.

4. Modeling – Create – Area – Rectangle – by dimensions – X1, X2, Y1, Y2 – 0, 10, 0, 20 – ok.

5. Meshing – Mesh Tool – Mesh Areas – Quad – Free – Mesh – pick all – ok. Mesh Tool – Refine – pick all – Level of refinement – 3 – ok.

6. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 1000 C lines – apply – DOFs to be constrained – TEMP – Temp value – 1000 C – ok.

7. Loads – Define loads – apply – Thermal – Temperature – on Lines – select 1000 C lines –

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

9. Read results-last set-ok

10. List results-nodal solution-select temperature-ok

11. Observe the nodal solution per node.

12. From the menu bar-plot ctrls-style-size and shape-display of the element-click on real constant multiplier=0.2, don't change other values-ok.

13. Plot results-contour plot-nodal solution-temperature-deformed shape only-ok

14. Element table-define table-add-enter user label item=HTRANS, select by sequence no SMISC, 1-ok-close.

15. Element table-list table-select HTRANS-ok

RESULT:

Thus the thermal stress analysis of a rectangular plate is done by using the ANSYS Software.

CONVECTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

EX.NO. 13

DATE:

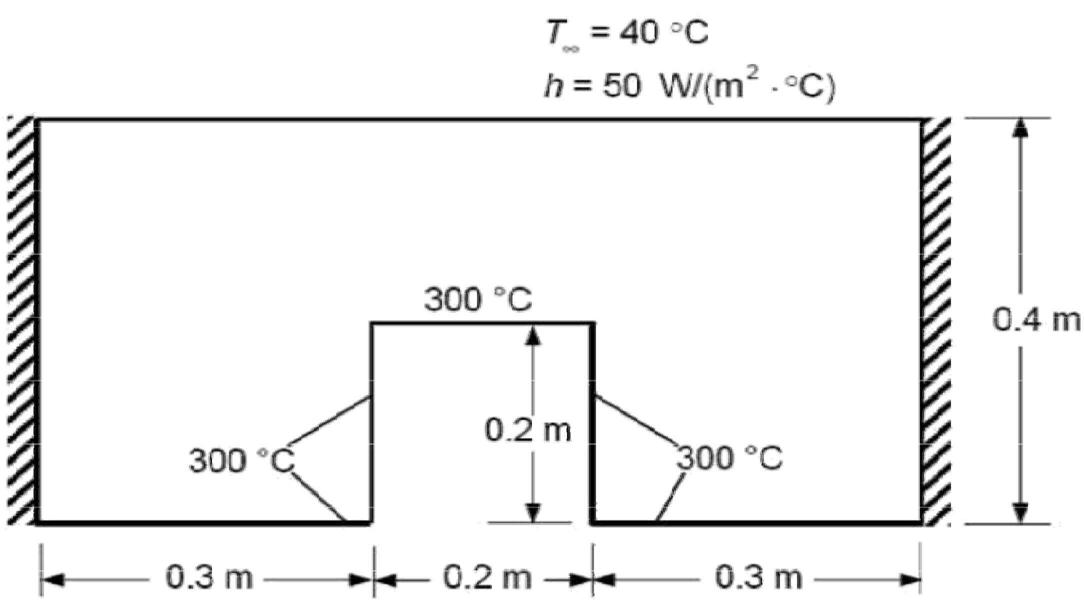
AIM:

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

SYSTEM CONFIGURATION

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

PROBLEM DESCRIPTION:



Thermal Conductivity of the material= 16 W/m.°C

PROCEDURE:

1. Preference – Thermal - 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. [We can create full object (or) semi-object if it is a symmetrical shape]

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

1. 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.

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

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

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

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

RESULT:

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

MODEL ANALYSIS OF CANTILEVER BEAM WITHOUT LOAD

EX.NO. 14

DATE:

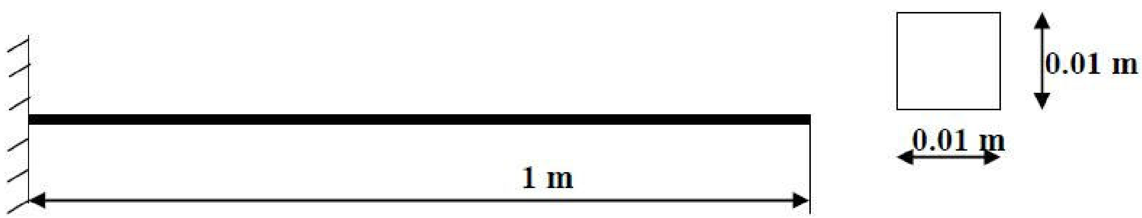
AIM:

To perform a Model Analysis of Cantilever beam without load using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram : 2 GB

Processor : intel CORE i3



Operating system : Window XP Service Pack 3

Software : ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Model Analysis of Cantilever beam for natural frequency determination.

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

PROCEDURE:

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node 188– ok- close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 200e9– PRXY – 0.27 – Density – 7800 – ok – close.

4. Sections-Beams-common sections- sub type- rectangle (1st element) - enter b=0.01, h=0.01- preview-ok.

5. Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations –1,0 – ok (Keypoints created).

6. Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.

7. Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1

   – ok. Mesh– Lines – Pick All – ok.

8. Solution – Analysis Type – New Analysis – Modal – ok.

9. Solution – Analysis Type – Subspace – Analysis options – no of modes to extract – 5

   – no of modes to expand – 5 – ok – (use default values) – ok.

10. Solution – Define Loads – Apply – Structural – Displacement – On Key points – Pick first key point – apply – DOFs to be constrained – ALL DOF – ok.

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

12. Result Summary

13. Read Results – First Set

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

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

16. Read Results – Next Set

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

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

RESULT:

Thus the Model Analysis of Cantilever beam without load for natural frequency is done by using the ANSYS Software.

MODEL ANALYSIS OF CANTILEVER BEAM WITH LOAD

EX.NO. 15

DATE:

AIM:

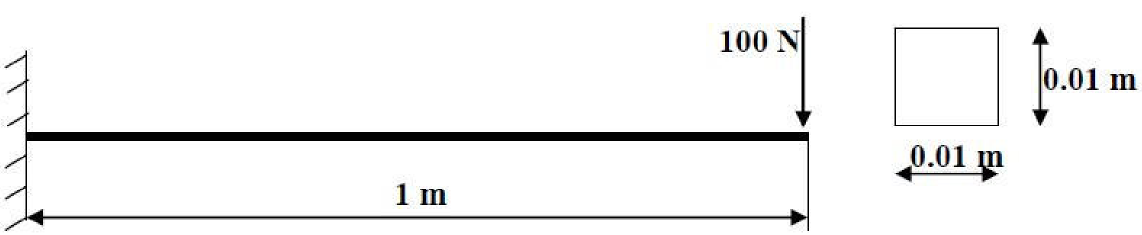
To perform a Model Analysis of Cantilever beam with load using analysis software ANSYS.

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:

Conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1 - 100 Hz. Modulus of elasticity = 200GPa, Poisson's ratio = 0.3, Density = 7800 Kg/m3.



PROCEDURE:

1. Ansys Main Menu – Preferences-select – STRUCTURAL- h method – ok

2. Element type – Add/Edit/Delete – Add – BEAM – 2 node BEAM 188 – ok – close.

3. Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX

   – 200e9– PRXY – 0.3 – Density – 7800 – ok.

4. Sections-Beams-common sections- sub type- rectangle (1^{st} element) - enter b=100, h=100- preview-ok.

5. Modeling – Create – Keypoints – in Active CS – x,y,z locations – 0,0 – apply – x,y,z locations –1,0 – ok (Keypoints created).

6. Create – Lines – lines – in Active Coord – pick keypoints 1 and 2 – ok.

7. Meshing – Size Cntrls – ManualSize – Lines – All Lines – element edge length – 0.1

   – ok. Mesh– Lines – Pick All – ok.

8. Solution – Analysis Type – New Analysis – Harmonic – ok.

9. Solution – Analysis Type – Subspace – Analysis options – Solution method – FULL – DOF printout format – Real + imaginary – ok – (use default values) – ok.

10. Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – Pick firstkeypoint – apply – DOFs to be constrained – ALL DOF – ok.

11. Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – Pick secondnode – apply – direction of force/mom – FY – Real part of force/mom – 100 – imaginary part of force/mom – 0 – ok.

12. Solution – Load Step Opts – Time/Frequency – Freq and Substps... – Harmonic frequency range– 0 – 100 – number of substeps – 100 – B.C – stepped – ok.

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

14. TimeHistPostpro

    Select 'Add' (the green '+' sign in the upper left corner) from this window – Nodal solution - DOF solution – Y component of Displacement – ok. Graphically select node 2 – ok.

    Select 'List Data' (3 buttons to the left of 'Add') from the window.

15. Time History Variables' window click the 'Plot' button, (2 buttons to the left of 'Add') Utility Menu – PlotCtrls – Style – Graphs – Modify Axis – Y axis scale – Logarithmic –ok. Utility Menu – Plot – Replot.

16. This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.

RESULT:

Thus the Model Analysis of Cantilever beam with load for natural frequency is done by using the ANSYS Software.

HARMONIC ANALYSIS OF A 2D COMPONENT

EX.NO. 16

DATE:

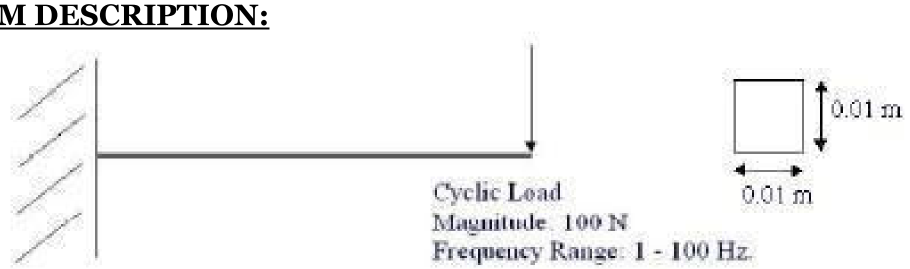
AIM:

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

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROBLEM DESCRIPTION:



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

PROCEDURE:

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

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

1. 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

RESULT:

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

MATLAB BASICS, DEALING WITH MATRICES, GRAPHING-FUNCTIONS OF ONE VARIABLE AND TWO VARIABLES

EX.NO. 17

DATE:

AIM:

To study the capabilities of MATLAB software

INTRODUCTION:

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

1. Math and computation
2. Algorithm development
3. Data acquisition
4. Modeling, simulation, and prototyping
5. Data analysis, exploration, and visualization
6. Scientific and engineering graphics
7. 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.

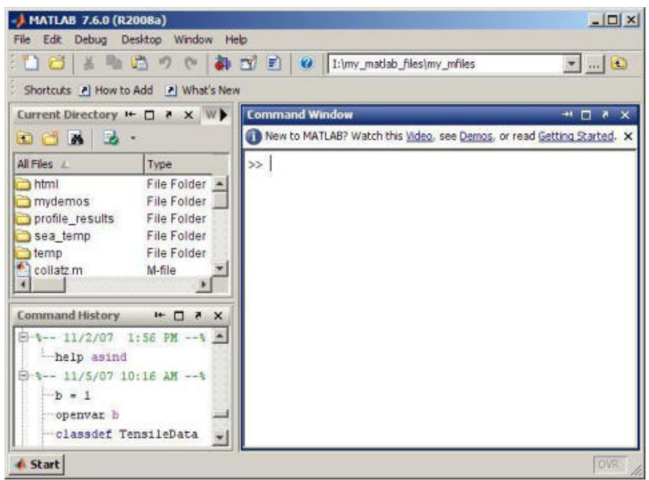
The name MATLAB stands for matrix laboratory. MATLAB was originally written to provide easy access to matrix software developed by the LINPACK and EISPACK projects. Today, MATLAB engines incorporate the LAPACK and BLAS libraries, embedding the state of the art in software for matrix computation

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

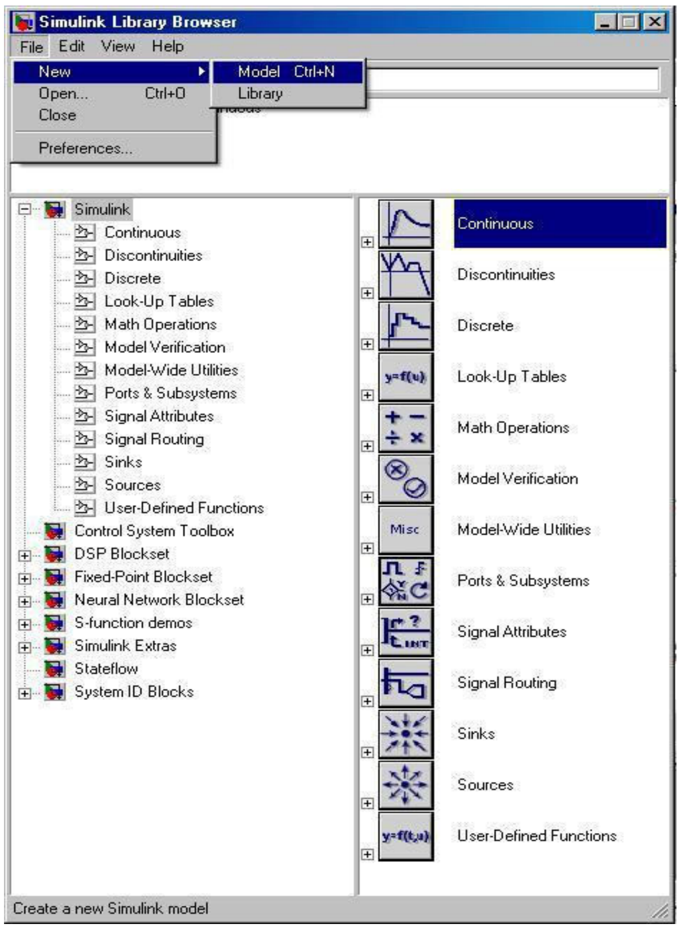
4. Running Simulations

5. Building Systems

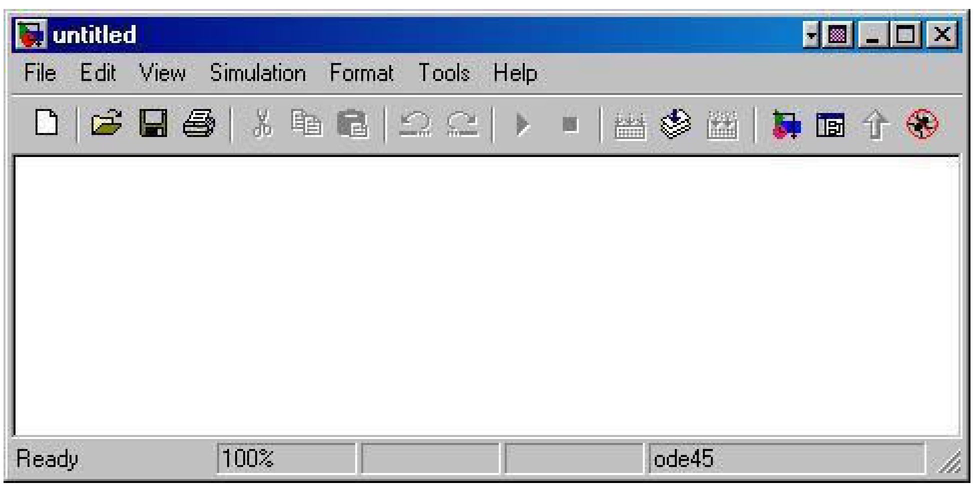
Starting Simulink:

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

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



Open the modeling window with New then Model from the File menu on the Simulink Library Browser as shown above. This will bring up a new untitled modeling window shown below.



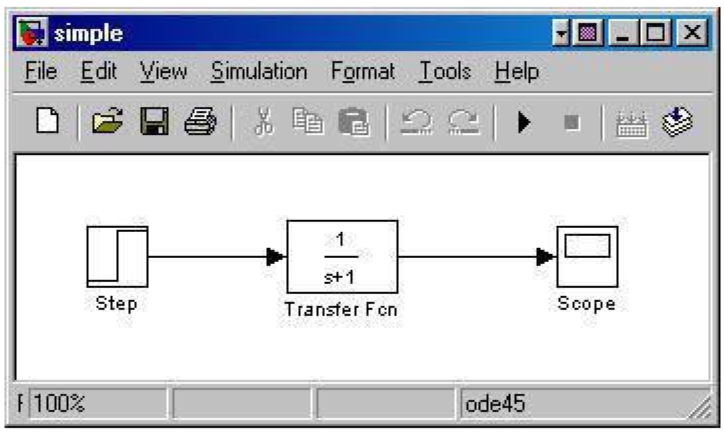
Model Files:

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

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

>> Filename

The following is an example model window.



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

Basic Elements:

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

Blocks:

There are several general classes of blocks:

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

1. Model-wide Utilities
2. Ports & subsystems
3. Signal attributes
4. Signal routing
5. Sinks: Used to output or display signals
6. Sources: Used to generate various signals

7. User-defined functions

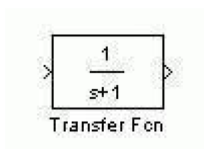
8. Discrete: Linear, discrete-time system elements (transfer functions, state-space models, etc.)

9. Linear: Linear, continuous-time system elements and connections (summing junctions, gains, etc.)

10. Nonlinear: Nonlinear operators (arbitrary functions, saturation, delay, etc.)

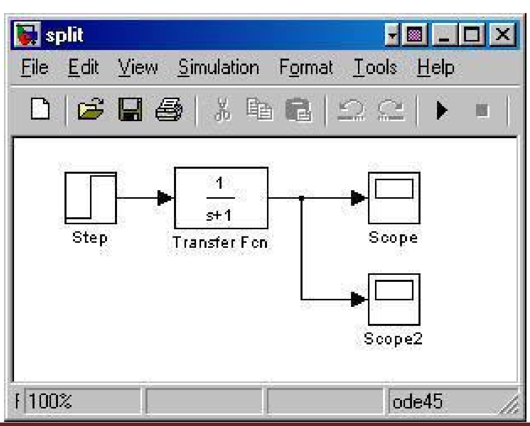
11. Connections: Multiplex; Demultiplex, System Macros, etc.

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.



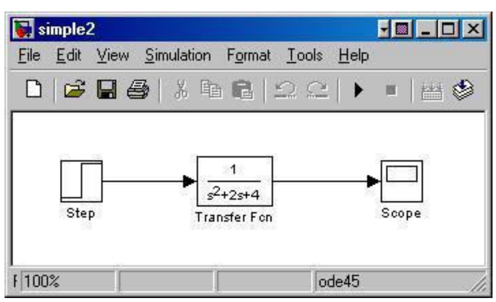
Lines can never inject a signal into another line; lines must be combined through the use of a block such as a summing junction.

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.

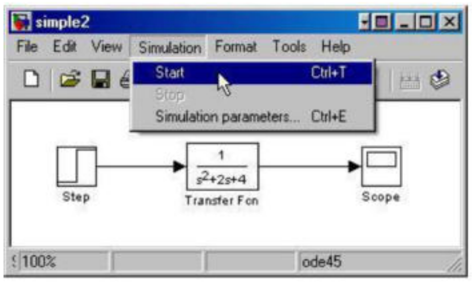
Running Simulations:

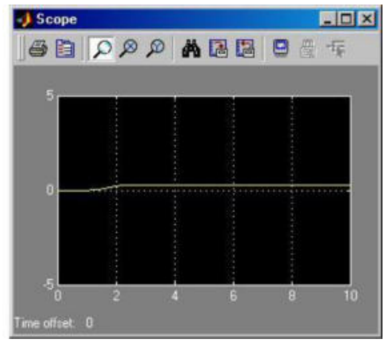
To run a simulation, we will work with the following model
file: simple2.mdl

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



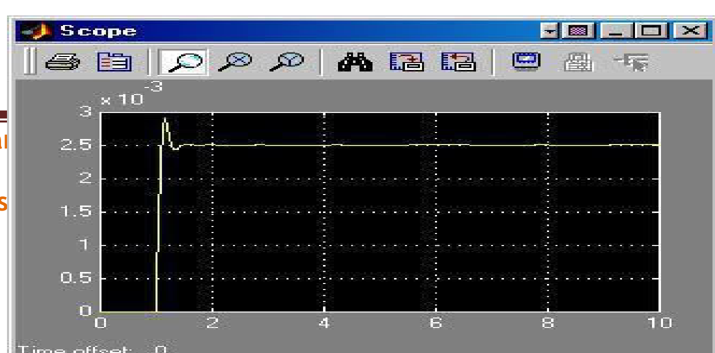
Before running a simulation of this system, first open the scope window by double-clicking on the scope block. Then, to start the simulation, either select Start from the Simulation menu (as shown below) or hit Ctrl-T in the model window.





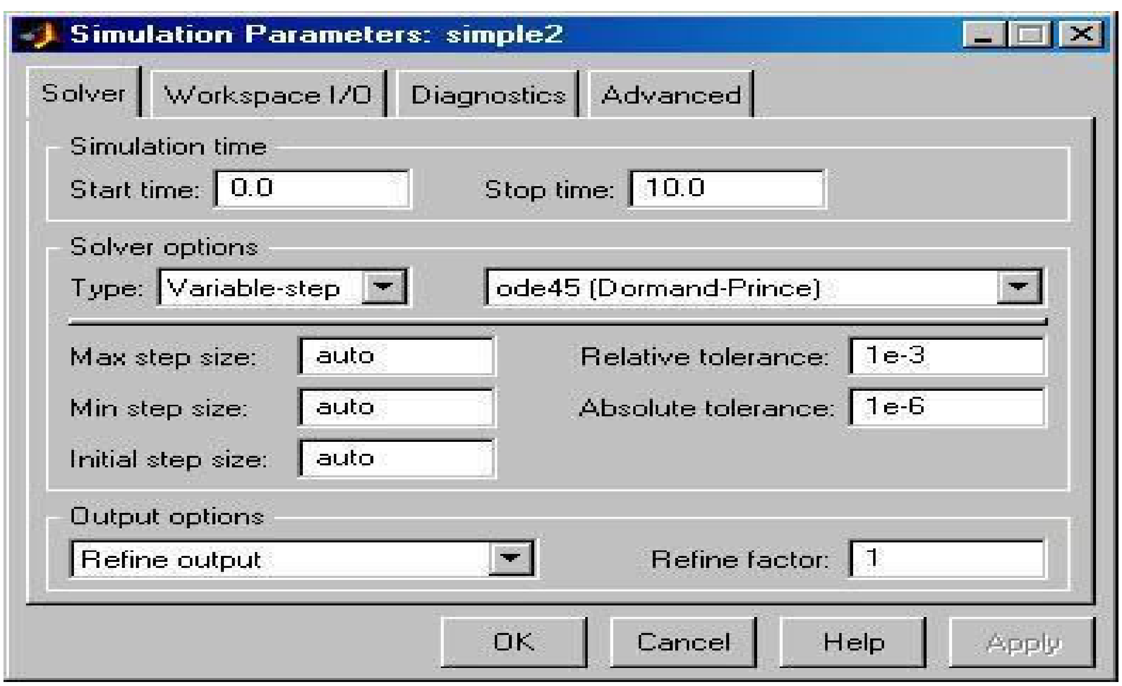
The simulation should run very quickly and the scope window will appear as shown below. If it doesn't, just double click on the block labeled "scope." Note that the simulation output (shown in yellow) is at a very low level relative to the axes of the scope. To fix this, hit the autoscale button (binoculars), which will rescale the axes as shown below.

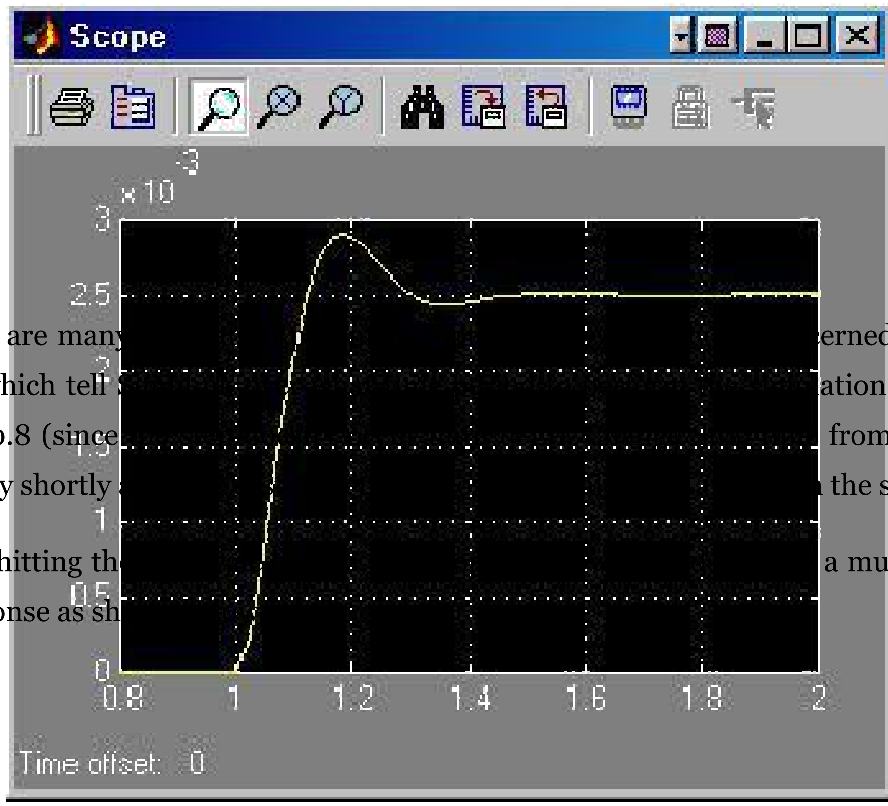
Note that the step response does not begin until t=1. This can be changed by double-clicking on the "step" block. Now, we will change the parameters of the system and simulate the system again. Double-click on the "Transfer Fcn" block in the model window and change the denominator to Re-run the simulation (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 auto scale 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 ... erned with the start and stop times, which tell ... ation. Change Start time from 0.0 to 0.8 (since ... from 10.0 to 2.0, which should be only shortly ... the simulation.

After hitting th... a much better display of the step response as sh...

RESULT:

Thus the features of MATLAB are studied.

SIMULATION OF SPRING-MASS SYSTEM USING MAT LAB

EX.NO. 18

DATE:

AIM:

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

SYSTEM CONFIGURATION:

Ram	: 2 GB
Processor	: intel CORE i3
Operating system	: Window XP Service Pack 3
Software	: ANSYS (Version10.0)

PROCEDURE:

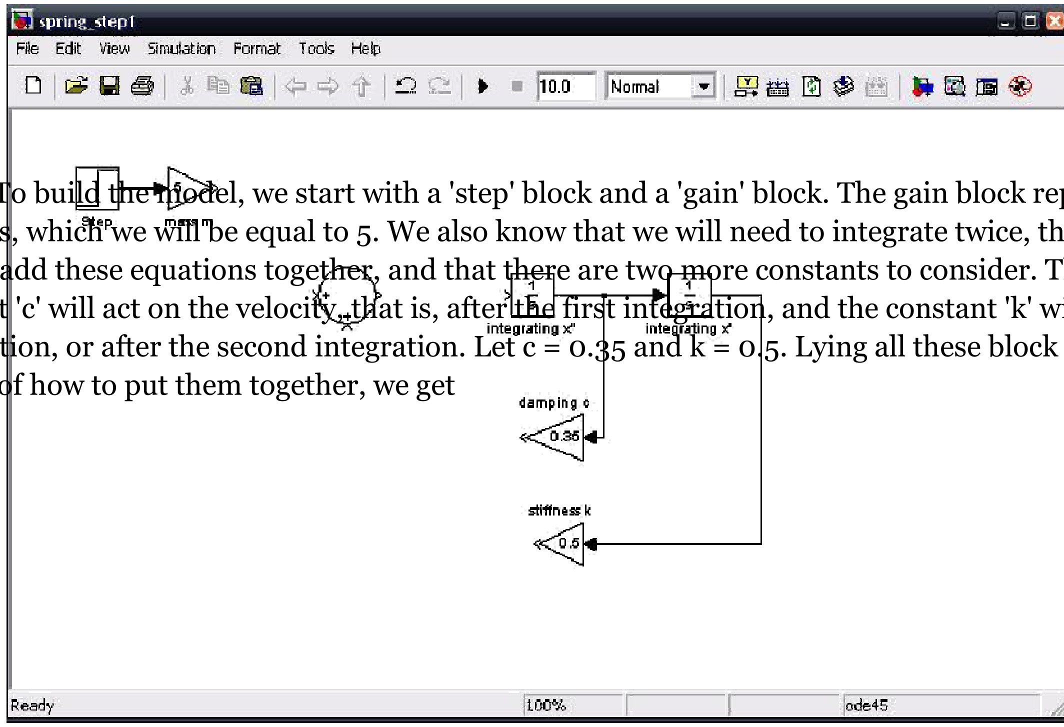
A mass on a spring with a velocity-dependent damping force and a time-dependent force acting upon it will behave according to the following equation:

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

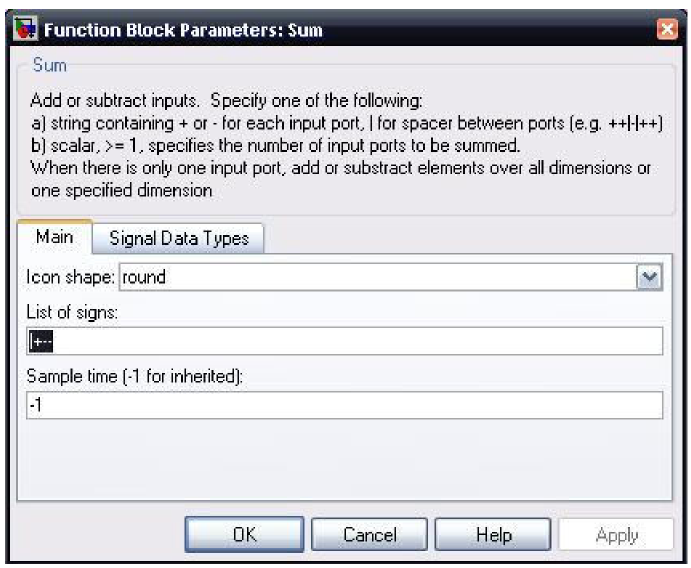
The model will be formed around this equation. In this equation, 'm' is the equivalent mass of the system; 'c' is the damping constant; and 'k' is the constant for the stiffness of the spring. First we want to rearrange the above equation so that it is in terms of acceleration; then we will integrate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

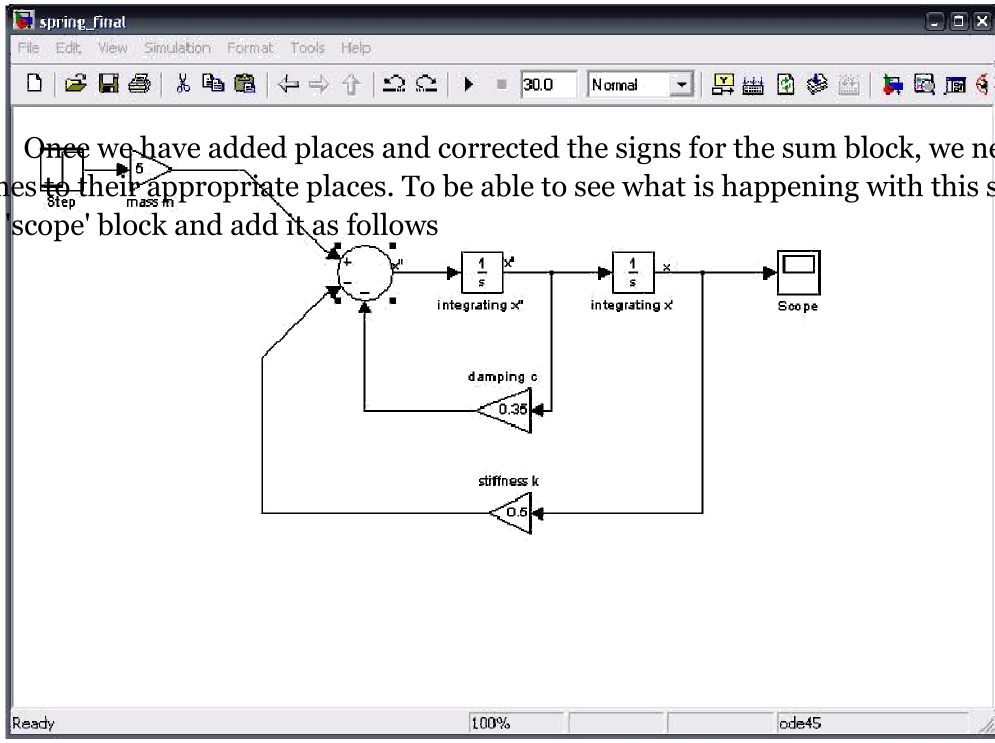
$$\ddot{x} = \frac{1}{m}(f(t) - c\dot{x} - kx)$$

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. Lying 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:



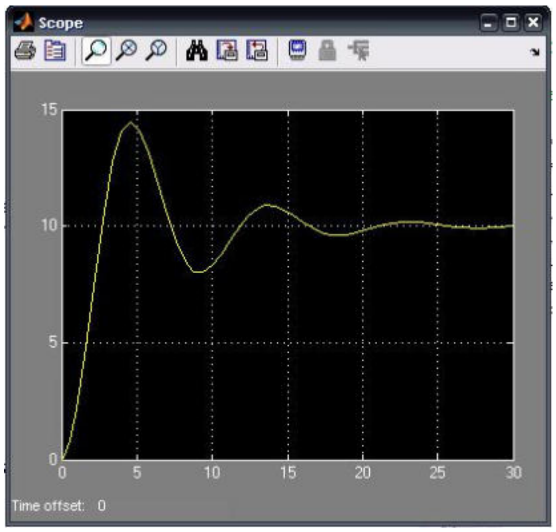


Once we have added places and corrected the signs for the sum block, we need only connect the lines to their appropriate places. To be able to see what is happening with this spring system, we add a 'scope' block and add it as follows

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.



SIMULATION OF CAM AND FOLLOWER MECHANISM USING MAT LAB

EX.NO. 19

DATE:

AIM:

To simulate the cam and follower mechanism using MATLAB software.

SYSTEM CONFIGURATION:

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

PROCEDURE:

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 displacements 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	0^0	0 mm
End of first half of the cycle	180^0	30 mm
End of the full cycle	360^0	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.

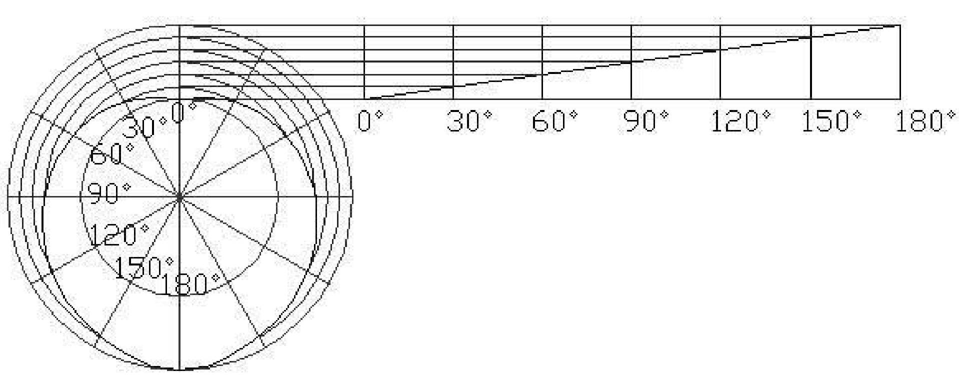
We shall take the cycle in steps. Firstly if the cam has to impart a displacement of 30mm on follower over half its cycle then it must impart a displacement of 30mm÷180° for every 1° turned by the cam i.e. it must move the follower 0.167mm per degree turn. This distance is to much to small to draw on 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 1^{o} is too small we will use increments of 30^{o} 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.