Chapter 1

Introduction to FEA and SOLIDWORKS Simulation

Learning Objectives

After completing this chapter, you will be able to:

- *Understand basic concepts and general working of FEA*
- *Understand advantages and limitations of FEA*
- *Understand important terms and definitions used in FEA*
- *Understand SOLIDWORKS simulation*
- *Understand the type of analysis performed using SOLIDWORKS Simulation*
- *Basic steps in SOLIDWORKS Simulation*

INTRODUCTION TO FEA

The Finite Element Analysis (FEA) is a computing technique that is used to obtain approximate solution to boundary value problems. It is a numerical procedure to find solution of engineering problems through various analyses such as structural, thermal, fluid flow, electrical, and so on. The most common use of FEA is in structural analysis. These problems are basically mathematical models for physical situation. These mathematical models are differential equations with a set of boundary values and initial conditions. The method used to derive these equations is called Finite Element Method (FEM).

To understand the concept of FEA, consider a circle whose perimeter is to be calculated. To measure the perimeter of the circle without using the conventional formula, divide it into equal segments, as shown in Figure 1-1. Next, join the start and end points of these segments with a straight line. Now, you can measure the length of the straight line very easily, and thus, the perimeter of the circle by adding the length of these straight lines.

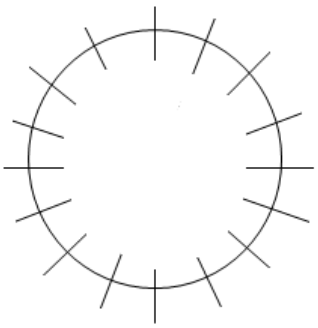

Figure 1-1 *The circle divided into small equal segments*

If the number of segments into which the circle is divided is less, you will not get accurate results. For accuracy, divide the circle into more number of segments. However, with more segments, the time required will be more. The same concept can be applied to FEA also, and therefore, there is always a compromise between accuracy and speed while using this method. This compromise between accuracy and speed makes it an approximate method.

The FEA was first developed to be used in the aerospace and nuclear industries where the safety of structures is critical. Today, even the simplest of products rely on FEA for design evaluation.

The FEA simulates the loading conditions of a design and determines the design response in those conditions. It can be used in new product design as well as in existing product refinement. A model is divided into a finite number of regions/divisions called elements. These elements can be of predefined shapes, such as triangular, quadrilateral, hexahedron, tetrahedron, and so on. The predefined shape of an element helps define the equations that describe how the element will respond to certain loads. The sum of the responses of all the elements in a model gives the total response of the complete model.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

GENERAL WORKING OF FEA

A better knowledge of FEA helps in creating more accurate finite element models. Also, it helps in understanding the backend working of SOLIDWORKS Simulation. Here, a simple model is discussed to give you a brief overview of the working of FEA.

Figure 1-2 shows a spring assembly that represents a simple two-spring element model. In this model, two springs are connected in series and one of the springs is fixed at the left most end point, refer to Figure 1-2. In this figure, the stiffness of the springs has been represented by the spring constants K_1 and K_2. The movement of end points of each spring is restricted in X direction only. The change in position from the undeformed state of each end point can be defined by X1 and X2 variables. The two forces acting on the end points of the springs are represented by F_1 and F_2.

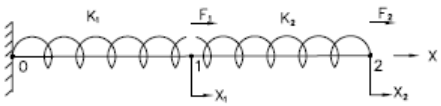

Figure 1-2 *Representation of a two-spring assembly*

To develop a model that can predict the state of this spring assembly, you can use the linear spring equation given below:

$$F = KX$$

where,
F = force applied,
X = displacement, and
K = spring constant

If you use the spring parameters defined above and assume a state of equilibrium, the following equations can be written for each endpoint:

$$F_1 - X_1K_1 + (X_2 - X_1)K_2 = 0$$
$$F_2 - (X_2 - X_1)K_2 = 0$$

Therefore,

$$F_1 = (K_1 + K_2)X_1 + (-K_2)X_2$$
$$F_2 = (-K_2)X_1 + K_2X_2$$

If the set of equation is written in matrix form, it will be represented as follows:

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \begin{bmatrix} K_1 + K_2 & -K_2 \\ -K_2 & K_2 \end{bmatrix} \begin{bmatrix} X_1 \\ X_2 \end{bmatrix}$$

In the above mathematical model, if the spring constants (K_1 and K_2) are known and the deformed shapes (X_1 and X_2) are defined, then the resulting forces (F_1 and F_2) can be determined. Alternatively, if the spring constants (K_1 and K_2) are known and the forces (F_1 and F_2) are defined,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

then the resulting deformed shapes (X_1 and X_2) can be determined. The terminologies that are used in the previous example are discussed next.

Stiffness Matrix

In the previous equation, the following part represents the stiffness matrix (K):

$$\begin{bmatrix} K_1 + K_2 & -K_2 \\ -K_2 & K_2 \end{bmatrix}$$

This matrix is relatively simple because it comprises only one pair of springs, but it turns complex when the number of springs increases.

Degrees of Freedom

Degrees of freedom is defined as the least number of independent coordinates required to define the configuration of a system in space. In the previous example, you were only concerned about displacement and forces. By making one endpoint fixed, you will restrict all degrees of freedom for that particular node. This means there will be no translational or rotational degrees of freedom for that node. But, still there are two nodes with some degrees of freedom. As these two nodes are allowed to translate along the X axis only, they have 1 degree of freedom each considering that no rotational degree of freedom exists in them. The number of degrees of freedom on free nodes in a model determines the number of equations required to solve a mathematical model.

Boundary Conditions

The boundary conditions are used to eliminate the unknowns in a system. A set of equations that is solvable is meaningless without the input. In the previous example, the boundary condition $X_0 = 0$, and the input forces are F_1 and F_2. Either ways, the displacements could have been specified in place of forces as boundary conditions and the mathematical model could have been solved for the forces. In other words, the boundary conditions help you reduce or eliminate the unknowns in the system.



Note

The solutions generated by using FEA are always approximate.

Elements and Element Shapes

Elements and element shapes are the building blocks of Finite Element Method (FEM). These are discussed next.

Elements

Element is an entity into which the system under study is divided. An element shape is specified by nodes. The shape (area, length, and volume) of an element depends on the nodes. A node and an element (triangular shaped) are shown in Figure 1-3.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

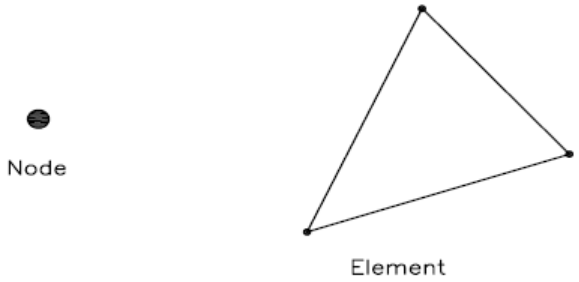

Figure 1-3 *A node and an element*

Element Shapes

There are many types of element shapes that are further divided into various classes depending on their usage. The following are some basic element shapes:

Line Element

A line element has the shape of a line or a curve. Therefore, a minimum of two nodes are required to define it. There can be higher order elements that have additional nodes (at the middle of the edge of an element). An element that does not have a node in between its edges is called a linear element. The elements that have nodes in between the edges are called quadratic or second order elements. Figure 1-4 shows some line elements.

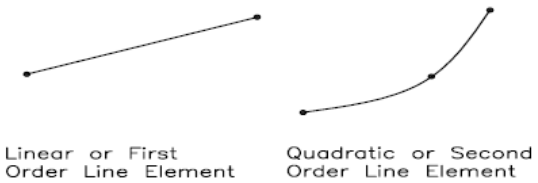

Figure 1-4 *Line elements*

Area Element

An area element has the shape of a triangle or a quadrilateral; therefore, it requires a minimum of three or four nodes to define it. Some area elements are shown in Figure 1-5.

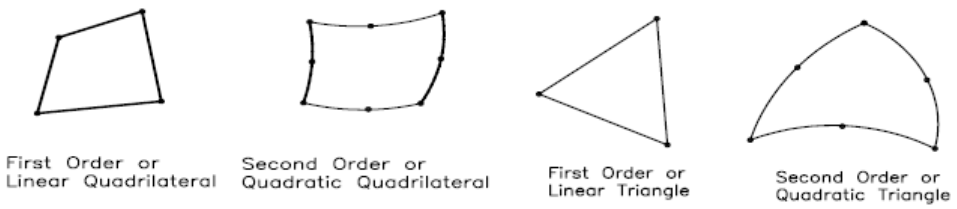

Figure 1-5 *The area elements*

Volume Element

A volume element has the shape of a hexahedron, wedge, tetrahedron, or a pyramid. Some of the volume elements are shown in Figure 1-6.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

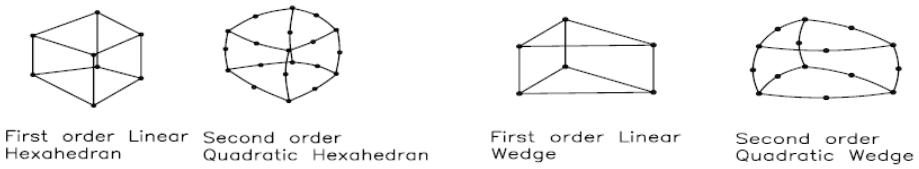

Figure 1-6 *The volume elements*

General Procedure to Conduct Finite Element Analysis

To conduct finite element analysis, you need to follow certain steps that are given next.

1. Set the type of analysis to be used.
2. Create the model.
3. Define the element type.
4. Divide the given geometry into nodes and elements (mesh the model).
5. Apply material properties and boundary conditions.
6. Derive element matrices and equations.
7. Assemble element equations.
8. Solve the unknown parameters at nodes.
9. Interpret the results.

The general process of FEA by using software is divided into three main phases: preprocessing, solution, and postprocessing, refer to Figure 1-7.

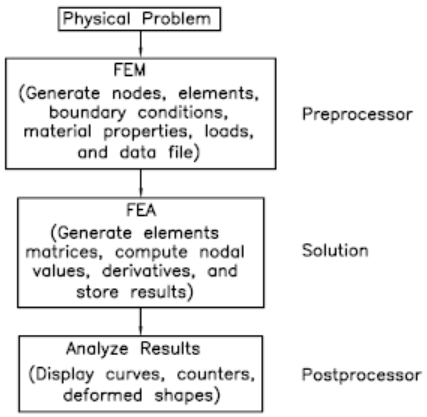

Figure 1-7 *FEA through software*

Preprocessor

In the preprocessor phase, the input data is processed to produce output which is further used as input in the subsequent phase (solution). The input data required for the preprocessor phase is given next:

1. Type of analysis (structural or thermal, static or dynamic, and linear or nonlinear)
2. Element type
3. Real constants for elements (Cross-sectional area, Moment of Inertia, Shell thickness, and so on)
4. Material properties (Young's Modulus, Poisson's ratio, Spring Constant, Thermal Conductivity, Coefficient of Thermal Expansion, and so on)
5. Geometric model (either created in the FEA software or imported from other CAD packages)
6. FEA model (discretizing the geometric model into small elements)
7. Loading and boundary conditions (defining loads, pressures, moments, temperature, conductivity, convection, constraints such as fixed, pinned, or frictionless/symmetrical), and so on

The input data are preprocessed for the output data and the preprocessor generates the data files automatically with the help of users. These data files are used in the subsequent phase (solution), refer to Figure 1-7.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Solution

The solution phase is completely automatic. The FEA software generates element matrices, computes nodal values and derivatives, and stores the result data in files. These files are further used in the subsequent phase (postprocessor) to review and analyze the results through the graphic display and tabular listings, refer to Figure 1-7.

Postprocessor

The output from the solution phase (result data files) is in numerical form and consists of nodal values of the field variable and its derivatives. For example, in structural analysis, the output of the postprocessor is nodal displacement and stress in elements. The postprocessor processes the result data and displays them in graphical form to check or analyze the result. The graphical output gives the detailed information about the required result data. The postprocessor phase is automatic and generates graphical output in the specified form, refer to Figure 1-7.

FEA SOFTWARE

A variety of commercial FEA software packages are available in the market. Every CAE software provides various modules for various analysis requirements. Depending on your requirement, you can select a required module for your analysis. Some firms use one or more CAE software and others develop customized version of commercial software to meet their requirements.

Advantages and Limitations of FEA Software

Following are some of the advantages and limitations of FEA software:

Advantages

1. It reduces the amount of prototype testing, thereby saving the cost and time.
2. It gives graphical representation of the result of analysis.
3. The finite element modeling and analysis are performed in the preprocessor and solution phases, which if done manually would consume a lot of time and in some cases, might be impossible to perform.
4. Variables such as stress and temperature can be measured at any desired point of the model.
5. It helps optimize a design.
6. It is used to simulate the designs that are not suitable for prototype testing.
7. It helps you create more reliable, high quality, and competitive designs.

Limitations

1. It does not provide exact solution.
2. FEA packages are costly.
3. An inexperienced user can deliver incorrect answers upon which expensive decisions will depend.
4. Results give solutions but not remedies.
5. Features such as bolts, welded joints, and so on cannot be accommodated in a model. This may lead to approximation and errors in the result.
6. For more accurate results, more hard disk space, RAM, and time are required.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

KEY ASSUMPTIONS IN FEA

There are four types of key assumptions that must be considered while performing the finite element analysis. These assumptions are not comprehensive but cover a wide variety of situations applicable to the problem. Moreover, by no means do all the following assumptions apply to all situations. Therefore, you need to consider only those assumptions that are applicable for analysis problem.

Assumptions Related to Geometry

1. Displacement values will be small so that a linear solution is valid.
2. Stress behavior outside the area of interest is not important. Therefore, geometric simplifications in those areas do not affect the outcome.
3. Only internal fillets in the area of interest will be included in the solution.
4. Local behavior at the corners, joints, and intersection of geometries is of primary interest, therefore, no special modeling of these areas is required.
5. Decorative external features will be assumed insignificant for the stiffness and performance of the part and these external features will be omitted from the model.
6. Variation in the mass due to suppressed features is negligible.

Assumptions Related to Material Properties

1. Material properties will remain in the linear region and the nonlinear behavior of the material property cannot be accepted.
2. Material properties are not affected by the load rate.
3. The component is free from surface imperfections that can produce stress concentration.
4. All simulations will assume room temperature unless otherwise specified.
5. The effect of relative humidity or water absorption on the material used will be neglected.
6. No compensation will be made to account for the effect of chemicals, corrosives, wears, or other factors that may have an impact on the long term structural integrity.

Assumptions Related to Boundary Conditions

1. Displacements will be small so that the magnitude, orientation, and distribution of the load remains constant throughout the process of deformation.
2. Frictional loss in the system is considered to be negligible.
3. All interfacing components will be assumed rigid.
4. The portion of the structure being studied is assumed as separate from the rest of the system, so that any reaction or input from the adjacent features is neglected.

Assumptions Related to Fasteners

1. Residual stresses due to fabrication, pre loading on bolts, welding, or other manufacturing or assembly processes will be neglected.
2. All welds between components will be considered as ideal and continuous.
3. The failure of fasteners will not be considered.
4. The load on the threaded portion of the part is supposed to be evenly distributed among the engaged threads.
5. The stiffness of bearings, both in radial and in axial directions, will be considered as infinite or rigid.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

IMPORTANT TERMS AND DEFINITIONS

There are some important terms and definitions used in FEA software that are discussed next.

Strength

When a material is subjected to an external load, the system undergoes deformation. The material, in turn, offers resistance against this deformation. This resistance is offered by the material by virtue of its strength.

Load

The external force acting on a body is called load.

Stress

The force of resistance offered by a body per unit area against the deformation is called stress. The stress is induced in the body while the load is being applied on the body. The stress is calculated as load per unit area.

$$p = F/A$$

Where,

p = Stress in N/mm^2
F = Applied Force in Newton
A = Cross-Sectional Area in mm^2

The material can undergo various types of stresses which are discussed next.

Tensile Stress

If the resistance offered by a body is against the increase in size, the body is said to be under tensile stress.

Compressive Stress

If the resistance offered by a body is against the decrease in size, the body is said to be under compressive stress. Compressive stress is just the reverse of tensile stress.

Shear Stress

The shear stress exists when two layers of materials tend to slide across each other in any typical plane of shear on the application of force parallel to that plane. In other words, shear stress is generated in the body when force is applied parallel to the cross-section of the body.

Shear Stress = Shear resistance (R) / Shear area (A)

Strain

When a body is subjected to a load (force), its length undergoes change. The ratio of change in length to the original length of the member is called strain. If the body returns to its original shape on removing the load, the strain is called elastic strain. If the body remains distorted after removing the load, the strain is called plastic strain. The strain can be of three types, tensile, compressive, and shear strain.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Strain (e) = Change in Length (*dl*) / Original Length (*l*)

Strain is further categorized as Longitudinal strain and Lateral strain. The ratio of deformation in the body to the original length in the direction of the applied load is known as longitudinal strain or linear strain.

The strain at right angle to the direction of applied load is known as lateral strain or transverse strain.

Elastic Limit

The maximum stress that can be applied to a material without causing permanent deformation is known as the elastic limit of the material. If the stress is within the elastic limit, the material returns to its original shape and size on removing the external load.

Hooke's Law

This law states that the stress is directly proportional to the strain within the elastic limit.

Stress / Strain = Constant (within the elastic limit)

This constant is known as Young's modulus or Modulus of Elasticity, and denoted by E.

Hence, **E = p/e**

Shear Modulus or Modulus of Rigidity

In case of shear loading, the ratio of shear stress to the corresponding shear strain is constant. This ratio is called shear modulus, and it is denoted by G.

Ultimate Strength

The maximum stress that a material withstands when subjected to an applied load is called its ultimate strength.

Yield Strength

The maximum stress that can be developed in a material without causing plastic deformation is called its yield strength.

Factor of Safety

The ratio of the absolute strength to the working stress (design stress) is known as factor of safety. It is necessary that the value of factor of safety should be equal or greater than unity for the design to be safe.

Factor of Safety (FOS) = Absolute strength/Working stress

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Poisson's Ratio

The ratio of the lateral strain to the longitudinal strain is constant within the elastic limit. This ratio is called the Poisson's ratio and is denoted by μ. The value of 'μ' lies between 0.0 to 0.5.

Poisson's ratio (μ) = Lateral Strain / Longitudinal Strain

Bulk Modulus

If a body is subjected to equal stresses along three mutually perpendicular directions, the ratio of the direct stresses to the corresponding volumetric strain is found constant for a given material, when the deformation is within a certain limit. This ratio is called the Bulk Modulus and is denoted by K.

Stress Concentration

The value of stress changes abruptly in the regions where the cross-section or profile of a structural member changes abruptly. The phenomenon of this abrupt change in stress is known as stress concentration and the region of the structural member affected by stress concentration is known as the region of stress concentration. The region of stress concentration needs to be meshed densely to get accurate results.

Bending

When a force is applied perpendicular to the longitudinal axis of a body, the body starts deforming. This phenomenon is known as bending. In case of bending, strains vary linearly from the center line of beam to the circumference. In case of pure bending, the value of strain is zero at the center line.

Bending Stress

When a non-axial force is applied on a structural member, some compressive and tensile stresses are developed in the member. These stresses are known as bending stresses.

Creep

At elevated temperature and constant load, many materials continue to deform but at a slow rate. This behavior of materials is called creep. At constant stress and temperature, the rate of creep is approximately constant for a long period of time. After a certain amount of deformation, the rate of creep increases, thereby causing fracture in the material. The rate of creep is highly dependent on both stress and temperature.

Degrees of Freedom (DOF)

The degrees of freedom is defined as the freedom allowed to a given object to move and rotate in any direction in space.

There are six DOFs for any object in 3-dimensional (3D) space: 3 translational DOFs (one each in the X,Y and Z directions) and 3 rotational DOFs (one rotation about each of the X,Y, and Z axes).

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

INTRODUCTION TO SOLIDWORKS SIMULATION

SOLIDWORKS Simulation is a Computer Aided Engineering(CAE) tool which works on Finite Element Analysis (FEA) technology. SOLIDWORKS Simulation was originally developed by Structural Research and Analysis Corporation (SRAC), USA and its first version came in 1995. SRAC joined with SOLIDWORKS and created COSMOSWorks. SRAC was acquired by Dassault Systemes in 2001. In 2008, COSMOSWorks was renamed to SOLIDWORKS Simulation.

SOLIDWORKS Simulation is fully integrated with SOLIDWORKS and supports multi-CAD environment products by providing direct file exchange with other CAD applications such as Creo Parametric, CATIA V5, NX (Unigraphic), Solid Edge, Autodesk Inventor, and so on. It makes iterative design change without redefining material, loads, constraints, or other simulation data when working with the native CAD format. You can also import geometry of universal file formats such as ACIS, IGES, STEP, and STL for solid models and CDL, DXF, and IGES for wireframe models.

Now, SOLIDWORKS Simulation is available in different packages and solvers which are discussed next.

SOLIDWORKS Simulation Standard

SOLIDWORKS Simulation Standard can perform static and stress analysis on assembly and parts only. It is limited to basic contacts and connections.

SOLIDWORKS Simulation Professional

SOLIDWORKS Simulation Professional covers the analysis portion discussed in SOLIDWORKS Simulation Standard. And apart from this, advanced contacts and connections can also be applied during the analysis. The frequency, buckling, thermal, drop test, fatigue, and pressure vessel design analysis can also be performed with professional suite.

SOLIDWORKS Simulation Premium

SOLIDWORKS Simulation Premium covers all analysis tools available in SOLIDWORKS Simulation Standard and Professional suite as well as the tools used for non linear and dynamic simulation.



Note

1. Note that all these three packages of SOLIDWORKS Simulation do not run as stand-alone software. They are integrated with SOLIDWORKS to run on the PC machine.

2. SOLIDWORKS package comes with a simulation tool known as SimulationXpress. But, SimulationXpress is limited to part analysis and simple load and connections can be applied.

Analysis Solvers

In SOLIDWORKS Simulation, two direct solvers and one iterative solver are available for the solution of the set of equations. In finite element analysis, a problem is represented by a set of algebraic equations that must be solved to find out solutions. There are two classes of solution methods; direct and iterative. Direct method solves the equations using exact numerical techniques. Iterative method solves the equations using approximate techniques where in each

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

iteration, a solution is assumed and the associated errors are evaluated. The iterations continue until the errors become acceptable.

Automatic
This solver option is the default option for static, frequency, buckling, and thermal studies.

Direct Sparse
In the case of multi-area contact problems where the area of contact is found through several contact iterations, the Direct Sparse solver is preferred.

FFEPlus
In general, FFEPlus is faster in solving problems with degrees of freedom (DOF) over 100,000. It becomes more efficient as the problem gets larger.

Large Problem Direct Sparse
The Direct Sparse solver requires about 10 times more RAM and is faster than the FFEPlus solver. It leverages multicore processing capability and improves solution speed for static and nonlinear studies. Note that it is available for static and nonlinear studies.

Types of Analyses Performed Using SOLIDWORKS Simulation
You can perform different types of analysis using SOLIDWORKS Simulation. Few of them are discussed next.

Static Analysis
In static analysis, the load or field conditions do not vary with respect to time, and therefore, it is assumed that the load or field conditions are applied gradually. When loads are applied to a body, the body deforms and the effect of loads is transmitted throughout the body. The external loads induce internal forces and reactions to make the body into a state of equilibrium. Linear Static Analysis is used to calculate the displacements, strains, stresses, and reaction forces under the effect of applied loads in equilibrium state. It helps to avoid failure due to high stress.

Dynamic Analysis
This type of study assumes that the materials are linear and the loads are either time dependent, frequency dependent or defined by limiting spectra. Also, the mass and inertia effects are included and damping is available. The options in SOLIDWORKS Simulation are Drop Test (also known as Direct Time History Analysis), Modal Time History Analysis (Mode Superposition Analysis), Harmonic Analysis (Harmonic Response Analysis), and Random Vibration Analysis (Response Spectra Analysis). The last three analysis types require a Frequency Analysis to be completed to supply the eigenvalues (natural frequencies) and eigenvectors (mode shapes) needed as inputs.

Thermal Analysis
Thermal study calculates temperatures, temperature gradients, and heat flow based on heat generation, conduction, convection, and radiation conditions. Thermal analysis can help you avoid undesirable thermal conditions like overheating and melting.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Frequency Analysis

When a structure is put under a dynamic load with a frequency that coincides with one of its natural frequencies, the structure undergoes large displacements and stresses. This phenomenon is known as resonance. For undamped systems, resonance theoretically causes infinite motion. Damping, however, puts a limit on the response of the structures due to resonant loads and calculates natural frequencies and associated mode shapes.

Buckling Analysis

Buckling refers to sudden large displacements due to axial loads. Slender structures subject to axial loads can fail due to buckling at load levels lower than those required to cause material failure. Buckling can occur in different modes under the effect of different load levels. Buckling studies can help to avoid failure due to buckling. Usually the lowest buckling load is the subject of interest and is lower than those required to cause material failure.

Fatigue

Fatigue study evaluates the consumed life of an object based on a very large number of fatigue events (cycles). Repeated loading weakens materials over a period of time even when the induced stresses are low. The number of cycles that can cause failure depends on the material and the stress fluctuations. The data is provided by the material S-N curve which depicts the number of cycles that cause failure due to different stress levels. Fatigue study determines the consumed life of an object, in relation to fatigue events, based on fatigue calculations of stress intensity, von-Mises stress or Maximum principal stress.

Pressure Vessel Design

Pressure vessel design study combines the results of multiple static studies with the desired load factors. This study combines the results algebraically using a linear combination or the square root of the sum of the squares. Engineering plastics have been used for pressure vessel applications for a long time. Lighter bodies, ballcock valves, and spray paint containers are few examples of successful developments in this area.

Drop Test

Drop test study evaluates the impact of dropping body on a rigid floor. You can specify the dropping distance or the velocity at the time of analysis in addition to gravity. In this test no other loads or restraints are allowed.

Nonlinear Study

Nonlinear study enables to quickly and efficiently analyze stresses and deformations under general conditions to ensure high quality, performance, and safety. Nonlinear stress analysis calculates the stresses and deformations under the most general loading and material conditions such as Dynamic (time dependent) loads, Large component deformation, and Nonlinear materials.

Topology Study

In SOLIDWORKS Simulation, topology study allow you to explore design iterations of a component that satisfy a given optimization goal and geometric constraints. A Topology study performs nonparametric topology optimization of parts. It starts with maximum design space which represents the maximum allowed size for a component with all applied loads, fixtures,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

and manufacturing constraints. The topology optimization seeks a new material layout within the boundaries of the maximum allowed geometry by redistributing the material. The optimized component satisfies all the required functional and manufacturing requirements.

Starting SOLIDWORKS Simulation

To start SOLIDWORKS Simulation, activate SOLIDWORKS Simulation by clicking on the downward arrow beside the **Options** button in the menu bar; a flyout will be displayed, as shown in Figure 1-8. Next, choose the **Add-Ins** options from the flyout; the **Add-Ins** dialog box will be displayed, as shown in Figure 1-9. You can also invoke this dialog box by choosing the **Add-Ins** option from the **Tools** menu. In this dialog box. if you select the check box located on the right of the add-ins name, then SOLIDWORK Simulation will be activated whenever SOLIDWORKS is started. If you prefer to activate the SOLIDWORKS Simulation program only when it is needed for an analysis, select the check box available on the left of **SOLIDWORKS Simulation** in the **Add-Ins** dialog box and choose the **OK** button.

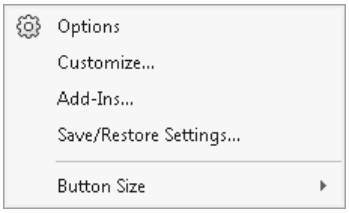

Figure 1-8 *Flyout displayed on clicking the* ***Options*** *button arrow*

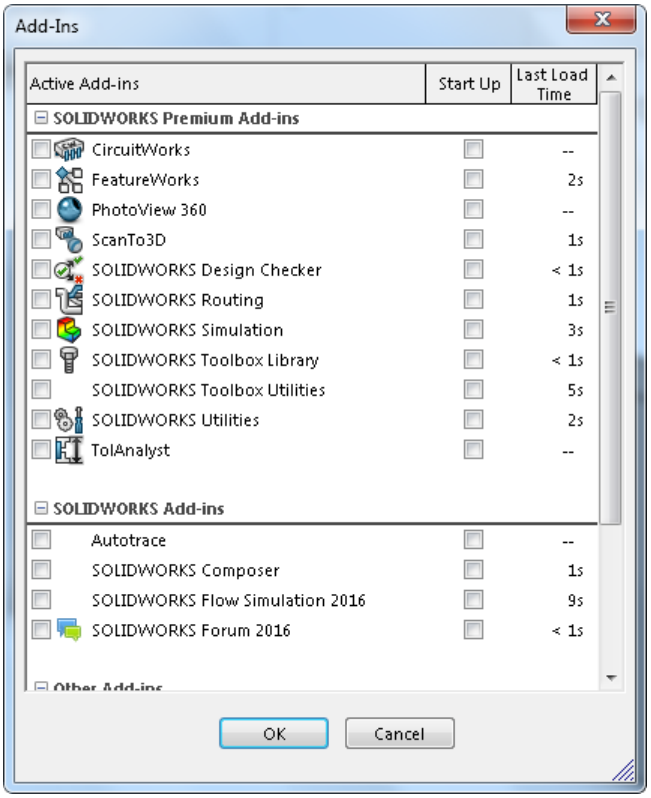

Figure 1-9 *The* ***Add-Ins*** *dialog box*

Next, start a new part document in SOLIDWORKS by using the **New SOLIDWORKS Document**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

dialog box; the **Simulation** menu is added in the SOLIDWORKS menus and the **Simulation** tab is added to the CommandManager, as shown in Figure 1-10.

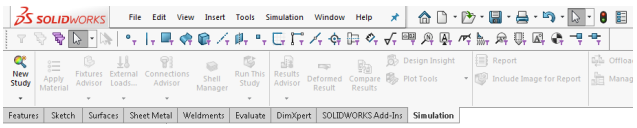

Figure 1-10 *Simulation CommandManager*

To start a new study, click on the downward arrow below the **New Study** button; a drop-down menu will be displayed. Now, choose the **New Study** option from this drop-down menu; the **Study PropertyManager** will be displayed, as shown in Figure 1-11. Next, select the type of study which you want to perform and then select the **OK** button from the **Study PropertyManager**; the Simulation Tree will be displayed, as shown in Figure 1-12.

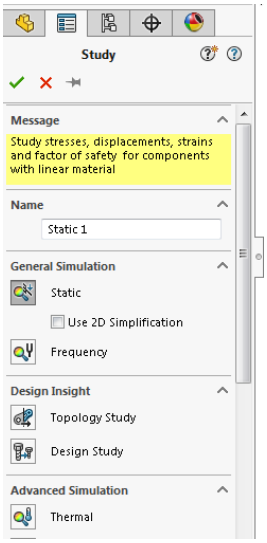

Figure 1-11 *The* ***Study PropertyMananger***

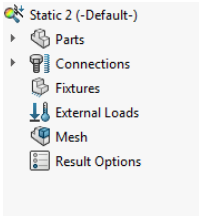

Figure 1-12 *Simulation Tree*

BASIC STEPS REQUIRED TO PERFORM SIMULATION IN SOLIDWORKS SIMULATION

You need to perform the following basic steps to complete the finite element analysis in SOLIDWORKS Simulation. The steps are discussed next.

Prepare the Model

Finite element analysis (FEA) requires a well described model for study, in order to perform correct analysis which in return, should offer plausible results. In FEA, different assumptions and simplifications are made such as 3D geometrical model and proposed type of study. Simplifications and repairing of design are required so that it take less computing time during the analysis. By combining the right assumptions with a good robust model, a successful technical analysis can be achieved.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Assign Material Properties

When you are modelling a component in SOLIDWORKS, you can assign material to the model. You can also assign material to the model imported in simulation environment. To apply the material in simulation environment, right-click on the name of the model in the Simulation Tree; a shortcut menu will be displayed, as shown in Figure 1-13. Choose the material from the **Apply Favorite Material** list or choose the **Apply/Edit Material** option; the **Material** dialog box will be displayed, as shown in Figure 1-14. Select the required material (Alloy Steel); the properties of the selected material will be displayed, as shown in Figure 1-14. You will notice that the material properties will be displayed in red, blue, and black colors. These colors describe the mandatory and optional properties for the selected study. The description in red color (Elastic Modulus, Poisson's Ratio, Mass Density, Yield Strength) indicates the property is mandatory for the active study type and the material of the model. A blue description (Tensile Strength, Compressive Strength, Thermal Expansion Coefficient) indicates an optional property. A black description (Shear Modulus, Thermal Conductivity, Specific Heat, Material Damping Ratio) indicates properties are not applicable to the current study.

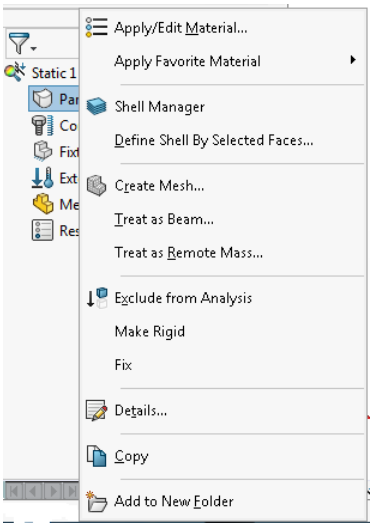

Figure 1-13 *The shortcut menu displayed*

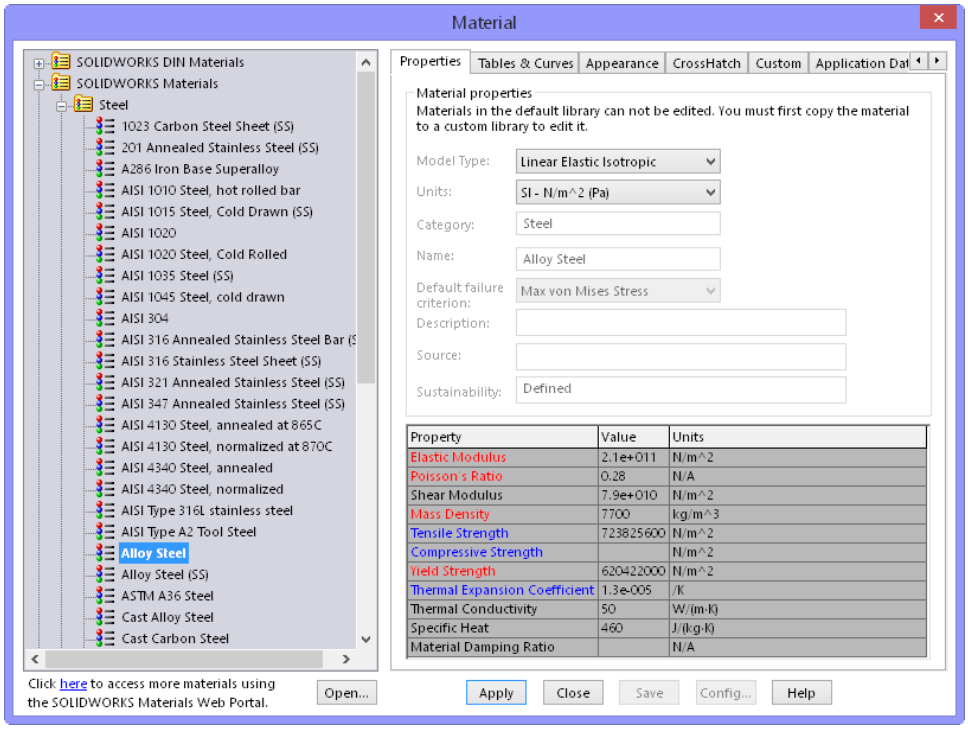

Figure 1-14 *The* ***Material*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Note

1. If you are working with an assembly or with composite material, you need to expand the part node and then apply material by right-clicking on the part.

2. Material applied in SOLIDWORKS Simulation will not affect the material applied in CAD model in SOLIDWORKS.

Apply Loads and Restraints

Loads and restraints are applied to define actual environmental conditions. The applied loads and restraints are represented by an icon in Simulation Tree. SOLIDWORKS Simulation provides context-sensitive options for defining restraints. For example, if reference axis is selected or the selected faces are cylindrical, the program expects you to define radial, circumferential, and axial restraints.

If you make geometry changes such that the entity is no longer defined, then after applying a restraint or load to the entity, the restraint becomes dangled. SOLIDWORKS Simulation gives you a message that lists dangled restraints or load for each study you defined.

Meshing

FEA provides a reliable numerical technique for analyzing engineering designs. The process starts with the creation of a geometric model. Then, the program subdivides the model into small pieces of simple shapes called elements, connected at common points called nodes. The process of subdividing a model into small pieces is called meshing.

Meshing is a crucial step in design analysis. SOLIDWORKS Simulation allow you to create a mesh of solid elements (3D tetrahedral), shells (2D triangular), or beam (1D linear). The solid mesh is appropriate for bulky or complex 3D models. Shell elements are suitable for thin parts (such as sheet metals). The accuracy of the results depends on the quality of the mesh. In general, accurate and finer the mesh, better the accuracy. The resultant mesh depends on the following factors:

- Type of mesh (solid, shell using midsurfaces, or shell using surfaces).
- Active mesh preferences.
- Mesh control.
- Contact conditions for static and thermal assembly problems.
- Global element size and mesh tolerance.

Run Studies

After assigning materials, defining loads and restraints, and meshing the model, you can run the study.

SOLIDWORKS Simulation has four options for solvers: Auto, FFEPlus, Direct Sparse, and Large Problem Direct Sparse. By default, SimulationXpress and SOLIDWORKS Simulation use FEEPlus solver. Solver is selected when you define the properties of a study. In some cases, the program switches to another solver automatically if the selected solver does not support all options used in a study. All solvers provide the same results provided that the same mesh is used. However, the performance and speed vary depending on the type and size of the problem.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

View Results

After the completion of study, you can automatically generate standard plots for each type of analyses. The standard plots for an analysis type are the most commonly used results. For example, after running a static study, SOLIDWORKS Simulation creates result folders containing default plots for stress, displacement, and strain. You can view a plot by double-clicking on its node in the Simulation Tree.

You can also view the other plots by right-clicking on the **Results** node and then selecting the required option from the shortcut menu displayed. You can also generate reports that include all available plots automatically. To do so, choose the **Report** tool in **Simulation Command Manager**; the **Report Options** dialog box will be displayed. Fill the required input and choose the **Publish** button; the report for the study will be generated in Word Document.



Note

It is not compulsory that the above sequence should be maintained to define the parameters. You can define these parameter in any order but make sure that you have applied all necessary parameters before you run the study.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Which of the following options represents the shape of a face of first order tetrahedral finite element?

   (a) Rectangle (b) Triangle
   (c) Pentagon (d) None of these

2. What is the maximum number of nodes that can be available in a hexahedral element?

   (a) 6 (b) 4
   (c) 8 (d) None of these

3. Stress induced in a model due to an external load, gets changed due to changing material of the model. (T/F)

4. Stiffness constant of a model does not depend upon the material of the model. (T/F)

5. Stress is defined as the resistance offered by a body against deformation. (T/F)

6. A design is assumed to be safe if the factor of safety value for that model lies between 0 to 1. (T/F)

7. A model can have infinite degrees of freedom. (T/F)

8. Material applied to a model in part environment of SOLIDWORKS can be used for SOLIDWORKS Simulation. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Review Questions

Answer the following questions:

1. How many faces does a tetrahedral element have?

   (a) 6 (b) 4
   (c) 3 (d) 8

2. What is the minimum number of nodes required to define a line element?

   (a) 2 (b) 4
   (c) 3 (d) None of these

3. What is the maximum number of degrees of freedom that a tetrahedral element can have?

   (a) 6 (b) 4
   (c) 3 (d) 8

4. Factor of safety of a model depends upon the material of the model.(T/F)

5. Factor of safety for mild steel is defined as the ratio of ultimate strength to working stress. (T/F)

6. An area element can have a maximum of six nodes. (T/F)

7. Yield point can not be defined for brittle materials. (T/F)

8. In Finite element Analysis you can assign materials, apply fixtures and loads and run the study to visualize the results. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Answers to Self-Evaluation Test

1. a, **2.** d, **3.** F, **4.** F, **5.** F, **6.** F, **7.** F, **8.** T