

Introduction to FEA and SOLIDWORKS Simulation

In this chapter, the following topics will be discussed:

- Introduction to SOLIDWORKS Simulation
 - Introduction to Finite Element Analysis (FEA)
 - Working with Different Phases of FEA
 - Important Terms and Definitions used in FEA
 - Different Application Areas of FEA
 - Installing SOLIDWORKS Simulation
 - Getting Started with SOLIDWORKS Simulation
 - Identifying SOLIDWORKS Documents
 - Adding CommandManager Tabs
 - Invoking SOLIDWORKS Simulation
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Welcome to the world of Computer Aided Engineering (CAE) with SOLIDWORKS Simulation. SOLIDWORKS Simulation is a product of Dassault Systemes SOLIDWORKS Corp., one of the biggest technology providers for engineering software solutions that lets you create, simulate, publish, and manage data. By providing advanced analysis techniques, SOLIDWORKS Simulation helps engineers to optimize performance of products and allows them to cut prototyping cost, create better and safer products, and save time as well as development costs.

Introduction to SOLIDWORKS Simulation

SOLIDWORKS Simulation is a Finite Element Analysis tool which enables critical engineering decisions to be made early on in the design process. With this software, engineers have the tools to easily study the initial design and predict the performance of the complete digital prototype. The automatic meshing tools of this software generate mesh with high-quality elements on the first pass. It also enables engineers to directly edit the mesh for accurate placement of loads and constraints, and simplifies geometry by using its powerful modeling capabilities. SOLIDWORKS Simulation makes it possible to quickly validate the design concepts before resources are invested in significant design changes or new products.

SOLIDWORKS Simulation provides a wide range of linear and non-linear materials that allow a better understanding of the real-world behavior of products and provides engineers the knowledge of how a product will perform in the real-world environment.

SOLIDWORKS Simulation is fully integrated with SOLIDWORKS and supports efficient workflow in today's multi-CAD environment by providing direct geometry exchange with other CAD applications such as Creo Parametric, CATIA V5, NX (Unigraphics), 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 as well as the CDL, DXF™, and IGES for wireframe models.

SOLIDWORKS Simulation provides a broad range of simulation tools to perform various types of analysis, which helps engineers to bring product performance knowledge into the early stages of the design cycle.

The various types of analysis that can be carried out by using SOLIDWORKS Simulation are discussed below.

Linear Static Analysis

Linear static analysis is used for calculating displacement, strain, stress, and reaction forces of an object under the impact of applied load. In the linear static analysis, the material properties of the object are assumed to behave linearly under the impact of applied load and the object returns to its original configuration once the load has been removed. Also, the load is assumed to be constant and does not vary with respect to time. Besides, in this analysis, the displacement is assumed to be smaller due to the applied load.

Frequency Analysis

Frequency analysis is used for calculating the natural or resonant frequencies and the associated mode shapes of a structure. Natural or resonant frequency is the frequency of an object at which it vibrates when disturbed from its rest position. By knowing the natural frequencies of an object, you can ensure that the actual operating frequency of the object will not coincide with any of its natural frequencies to avoid the failure of the object due to resonance.

Buckling Analysis

Buckling analysis is used for calculating the buckling load, which is also known as the critical load, when the model can start buckling, even if the maximum stress developed in the model is within the yield strength of the material. Buckling refers to a larger deformation occurring due to axial loads acting on the structures such as long slender columns and thin sheet components.

Thermal Analysis

Thermal analysis is used for calculating temperature distribution in an object as a result of conduction, convection, and radiation. It helps to avoid over-heating and melting conditions. In addition to calculating temperature distribution, this analysis also determines the related thermal quantities such as thermal distribution, amount of heat loss and gain, thermal gradients, and thermal fluxes.

Drop Test Analysis

Drop test analysis is used for calculating the response of an object when it is dropped on a rigid or flexible floor.

Fatigue Analysis

Fatigue analysis is used for calculating the stress at which an object fails, when it undergoes repeated loading and unloading which weakens the object after a period of time and causes failure of the object even at a stress value within the allowable stress limits. You can also predict the total life and total damage of the object due to repeated loading and unloading.

Non-linear Analysis

Non-linear analysis is used for calculating displacement, strain, stress, and reaction forces of the non-linear mechanical problems which includes large deformation, plasticity, creep and so on. In this analysis, the material properties of the object are assumed to exceed its elastic region under the impact of applied load and experience plastic deformation. This means that the object will not return to its original configuration even after removing the applied load. Non-linear analysis can undergo static analysis (applied load or field conditions do not vary with respect to time) and dynamic analysis (applied load or field conditions vary with respect to time).

Linear Dynamic Analysis

Linear dynamic analysis is used for calculating the response of an object to a dynamic loading environment. In this analysis, the load or boundary conditions vary with time due to the sudden loading. Also, the material of the object is assumed to behave linearly under the impact of the applied load and that it will return to its original shape once the load has been removed. This analysis includes oscillating loads, impacts, collisions, and random loads. The linear dynamic analysis is classified into four main categories which are discussed below.

Modal time history

The modal time history analysis is used for analyzing the response to load as the function of time.

Harmonic

The harmonic analysis is used for analyzing the response of an object to harmonically time varying loads.

Random vibration

The random vibration analysis is used for calculating maximum stresses due to the vibration which occurs in response to the non-deterministic loads, which include loads generated on the wheel of a car traveling on a rough road, base accelerations generated by earthquakes, pressure generated by air turbulence, and other similar types of load. In the random vibration analysis, the input is provided to the system in the form of 'Power Spectral Density (PSD)', which is represented as vibration frequencies.

Response Spectrum

The Response spectrum analysis is used for calculating the response of a structure which undergoes sudden forces or shocks due to earthquakes, wind loads, ocean wave loads and so on. It is also assumed that the shocks or forces occur at an area which is fixed.

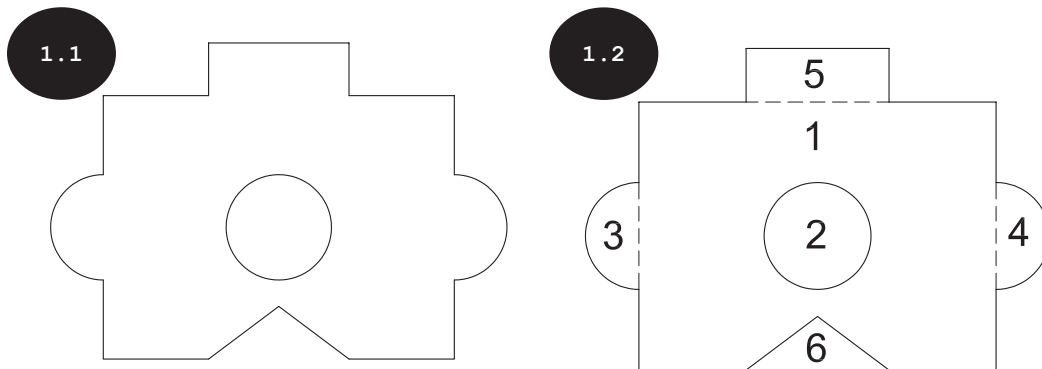
Pressure Vessel Design Analysis

The pressure vessel design analysis is used for analyzing pressure vessels. In this analysis, you can combine the results of static analysis with a different set of loads such as dead loads, live loads, thermal loads, seismic loads, and so on.

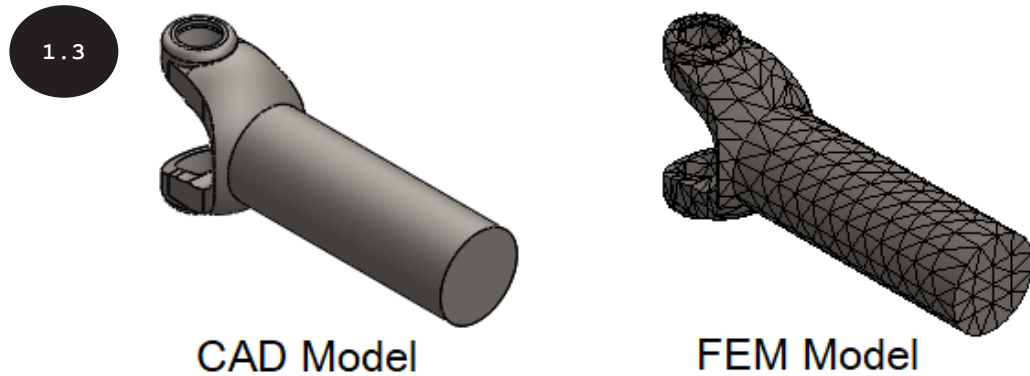
Introduction to Finite Element Analysis (FEA)

Finite element analysis (FEA) uses a numerical technique known as finite element method (FEM) to solve engineering problems. The finite element method (FEM) is the most widely used and accepted method to solve engineering problems involving stress analysis, deflections, reactions, vibrations, fluid flow, heat transfer, electrical, magnetic fields, and so on due its suitability, numerical efficiency, and generality for computer application.

The whole concept of FEM can be explained with a small example of measuring the area of an unknown geometry of a plate, see Figure 1.1. There are many ways to measure the area of an unknown geometry, but the best way is to divide the entire geometry into different known geometries whose area can be easily calculated, see Figure 1.2. After measuring the area of each individual known geometry, assemble them together to get the total area of the geometry.



The same concept is used in FEM to calculate stresses, displacements, strains, reaction forces, temperature, frequency, vibrations, and so on of a complex structure. FEM divides the entire complex structure into a finite number of pieces of simple geometric shapes called elements, see Figure 1.3. It replaces a complex engineering problem with multiple simple problems that can be easily solved.



It is clear from the above figure that a FEM model consists of a number of finite elements which collectively represent the entire structure. Note that the geometries of the real-world mechanical structures are complex and to accurately represent their shapes, more number of finite elements are required due to which the computational time to calculate the response of all elements gets increased. Therefore, in finite element analysis (FEA), there needs to be proper balance between the accuracy of results and the computational time. This makes the finite element analysis (FEA) a suitable method for finding approximate solutions to the engineering problems.

Working with Different Phases of FEA

Before you start performing an analysis, it is important to understand the different phases of finite element analysis (FEA). As discussed, SOLIDWORKS Simulation is a finite element analysis (FEA) tool which uses a numerical technique known as finite element method (FEM) to solve engineering problems. In finite element analysis (FEA), the entire process of analyzing the engineering design is divided into three phases: Pre-processing, Solution, and Post-processing. The phases are discussed below.

Pre-processing

The Pre-processing phase involves creating/importing CAD models, simplifying geometry, selecting analysis type, assigning material properties, defining boundary conditions (external loads and supports), and meshing the model, see Figure 1.4.

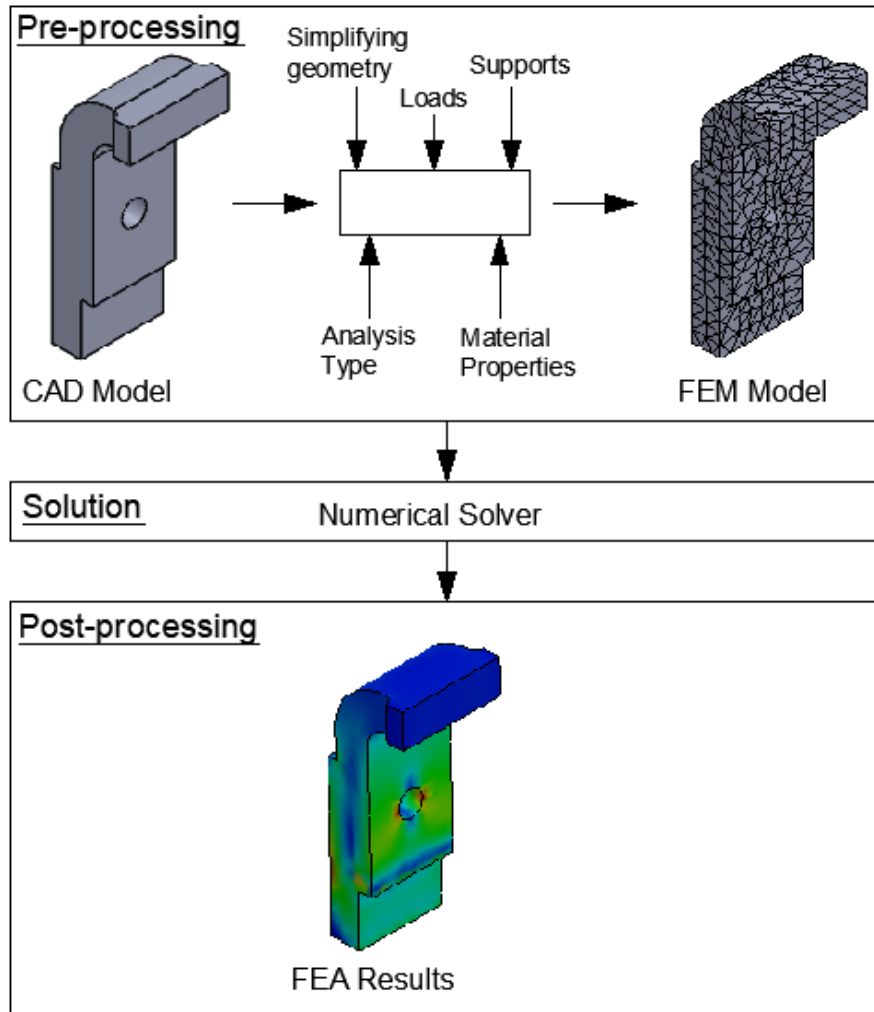
Solution

The Solution phase is completely automatic in SOLIDWORKS Simulation. In this phase, the system generates matrices for individual finite elements, which are then assembled to generate a global matrix equation for the structure. Further, it solves the global matrix equation to compute displacement, which is then used to compute strain, stress, and reaction forces. Note that in this phase the computed results are stored in numerical form.

Post-processing

In the Post-processing phase, the results generated in the Solution phase appear in graphical form for checking or analyzing them, see Figure 1.4. You can also animate the structure response based on the results obtained in the Solution phase. The graphical representation of results is very useful in understanding the correct behavior of the structure.

1.4



Important Terms and Definitions used in FEA

Some of the important terms and definitions used in finite element analysis (FEA) are discussed below.

Stress

Stress is defined as the force per unit area. When an object is subjected to an external force, the internal resistance offered by the object is known as stress.

$$\sigma = F/A$$

Where,

σ = Stress

F = External force acting on the object

A = Cross sectional area of the object

Stress has various forms, but mainly categorized into three types: Tensile Stress, Compressive Stress, and Shear Stress, all of which are discussed below.

Tensile Stress

When an object is subjected to tensile forces, the internal resistance applied by the object against the increase in its length is known as tensile stress, see Figure 1.5.

1.5

Tensile Force



Compressive Stress

When an object is subjected to compressive forces, the internal resistance applied by the object against the decrease in its length is known as compressive stress, see Figure 1.6.

1.6

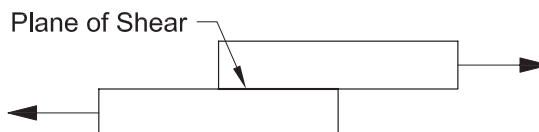
Compressive Force



Shear Stress

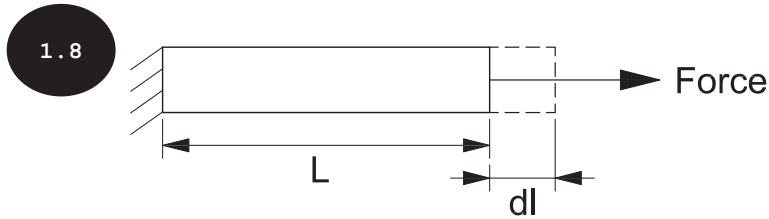
Shear stress occurs when two objects tend to slide over one another due to the application of external forces which are parallel to the plane of shear, see Figure 1.7.

1.7



Strain

Strain is defined as the ratio of change in length to the original length of the object when it undergoes deformation due to the application of an external force, see Figure 1.8.



$$\varepsilon = dl/L$$

Where,

ε = Strain

dl = Change in length of the object

L = Original length of the object

Load

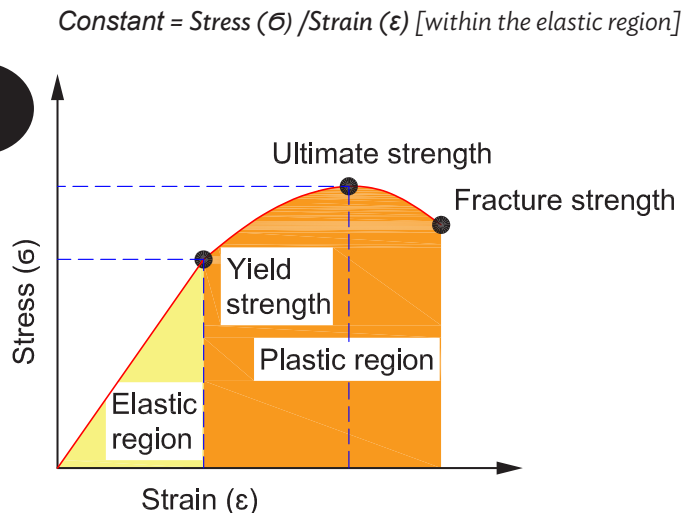
Load is defined as the external force acting on an object.

Displacement

Displacement is defined as the change in length or position of an object.

Hooke's Law

Hooke's Law defines that the ratio of stress to strain is constant. It states that stress is directly proportional to strain within the elastic region of the stress-strain curve of a material, when the material is subjected to an external load, see Figure 1.9.



Yield Strength

Yield strength is defined as the maximum stress (yield stress) up to which a material deforms elastically under the impact of an applied load and returns to its original configuration once the load is removed. It is also defined as the stress under which the material begins to deform plastically.

Ultimate Strength

Ultimate strength is defined as the maximum stress that a material can withstand when subjected to an external load. It is also defined as the stress beyond which the material begins to fail or weaken.

Fracture Strength

Fracture strength is defined as the breaking stress under which a material fails due to fracture or breakage.

Young's Modulus

Young's modulus is also known as the modulus of elasticity or the elastic modulus, and defines the relationship between stress and strain of a material as per Hooke's law. It measures the stiffness of a material.

$$E = \text{Stress } (\sigma) / \text{Strain } (\epsilon)$$

$$E = F \cdot L / A \cdot dl$$

$$F = E \cdot A \cdot dl / L$$

$$F = (E \cdot A / L) \cdot dl$$

$$F = K \cdot X$$

Where,

$$K = \text{Stiffness } (E \cdot A / L)$$

$$E = \text{Young's modulus}$$

$$A = \text{Cross section area}$$

$$L = \text{Original length}$$

$$X = \text{Change in length } (dl)$$

$$F = \text{Applied force}$$

Stiffness

Stiffness is defined as the property of a material which offers resistance against its deformation when it is subjected to an external force.

$$K = F / dl$$

Where,

$$K = \text{Stiffness}$$

$$F = \text{Applied external force}$$

$$dl = \text{Displacement (change in length)}$$

Poisson's Ratio

Poisson's ratio is defined as the ratio of lateral strain to the axial or longitudinal strain of a material in the direction of the applied load, see Figure 1.10. The Poisson's ratio of a material within the elastic limit is constant. This implies that the ratio of lateral strain to the axial or longitudinal strain of a material within the elastic limit is constant.

$$\mu = \epsilon_{\text{lateral}} / \epsilon_{\text{axial}}$$

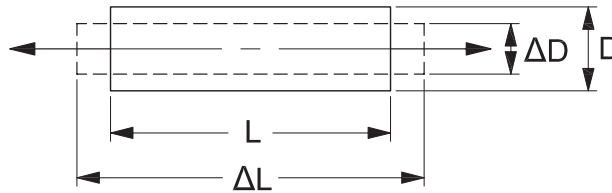
Where,

μ = Poisson's ratio

$\epsilon_{\text{lateral}}$ = Lateral strain

ϵ_{axial} = Axial or longitudinal strain

1.10



$$\text{Axial or Longitudinal strain } (\epsilon_{\text{axial}}) = \Delta L - L / L = dl_a / L$$

Where,

L = Initial/original length

ΔL = Final length

dl_a = Change in length in axial direction

$$\text{Lateral strain } (\epsilon_{\text{lateral}}) = \Delta D - D / D = dl_l / D$$

Where,

D = Initial/original width

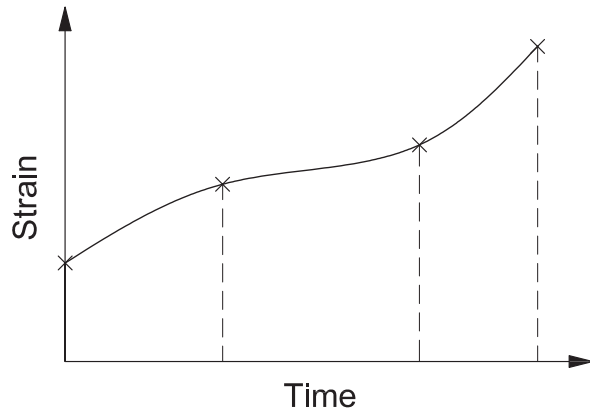
ΔD = Final width

dl_l = Change in width in lateral direction

Creep

Creep is defined as the tendency of a material to deform slowly or gradually. It increases with time under the impact of stress which is below the yield strength of the material, see Figure 1.11. It is a material property which depends on both stress and temperature.

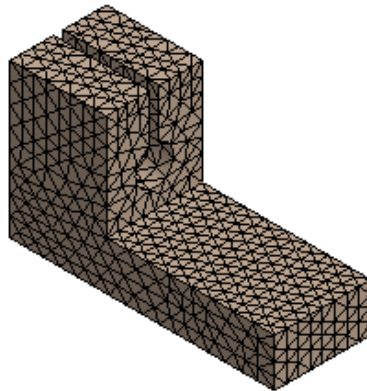
1.11



Meshing


Meshing is defined as the process of dividing an object into a finite number of pieces with simple geometric shapes called elements, see Figure 1.12.

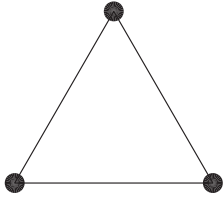
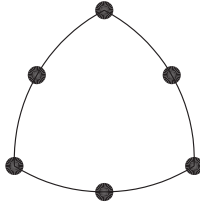
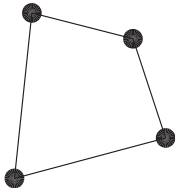
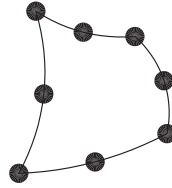
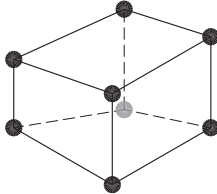
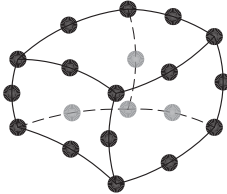
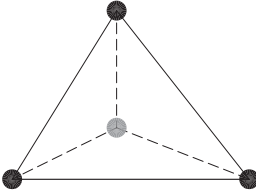
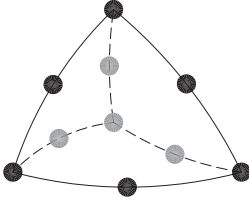
1.12



Elements

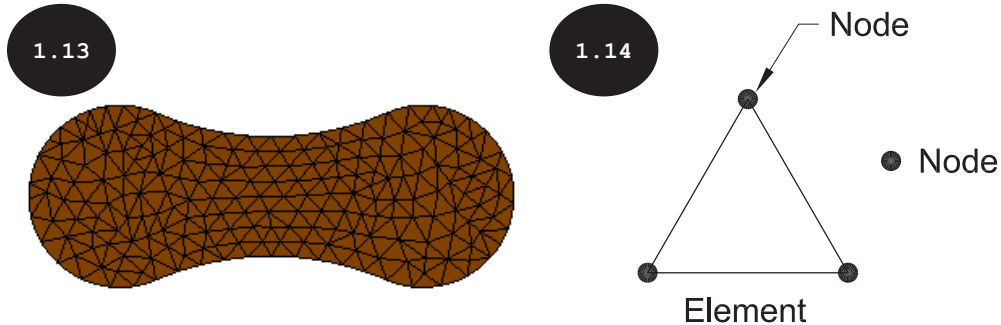
Elements are small pieces of simple geometric shapes into which an object is divided while meshing. Elements are mainly categorized into three types: 1D elements, 2D elements, and 3D elements, see the table given below:

Element Type	Element Shape
1D Element	

<p>2D Element</p>	<div style="display: flex; justify-content: space-around; align-items: center;"> <div style="text-align: center;">  <p>First Order Triangular element</p> </div> <div style="text-align: center;">  <p>Second Order Triangular element</p> </div> </div> <div style="display: flex; justify-content: space-around; align-items: center; margin-top: 20px;"> <div style="text-align: center;">  <p>First Order Quadrilateral element</p> </div> <div style="text-align: center;">  <p>Second Order Quadrilateral element</p> </div> </div>
<p>3D Element</p>	<div style="display: flex; justify-content: space-around; align-items: center;"> <div style="text-align: center;">  <p>First Order Hexahedral element</p> </div> <div style="text-align: center;">  <p>Second Order Hexahedral element</p> </div> </div> <div style="display: flex; justify-content: space-around; align-items: center; margin-top: 20px;"> <div style="text-align: center;">  <p>First Order Tetrahedral element</p> </div> <div style="text-align: center;">  <p>Second Order Tetrahedral element</p> </div> </div>

Nodes

Elements are connected to each other at common points called nodes, see Figures 1.13 and 1.14. The nodes define the shape of elements. If you move a node of an element, the shape of the element will change depending on the new position of the node.



Different Application Areas of FEA

The finite element analysis (FEA) was developed for the nuclear research, and aerospace industries. However, currently, it is a widely used and accepted method in all engineering disciplines (mechanical, civil, electrical, and automobile etc). The areas of applications of FEA include:

1. Structure analysis
2. Thermal analysis
3. Buckling analysis
4. Fluid flows analysis
5. Frequency analysis
6. Mould flow analysis
7. Drop test
8. Pressure vessel design
9. Fatigue analysis
10. Vibrations
11. Electromagnetic analysis
12. Biomechanics, and many more

Installing SOLIDWORKS Simulation

SOLIDWORKS Simulation is fully integrated with SOLIDWORKS, therefore to install SOLIDWORKS Simulation, you must have SOLIDWORKS installed on your system. If you do not have these softwares installed in your system, you first need to install them. However, before that, you need to evaluate the system requirements and ensure that you have a system capable of running SOLIDWORKS and SOLIDWORKS Simulation adequately. The system requirements for installing the same are listed below:

1. Operating Systems: Windows 10 - 64-bit
2. RAM: 8 GB or more (16 GB or more recommended)
3. Disk Space: 10 GB or more

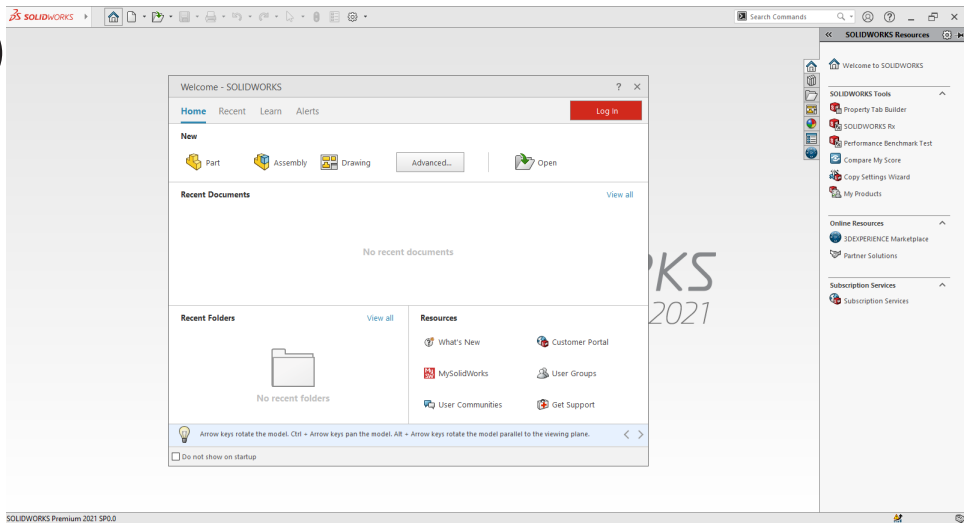
4. Processor: 3.3 GHz or higher
5. Graphics Card: SOLIDWORKS certified graphics card drivers

For more information about the system requirements for SOLIDWORKS, visit the SOLIDWORKS website at <https://www.solidworks.com/sw/support/SystemRequirements.html>

Getting Started with SOLIDWORKS Simulation


Once SOLIDWORKS 2021 and SOLIDWORKS Simulation 2021 are installed on your system, start by double-clicking on the SOLIDWORKS 2021 icon on your desktop. The system prepares for starting SOLIDWORKS and SOLIDWORKS Simulation by loading all required files. Once all the required files have been loaded, the initial screen of SOLIDWORKS 2021 appears along with a **Welcome** dialog box, see Figure 1.15. If you are starting SOLIDWORKS for the first time after installing the software, the **SOLIDWORKS License Agreement** window appears. Click on the **Accept** button in the **SOLIDWORKS License Agreement** window to accept the license agreement and start SOLIDWORKS.

1.15




In SOLIDWORKS 2021, the **Welcome** dialog box appears every time you start SOLIDWORKS and is a convenient way to invoke new SOLIDWORKS documents, open existing documents, view recent documents and folders, access SOLIDWORKS resources, and stay updated on SOLIDWORKS news.

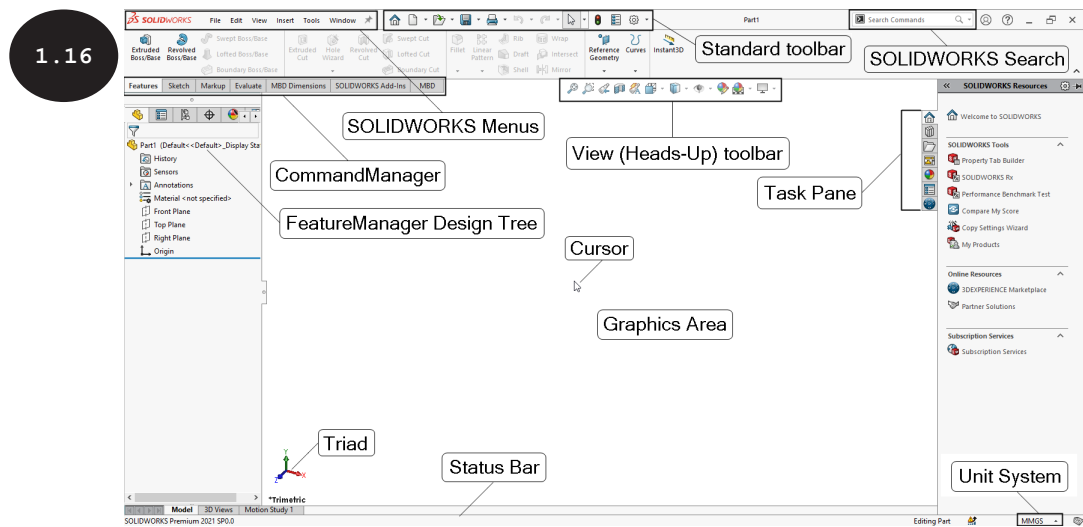
The **Welcome** dialog box has four tabs: **Home**, **Recent**, **Learn**, and **Alerts**. The options in the **Home** tab are used for invoking new SOLIDWORKS documents, opening existing documents, viewing recent documents and folders, and accessing SOLIDWORKS resources. The **Recent** tab displays a longer list of recent documents and folders. The **Learn** tab is used for accessing instructional resources such as tutorials, sample models, access to 3D Content Center, certification program, and so on, to help you learn more about SOLIDWORKS. The **Alerts** tab is used for updating you with SOLIDWORKS news and providing different types of alerts in different sections including **Troubleshooting** and **Technical Alerts**.

Tip: If the **Welcome** dialog box does not appear on the screen, then you can invoke it by clicking on the **Welcome to SOLIDWORKS** tool  in the **Standard toolbar** or by pressing the **CTRL + F2** keys. Alternatively, you can click on **Help > Welcome to SOLIDWORKS** in the **SOLIDWORKS Menu** or the **Welcome to SOLIDWORKS** option in the **SOLIDWORKS Resources Task Pane** to invoke the **Welcome** dialog box. You will learn more about **Standard toolbar**, **SOLIDWORKS Menu**, and **Task Pane** later in this chapter.

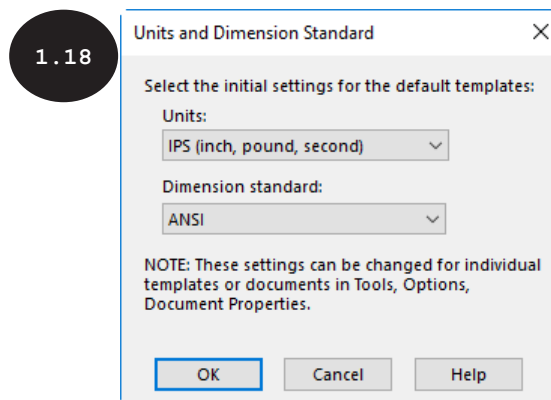
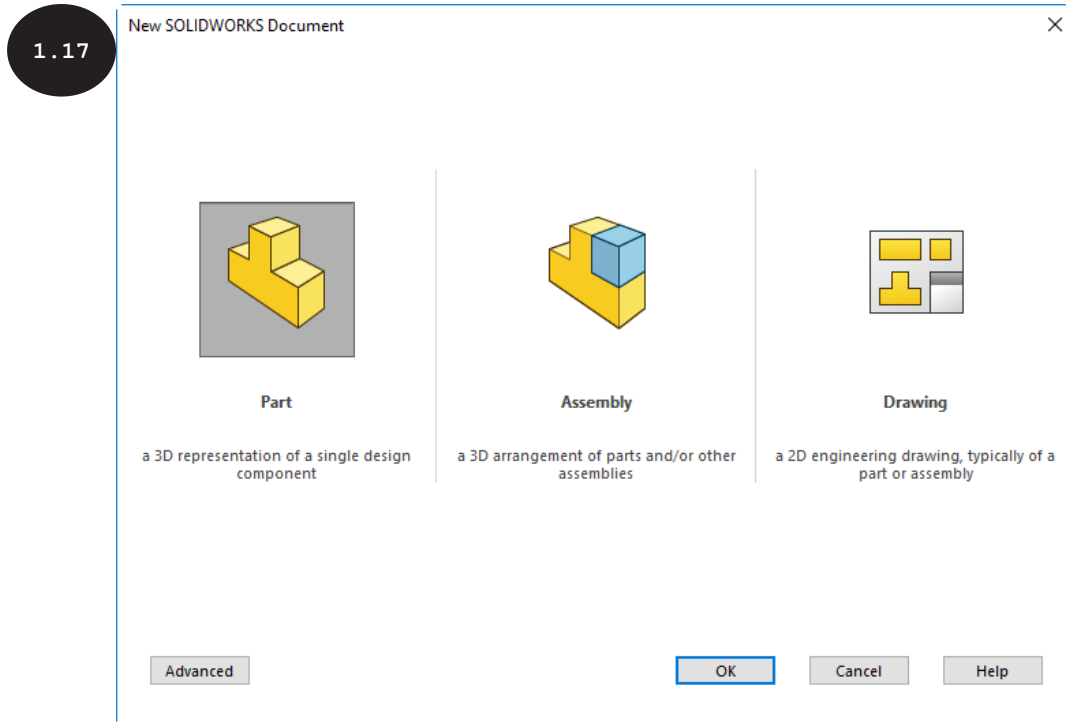
Note that **SOLIDWORKS Simulation** can be invoked within the **Part modeling** and **Assembly** environments of **SOLIDWORKS**, therefore, it is important to get familiar with different environments of **SOLIDWORKS**. The methods for invoking different environments of **SOLIDWORKS** are discussed below.

Invoking the Part Modeling Environment

The **Part modeling** environment is used for creating 3D solid models, surface models, and sheet metal models. Further, you can access **SOLIDWORKS Simulation** within the **Part modeling** environment of **SOLIDWORKS** to perform finite element analysis (FEA) on a part/component. To invoke the **Part modeling** environment, click on the **Part** button in the **Welcome** dialog box. The **Part modeling** environment gets invoked and the initial screen of the **Part modeling** environment appears, as shown in Figure 1.16. Alternatively, to invoke the **Part modeling** environment, click on the **New** tool  in the **Standard toolbar**. A **New SOLIDWORKS Document** dialog box appears, see Figure 1.17. In this dialog box, make sure that the **Part** button is activated and then click on the **OK** button. You will learn how to invoke **SOLIDWORKS Simulation** within the **Part modeling** environment later in this chapter.



Note: If you are invoking the **Part modeling** environment for the first time after installing the software, the **Units and Dimension Standard** dialog box appears, see Figure 1.18. You can specify the unit system as the default unit system for **SOLIDWORKS** by using this dialog box.



It is evident from the initial screen of the Part modeling environment that SOLIDWORKS is very user-friendly and easy to operate. Some of the components of the initial screen are discussed below.

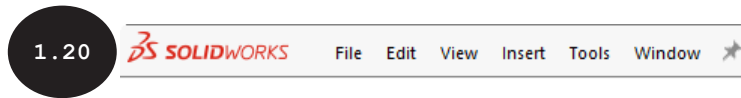
Standard Toolbar


The Standard toolbar contains a set of the most frequently used tools such as New, Open, and Save, see Figure 1.19.



SOLIDWORKS Menus

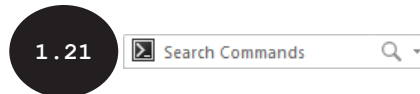
The SOLIDWORKS Menus contain different menus such as **File**, **View**, and **Tools** for accessing different tools of SOLIDWORKS, see Figure 1.20.



Note that the SOLIDWORKS Menus appear when you move the cursor over the SOLIDWORKS logo, which is available at the top left corner of the screen. You can keep the SOLIDWORKS Menus visible all the time by clicking on the push-pin button  available at the end of the SOLIDWORKS Menus. Note that the tools in different menus are dependent upon the type of environment invoked.

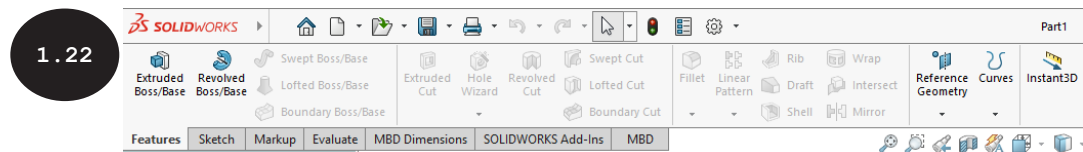
SOLIDWORKS Search

The SOLIDWORKS Search is a search tool for searching commands, tools, knowledge database (help topics), community forum, files, models, and so on, see Figure 1.21.



CommandManager

CommandManager is available at the top of the graphics area. It provides access to different SOLIDWORKS tools. There are various CommandManagers such as **Features CommandManager**, **Sketch CommandManager**, **Evaluate CommandManager**, and so on, which are available in the Part modeling environment. When the **Features** tab is activated in the CommandManager, the **Features CommandManager** appears, which provides access to different tools for creating 3D solid models, see Figure 1.22. On clicking on the **Sketch** tab, the **Sketch CommandManager** appears, which provides access to different tools for creating sketches.



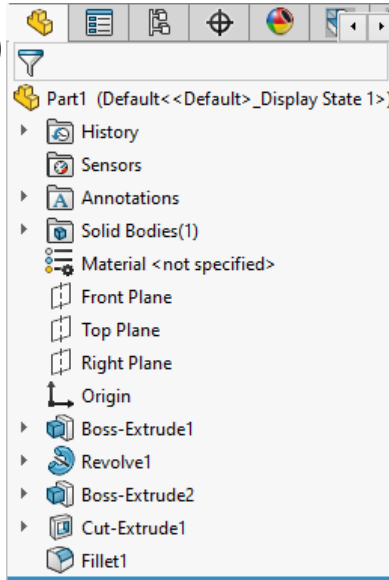
Note that tabs of some of the CommandManagers such as **Surfaces CommandManager** and **Sheet Metal CommandManager** are not available in the CommandManager by default. You will learn about adding these tabs later in this chapter.

Note: Different environments (Part, Assembly, and Drawing) of SOLIDWORKS are provided with different sets of CommandManagers.

FeatureManager Design Tree

FeatureManager Design Tree appears on the left side of the graphics area and keeps a record of all operations or features used for creating a model, see Figure 1.23. Note that the first created feature appears at the top and the following created features appear one after the other in the order of creation in the FeatureManager Design Tree. Also, in the FeatureManager Design Tree, three default planes and an origin appear by default, see Figure 1.23.

1.23



Tip: Features are logical operations that are performed to create a component. In other words, a component can be designed by creating a number of features such as extrude, sweep, hole, fillet, draft, and so on.

View (Heads-Up) Toolbar

The View (Heads-Up) toolbar is available at the top center of the graphics area, see Figure 1.24. It is provided with different sets of tools that are used for manipulating the view and display of a model available in the graphics area.

1.24



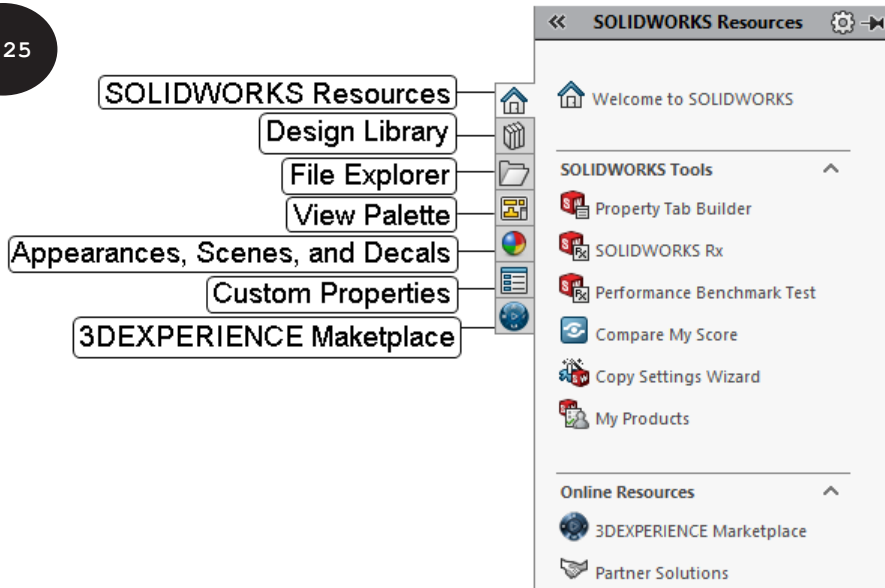
Status Bar

The Status Bar is available at the bottom of the graphics area and provides information about the action to be taken based on the currently active tool. It also displays the current state of a sketch being created, coordinate system, and so on.

Task Pane

Task Pane appears on the right side of the screen with various tabs such as SOLIDWORKS Resources, Design Library, File Explorer, View Palette, Appearances, Scenes, and Decals, Custom Properties, and 3DEXPERIENCE Marketplace for accessing various online resources of SOLIDWORKS, several applications, subscription services, library, and so on, see Figure 1.25.

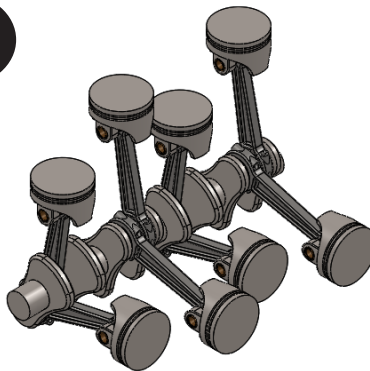
1.25



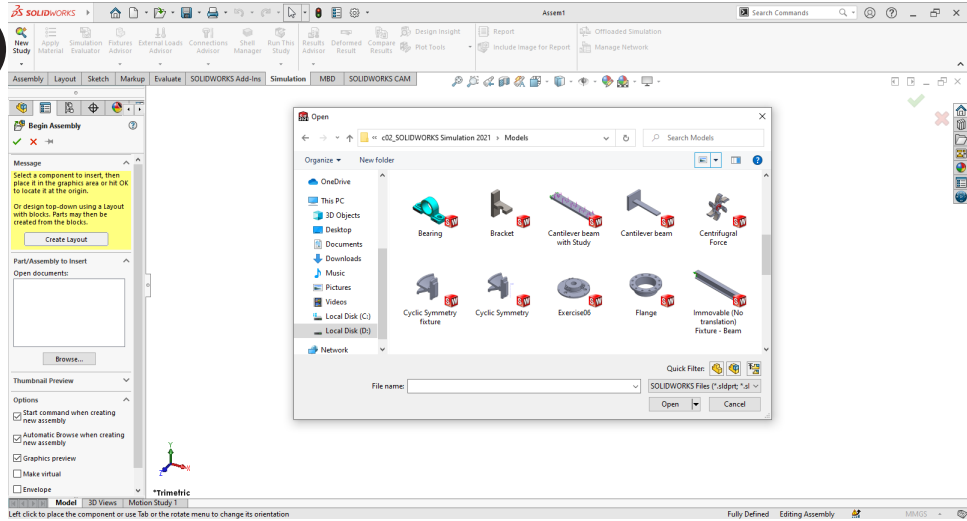
Invoking the Assembly Environment


The Assembly environment is used for assembling different components of an assembly with respect to each other by applying the required mates, see Figure 1.26. Further, you can access SOLIDWORKS Simulation within the Assembly environment to perform finite element analysis (FEA) on an assembly or assembly components. To invoke the Assembly environment, click on the **Assembly** button in the **Welcome** dialog box. The Assembly environment is invoked with the display of the **Open** dialog box along with the **Begin Assembly PropertyManager**, see Figure 1.27. You will learn how to invoke SOLIDWORKS Simulation within the Assembly environment later in this chapter.

1.26



1.27

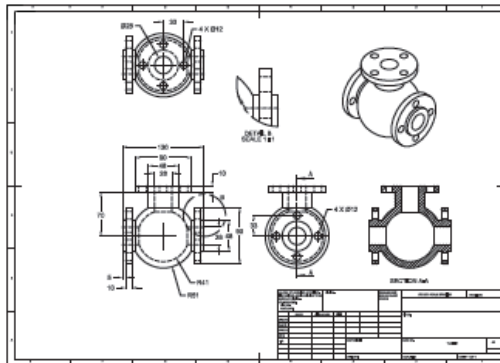


Alternatively, to invoke the Assembly environment, click on the **New** tool  in the **Standard** toolbar. A **New SOLIDWORKS Document** dialog box appears. In this dialog box, click on the **Assembly** button and then click on the **OK** button.

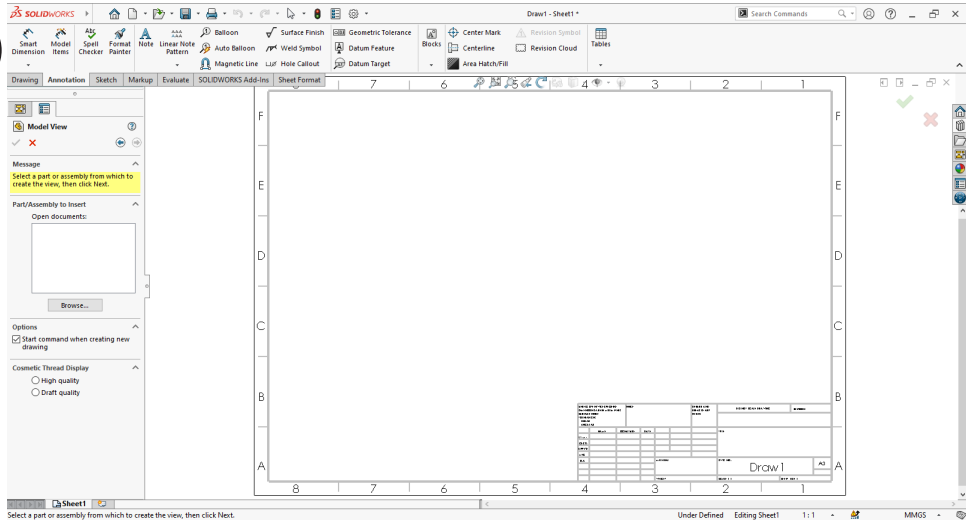
Invoking the Drawing Environment

The Drawing environment of SOLIDWORKS is used for creating 2D drawings of a part or an assembly, see Figure 1.28. To invoke the Drawing environment, click on the **New** tool in the **Standard** toolbar. A **New SOLIDWORKS Document** dialog box appears. In this dialog box, click on the **Drawing** button and then click on the **OK** button. The **Sheet Format/Size** dialog box appears. The options in this dialog box are used for selecting sheet size/format to be used for creating drawings. You can also invoke the **Sheet Format/Size** dialog box by clicking on the **Drawing** button in the **Welcome** dialog box. Once you have defined sheet size and format in the **Sheet Format/Size** dialog box, click on the **OK** button. The initial screen of the Drawing environment appears with the display of the **Model View PropertyManager** on the left of the drawing sheet, see Figure 1.29. The **Model View PropertyManager** is used for inserting a component or an assembly into the Drawing environment to create its drawing views.

1.28



1.29



Identifying SOLIDWORKS Documents

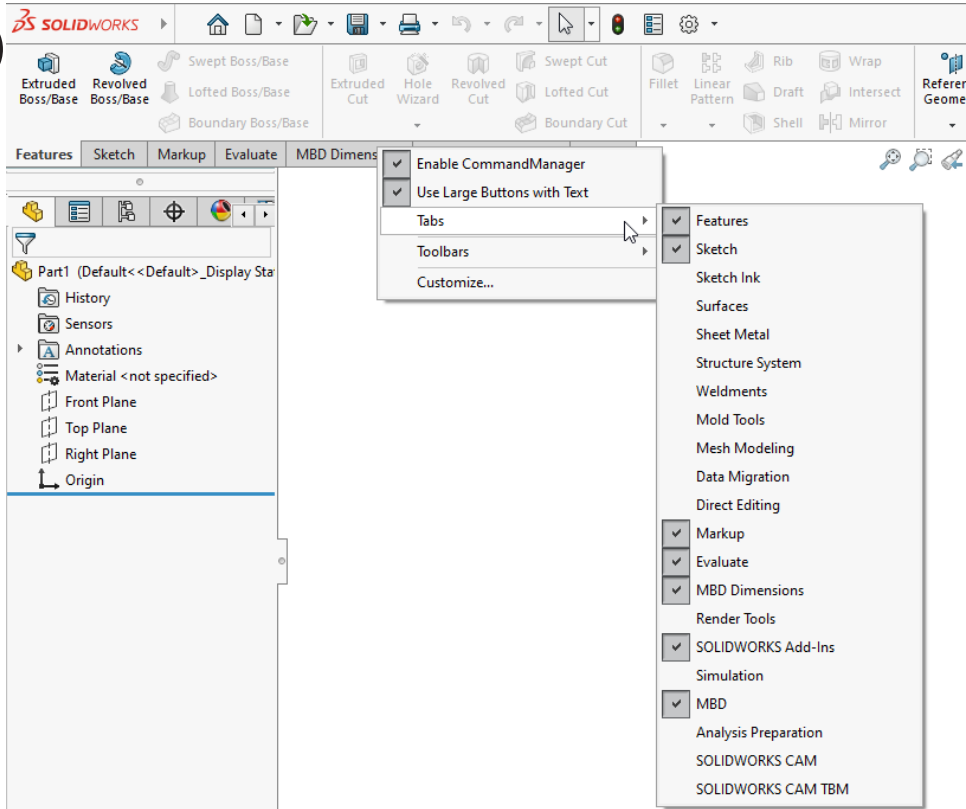
The documents created in different environments (Part, Assembly, and Drawing) of SOLIDWORKS have a different file extension, see the table given below:

Environments	File Extension
Part Modeling Environment	*.sldprt
Assembly Modeling Environment	*.sldasm
Drawing Modeling Environment	*.slddrw

Adding CommandManager Tabs

In addition to the default CommandManagers such as **Features CommandManager** and **Sketch CommandManager**, you can also add additional CommandManagers that are not available by default. To add a CommandManager, right-click on any of the available CommandManager tabs. A shortcut menu appears. In this shortcut menu, move the cursor over the **Tabs** option. A cascading menu appears, see Figure 1.30. This cascading menu displays a list of the available CommandManagers. A tick-mark in front of the CommandManager indicates that the respective CommandManager is already added. Click on the required CommandManager in the cascading menu, the respective CommandManager tab gets added to the CommandManager.

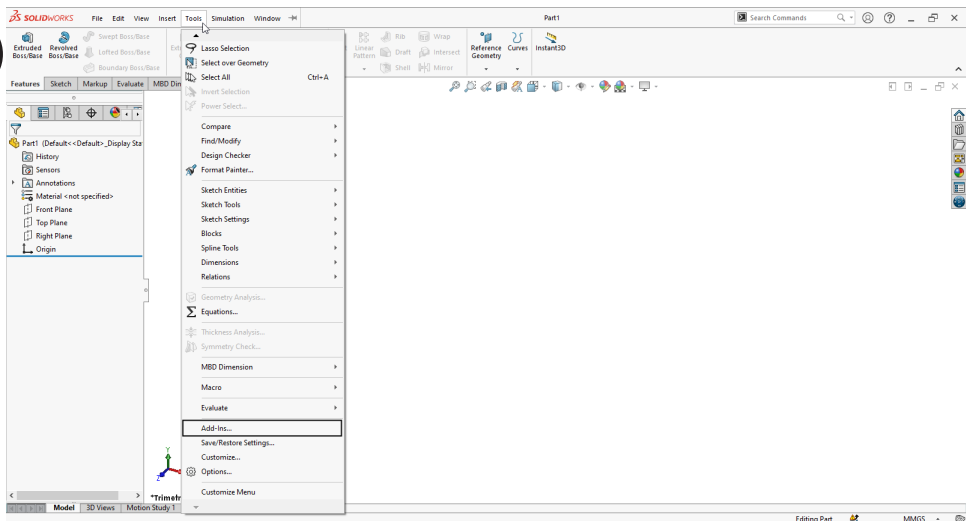
1.30



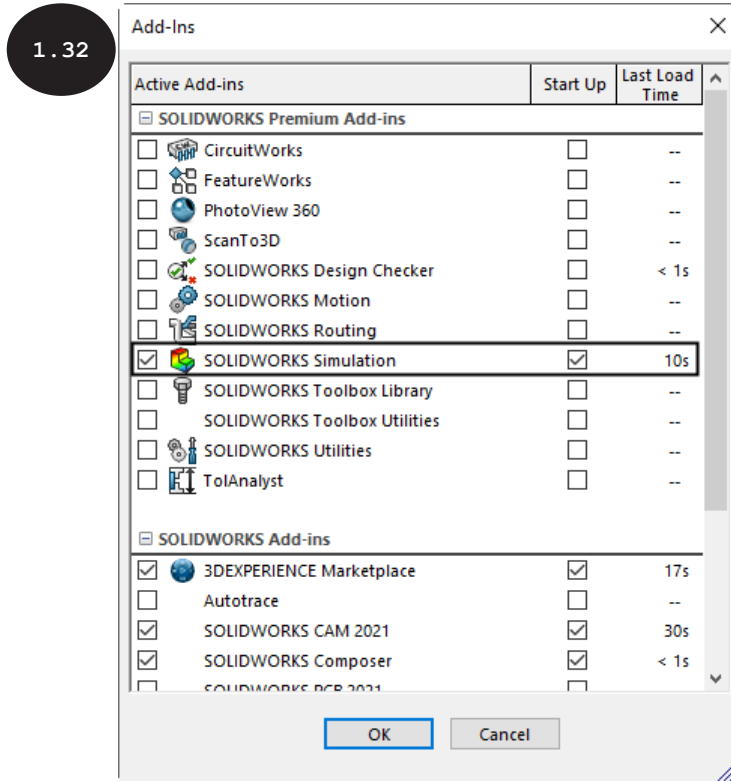
Invoking SOLIDWORKS Simulation

SOLIDWORKS Simulation can be invoked within the Part modeling and Assembly environments of SOLIDWORKS to perform various types of finite element analysis (FEA). To start SOLIDWORKS Simulation, click on the Tools > Add-Ins in the SOLIDWORKS Menus, see Figure 1.31. The Add-Ins dialog box appears, see Figure 1.32.

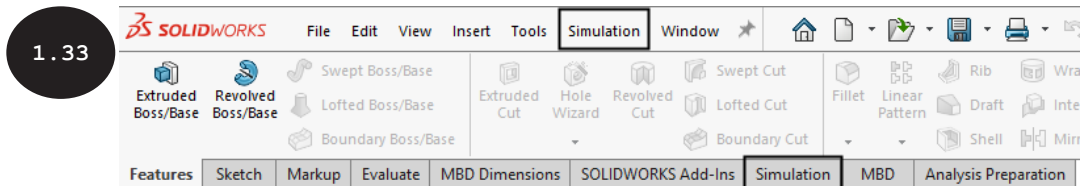
1.31



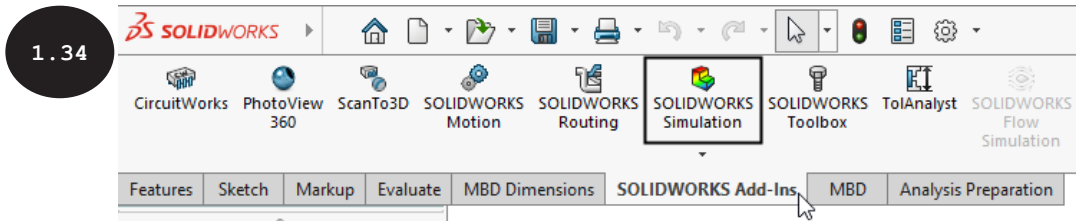
NOTE You may need to expand the **Tools** menu of the SOLIDWORKS Menus by clicking on the arrow at its bottom to display the **Add-Ins** tool as shown in Figure 1.31.



In the **Add-Ins** dialog box, click on the check boxes available on the left and right of the **SOLIDWORKS Simulation** option, see Figure 1.32. Next, click on the **OK** button. **SOLIDWORKS Simulation** is invoked and the **Simulation** menu gets added to the **SOLIDWORKS Menus**, see Figure 1.33. Also, the **Simulation** tab gets added to the **CommandManager**, see Figure 1.33. The **Simulation** menu and **Simulation** tab are provided with different sets of simulation tools to perform various types of finite element analysis (FEA).



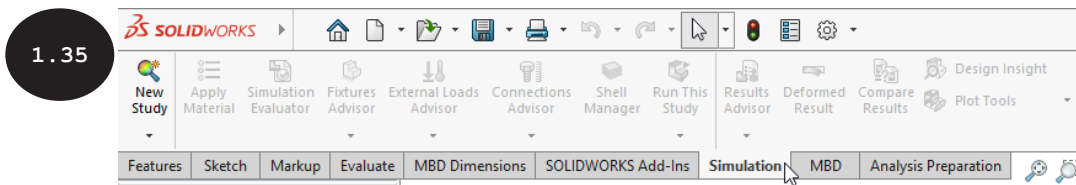
If the **Simulation** tab is not added in the CommandManager by default, then you need to add it manually. For doing so, right-click on any of the tabs of the CommandManager. A shortcut menu appears. In this shortcut menu, move the cursor over the **Tabs** options and then click on the **Simulation** option in the cascading menu that appears. Alternatively, click on the **SOLIDWORKS Add-Ins** tab in the CommandManager to display the tools of the **SOLIDWORKS Add-Ins CommandManager**, see Figure 1.34. Next, click on the **SOLIDWORKS Simulation** tool.



NOTE If you select the check box available to the right of the **SOLIDWORKS Simulation** option in the **Add-Ins** dialog box then **SOLIDWORKS Simulation** will be invoked every time on starting **SOLIDWORKS**, automatically. However, if you select the check box available to the left of the **SOLIDWORKS Simulation** option then **SOLIDWORKS Simulation** will be invoked only for the current session of the **SOLIDWORKS**. To invoke **SOLIDWORKS Simulation** for the current **SOLIDWORKS** session as well as for every session of **SOLIDWORKS**, you need to select both these check boxes in the **Add-Ins** dialog box.

If the **SOLIDWORKS Simulation** option is not available in the **Add-Ins** dialog box then you first need to install **SOLIDWORKS Simulation**.

After invoking **SOLIDWORKS Simulation**, click on the **Simulation** tab in the CommandManager. The **Simulation CommandManager** appears, see Figure 1.35. It contains tools to perform various types of finite element analysis (FEA). Note that initially, most of the tools of the **Simulation CommandManager** are not enabled. These tools will get enabled only after defining the type of finite element analysis (FEA) to be performed. You can define the type of analysis by using the **New Study** tool of the **Simulation CommandManager**.



Note that before you define the type of analysis to be performed by using the **New Study** tool, you need to create or import a geometry in SOLIDWORKS to perform the analysis on it. You will learn about performing different types of analysis in the later chapters.

Summary

This chapter provided an introduction to SOLIDWORKS Simulation, various types of analysis, introduction to finite element analysis (FEA), and different phases of finite element analysis (FEA): Pre-processing, Solution, and Post-processing. It explained various terms and definitions used in finite element analysis (FEA). The chapter also discussed different types of elements, the application areas of FEA, system requirements for installing SOLIDWORKS Simulation, and SOLIDWORKS interface, invoking different SOLIDWORKS documents and getting started with SOLIDWORKS Simulation.

Questions

- In the linear static analysis, the material properties of an object are assumed to behave _____ under the impact of applied load.
- The _____ analysis is used for calculating the natural or resonant frequencies and the associated mode shapes of a structure.
- The _____ analysis is used for calculating the stress at which an object fails, when it undergoes repeated loading and unloading process.
- The finite element analysis (FEA) uses a numerical technique known as _____ to solve engineering problems.
- In finite element analysis (FEA), the entire process of analyzing the engineering design is divided into three phases _____, _____, and _____.
- _____ is defined as the process of dividing an object into a finite number of pieces with simple geometric shapes called elements.
- The elements are mainly categorized into _____, _____, and _____.
- _____ strength is defined as the maximum stress that a material can withstand when subjected to an external load.
- _____ is defined as the ratio of change in length to the original length of the object when it undergoes deformation due to the application of external load.

- _____ strength is defined as the maximum stress up to which a material deforms elastically under the impact of applied load and returns to its original configuration once the load is removed.
 - _____ is defined as the property of a material which offers resistance against the deformation of the material when it is subjected to an external force.
-