Chapter 2

Introduction to Autodesk Simulation Mechanical

Learning Objectives

After completing this chapter, you will be able to:

- *Introduction to Autodesk Simulation Mechanical*
- *Understand the System Requirements to run Autodesk Simulation Mechanical*
- *Start Autodesk Simulation Mechanical*
- *Understand various screen components of Autodesk Simulation Mechanical*
- *Set Unit System*
- *Understand various Environments of Autodesk Simulation Mechanical*
- *Understand about various shortcut menus*
- *Modify the background color scheme in Autodesk Simulation Mechanical*
- *Understand the use of various hotkeys*

INTRODUCTION TO Autodesk Simulation Mechanical

Welcome to the world of Computer Aided Engineering (CAE) with Autodesk Simulation Mechanical. If you are a new user of this software package, you will be joining hands with thousands of users of this software package which provides a broad range of simulation tools to help designers and engineers to know the product performance in early stages of the design cycle. If you are familiar with the previous releases of this software, you will be able to upgrade your analysis skills with the tremendously improved version of this software.

Autodesk Simulation Mechanical is the product of Autodesk, a leading supplier of various engineering software packages that provide various tools to design, visualize, and simulate different ideas. By putting powerful Digital Prototyping technology within the reach of mainstream manufacturers, Autodesk is changing the way manufacturers think about their design processes and is helping them create more productive workflows. Autodesk approach to Digital Prototyping is unique because it is scalable, attainable, and cost-effective. These factors allow a broader group of manufacturers to realize the benefits with minimal disruption to existing workflows. It also provides the most straight forward path to create and maintain a single digital model in a multidisciplinary engineering environment.

Autodesk Simulation Mechanical is a Finite Element Analysis tool which enables critical engineering decisions to be made earlier in the design process. Using the tools in this software, designers and engineers can easily study the initial design and predict the performance of the complete digital prototype. The automatic meshing tools of this software produce high-quality elements on the first pass while working with CAD geometry. It also enables designers and engineers to directly edit a mesh for accurate placement of loads and constraints or to simplify the geometry using its modeling capabilities. Autodesk Simulation Mechanical software makes it possible to quickly validate design concepts before resources are invested in design changes or new products.

Autodesk Simulation Mechanical provides wide range of linear and nonlinear materials that allows better understanding of the real-world behavior of products and let engineers know how a product will perform in the real-world environment.

Autodesk Simulation Mechanical supports efficient workflows with today's multi-CAD environment by providing direct geometry exchange and full associativity with Autodesk Inventor, Pro/ENGINEER Creo Parametric, and other such software packages. It allows iterative design changes without redefining material, loads, constraints, or other simulation data when working with the native CAD format. You can also import 2D and 3D geometry of CAD universal file formats such as ACIS®, IGES, STEP, and STL for solid models and CDL, DXF™, and IGES for wireframe models.

As mentioned earlier, Autodesk Simulation Mechanical software provides a broad range of simulation tools which help engineers to ascertain product performance knowledge during early stages of the design cycle. It improves the collaboration, helps design better and safer products, saves time, and reduces manufacturing costs. Some of the most important types of analysis carried out using this software package are discussed next.

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

Linear Analysis

Linear analysis is undertaken when an object is expected to behave linearly. In other words, it follows the Hooke's Law which states that within the elastic limit stress is directly proportional to strain, refer to Figure 2-1. Also, the material that undergoes linear analysis experiences elastic deformation and will return to its original configuration or shape after the load has been removed from it. In linear analysis, the following factors are calculated:

1. Displacements and stresses due to static loads
2. Natural frequency and mode shapes of the model
3. Natural frequency and mode shapes of the model with load stiffening
4. Maximum displacement and stress due to a spectrum-type load
5. Statistical response of a system (displacements and stresses) due to random vibration, noise, or spectrum density
6. Steady state response (displacements and stresses) due to harmonic or sinusoidal load, or acceleration
7. Displacements and stresses over time due to loads that will vary in a known fashion
8. Load that causes the model to buckle due to geometric instability
9. Response of the model due to shock

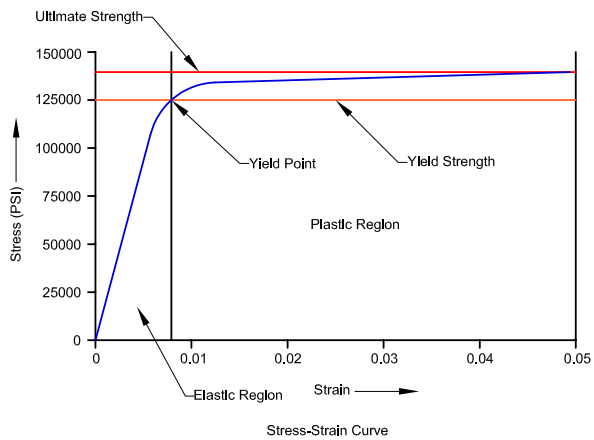

Figure 2-1 *The Stress and Strain Curve*

Non-linear Analysis

Non-linear analysis is used to analyze the behavior of an object that is loaded beyond its elastic limits and exceeds the yield strength on the stress-strain curve, refer to Figure 2-1. Also, the material that undergoes non-linear analysis experiences plastic deformation and will not return to its original configuration or shape even after the load has been removed from it. In non-linear analysis, you can calculate:

1. Displacements and stresses due to static loads
2. Displacements, velocities, accelerations, and stresses over time due to dynamic loads
3. Natural frequencies and mode shapes of the model
4. Displacements and stresses before and after the model has collapsed

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

Thermal Analysis

Thermal Analysis is used to determine the temperature distribution and related thermal quantities in a model. As there are three mechanisms of heat transfer, namely conduction, convection, and radiation, the thermal analysis is used to calculate the temperature distribution in a body due to all of these mechanisms. In thermal analysis, you can calculate:

1. Temperature and heat fluxes after an infinite period under steady-state conditions
2. Temperature and heat fluxes over time due to the thermal loads

SYSTEM REQUIREMENTS

The system requirements to ensure the smooth functioning of Autodesk Simulation Mechanical 2016 on 32 bit and 64 bit windows are as follows:

- Windows® 7 Home Premium, Professional, Enterprise (SP1), Ultimate; Windows® 8; Windows® 8.1; Windows Server® 2008 R2 SP1; Windows® Server 2012 operating system
- Intel® Pentium® 4, Intel Xeon®, Intel Core™, AMD Athlon™ II, or AMD Opteron™ or later (2 GHz CPU speed or higher) processor
- 2 GB RAM minimum (8 GB recommended)
- 500 GB or larger hard drive, 30 GB minimum free disk space (8 GB free disk space for installation)
- 512 MB DRAM or greater OpenGL®-capable graphics card
- 24-bit color setting at 1,280 x 1,024 or higher screen resolution
- Microsoft Office Excel, either version 2007 or higher
- Adobe Acrobat 7.0.7 or higher
- DVD drive and Mouse or any other compatible pointing device
- Internet Explorer version 7 or higher

GETTING STARTED WITH Autodesk Simulation

Install Autodesk Simulation Mechanical 2016 on your system and then double-click on the Autodesk Simulation Mechanical 2016 icon on the desktop of your computer, the system will prepare to start Autodesk Simulation Mechanical by loading all the required files. After all the required files have been loaded, the **New** or **Open** dialog box along with the initial screen of Autodesk Simulation Mechanical 2016 will be displayed, refer to Figure 2-2.

Note that if you are starting Autodesk Simulation Mechanical first time after installing it, the **Autodesk Simulation Mechanical** message window may be displayed informing you that no default modeling unit has been selected, refer to Figure 2-3. Choose the **OK** button from this window; the **Unit System** dialog box will be displayed, refer to Figure 2-4. Specify the default unit system for measurement by using the options of this dialog box. You will learn more about specifying unit system by using this dialog box later in this chapter. After specifying the unit system, choose the **OK** button; the **New** dialog box will be displayed along with the initial interface of Autodesk Simulation Mechanical, as shown in Figure 2-2.

Note

*1. The automatic display of the **New** or **Open** dialog box along with the initial screen of Autodesk Simulation Mechanical depends upon which of these dialog boxes was used in the last session of Autodesk Simulation Mechanical.*

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

*2. You can turn off the automatic display of the **New** or **Open** dialog box: at startup. To do so, first close the opened dialog box, if any, and then choose the **Tools** tab from the **Ribbon**. Next, choose the **Application Options** button from the **Options** panel of the **Tools** tab; the **Options** dialog box will be displayed. In this dialog box, choose the **General Information** tab and then clear the **Show file dialog on startup** check box from the **Miscellaneous options** area of the dialog box. Then choose the **Apply** button from the dialog box. Next, close the dialog box by choosing the **Close** button.*

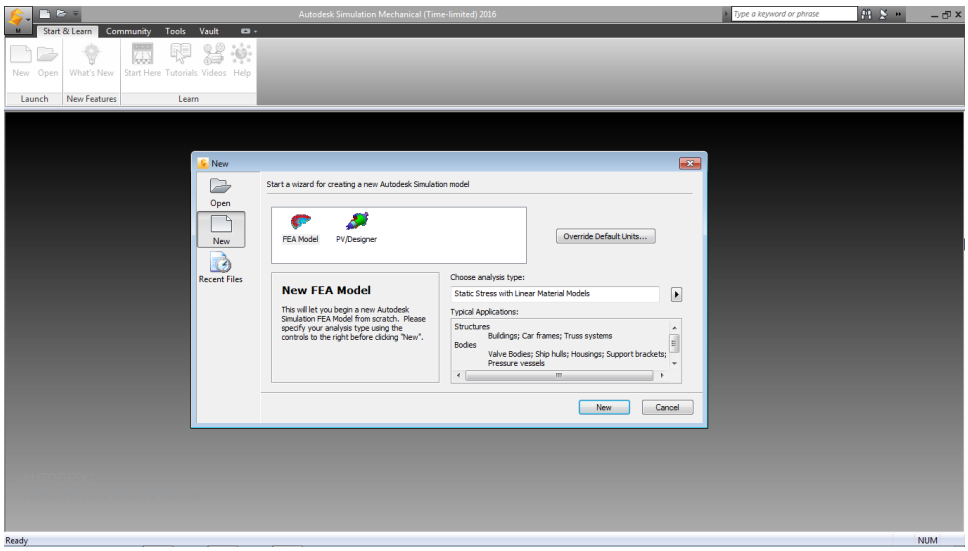

Figure 2-2** Initial Inteface display of **Autodesk Simulation Mechanical 2016

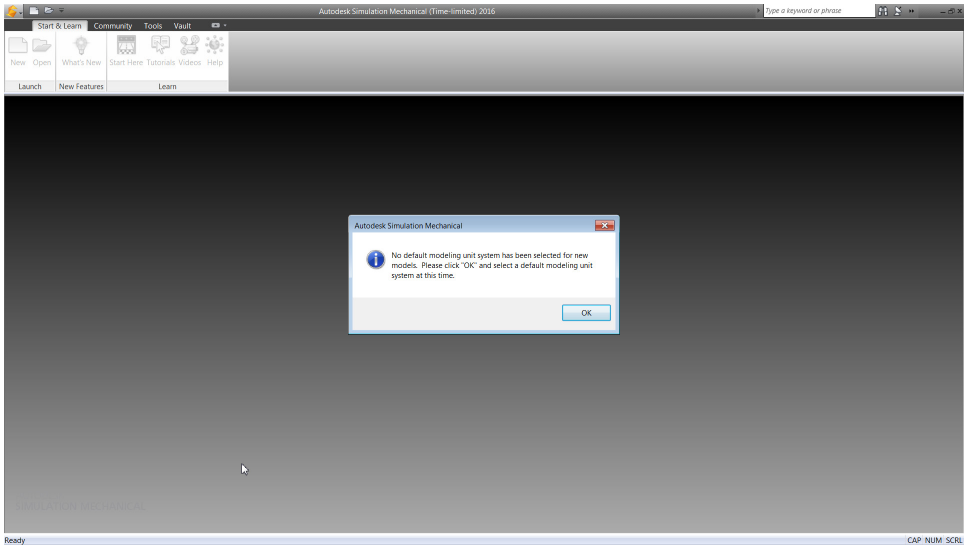

***Figure 2-3** The **Autodesk Simulation Mechanical** message window*

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

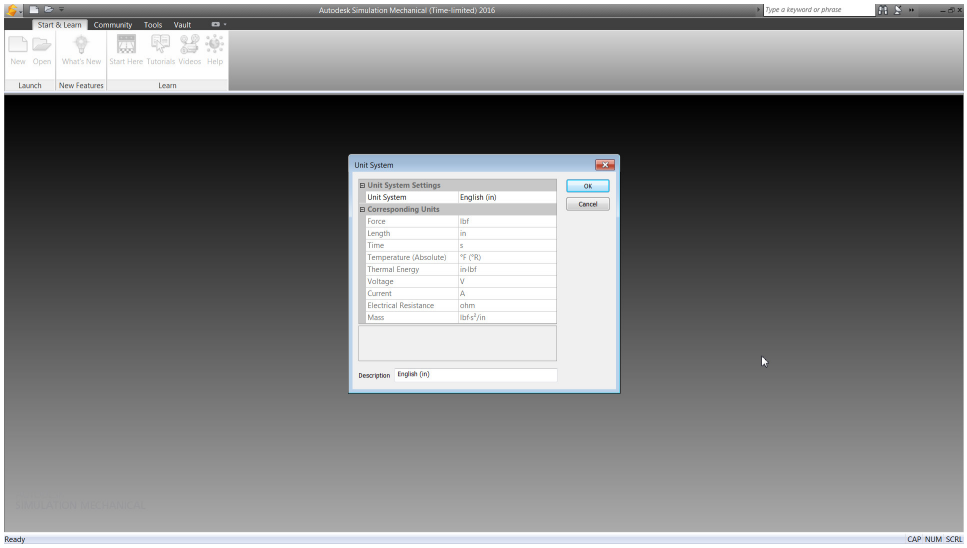

Figure 2-4 *The* ***Unit System*** *dialog box*

By using the **New** dialog box, you can create a new FEA model. In addition to creating new FEA model, you can also invoke the **Open** dialog box by choosing the **Open** button and the **Unit System** dialog box by choosing the **Override Default Units** button of the **New** dialog box. Note that if the **New** dialog box is not invoked by default, you can invoke it by choosing the **New** tool from the **Launch** panel of the **Start & Learn** tab in the **Ribbon**.

The **Open** dialog box is used to open an existing Autodesk Simulation FEA or CAD model in the Autodesk Simulation Mechanical 2016 to run the analysis. By default, the **Autodesk Simulation FEA Model (*.fem)** is selected in the **Files of type** drop-down list of the **Open** dialog box. As a result, you can open the existing Autodesk simulation FEA model by selecting the FEA model to be opened from the location where it is saved and then choosing the **Open** button from the dialog box. The *.fem* is the file extension of the Autodesk Simulation FEA model. To open a CAD model in the Autodesk Simulation Mechanical, you need to select their respective file extension from the **Files of type** drop-down list of the **Open** dialog box. You will learn more about opening the FEA and CAD models in the later chapters.

Similar to invoking the **Open** dialog box from the **New** dialog box, you can also invoke the **New** dialog box from the **Open** dialog box. Additionally, you can also invoke these dialog boxes by choosing their respective button from the **Launch** panel of the **Start & Learn** tab in the **Ribbon**.

To create a new FEA model by using the **New** dialog box, select the **FEA Model** button and then select the type of analysis to be performed on the model being created. To select the type of analysis, click on the arrow on the right in the **Choose analysis type** area of the **New** dialog box; a flyout will be displayed, refer to Figure 2-5. Now, move the cursor over the type of analysis that you want to carry out for the model; all the analyses that lie under the selected analysis type will be displayed in another flyout. Select the required type of analysis from this flyout.

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

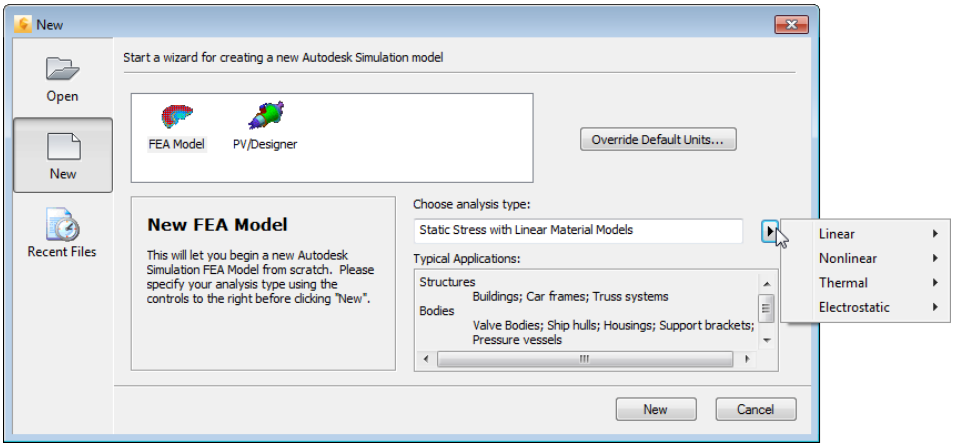

Figure 2-5 *The flyout displayed after clicking on the arrow*

After specifying the type of analysis, choose the **New** button available at the lower right corner the **New** dialog box; the **Save As** dialog box will be displayed. Browse to the location where you want to save the FEA model in your computer. Next, enter the name of the FEA model in the **File name** edit box of the dialog box and choose the **Save** button; the new FEA file is created with the specified name at the specified location. Also, the FEA Editor environment of Autodesk Simulation Mechanical is invoked for creating the FEA model, refer to Figure 2-6. Now, you can start working on creating the FEA model.

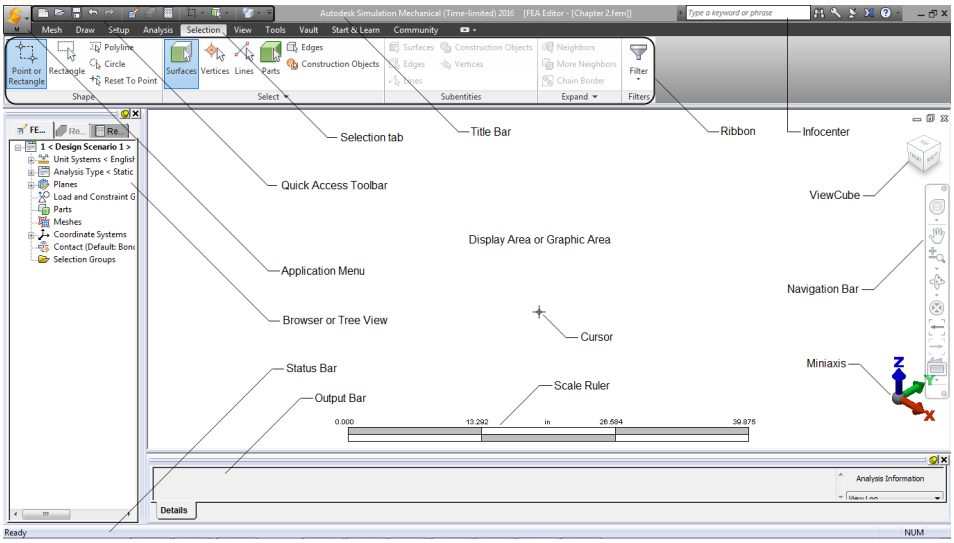

Figure 2-6 *The FEA Editor environment of Autodesk Simulation Mechanical*

It is evident from the Figure 2-6 that the interface of Autodesk Simulation Mechanical is very user-friendly. Apart from the components shown in Figure 2-6, you are also provided with various shortcut menus that are displayed on right-clicking. The type of the shortcut menu and its options will depend on where or when you are trying to access the shortcut menu. In

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

this textbook, the shortcut menus will be discussed when they are used. Some of the important components of Autodesk Simulation Mechanical interface are discussed next.

Quick Access Toolbar

This toolbar is common to all environments of Autodesk Simulation Mechanical. Note that some of the options in this toolbar may not be available when you start Autodesk Simulation Mechanical for the first time. Figure 2-7 shows the **Quick Access Toolbar**. Some of the important tools of this toolbar are discussed next.

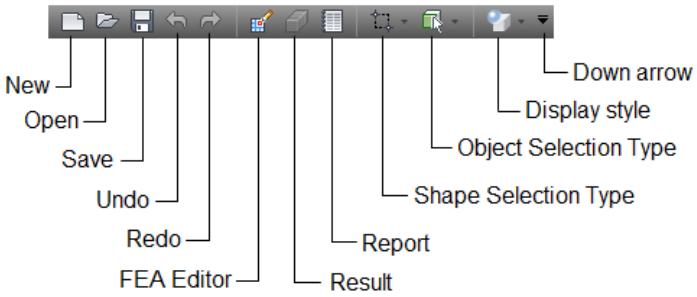

Figure 2-7 *The* ***Quick Access Toolbar***

Tip. *You can add or remove the tools in the* ***Quick Access Toolbar****. To add a tool to the* ***Quick Access Toolbar****, right-click on the tool to be added in the* ***Ribbon****; a shortcut menu will be displayed. Choose the* ***Add to Quick Access Toolbar*** *option from the shortcut menu; the chosen tool will be added. Similarly, to remove a tool from the* ***Quick Access Toolbar****, right-click on the tool to be removed and select the* ***Remove from Quick Access Toolbar*** *option from the shortcut menu displayed.*

Shape Selection Type Drop-down

The tools available in the **Shape Selection Type** drop-down are used to select objects by using different methods, refer to Figure 2-8. The **Point or Rectangle Select** tool is chosen by default in this drop-down. As a result, you can select an object by clicking on it. Also, you can draw a rectangular window around the objects to be selected by using this tool. The **Rectangle Select** tool is used to select multiple objects by creating a rectangle around the objects. To do so, you need to specify the two diagonally opposite corners of the rectangle in the drawing area around the objects to be selected. If you choose the **Polyline Select** tool, you can select multiple objects by creating a boundary of polyline around the objects. The **Circle Select** tool is used to select multiple objects by creating a circle around the object to be selected in the drawing area by specifying its center point and the radius. The **Reset To Point** tool is used to automatically reset the type of selection back to point selection, once you are done with selection by using the other selection method.

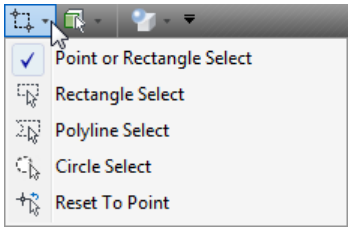

Figure 2-8 *The* ***Shape Selection Type*** *drop-down*

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

Object Selection Type Drop-down

The tools available in the **Object Selection Type** drop-down are used to specify the type of object to be selected, refer to Figure 2-9. The **Select Construction Objects** tool of this drop-down is used to select construction entities such as circles, arcs, lines, and splines. If you choose the **Select Surfaces** tool, you can select surfaces from the drawing area or from the **Tree View**. The **Select Vertices** tool is used to select the vertices. Similarly, you can select the entire part body by choosing the **Select Parts** tool, the edges of the part by choosing the **Select Edges** tool, and lines by choosing the **Select Lines** tool.

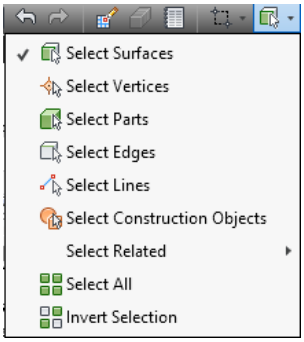

Figure 2-9 *The* ***Object Selection Type*** *drop-down*

In addition, you can also select the sub-entities of the selected entities or objects. For example, if a part body is selected in the drawing area using the **Select Parts** tool, you can further select its surfaces, edges, lines, construction object, or vertices. To do so, select the object whose sub-entities are to be selected and then choose the **Select Related > Surfaces/Edges/Lines/Construction Objects/Vertices** from the **Selection Type** drop-down of the **Quick Access Toolbar**.

FEA Editor

This tool is used to invoke the FEA Editor environment of Autodesk Simulation Mechanical. In this environment, you can set or create model for analysis. By default this environment is invoked automatically on starting a new FEA file or on opening a model for analysis.

Results

This tool is used to invoke the Results environment of Autodesk Simulation Mechanical. In the Results environment, you can view the result of the analysis. Also, you can review element types, nodes, properties, loads, and so on of the model to be analyzed in this environment. You will learn more about the Results environment in later chapters.

Report

This tool is used to invoke the Report environment of Autodesk Simulation Mechanical. In the Report environment, you can review the log and summary files of the analysis report and save the analysis report in different formats such as HTML, Word, PDF, and so on. You will learn more about the Report environment in later chapters.

Ribbon and Tabs

You might have noticed that there is no command prompt in Autodesk Simulation Mechanical. The complete analysis process is carried out by invoking the tools from the tabs in the **Ribbon**. The **Ribbon** is a long bar available below the **Quick Access Toolbar**. You can customize the tabs and panels of the **Ribbon** as per your need. To do so, right-click on the **Ribbon**; a shortcut menu will be displayed. To customize the tabs, choose **Show Tabs** from this shortcut menu; a cascading menu will be displayed. Now, you can add or remove the tabs from the **Ribbon** as per your need. Note that the tick mark on the name of a tab indicates that it is available in the **Ribbon**. Similarly, to customize the panels of the tabs in the **Ribbon**, choose **Show Panels** from the shortcut menu; a cascading menu will be displayed with a list of all the panels of the active tab. You can add or remove the panels of the tabs using this list.

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

Autodesk Simulation Mechanical provides you with different tabs while working with different environments. This means that the tabs available from the **Ribbon** while working with the FEA Editor, Report, and Results environments will be different. You can also change the location of the **Ribbon**. By default, the location of the **Ribbon** is at the top of the screen. To change the current location of the **Ribbon**, right-click on the empty area in the **Ribbon**; a shortcut menu will be displayed. Next, choose the **Ribbon Location > Left** or **Ribbon Location > Right** from the shortcut menu displayed.

Note

*In the tabs, the tools are arranged in different panels in the **Ribbon**. Some of the panels and tools have arrows on their right, refer to Figure 2-10. These arrows are called down arrows. When you click on the down arrow of a tool, a list of additional tools will be displayed, refer to Figure 2-10.*

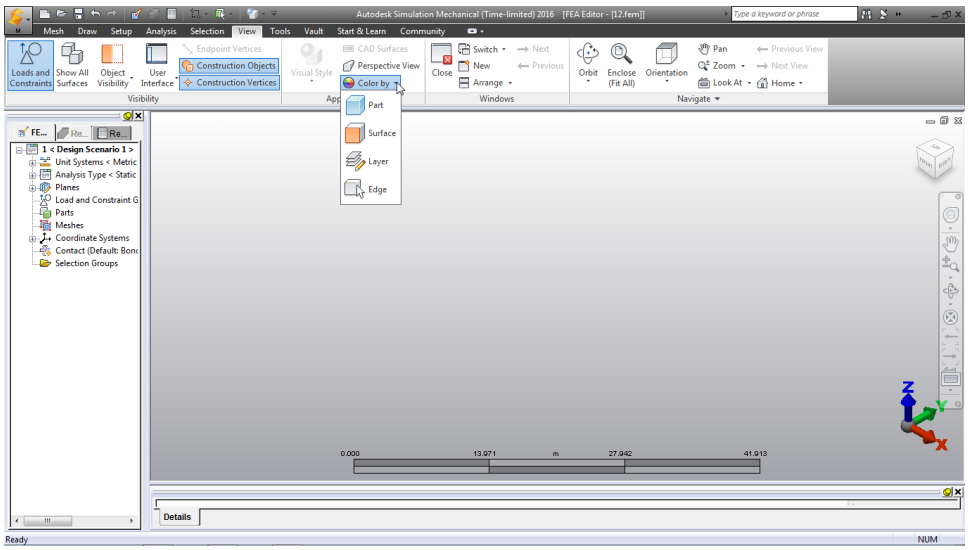

Figure 2-10 *Additional tools displayed on clicking the down arrow of a tool*

Selection Tab

This is one of the most important tabs in the **Ribbon**. All the selection tools are available in this tab. By using the tools available in this tab, you can specify the type of object you want to select and the method of selection. You will learn more about the usage of these tools in the later chapters. The **Selection** tab is shown in Figure 2-11.

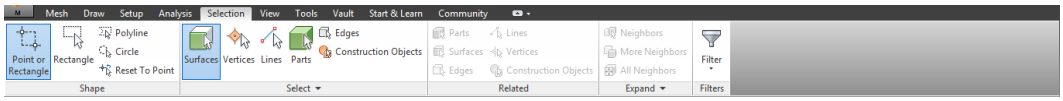

Figure 2-11 *The **Selection** tab*

Setup Tab

This is another important tab of the Autodesk Simulation Mechanical. This tab will be available only in the FEA Editor environment. The tools available in this tab are used to define the boundary conditions by specifying the required constraints and the applied loads for the model.

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

You will learn more about the usage of these tools in the later chapters. The tools in the **Setup** tab are shown in Figure 2-12.

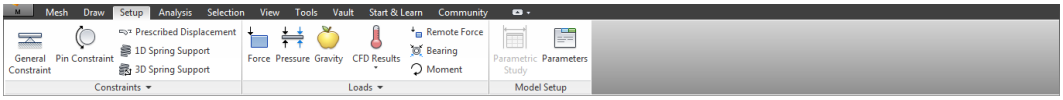

*Figure 2-12 The **Setup** tab*

Mesh Tab

This tab contains the tools that are required for generating mesh. Generating mesh on a component is one of the important steps carried out before proceeding with the analysis process. You will learn more about these tools in the later chapters. The tools in the **Mesh** tab are shown in Figure 2-13.

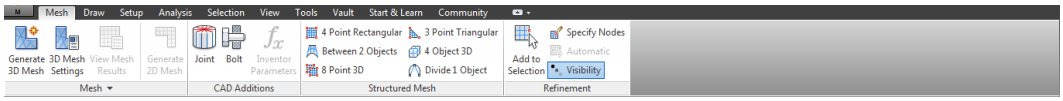

*Figure 2-13 The **Mesh** tab*

Note

*Most of the tools in the **Mesh** tab will be enabled if the model to be meshed is available or displayed in the graphic area of Autodesk Simulation Mechanical.*

Draw Tab

This tab contains the tools that are used to create and modify the geometry to be analyzed. This tab will be available only when you are in the FEA Editor environment. The tools in the **Draw** tab are shown in Figure 2-14.

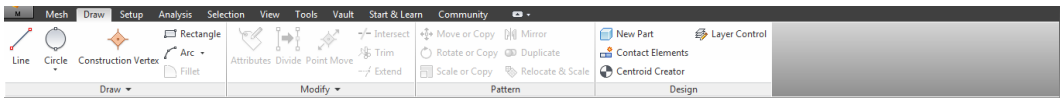

*Figure 2-14 The **Draw** tab*

View Tab

The tools in this tab enable you to the control the view, orientation, appearance, and visibility of objects and view windows. This tab is available in almost all environments. The tools in the **View** tab are shown in Figure 2-15.

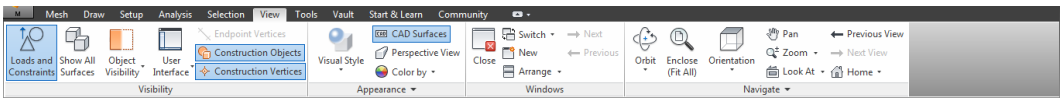

*Figure 2-15 The **View** tab*

Tools Tab

This tab contains tools that are mainly used for setting the preferences and for customizing the Autodesk Simulation Mechanical interface. This tab is available in almost all environments. The tools in the **Tools** tab are shown in Figure 2-16.

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

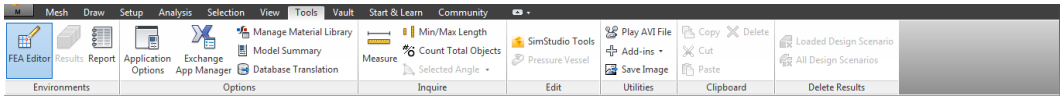

Figure 2-16 *The* ***Tools*** *tab*

Analysis Tab

This tab provides tools that are used to check the model for analysis, run the analysis, optimize the geometry, specify the type of analysis, and so on. The tools in the **Analysis** tab are shown in Figure 2-17.

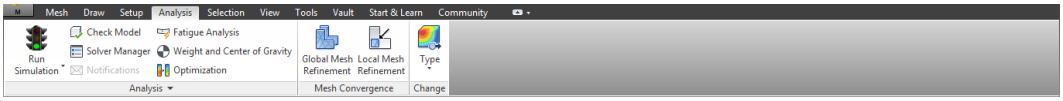

Figure 2-17 *The* ***Analysis*** *tab*

Status Bar

The Status Bar is located at the lower most of the Autodesk Simulation Mechanical screen and is used to display status messages and various command prompts.

Output Bar

The Output Bar is located at the bottom of the graphic area and is used to view the solid meshing results, analysis log, analysis summary, and convergence plot, refer to Figure 2-18. Note that the display of convergence plot will be available on performing nonlinear analyses. You can increase the display area of the Output Bar by dragging it from its top edge.

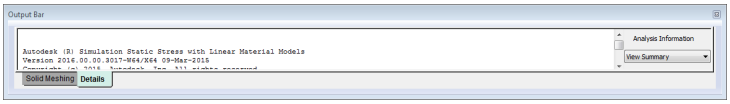

Figure 2-18 *The Output Bar*

Navigation Bar

The Navigation Bar is located on the right of the graphics area and contains tools that are used to make the designing process easier and quicker. The navigation tools also help you to control the view and orientation of the components in the drawing window. The Navigation Bar is shown in Figure 2-19.

ViewCube

The ViewCube is available at the upper right corner of the drawing window. It is used to freely change the view of a model in 3D space. A ViewCube is a 3D navigation tool, which allows you to switch between the standard and isometric views in a single click. The ViewCube is shown in Figure 2-20.

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

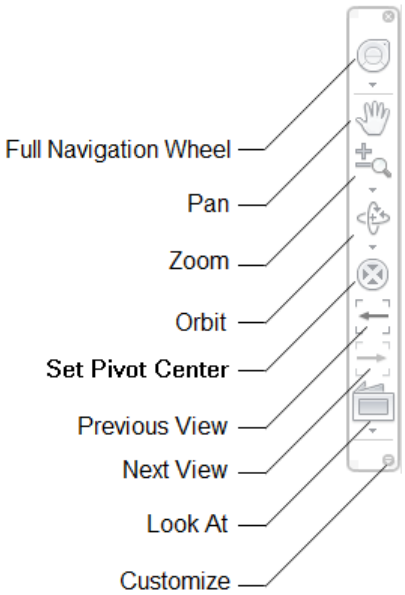

Figure 2-19 *The Navigation Bar*

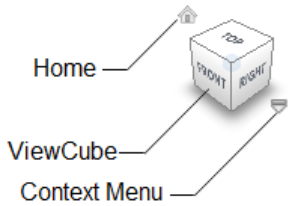

Figure 2-20 *The ViewCube*

Browser / Tree View

The Browser or Tree View is the most important component of the Autodesk Simulation Mechanical screen and is available below the **Ribbon** on the left in the drawing window. It keeps record of the operations performed during the analysis process in a sequence. All these operations are displayed in the form of a tree view. You can undock the Tree View by dragging it from its position. The content displayed in the Tree View is different for different environments in the Autodesk Simulation Mechanical.

Miniaxis

The Miniaxis is provided at the bottom right corner of the drawing window of the Autodesk Simulation Mechanical. It is used to display the current viewing direction of the model with respect to the three-dimensional work area. The Miniaxis is shown in Figure 2-21.

Figure 2-21 *The Miniaxis*

Scale Ruler

The Scale Ruler is provided at the bottom center of the drawing window of the Autodesk Simulation Mechanical. It is used to measure or display the length of the model relative to its current display size. The Scale Ruler is shown in Figure 2-22.

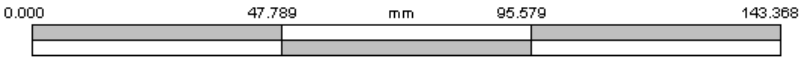

Figure 2-22 *The Scale Ruler*

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

SETTING THE UNIT SYSTEM

As discussed, you can set a unit system for the new FEA model by using the **New** dialog box. To do so, choose the **Override Default Units** button from the **New** dialog box; the **Unit System** dialog box will be displayed, as shown in Figure 2-23. Now, click on the cell next to the **Unit System** field in the **Unit System Settings** node of the dialog box; a down arrow will be displayed on the right of the cell. Next, click on this down arrow; a drop-down list will be displayed with a list of unit systems. You can select the required unit system from this drop-down list. If you select the **Custom** unit system, you will be allowed to customize the units. After setting the required unit system, choose the **OK** button.

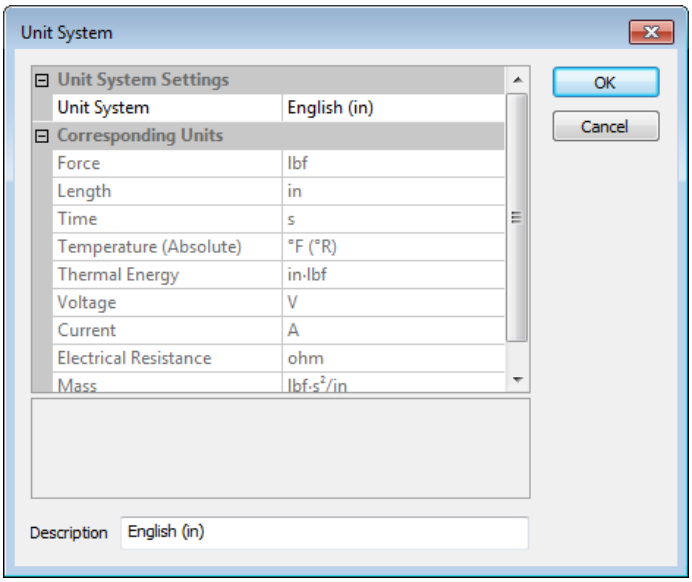

Figure 2-23 *The* ***Unit System*** *dialog box*

In Autodesk Simulation Mechanical, you can also change the current model unit system at any stage of the analysis process by using the Browser / Tree View. To change the model unit system, expand the **Unit System** node in the Tree View by clicking on its +sign, a list of available model and display unit systems is displayed. Select **Model Units** from the expanded **Unit System** node and right-click; a shortcut menu will be displayed. Next, select **Edit > Current Modeling Units** from the shortcut menu; the **Autodesk Simulation Mechanical** message window will be displayed. Choose the **Yes** button from this window; the **Unit System** dialog box will be displayed. Now, change the current model unit system as per your requirement and choose the **OK** button; the current model unit system will be changed.

In Autodesk Simulation Mechanical, all the inputs and results are stored in the model unit system only. However, you can display the results based on display unit system at any stage of the analysis process. By default, various display unit systems are available under the Tree View. To activate a unit system, double-click on the required display unit system available under the **Unit System** node of the Tree View. Alternatively, right-click on the required display unit system and then choose the **Activate** option from the shortcut menu. Note that in addition to the default available display unit systems, you can also add more as required. To do so, right-click on the

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

Unit Systems node of the Tree View and then choose the **New** option to display the **Unit System** dialog box. After specifying the required units by using this dialog box, choose the **OK** button; the new display unit system will be added and will get activated by default.

Note

When you import a CAD model, the Autodesk Simulation Mechanical identifies the unit specified for the length of the model being imported and on that basis it automatically chooses the most suitable unit system. As a result, the unit for the length dimensions remains the same as that of the CAD model, but the units for force, time, and so on may be different.

IMPORTANT ENVIRONMENTS OF AUTODESK SIMULATION MECHANICAL

Before you proceed with the analysis process in Autodesk Simulations Mechanical, it is important to understand the different environments of Autodesk Simulation Mechanical that are used in this book.

FEA Editor Environment

The FEA Editor environment is the primary environment of Autodesk Simulation Mechanical and is used to set up model for analysis. In this environment, you can create or modify the model to be analyzed. You can also change its display style, generate different mesh, define the type of analysis to be carried out, define boundary conditions, apply material, load, and so on.

Result Environment

The Result environment is used to view all the analysis results of the model. In this environment, you can also run animations to view the performance of the model. This environment is also used to check the FEA model before running an analysis by displaying its elements, nodes, properties, loads, and so on.

Note

*Autodesk Simulation Mechanical stores date and time of the last modification and analysis done on the model. If you make any change in the model geometry that can affect its analysis results then the date and time of this latest modification will be updated. In this case, on invoking the Result environment, a message window will be displayed with the message "**The existing results do not match the current model. If the changes have no affect on the results, then it is reasonable to proceed. (Changes include loads, mesh, materials, node numbering, and element numbering.) If the model has changed, the old results should not be used. Do you wish to continue?**". Choose the **No** button from this message window and run the analysis again to get the updated results based on the model modifications.*

Report Environment

The Report environment of the Autodesk Simulation Mechanical is used to create documentation for the analysis results of the FEA model. You can save the report in HTML, PDF, .doc, and so on. You can also insert the images and animations created in the Result environment into the report.

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

SHORTCUT MENUS

Shortcut menu is type of menu that consists some of the commonly used options in Autodesk Simulation Mechanical software. The shortcut menu will be displayed on right-clicking in the graphics area. Note that the shortcut menu contains different options for different environments of the Autodesk Simulation Mechanical. Also, if any object or entity is selected in the graphics area, the shortcut menu displays options related to the object or entity selected.

Figure 2-24 shows a shortcut menu invoked in the FEA Editor environment and Figure 2-25 shows a shortcut menu invoked in the Result environment of the Autodesk Simulation Mechanical.

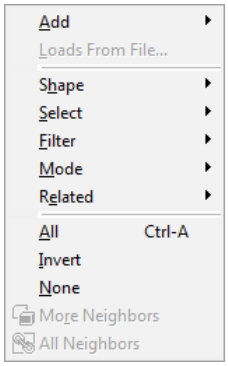

Figure 2-24 *The shortcut menu displayed in the FEA Editor environment*

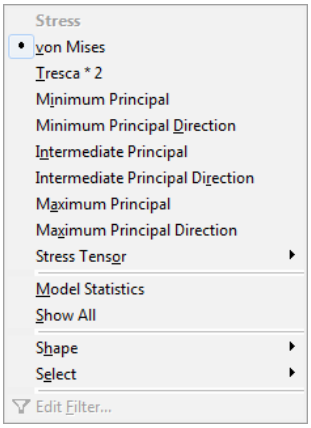

Figure 2-25 *The shortcut menu displayed in the Result environment*

COLOR SCHEMES

Autodesk Simulation Mechanical allows you to use various color schemes to set the background color of the screen and for displaying the entities on the screen. Note that this book uses the white color background. To change the background color scheme to white, invoke the **Options** dialog box by choosing the **Application Options** tool from the **Options** panel of the **Tools** tab in the **Ribbon**, refer to Figure 2-26. Next, choose the **Graphics** tab from the **Options** dialog box, refer to Figure 2-26.

Select the **Background** option from the list box of the **Options** dialog box; the options used for applying the background color of the screen will be displayed in the **Background** area of the dialog box. Next, click on the **Primary background color** swatch box; the color swatches window will be displayed. Select the **White** color swatch from the color swatches window. Next, choose **Apply** to apply the new background color scheme and then choose **OK** from the dialog box. Note that all the files you open henceforth will use this color scheme. You can also apply gradient to the background color of the screen by selecting the **Gradient background color** check box and then defining the gradient background color.

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

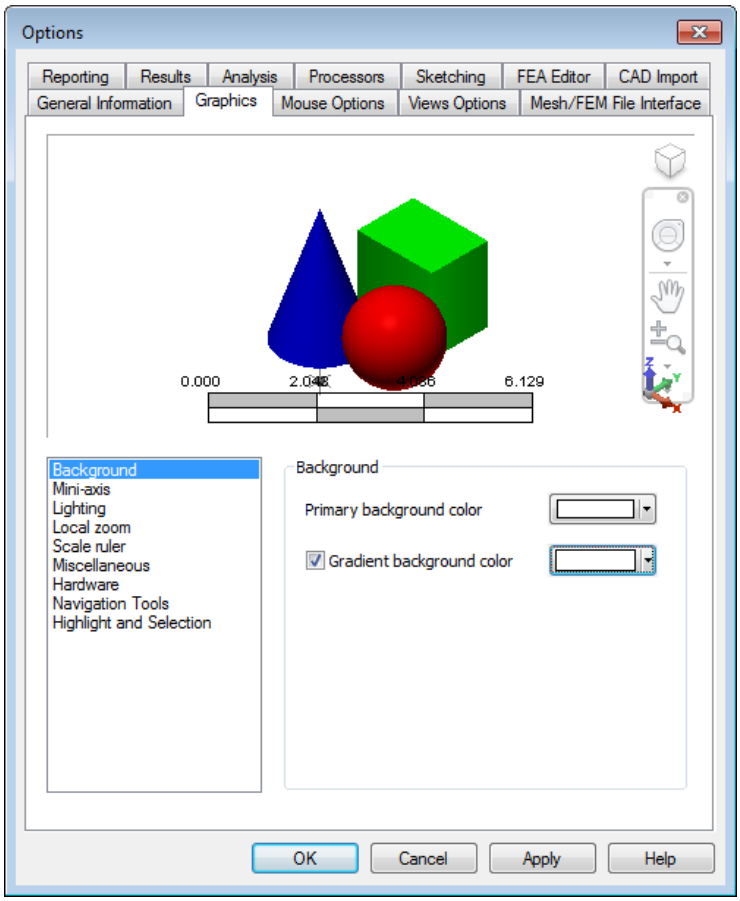

Figure 2-26 *The* ***Options*** *dialog box with* ***Graphics*** *tab chosen*

HOTKEYS

As mentioned earlier, there is no command prompt in Autodesk Simulation Mechanical. However, you can use the keys on the keyboard to invoke some tools. The keys that can be used to invoke the tools are called hotkeys. Some of the hotkeys that can be used in Autodesk Simulation Mechanical environments and their functions are mentioned in the table given next.

Table 2-1 *The hotkeys and their functions*

Hotkey	Function
CTRL+O	Invokes the **Open** dialog box
CTRL+P	Invokes the **Print** dialog box
CTRL+S	Saves the selection
CTRL+V	Pastes the selection
CTRL+X	Cuts the selection
CTRL+Y	Redo the modification

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

CTRL+Z	Undo the modification
CTRL+Num Pad 1	Invokes the Front view
CTRL+Num Pad 2	Invokes the Bottom view
CTRL+Num Pad 3	Invokes the Isometric view
CTRL+Num Pad 4	Invokes the Left view
CTRL+Num Pad 5	Invokes the **Enclose (Fit All)** tool
CTRL+Num Pad 6	Invokes the Right view
CTRL+Num Pad 7	Invokes the Back view
CTRL+Num Pad 8	Invokes the Top view
CTRL+Num Pad 9	Invokes the Axonometric view
Right or left arrow key	In the display area, these keys rotate the model about the vertical axis of the screen
Up or down arrow key	In the display area, these keys rotate the model about the horizontal axis of the screen
S+Arrow keys	In the display area, these keys rotate the model about the axis perpendicular to the screen
F1	Invokes the help file

Self-Evaluation Test

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

1. The _____ tool is used to select construction circles, arcs, lines, splines, rectangles, and vertices that can be created by using the tools available in the **Draw** panel of the **Draw** tab in the **Ribbon**.

2. In Autodesk Simulation Mechanical, the messages and prompts are displayed at the _____.

3. In Autodesk Simulation Mechanical, you can specify the type of analysis for a model by using the **New** dialog box. (T/F)

4. In Autodesk Simulation Mechanical, you can specify a unit system for a model by using the **New** dialog box. (T/F)

5. In Autodesk Simulation Mechanical, you cannot open an existing Autodesk Simulation FEA or CAD model to run the analysis by using the **New** dialog box. (T/F)

6. The **Quick Access Toolbar** is common to all environments of Autodesk Simulation Mechanical. (T/F)

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

7. You can add or remove the tools in the **Quick Access Toolbar**. (T/F)

8. You cannot customize the tabs and panels of the **Ribbon**. (T/F)

Review Questions

Answer the following questions:

1. All the selection tools are available in the _____ tab.

2. The tools provided in the _____ tab are used to define the boundary conditions by specifying the constraints and loads for the model.

3. The _____ tab has the tools that are required for generating the mesh for a model.

4. The _____ environment is the primary environment of Autodesk Simulation Mechanical and is used to set up a model for analysis.

5. In Autodesk Simulation Mechanical, you can change the default or current Model unit system at any stage of the analysis process by using the Tree View. (T/F)

6. In Autodesk Simulation Mechanical, the options available in the shortcut menus depend upon the type of environment invoked. (T/F)

7. The **Draw** tab of the **Ribbon** provides the tools that are used to create and modify the geometry to be analyzed. (T/F)

8. The **Setup** tab of the **Ribbon** will be available only in the FEA Editor environment. (T/F)

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

Answers to Self-Evaluation Test

1. Select Construction Objects, **2. Status Bar**, **3.** T, **4.** T, **5.** F, **6.** T, **7.** T, **8.** F

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